User's Guide 10/2004 Edition

sinumerik

SINUMERIK 840D/840Di/810D
Measuring Cycles

SIEMENS

SIEMENS

SINUMERIK 840D/840Di/810D

Measuring Cycles

User's Guide

Valid for

Control
SINUMERIK 840D powerline
SINUMERIK 840DE powerline (export version)
SINUMERIK 840Di
SINUMERIK 840DiE (export version)
SINUMERIK 810D powerline
SINUMERIK 810DE powerline (export version)

Drive
SIMODRIVE 611 digital

Software	*Version*
Measuring Cycles	6

10.04 Edition

SINUMERIK® Documentation

Printing history

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

The status of each edition is indicated by the code in the "Remarks" columns.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint 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
01.04	6FC5298-6AA70-0BP3	C
03.04	6FC5298-7AA70-0BP0	C
10.04	6FC5298-7AA70-0BP1	C

Trademarks

SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are Siemens trademarks. Other product names used in this documentation may be trademarks which, if used by third parties, could infringe the rights of their owners.

More information is available on the Internet at:
http://www.siemens.com/motioncontrol

This document has been created using with WinWord V 2000 and Designer V 7.0.

© Siemens AG, 1995-2004. All rights reserved

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 the contents of this manual for agreement with 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 all recommendations and suggestions.

Subject to change without prior notice

Order No. 6FC5298-7AA70-0BP1
Printed in Germany
Siemens Aktiengesellschaft

Preface

Structure of the Documentation

The SINUMERIK documentation is organized in 3 parts:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Audience

This Manual is intended for machine-tool users. This publication provides detailed information that the user requires for operating the SINUMERIK 810D and 840D controls.

Standard scope

This Operator's Guide describes only the functionality of the standard scope. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

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:
Measuring cycles, version 6.3.
Software versions stated in the User's Guide refer to the 840D and their 810D equivalent, e.g. SW 5 (840D) corresponds to SW 3 (810D).

SINUMERIK 840D powerline

Since 09/2001

- SINUMERIK 840D powerline and
- SINUMERIK 840DE powerline

have been 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

Since 12/2001

- SINUMERIK 810D powerline and
- SINUMERIK 810DE powerline

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

© Siemens AG, 2004. All rights reserved

Hotline

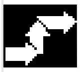
If you have any questions, please get in touch with our hotline:
A&D Technical Support Phone: +49 (0) 180 5050-222
Fax: +49 (0) 180 5050-223
http://www.siemens.com/automation/support-request

Please send any queries about the documentation (suggestions or corrections) to the following fax number or email address:
Fax: +49 (0) 0131 98-2176
E-mail: motioncontrol.docu@erlf.siemens.de
Fax form: See the reply form at the end of the document.

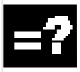
Internet address

http://www.siemens.com/motioncontrol

Explanation of symbols



Sequence



Explanation

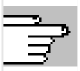
Function

Parameters

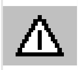
Programming example

Programming

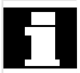
Other Information



Cross-references to other documentation or sections



Notes and warnings



Additional notes or background information

© Siemens AG, 2004. All rights reserved

Warnings

The following warnings with varying degrees of severity appear in this document.

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.

© Siemens AG, 2004. All rights reserved

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

Supplementary devices
The applications of SIEMENS controls can be expanded for specific purposes through the addition of special add-on devices, equipment and expansions supplied by SIEMENS.

Personnel
Only **properly trained, authorized, reliable personnel** must be allowed to use this equipment. No-one without the necessary training must be allowed to operate the control, even temporarily.

The **areas of responsibility** assigned to personnel involved in setting up, operating and maintaining the equipment must be clearly **specified** and their compliance **verified**.

Procedure
Before the control is started up, personnel responsible for its operation must have read and understood the Operator's Guides. The company used this equipment is also obliged to carry out **continuous monitoring** of the overall technical condition of the equipment (with a view to identifying externally visible defects and damage as well as changes in the operating behavior of the control).

© Siemens AG, 2004. All rights reserved

Servicing
Repairs to equipment may only be carried out by **personnel specially trained and qualified** for the application in question in accordance with the provisions specified in the maintenance and servicing guides. All relevant safety regulations must be followed.

Note
The following are deemed as **improper usage** and **exclude the manufacturer from all liability**:

Any usage or application incompatible with or beyond the scope of the items specified above.

Cases where the control is operated **in a technically imperfect condition**, without due provision for safety considerations and/or hazards, or in contravention with any of the instructions in the Instruction Manual.

Cases where faults which could affect safety are not remedied **before** starting up the control.

Any **modification, bypassing** or **decommissioning** of equipment on the control whose intended purpose is to ensure proper functioning, unrestricted use of equipment and/or active and passive safety.

Unforeseen danger can arise with reference to:
- life and limb of personnel,
- the control, machine or other assets of the owner and the user.

© Siemens AG, 2004. All rights reserved

Notes

© Siemens AG, 2004. All rights reserved

Contents

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

Introduction

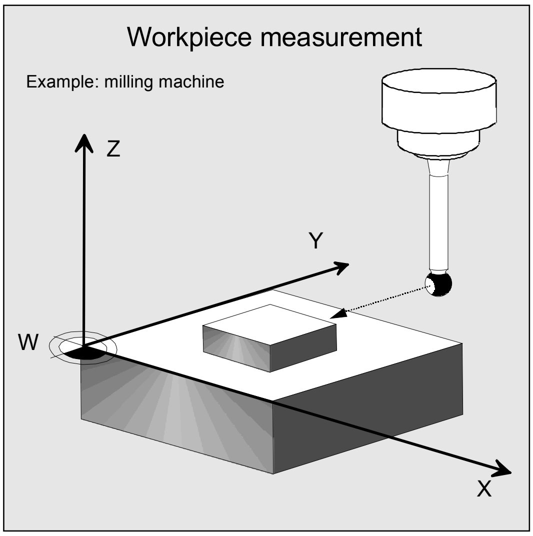
© Siemens AG, 2004. All rights reserved

1.1 Fundamentals

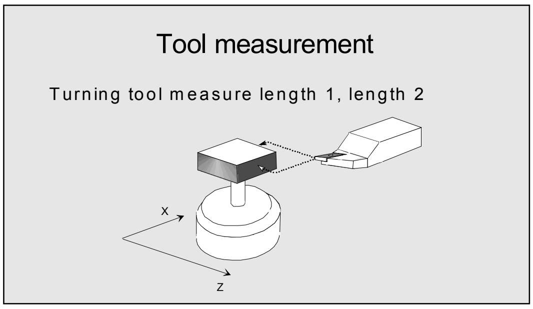
Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be adapted to specific problems via parameter settings.

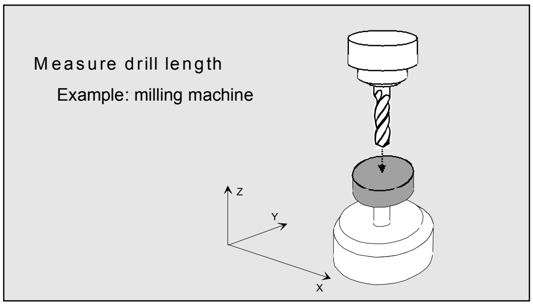
In measurement applications, a general distinction is made between **tool measurement** and **workpiece measurement**.

Workpiece measurement
In workpiece measurement, a measuring probe is moved up to the clamped workpiece in the same way as a tool and the measured values are acquired. The flexibility of measuring cycles makes it possible to perform nearly all measurements required on a milling or turning machine.
An automatic tool offset or ZO correction can be applied to the workpiece measurement result.



Tool measurements
In tool measurement, the selected tool is moved up to the probe and the measured values are acquired. The probe is either in a fixed in position or is swung into the working area with a mechanism. The tool geometry measured is entered in the appropriate tool offset data set.





© Siemens AG, 2004. All rights reserved

1.2 General prerequisites

Certain preconditions must be met before measuring cycles can be used.
These are detailed in Part 2 Description of Functions (Chapter 8 ff.).

The following checklist is useful for checking which preconditions have been met:

Machine

- All machine axes are designed in accordance with DIN 66217.
- Machine data have been adapted.

Existence of cycles, data blocks

The following have been loaded into the control and activated:

- The data blocks into the "Definitions" directory:
  GUD5.DEF
  GUD5.DEF and
  GUD7.DEF (for measuring in JOG)
- The measuring cycles into the standard cycle directory.

Starting position

- The reference points have been approached.
- The starting position can be reached by linear interpolation without collision.

Display functions of the measuring cycles

An MMC/PCU is required for showing the measuring result displays and measuring cycle support.

© Siemens AG, 2004. All rights reserved

Please observe the following when programming:

- Tool radius compensation is deselected before it is called (G40).
- All parameters for the cycle call have been defined beforehand.
- The cycle is called no later than at the 5th program level.
- The system of units allows measuring in the programmed unit system that deviates from the basic system with switchable technology data. This means in a metric basic system with active G70, G700 and in an inch basic system with active G71, G710.

1.3 Behavior on block search, dry run, program testing, simulation

Function

Up to meas. cycles SW 6.2
The measuring cycles are skipped during execution if one of the following execution types is active:

- "Block search" ($P_SEARCH=1)
- "Dry run" ($P_DRYRUN=1)
- "Simulation" ($P_SIM=1)
- "Program test" ($P_ISTEST=1) (from SW 6.2)

Function

From meas. cycles SW 6.3)
The measuring cycles are skipped during execution if one of the following execution modes is active:

- "Dry run" ($P_DRYRUN=1)
- "Program test" ($P_ISTEST=1)
- "Block search" ($P_SEARCH=1), only if $A_PROTO=0.

Simulation

- **on HMI Advanced** ($P_SIM=1)
  The measuring cycle programs are executed on selection of "Simulation" on the HMI.
- **on Jobshop products** ($P_SEARCH=1 and $A_PROTO=1)

© Siemens AG, 2004. All rights reserved

The measurements are simulated. No selected tool or zero offset is applied. Active functions such as "measuring result display", "travel with collision monitoring", "measuring cycle logging" are not executed.

Suppression of simulation execution
Execution of the measuring cycles in simulation can be suppressed in simulation by resetting the variable **_MC_SIMSIM=0** in data block GUD6. The measuring cycles are then skipped.

Specifying differences for simulation
The variable **_MC_SIMDIFF** of data type REAL permits specification of simulated measurement deviations at measuring points. The value is a dimension in the basic system of the control.
Excessive values of _MC_SIMDIFF with corresponding value assignment of the defining parameters cause cycle alarms to be output.

Note:
It is not ensured that the correction value contains the correct sign of _MC_SIMDIFF. This depends on the measurement or calibration task and the direction of measurement. The sign is influenced in such a way that the overall result makes sense (e.g. "Measure hole"). Changing the sign always causes the sign of the result to change.

Demo programs for simulation on HMI Advanced
The measuring cycle software supplied includes programs for demonstrating how the measuring cycles work when simulating on HMI Advanced. The programs are divided into turning technology and milling technology. When a program is loaded, the "workpieces" are created whose directory contains all data and settings required for simulation.

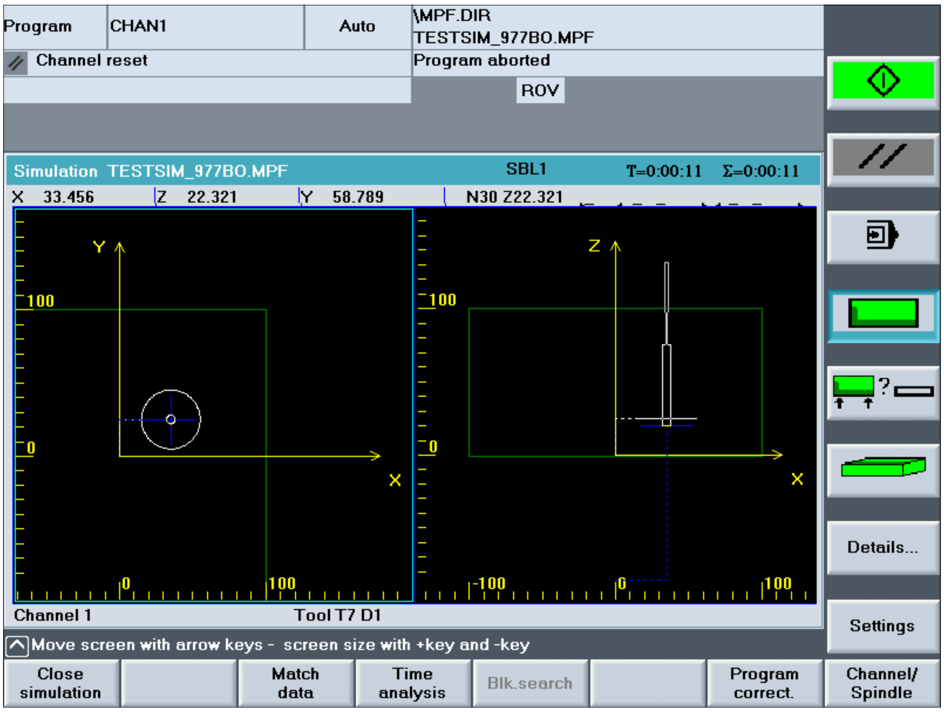
© Siemens AG, 2004. All rights reserved

Examples of demo programs for simulation on HMI Advanced

Example 1:

Measuring a hole

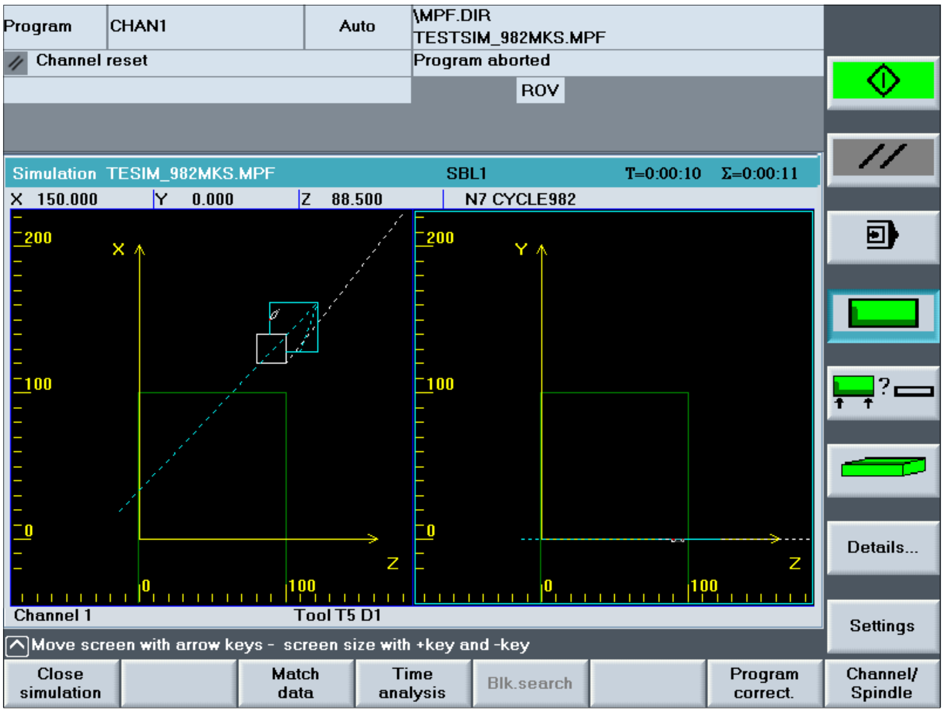
(TESIM_977BO with CYCLE977, milling technology)



Example 2:

Calibrating a tool probe

(TESIM_982MKS with CYCLE982, turning technology)



© Siemens AG, 2004. All rights reserved

1.4 Reference points on the machine and workpiece

Function

Depending on the measuring task, measured values may be required in the machine coordinate system or in the workpiece coordinate system.
e.g.: It may be easier to ascertain the tool length in the machine coordinate system.
Workpiece dimensions are measured in the workpiece coordinate system.

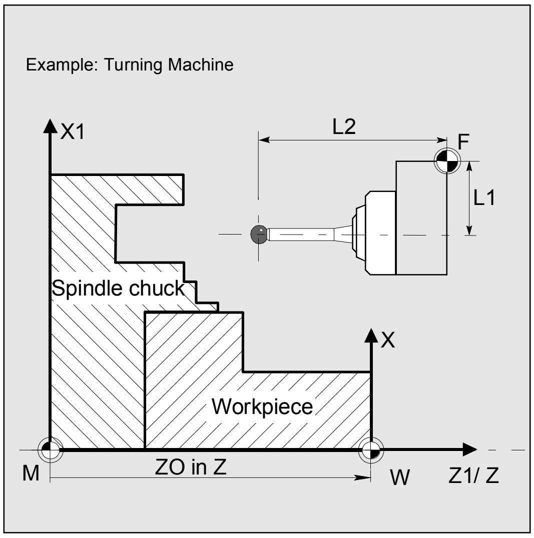
The position of tool reference point F in the machine coordinate system is displayed with machine zero M as the **machine actual value**.

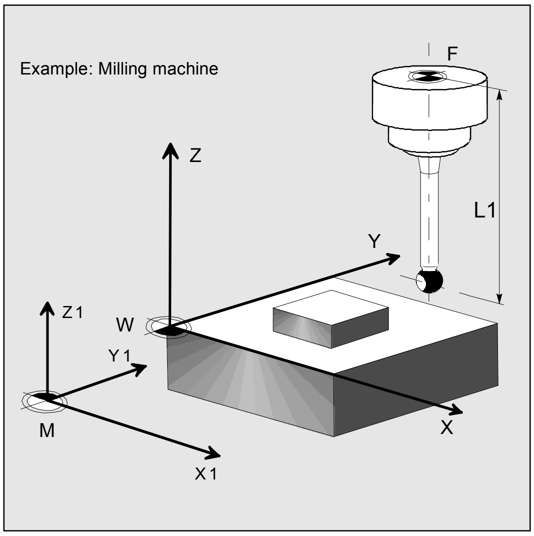
The position of the tool tip (active tool) in the workpiece coordinate system is displayed with workpiece zero W as the **workpiece actual value**.
If a workpiece probe is active, the position usually refers to the center point of the probe ball.

The **zero offset** (ZO) characterizes the position of the workpiece on the machine. The ZO is
the position of workpiece zero W in the machine coordinate system.
In addition to the pure offset, a ZO might also include rotation, mirroring, and a dimension factor. Together these are termed a **frame**.
SINUMERIK controls use numerous frames: various basic frames, system frames, settable frames (e.g. G54), programmable frames: They interact in a frame chain to produce the overall frame and the workpiece coordinate system.
Measuring cycles do not support any frames with an active scale factor. Rotation or mirroring is not supported in some cycles and measuring variants.

The machine and workpiece coordinate system can be set and programmed separately in the "inch" or "metric" measuring system.

Reference points





M = Machine zero
W = Workpiece zero
F = Tool reference point

© Siemens AG, 2004. All rights reserved

If kinematic transformation is active, the control makes a distinction between the **basic coordinate system** and the **machine coordinate system**.
If kinematic transformation is deactivated, this distinction is made.
All descriptions provided below assume that kinematic transformation is deactivated and therefore refer to the machine coordinate system.

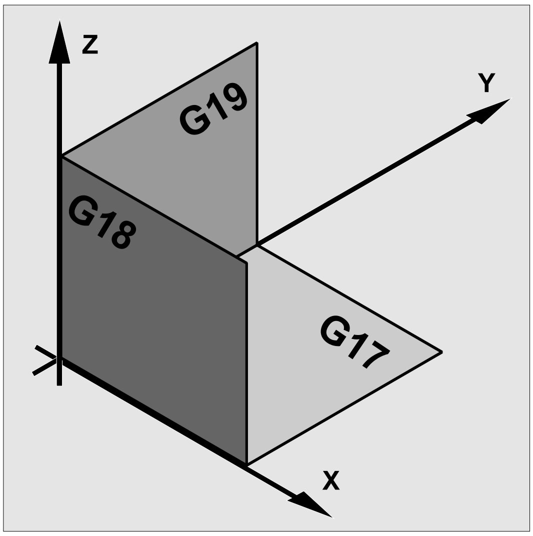
© Siemens AG, 2004. All rights reserved

1.5 Definition of the planes, tool types

Tool radius compensation planes:
G17, G18, or G19 can be selected.
Depending on the tool type, the tool lengths are assigned to the axes as follows:

- Milling cutter, workpiece probe for milling: 1xy or workpiece probe for milling: 710
- Drill: 2xy
- Turning tool, workpiece probe for turning: 5xy

Milling



G17 plane

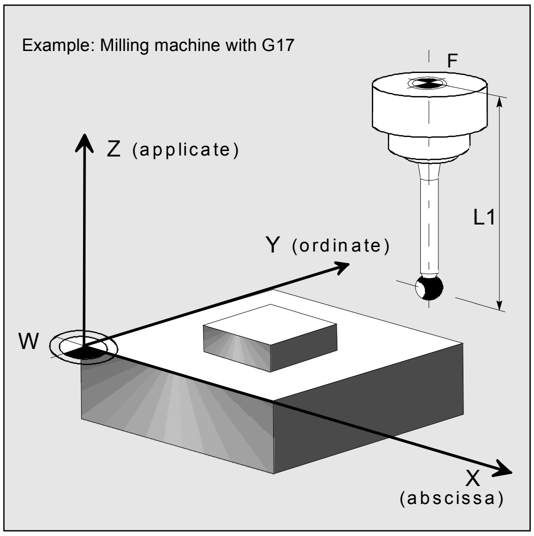
Tool type	1xy / 2xy / 710
Length 1	active in Z (applicate)
Length 2	active in Y (ordinate)
Length 3	active in X (abscissa)

G18 plane

Tool type	1xy / 2xy / 710
Length 1	active in Y (applicate)
Length 2	active in X (ordinate)
Length 3	active in Z (abscissa)

G19 plane

Tool type	1xy / 2xy / 710
Length 1	active in X (applicate)
Length 2	active in Z (ordinate)
Length 3	active in Y (abscissa)



Lengths 2 and 3 are used in special cases, for example, if an angle head is attached.

© Siemens AG, 2004. All rights reserved

Turning

Turning machines generally only use axes Z and X and therefore:

G18 plane

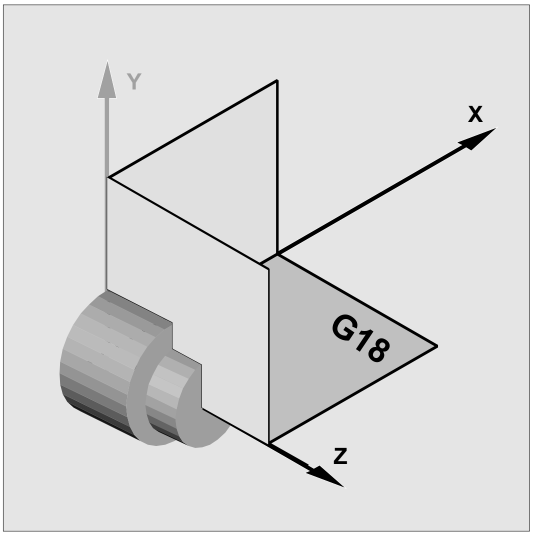
Tool type	5xy (turning tool, workpiece probe)
Length 1	active in X (ordinate)
Length 2	active in Z (abscissa)

G17 and G19 are used for milling on a turning machine. If there is no machine axis Y, milling can be implemented with kinematic transformations. TRANSMIT, TRACYL.

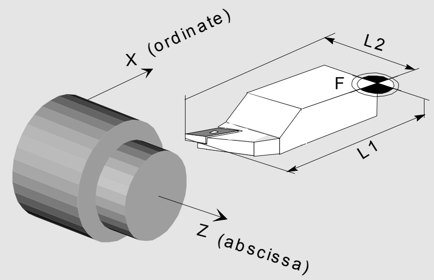
In principle, measuring cycles support kinematic transformations. This is stated more clearly in the individual cycles, measuring variants.

Notes:
If a drill or milling cutter is measured on a turning machine, in most cases, setting data SD 42950: TOOL_LENGTH_TYPE=2 is set. These tools are then length-compensated like a turning tool.

SINUMERIK controls have other machine and setting data that can influence calculation of a tool.



G18 - Standard setting for turning machines



References: /FB1/, Description of Functions – Basic Machine
/FB2/, Description of Functions – Extended Functions
/FB3/, Description of Functions – Special Functions

© Siemens AG, 2004. All rights reserved

1.6 Probes that can be used

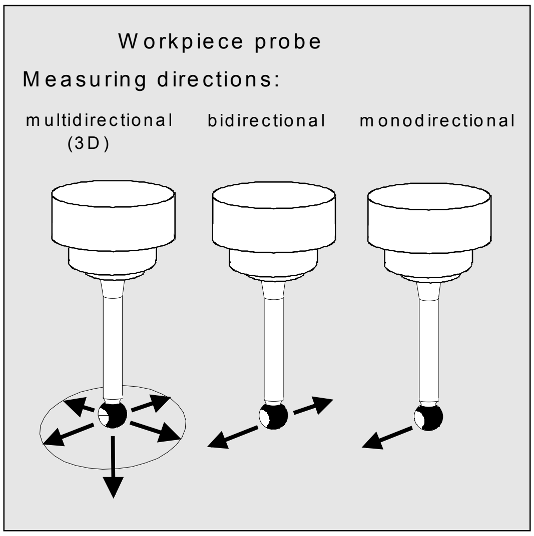
Function

To measure tool and workpiece dimensions, a touch-trigger probe is required that provides a signal change (edge) when deflected.
The probe must operate virtually bounce-free.

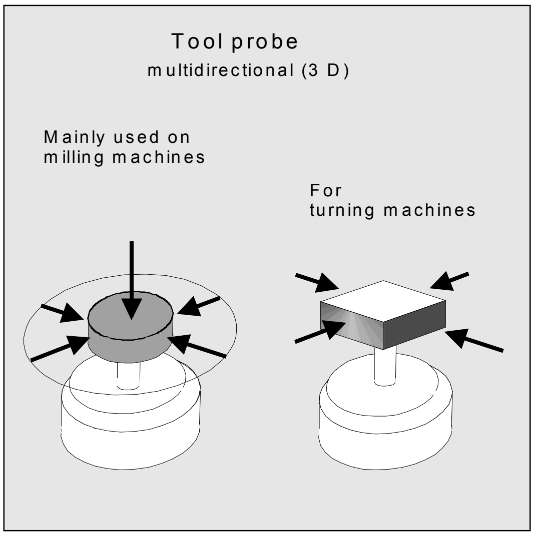
Different types of probe are offered by different manufacturers. Probes are distinguished according to the number of measuring directions.

- Monodirectional (mono probe)
- Bidirectional
- Multidirectional (multi probe)

The probe type is defined by a parameter (_PRNUM) in measuring cycles (see Section 2.10).



Monodirectional probe
This type of probe can only be used if the spindle can be positioned with NC function SPOS and the switching signal of the probe can be transmitted through 360° to the receiving station (at the machine column).



The probe must be mechanically aligned in the spindle to permit measurements 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 takes longer with a mono probe since the spindle must be positioned in the cycle several times by means of SPOS.

Probe assignment:

Probe type	**Turning machines**		**Milling and machining centers**
	Tool measurements	**Workpiece measurement**	**Workpiece measurement**
Multidirectional	X	X	X
Bidirectional	–	X	X
Monodirectional	–	–	X

In workpiece measurement, a bidirectional probe is treated like a mono probe.

© Siemens AG, 2004. All rights reserved

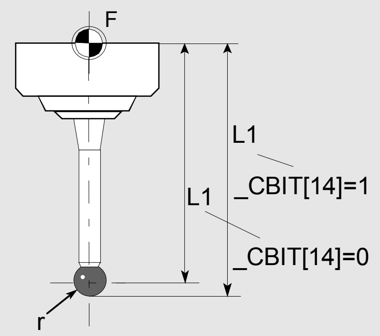
If a workpiece probe is used, both the direction of deflection and transmission of switching signal to the machine column (radio, infrared light or cable) must be taken into account.
In some versions, transmission is only possible in particular spindle positions or in particular ranges.
This may further limit the use of the probe.
In any case, please follow the advice of the probe or machine manufacturer.

© Siemens AG, 2004. All rights reserved

1.7 Probe, gauging block, calibration tool

1.7.1 Measuring workpieces on milling machines, machining centers

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



Entry in tool memory

Tool type (DP1):	710 or 1xy
Length 1 – Geometry (DP3):	L1
Radius (DP6):	r
Length 1 – Tool base dimension (DP21):	Only if necessary

The wear and other tool parameters must be assigned the value 0.
In _CBIT[14] you can set whether length L1 refers to the ball center point or the ball circumference.

_CBIT[14] see Subsection 9.2.4 (central bits)

Calibration
A probe must be calibrated before it can be used.
Calibration involves determining the triggering points (switching points), skew (positional deviation), and active ball radius of the workpiece probe and then entering them in special data fields _WP[ ] in data block GUD6.DEF.
The default setting has data fields for 3 probes. Up to 99 are possible.

Calibration can be performed on holes of a known size or workpiece surfaces with a sufficient form precision and low surface roughness.
Use of special gauging blocks is not supported on milling and machine centers.
Use the same measuring velocity for calibrating and measuring.

A special cycle is available for calibration.

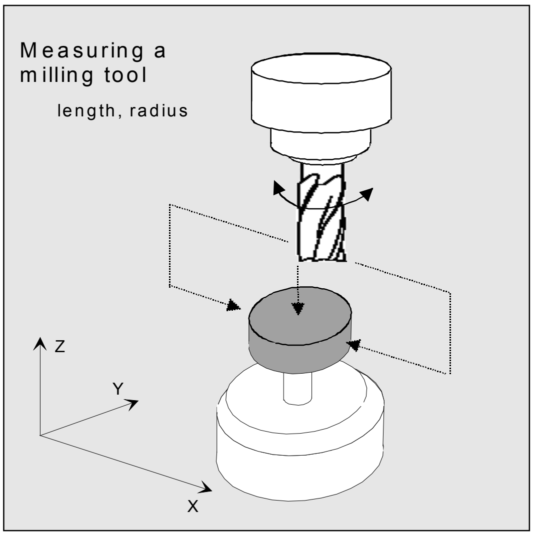
© Siemens AG, 2004. All rights reserved

1.7.2 Measuring tools on milling machines, machining centers

Tool probe
The tool probes have dedicated data fields _TP[ ] and _TPW[ ] in data block GUD6.DEF.
The triggering points (switching points), upper disk diameter and edge length are entered here.

Approximate values must be entered here before calibration – if cycles are used in automatic mode. The cycle will then recognize the position of the probe.

In the default setting there are data fields for 3 probes. Up to 99 are possible.



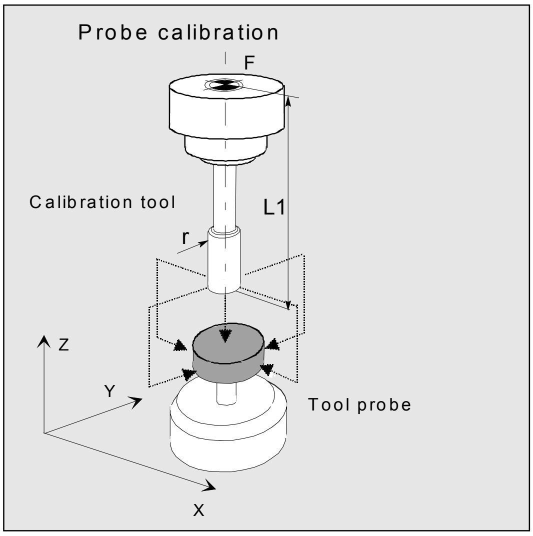
Calibration, calibrating tool
A probe must be calibrated before it can be used. Calibration involves precisely determining the triggering points (switching points) of the tool probe and entering them in special data fields.
Calibration is performed with a calibration tool. The precise dimensions of the tool are known.
Use the same measuring velocity for calibrating and measuring.

A special cycle is available for calibration.

Entry of calibration tool in tool memory:

Tool type (DP1):	1xy
Length 1 – Geometry (DP3):	L1
Radius (DP6):	r
Length 1 – Tool base dimension (DP21):	Only if necessary

The wear and other tool parameters must be assigned the value 0.



© Siemens AG, 2004. All rights reserved

1.7.3 Measuring workpieces at the lathe

Workpiece probe
On turning machines, the workpiece probes are treated as tool type 5xy with permissible cutting edge positions (SL) 5 to 8 and must be entered in the tool memory accordingly.

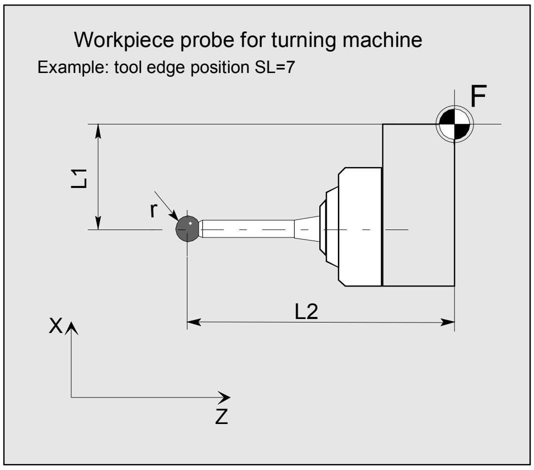
Lengths specified for turning tools always refer to the tool tip, except in the case of workpiece probes on turning machines where they refer to the probe center.

Probes are classified according to their position:

Workpiece probe SL 7
Entry in tool memory

Tool type (DP1):	5xy
Tool edge position (DP2):	7
Length 1 – Geometry:	L1
Length 2 – Geometry:	L2
Radius (DP6):	r
Length 1 – Tool base dimension (DP21):	Only if necessary
Length 2 – T. base dim. (DP22):	Only if necessary

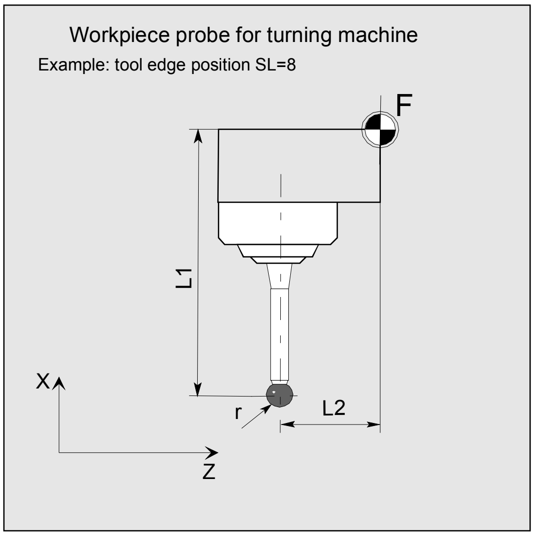
The wear and other tool parameters must be assigned the value 0.



Workpiece probe SL 8
Entry in tool memory

Tool type (DP1):	5xy
Tool edge position (DP2):	8
Length 1 – Geometry:	L1
Length 2 – Geometry:	L2
Radius (DP6):	r
Length 1 – Tool base dimension (DP21):	Only if necessary
Length 2 – T. base dim. (DP22):	Only if necessary

The wear and other tool parameters must be assigned the value 0.



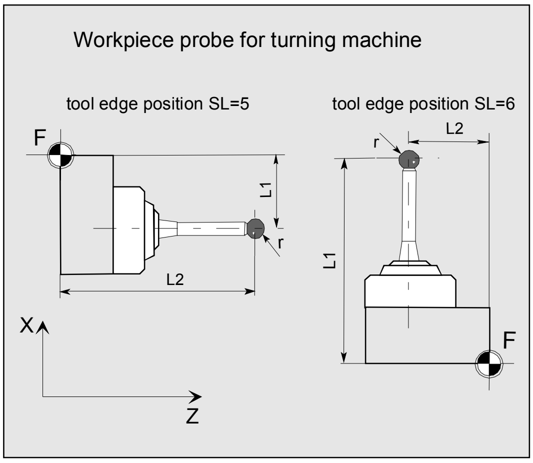
© Siemens AG, 2004. All rights reserved

Workpiece probe SL 5 or SL 6

Entry in tool memory

Tool type (DP1):	5xy
Tool edge position (DP2):	5 or 6
Length 1 – Geometry:	L1
Length 2 – Geometry:	L2
Radius (DP6):	r
Length 1 – Tool base dimension (DP21):	Only if necessary
Length 2 – T. base dim. (DP22):	Only if necessary

The wear and other tool parameters must be assigned the value 0.



Calibration, gauging block

A probe must be calibrated before it can be used. During calibration the triggering points (switching points), skew (positional deviation), and precise ball radius of the workpiece probe are determined and then entered in special data fields _WP[ ] in data block GUD6.DEF.

In the default setting there are data fields for 3 probes. Up to 99 are possible.

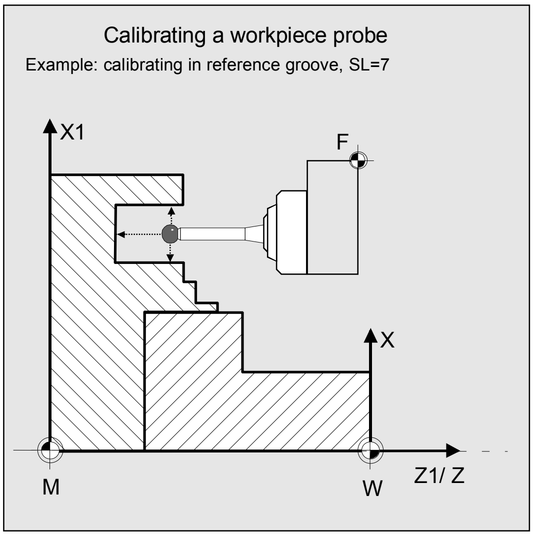
Calibration of the workpiece probe on turning machines is usually performed with gauging blocks (reference grooves). The precise dimensions of the reference groove are known and entered in the relevant data fields _KB[ ] in data block GUD6.DEF.

In the default setting there are data fields for 3 gauging blocks. The gauging block is selected in the program with variable _CALNUM.

It is also possible to calibrate on a known surface.

Use the same measuring velocity for calibrating and measuring.

A cycle with different measuring versions is provided for calibration.



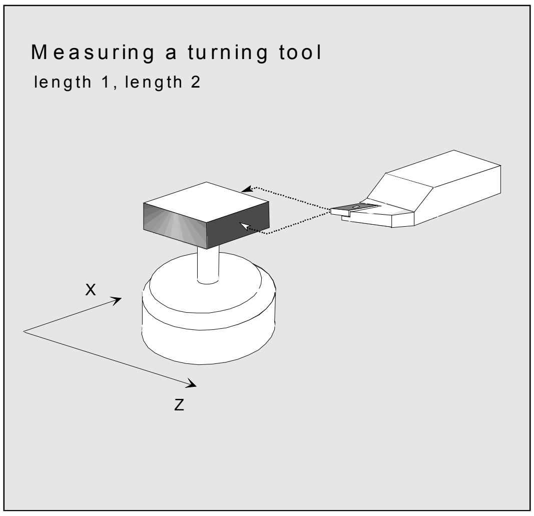
© Siemens AG, 2004. All rights reserved

1.7.4 Measuring tools at lathes

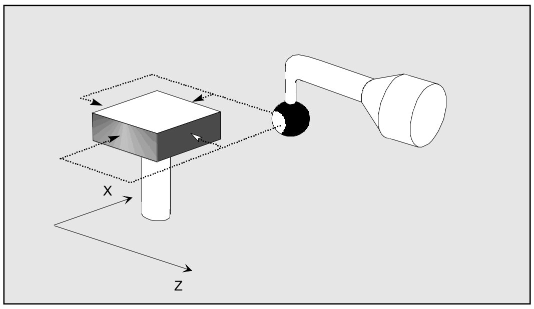
Tool probe
The tool probes have dedicated data fields _TP[ ] and _TPW[ ] in data block GUD6.DEF.
The triggering points (switching points) are entered here. Approximate values must be entered here before calibration – if cycles are used in automatic mode. The cycle will then recognize the position of the probe.

The default setting has data fields for 3 probes. Up to 99 are possible.

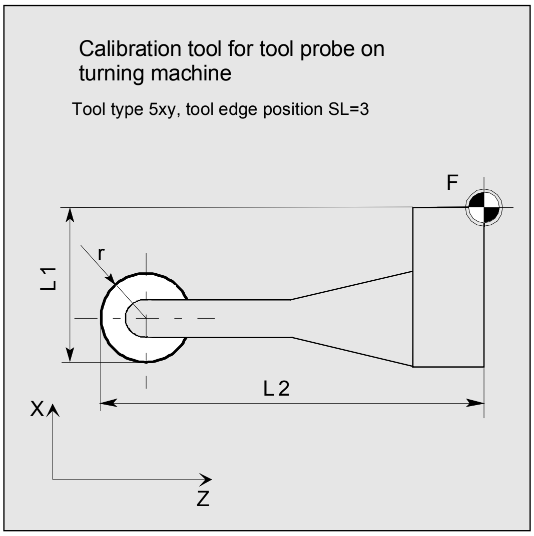
In addition to turning tools, drills and mills can also be measured.



Calibration, calibrating tool
A probe must be calibrated before it can be used.
Calibration involves precisely determining the triggering points (switching points) of the tool probe and entering them in special data fields.
Calibration is performed with a calibration tool. The precise dimensions of the tool are known.
Use the same measuring velocity for calibrating and measuring.
A special measuring variant in a cycle is available for calibration.



On turning machines, the calibration tool is treated like a turning tool with cutting edge position 3.
The lengths refer to the ball circumference, not to the ball center.



Entry in tool memory

Tool type (DP1):	5xy
Tool edge position (DP2):	3
Length 1 – Geometry:	L1
Length 2 – Geometry:	L2
Radius (DP6):	r
Length 1 – Tool base dimension (DP21):	Only if necessary
Length 2 – T. base dim. (DP22):	Only if necessary

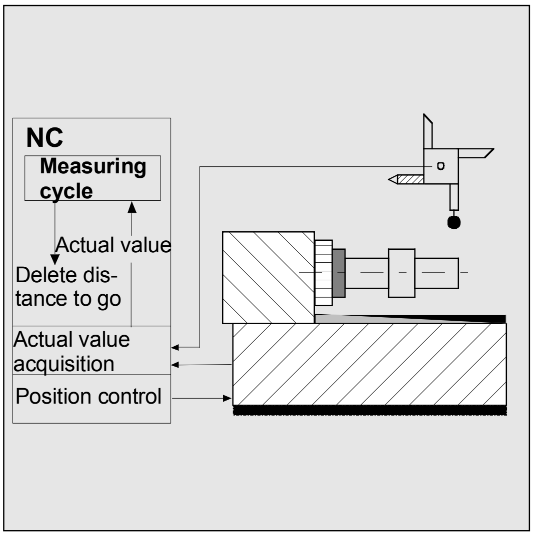
The wear and other tool parameters must be assigned the value 0.

© Siemens AG, 2004. All rights reserved

1.8 Measuring principle

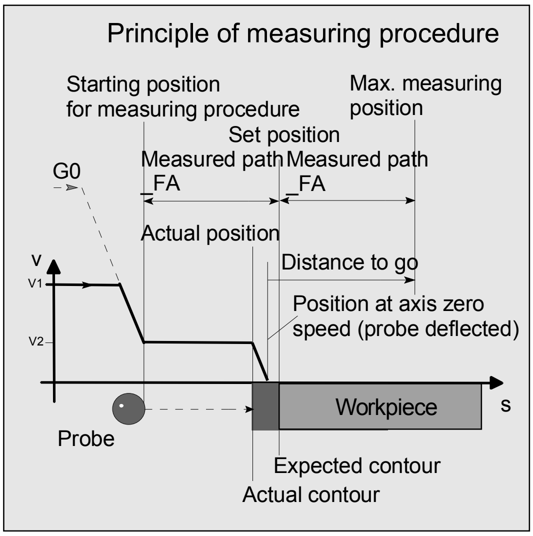
Function

"On-the-fly" measurement
The principle of "on-the-fly" measurement is implemented in the SINUMERIK control. The probe signal is processed directly on the NC so that the delay when acquiring measured values is minimal. This permits a higher measuring speed for the prescribed measuring precision and time needed for measuring is reduced.



Connecting probes
Two inputs for connecting touch trigger probes are provided on the I/O device interface of the SINUMERIK control systems.

Measuring procedure
The procedure is described with **workpiece measurement**. The procedure is the same for tool measurement. In this case, however, the tool is moved and the probe is fixed.
Depending on its design, the actual movements of a machine may be different anyway. Workpiece measurement is described as follows:
The workpiece is stationary and the probe moves.



The **starting position** for the measuring procedure is a position **_FA** in front of the specified **set position** (expected contour).
The starting position is calculated in the cycle based on parameter entries and probe data. The starting position is approached either with rapid traverse G0 or with positioning velocity G1; then from the starting position with **measuring velocity**.
The switching signal is expected along path 2 · _FA as from the starting position. Otherwise, an alarm will be triggered or the measurement repeated.
The resulting **maximum measuring position** is in the measuring block of the cycle.
At the instant the switching signal is output by the probe, the current **actual position** is stored internally "on-the-fly" as the actual value, the measuring axis is stopped and then the "**Delete distance-to-go**" function is executed.

© Siemens AG, 2004. All rights reserved

The distance-to-go is the path not yet covered in the measuring block. After deletion, the next block in the cycle can be processed. The measuring axis travels back to the starting position. Any measurement repetitions selected are restarted from this point.

Measurement path _FA
Measurement path _FA defines the distance between the starting position and the expected switching position (setpoint) of the probe.
(see Chapter 2).

Measuring velocity
The measuring velocity is dependent on the measurement path _FA and its default setting is 150 mm/min if _FA=1; if FA>1: 300 mm/min. Cycles parameter _VMS is then =0.

Other measuring velocities can be set by the user to a value of >0 in _VMS (see Chapter 2).

The **maximum permissible measuring velocity** is derived from:

- the deceleration behavior of the axis
- the permissible deflection of the probe
- the signal processing delay

Deceleration distance, deflection of probe
Caution
Safe deceleration of the measuring axis to standstill within the permissible deflection path of the probe must always be ensured.
Otherwise damage will occur!

A delay "t" typical of the control is taken into account in signal processing (IPO cycle) for the time between detection of the switching signal and output of the deceleration command to the measuring axis MD 10050: SYSCLOCK_CYCLE_TIME and MD 10070: IPO_SYSCLOCK_TIME_RATIO). This results in the deceleration path component.

The following error of the measuring axis is reduced. The following error is velocity dependent and at the same time dependent on the control factor of the measuring axis (servo gain of the associated machine axis: servo gain factor).

© Siemens AG, 2004. All rights reserved

The deceleration rate of the axis must also be taken into account.
Together they produce an axis-specific velocity-dependent deceleration distance.

The servo gain factor is MD 32200: POSCTRL_GAIN.

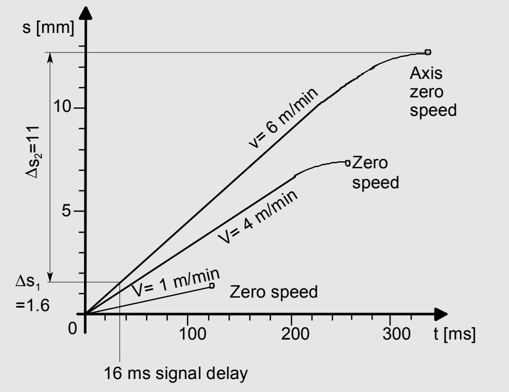
Axis acceleration / deceleration rate a is stored in MD 32300: MAX_AX_ACCEL. It may have a lesser effect due to other influences.
Always used the lowest values of the axes involved in the measurement.

Calculation of the deceleration path
The deceleration path to be considered is calculated as follows:

$$s_b = \underbrace{1000 \cdot v \cdot t}_{\Delta s_1} + \underbrace{\frac{1000 \cdot v^2}{2a} + \Delta s}_{\Delta s_2}$$

s_b	Deceleration path	in mm
v	Measuring velocity	in m/s
t	Delay signal	in s
a	Deceleration	in m/s^2

Δs following error	in mm
Δs = v / Kv	v here in m/min
Kv servo gain factor	in (m/min)/mm



Example of calculation:

- v = 6 m/min = 0.1 m/ s measuring velocity
- a = 1 m/s^2 deceleration rate
- t = 16 ms signal delay
- Kv = 1 in (m/min)/mm

The deflection of the probe = deceleration distance to zero speed of axis is: **s_b = 12.6 mm**.
The deceleration distance components are:

Δs = 6/ 1 = 6 mm	Following error
Δs2 = 1000 · 0.01 / 2 + 6 = 11 mm	Axis spec. percentage
Δs1 = 1000 · 0.1 · 0.016 = 1.6 mm	Percentage due to signal delay

© Siemens AG, 2004. All rights reserved

Measuring accuracy

A delay occurs between detection of the switching signal from the probe and transfer of the measured value to the control. This is caused by signal transmission from the probe and the hardware of the control. In this time a path is traversed that falsifies the measured value. This influence can be minimized by reducing the measuring speed.
The rotation when measuring a mill on a rotating spindle has an additional influence. This can be compensated for by compensation tables.
(see Subsection 5.2.1)

The measurement accuracy that can be obtained is dependent on the following factors:

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

A test program for determining the overall repeatability of a machine is described in Section 10.4.

© Siemens AG, 2004. All rights reserved

1.9 Measuring strategy for measuring workpieces with tool offset

The actual workpiece dimensions must be measured exactly and compared with the setpoint values to be able to determine and compensate the actual dimensional deviations on the workpiece. An offset value can then be ascertained for the tool used for machining.

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 fall into 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 machine, etc.).

  In this case, it is possible to apply **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.

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

  Assuming the ideal case, only those dimensional deviations that 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 (sliding averaging) is needed that derives a compensation value from the actual/set difference measured.

© Siemens AG, 2004. All rights reserved

Mean value calculation

Mean value calculation in conjunction with measurement weighting has proven a suitable method.

The formula of the mean value generation chosen is:

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

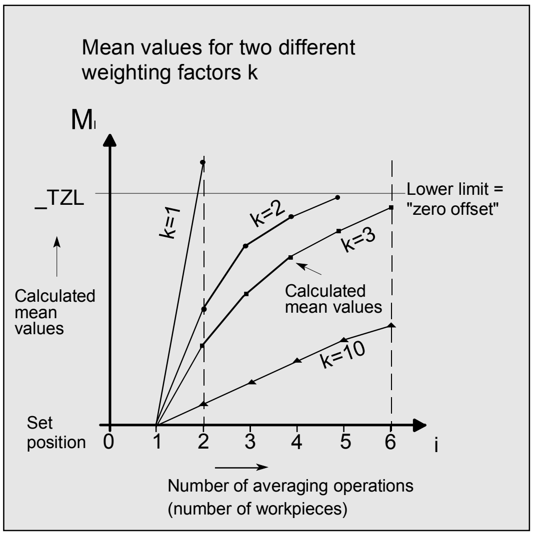
Mv_{new} Mean value new = amount of compensation
Mv_{old} Mean value prior to last measurement
k Weighting factor for average value calculation
D_i Actual/set difference measured
(minus empirical value, if any)

The mean value calculation takes account of the trend of the dimensional deviations of a machining series. The **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.

© Siemens AG, 2004. All rights reserved

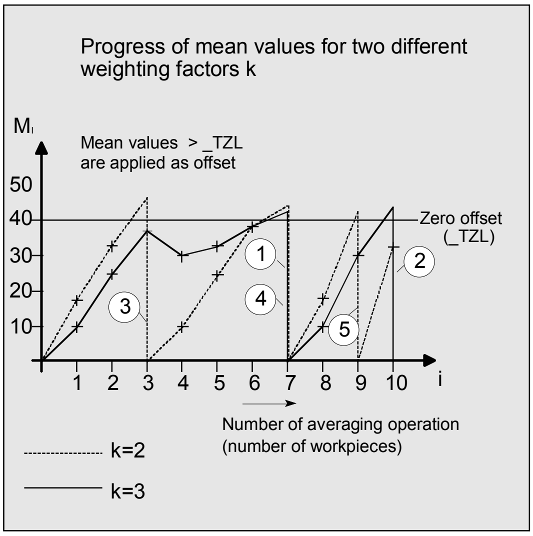
Computational characteristic of the mean value with different weightings k



- 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 Mi is calculated starting at 0 over the number of workpieces i, until the calculated mean value exceeds the range of "zero compensation" (cycle parameter _TZL, see Chapter 2). From this limit on, the calculated mean value is applied as an offset.
- Once the mean value has been used for the offset, it is deleted from the memory. The next measurement then starts again with Mi_{old} = 0.

Example of mean value calculation and offset

	Lower limit = 40 µm (_TZL= 0.04)		
	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



In the measurements with marked fields, tool compensation is performed with the mean value (calculated mean value >_TZL):

If k=3 in the 7th and 10th measurement (① and ②),

If k=2 in the 3rd, 7th, and 9th meas.(③ , ④ and ⑤).

© Siemens AG, 2004. All rights reserved

1.10 Parameters for checking the measurement result and offset

Explanation

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

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

If you do not want to apply any empirical values, _EVNUM=0 must be set.

The empirical values themselves are stored in data block (GUD5) in 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 single-point measurement with automatic zero offset and
- tool measurement

Mean value _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 data block (GUD5) in field **_MV[ ] Mean values**. **_EVNUM** specifies the number of the mean value memory in this field.

© Siemens AG, 2004. All rights reserved

Safe area _TSA
The safe area is effective for almost all measuring variants and does not affect the offset value; it is used for diagnostics.
If this limit is reached then the following can be assumed:

- a probe defect, or
- an incorrect setpoint position, or
- an illegal deviation from the setpoint position can be assumed.

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.
This enables a fast response to major dimensional deviations.

When the tolerance limit of the workpiece is exceeded, this is indicated to the user depending on the tolerance position "oversize" or "undersize".

© Siemens AG, 2004. All rights reserved

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.

Weighting factor for mean value calculation _K
_K is active only workpiece measurement with automatic tool offset. The weighting factor can be used to give a different weighting for each measurement.

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

Lower limit (zero compensation area) _TZL
_TZL active for

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

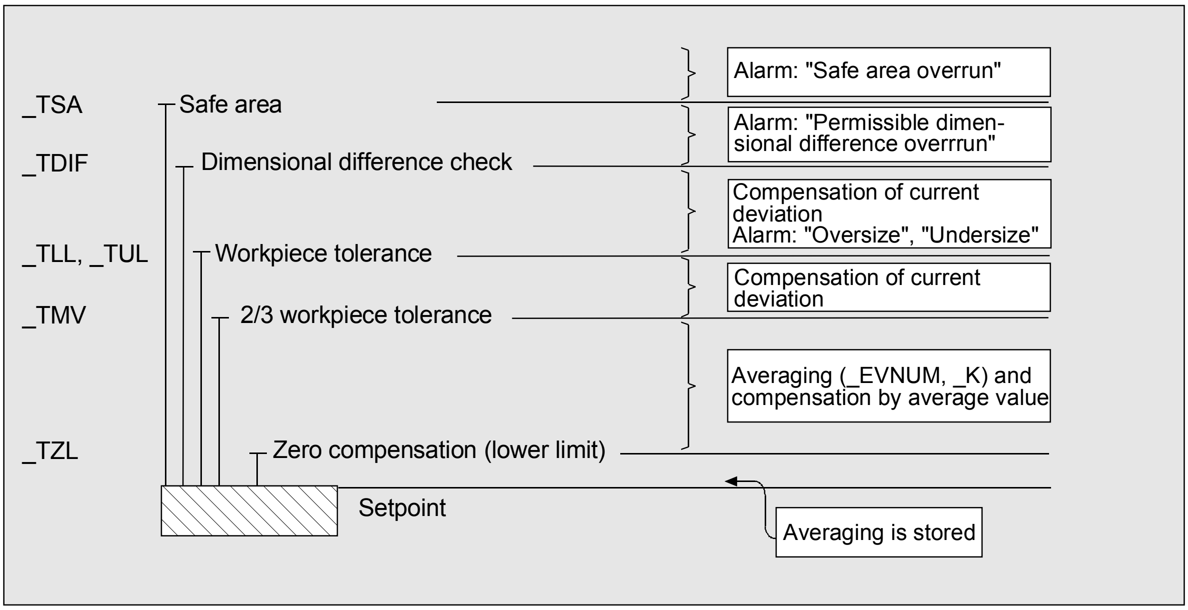
This tolerance range corresponds to the amount of maximum accidental dimensional deviations. It has to be determined for each machine.
No tool compensation is made within these limits.

In workpiece measurement with automatic tool offset, however, the mean value of this measuring point is updated and re-stored with the measured actual/set difference, possibly compensated by an empirical value.

© Siemens AG, 2004. All rights reserved

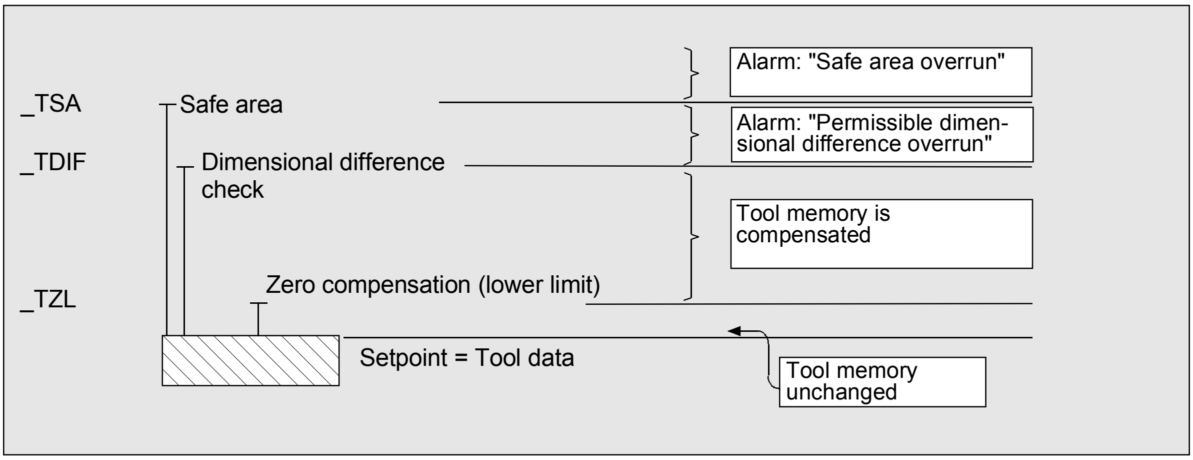
The tolerance bands (range of permissible dimensional tolerance) and the responses derived from them are as follows:

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



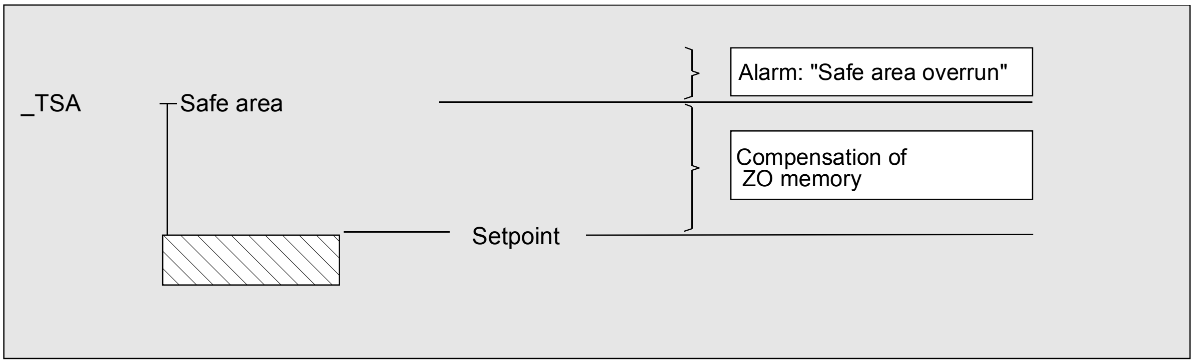
In measuring cycles, the workpiece setpoint dimension is placed in the middle of the permitted ± tolerance range for reasons of symmetry.
See Subsection 2.3.11 "Tolerance parameters..."

- **For tool measurement**

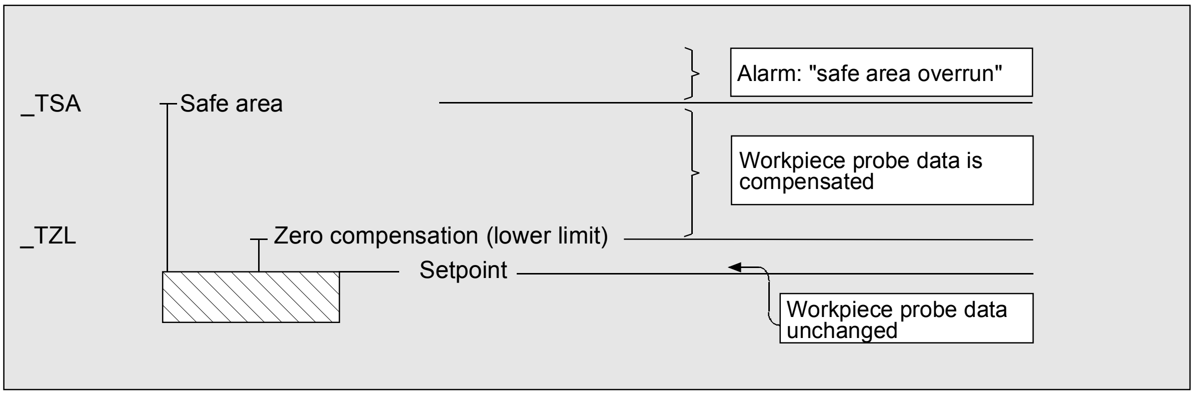


© Siemens AG, 2004. All rights reserved

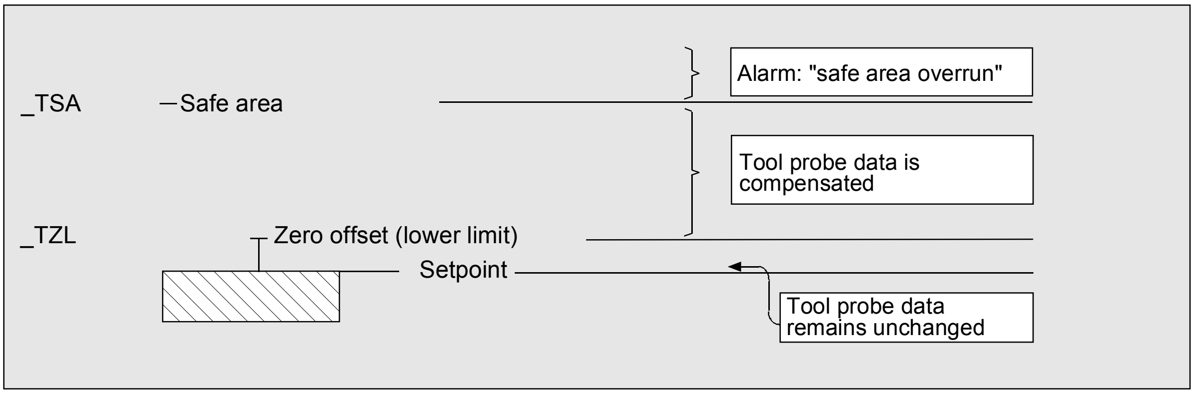
- **For workpiece measurement with zero offset**



- **For workpiece probe calibration**



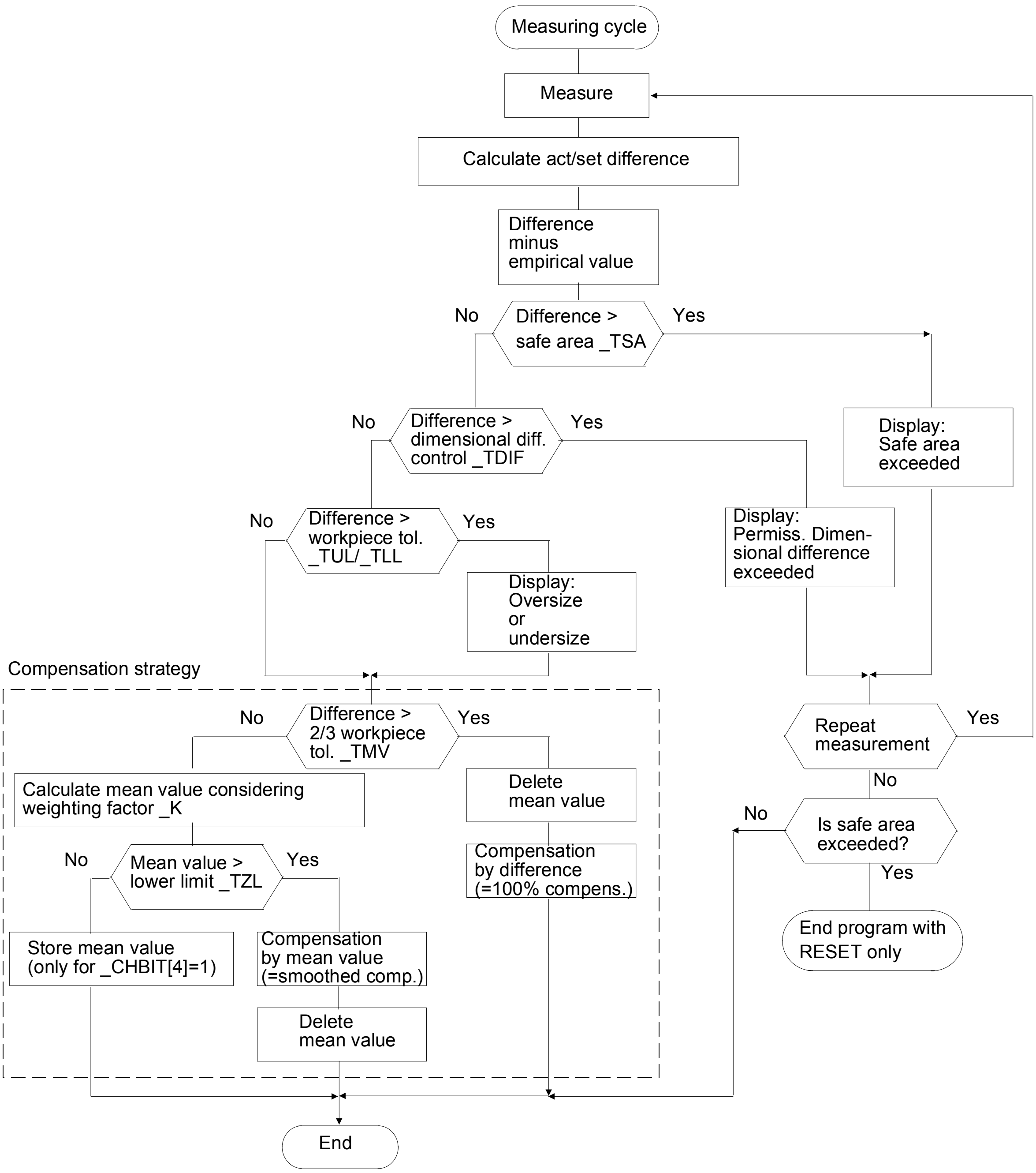
- **For tool probe calibration**



© Siemens AG, 2004. All rights reserved

1.11 Effect of empirical value, mean value, and tolerance parameters

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



© Siemens AG, 2004. All rights reserved

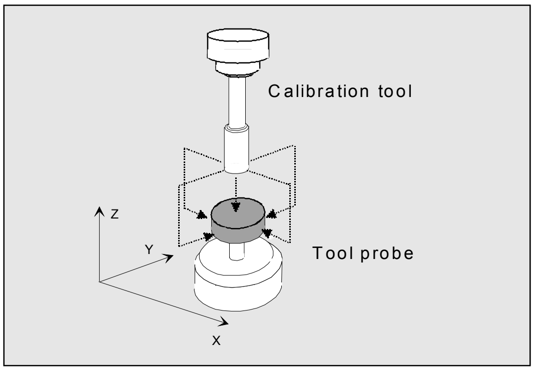
1.12 Overview of measuring cycle functions for milling technology

1.12.1 Tool measurement on milling machines, machining centers

Function

Measuring cycle CYCLE971 can be used to calibrate a tool probe and measure the tool length and/or radius for milling tools.

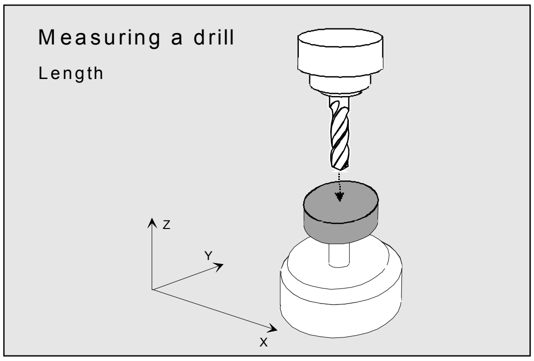
Calibrating a tool probe

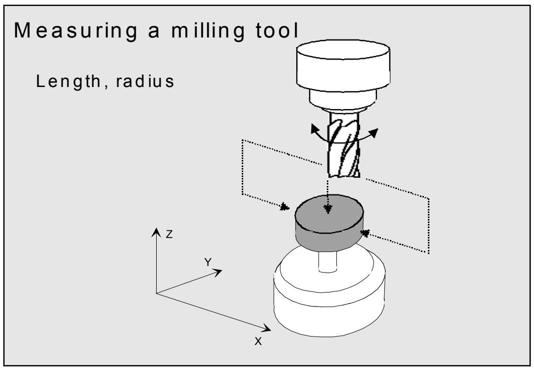


Result:
Probe switching point with reference to machine zero

From measuring cycles SW 6.3 onwards reference to the workpiece zero is possible.

Measuring the tool





Result:
Tool length
Tool radius

© Siemens AG, 2004. All rights reserved

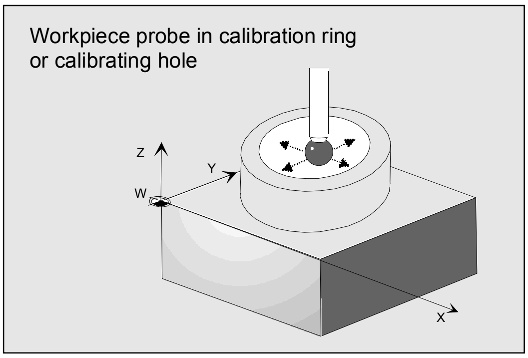
1.12.2 Calibrating workpiece probes

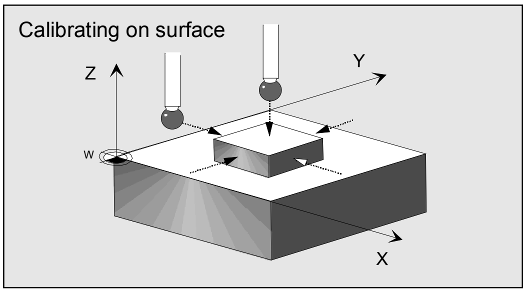
Function

With cycle CYCLE976 a workpiece probe can be calibrated in a hole (calibration ring) or on a surface for a particular axis and direction.

Calibrating a workpiece probe

Result:
Probe switching point (trigger value),
possibly an additional skew, active ball diameter of probe





© Siemens AG, 2004. All rights reserved

1.12.3 Workpiece measurement at one point

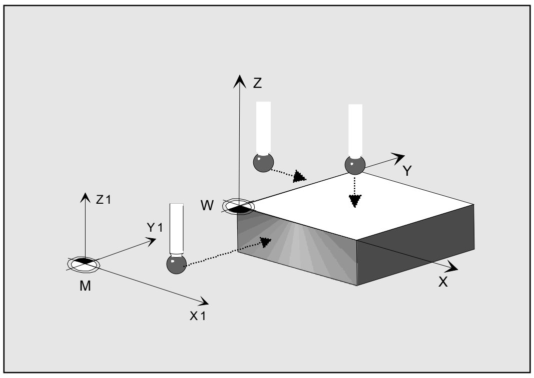
Function

CYCLE978 permits measurement at one point of a surface.

The measuring point is approached paraxially in the active workpiece coordinate system.

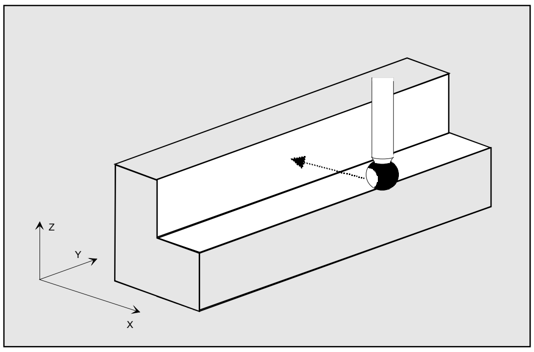
Depending on the measuring variant, the result may influence the selected tool offset or zero offset.

Workpiece measurement: Blank measurement



Result:
- Position
- Deviation
- zero offset

Workpiece measurement: 1-point measurement



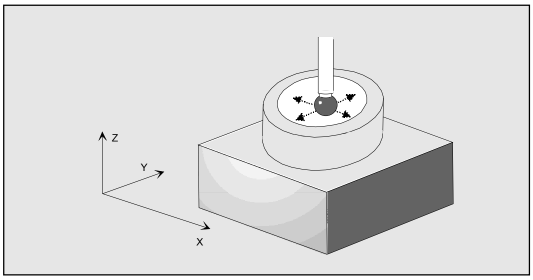
Result:
- Actual dimension
- Deviation
- Tool offset

1.12.4 Measuring the workpiece parallel to the axis

Function

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

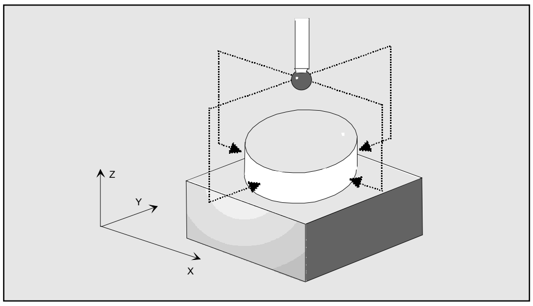
Workpiece measurement: Measuring a hole



Result:

- Actual dimension, deviation: Diameter, center point
- Deviation: Tool offset of the zero offset

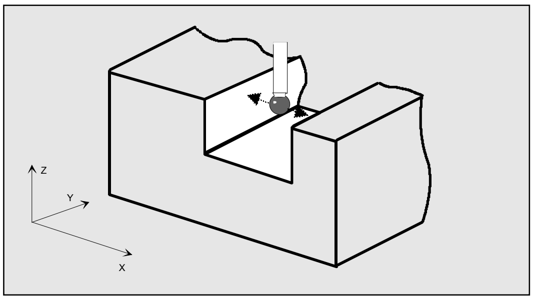
Workpiece measurement: Measuring a shaft



Result:

- Actual dimension, deviation: Diameter, center point
- Deviation: Tool offset of the zero offset

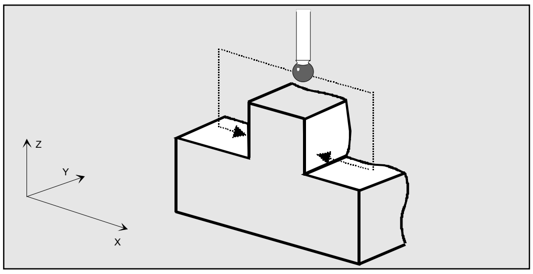
Workpiece measurement: Measuring a groove



Result:

- Actual dimension, deviation: Groove width, groove center
- Deviation: Tool offset of the zero offset

Workpiece measurement: Measuring a web



Result:

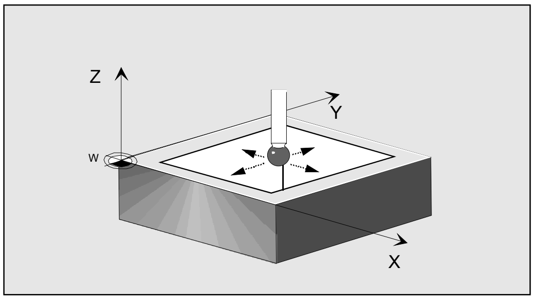
- Actual dimension, deviation: Web width, web center
- Deviation: Tool offset of the zero offset

© Siemens AG, 2004. All rights reserved

Workpiece measurement: Inside rectangle

Result:

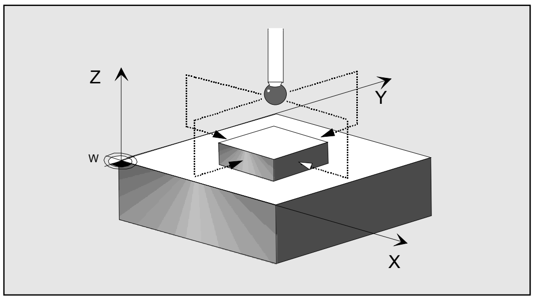
- Actual dimension, deviation: Rectangle length and width, rectangle center
- Deviation: Tool offset of the zero offset



Workpiece measurement: Outside rectangle

Result:

- Actual dimension: Rectangle length and width, rectangle center
- Deviation: Rectangle length and width, rectangle center
- Deviation: Tool offset of the zero offset



© Siemens AG, 2004. All rights reserved

1.12.5 Measuring a workpiece at an angle

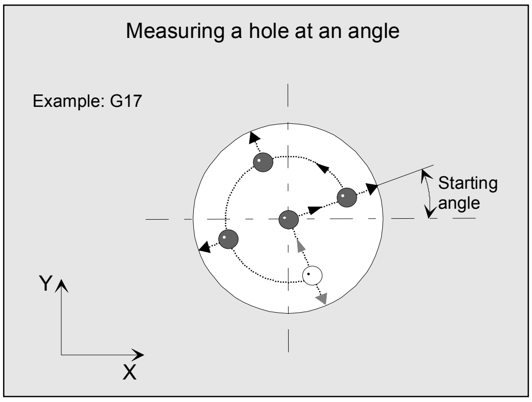
Function

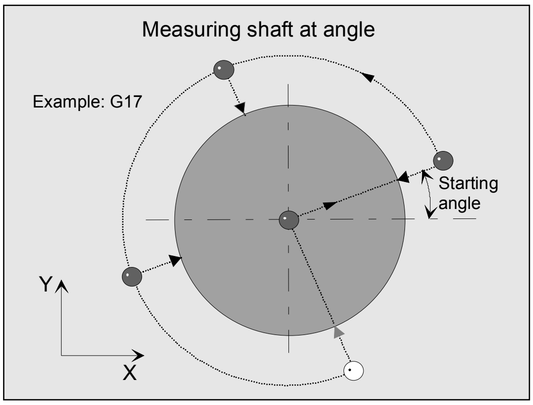
The following measuring variants are provided for the measurement of a hole, shaft, groove, or web at an angle. They are executed by CYCLE979.

Three- or four-point measurement at an angle

Result:

- Actual dimension, deviation: Diameter, center point
- Deviation: Tool offset of the zero offset

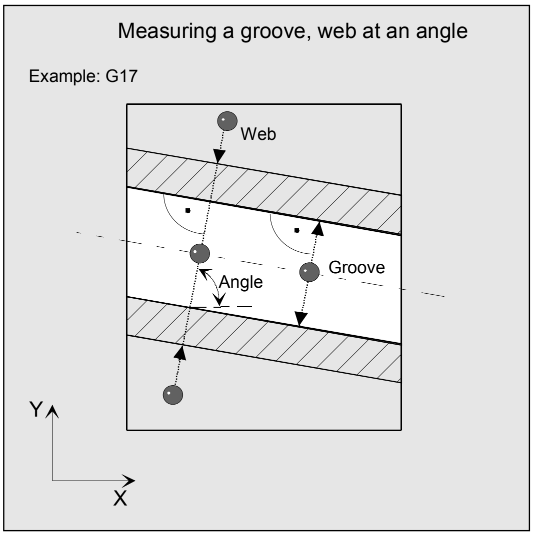




Two-point measurement at an angle

Result:

- Actual dimension, deviation: Groove, web width, groove, web center
- Deviation: zero offset



© Siemens AG, 2004. All rights reserved

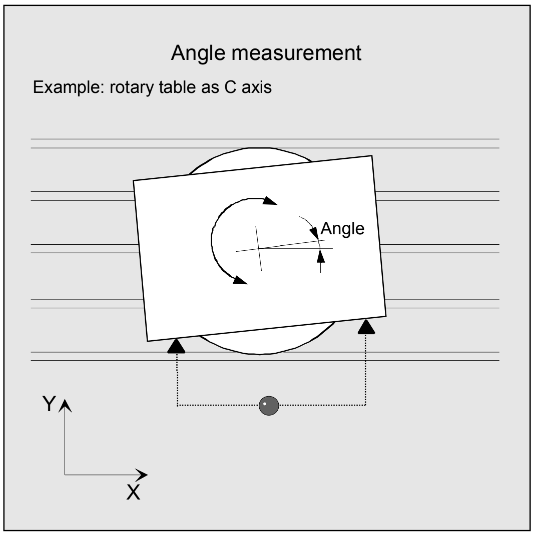
1.12.6 Measuring a surface at an angle

Function

CYCLE998 permits correction of the zero offset after measurement of a surface at an angle.
It is still possible the determine the angles on an oblique surface in space.

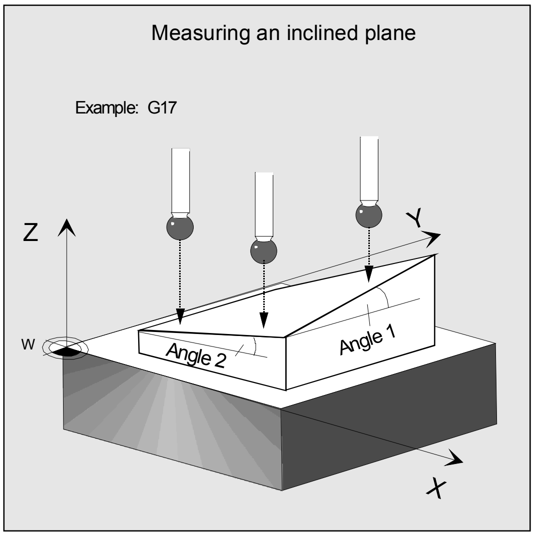
Workpiece measurement: Angle measurement

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



Workpiece measurement: Two-angle measurement

Result:
Actual dimension (two angles),
deviation,
zero offset



© Siemens AG, 2004. All rights reserved

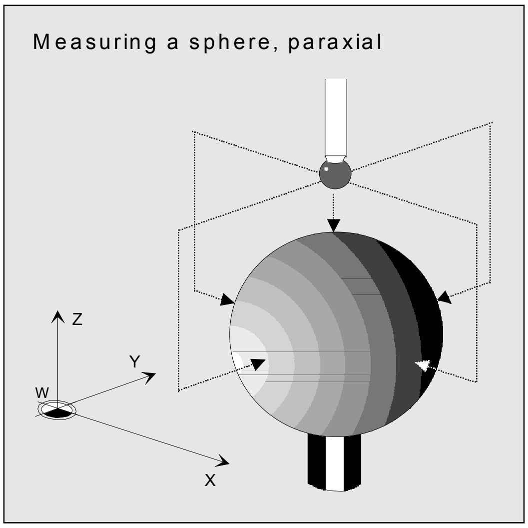
1.12.7 Measuring spheres (from measuring cycles SW 6.3 onwards)

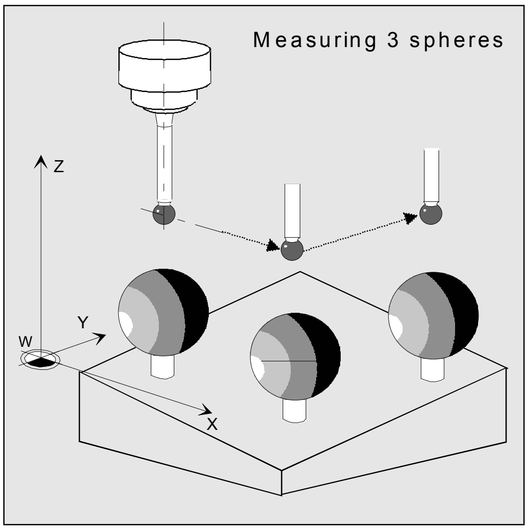
Function

CYCLE997 permits correction of the zero offset after measurement of a sphere or of three identically sized spheres on a common base (workpiece). Either paraxial measurement or measurement at an angle can be selected.

Workpiece measurement: Sphere

Result:
Actual dimension (position of center, diameter), deviation,
Zero offset (for one sphere translation only, for three spheres also rotation in space)





© Siemens AG, 2004. All rights reserved

1.12.8 Workpiece measurement: Setting-up a corner

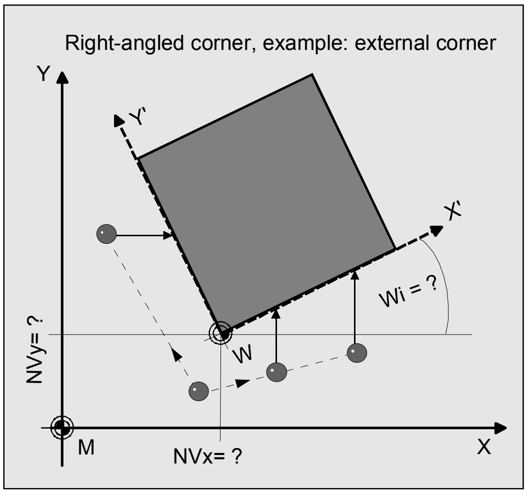
Function

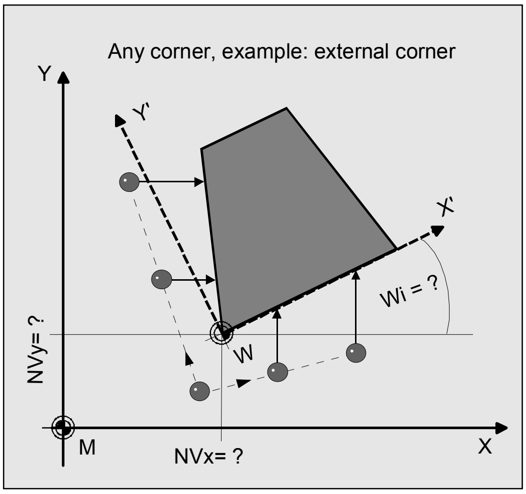
Using the CYCLE961 cycle, it is possible to determine the position of a workpiece corner (inner or outer) and use this as zero offset.

Measuring a corner by specifying clearances and angles

Result:

- Actual position of the corner with angle
- Zero offset, rotation

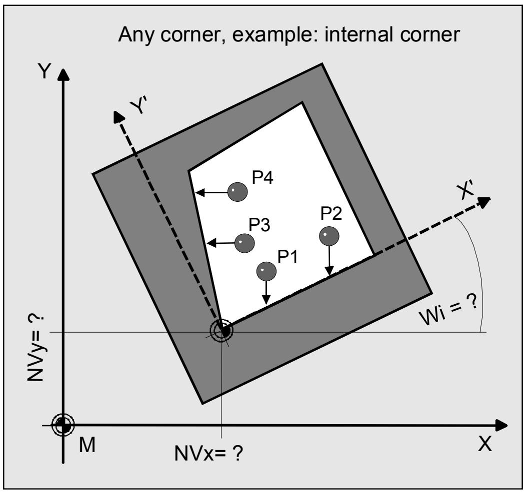




Measuring a corner, specifying 4 points

Result:

- Actual position of the corner with angle
- Zero offset, rotation



© Siemens AG, 2004. All rights reserved

1.13 Overview of measuring cycle functions for turning technology

1.13.1 Measuring tools at lathes

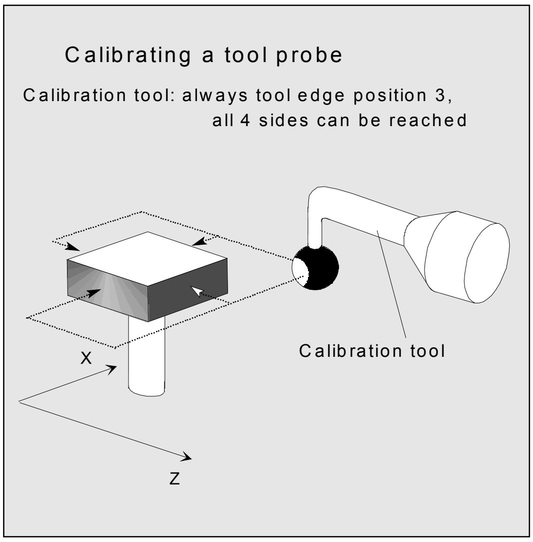
Function

Cycle CYCLE982 is used to calibrate a tool probe and measure turning, drilling, and milling tools on turning machines.

Calibrating a tool probe

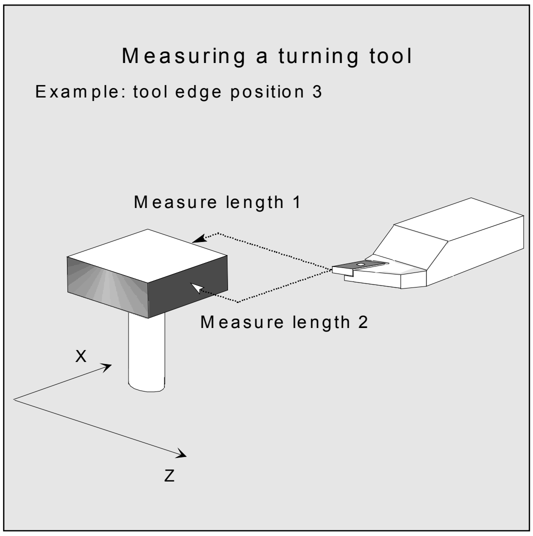
Result:
Probe switching point with reference to machine zero

From measuring cycles SW 6.3 onwards reference to the workpiece zero is possible.

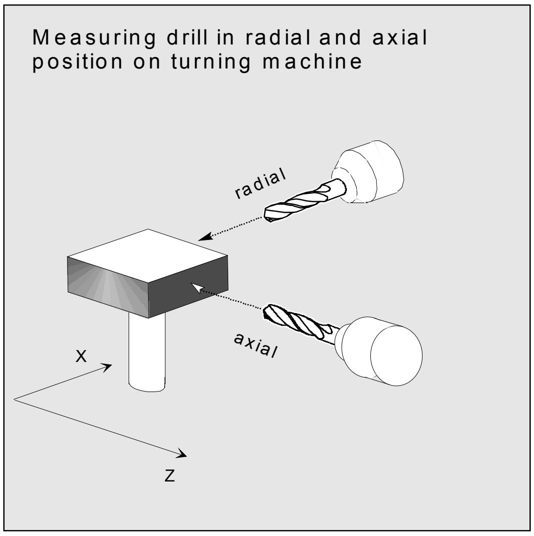


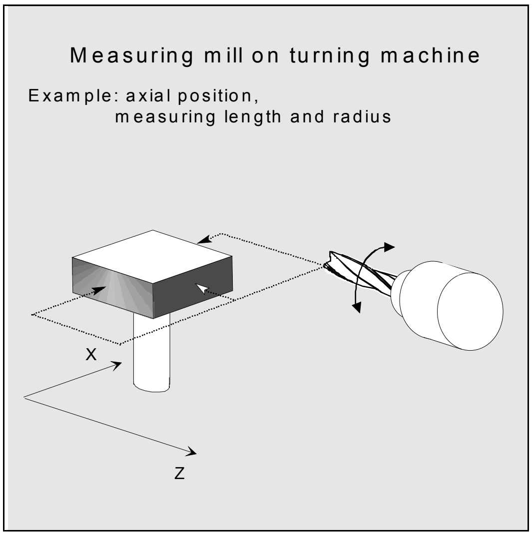
Measuring the tool

Result:
Tool length (length1, length2)
Milling radius (R - for milling tools)



© Siemens AG, 2004. All rights reserved


Measuring drill in radial and axial
position on turning machine
radial
axial
X
Z


Measuring mill on turning machine
Example: axial position,
measuring length and radius
X
Z

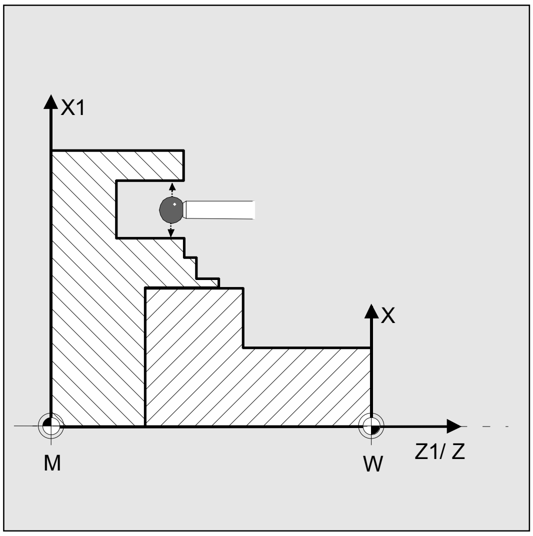
1.13.2 Calibrating workpiece probes

Function

CYCLE973 permits calibration of a probe on a surface of the workpiece or in a calibration groove.

Calibrating workpiece probes

Example:
Calibrate probe with cutting edge position 7, in X axis in both directions in a calibration groove.



© Siemens AG, 2004. All rights reserved

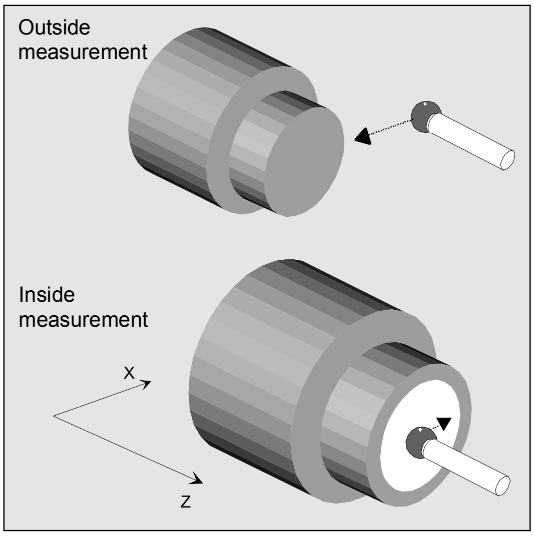
1.13.3 Measuring tools at lathes: 1-point measurement

Function

CYCLE974 is used to determine the actual value of the workpiece in the selected measuring axis with reference to the workpiece zero with 1-point measurement.

1-point measurement outside

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

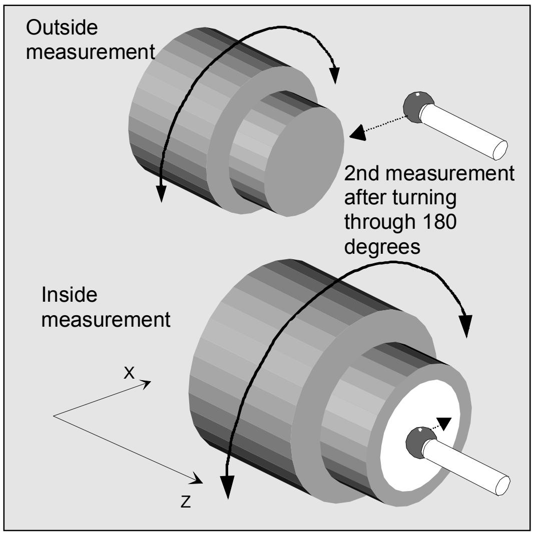


1-point measurement inside

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

1-point measurement outside with 180° reversal spindle

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



1-point measurement inside with 180° reversal spindle

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

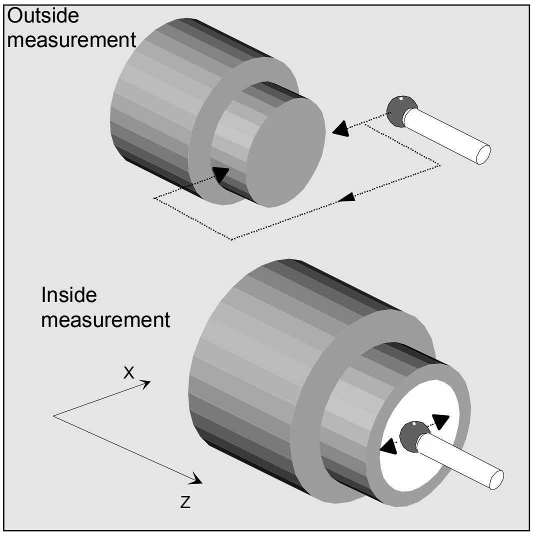
© Siemens AG, 2004. All rights reserved

1.13.4 Measuring workpieces at lathes: 2-point measurement

Function

CYCLE994 is used to determine the actual value of the workpiece in the selected measuring axis with reference to the workpiece zero with 2-point measurement. This is done automatically by approaching two opposite measuring points on the diameter.

2-point measurement on outside diameter



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

2-point measurement on inside diameter

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

■

© Siemens AG, 2004. All rights reserved

Parameter Description

© Siemens AG, 2004. All rights reserved

2.1 Parameter concept of the measuring cycles

Function

Measuring cycles are general subroutines designed to solve specific measurement tasks, which are suitably adapted to the problem at hand with parameter settings. They can be adapted for this purpose via **defining parameters**.
The measuring cycles also return data such as measuring results. They are stored in **result parameters**.
These measuring cycle parameters are called **G**lobal **U**ser **D**ata (abbreviated to GUDs).
They are located in the battery-backed memory of the control. These values are therefore not lost when the control is switched off and on.

This data is kept in data blocks.

- GUD5.DEF
- GUD5.DEF and
- GUD7.DEF (for measuring in JOG)

On delivery these data have default settings (see Section 2.2). These must be adapted by the user or machine manufacturer on installation (see Chapter 9).
The defining parameters must be assigned values before the measuring cycle is called:

- Either in the program or
- Operator input in the measuring cycle support

The data in the operating area "Parameters" can be displayed by means of "User data", "Global user data", "GUD…" or "Channel-specific user data", "GUD…".. As an alternative, parameters that are not assigned values in the program or in the measuring cycles support can be assigned values directly by experts.

Measuring cycles also require **internal parameters** for calculations. **L**ocal **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 exist only during runtime.

© Siemens AG, 2004. All rights reserved

2.2 Parameter overview

2.2.1 Supply parameters

Explanation

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

- **Mandatory parameters**
- **Auxiliary 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 additional measuring cycle call** until modified by programming or operation.

In addition to defining parameters for calculation or character string input, there are also BOOLEAN type variables. These bits can be used to vary planned cycle sequences or enable or disable certain settings. These **cycle bits** are arrays of variables and of two types:

- **Central bits: _CBIT[ ]**
- **Channel-oriented bits: _CHBIT[ ]**

Their name defines their validity and occurrence:

- Central bits --> NCK: Occur once, apply to all channels
- Channel bits --> CHAN: Defined separately for each channel.

An overview of central and channel-oriented bits is given in Section 9.2.
These bits can also be changed by programming or operation.

© Siemens AG, 2004. All rights reserved

Mandatory parameters

Parameters	*Type*	*Validity*	*Default:*	*Meaning*
_SETVAL[1]	REAL	CHAN	–	Setpoint
_SETV[ ] [1]	REAL	CHAN	–	Setpoints – additional, e.g. for measuring 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	–	Start angle
_INCA	REAL	CHAN	–	Indexing angle
_MVAR	INT	CHAN	–	Measuring 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	–	Correction number (D No. or ZO No.)
_RA	INT	CHAN	–	Number of rotary axis at angle measurement
_TENV	STRING[32]	CHAN	–	Name of tool environment
_DLNUM	INT	CHAN	–	DL number for setup or additive offset

Auxiliary parameters

Parameters	*Type*	*Validity*	*Default* (SW 6.3)	*Meaning*
_VMS	REAL	CHAN	0	Variable measuring speed
_RF	REAL	CHAN	0	Feedrate in circular-path programming
_CORA	REAL	CHAN	0	Compensation angle, e.g. for mono probe
_TZL	REAL	CHAN	0.001	Zero offset area
_TMV	REAL	CHAN	0.7	Mean value generation with compensation
_TNVL	REAL	CHAN	1.2	Limit value for distortion of triangle
_TUL[1]	REAL	CHAN	1.0	Tolerance upper limit
_TLL[1]	REAL	CHAN	–1.0	Tolerance lower limit
_TDIF	REAL	CHAN	1.2	Dimension difference check
_TSA	REAL	CHAN	2	Safe area
_FA[2]	REAL	CHAN	2	Measurement path in mm
_CM[ ]	REAL	NCK	100, 1000, 1, 0.005, 20, 4, 10, 0	Monitoring parameters for tool measurement for a rotating spindle
_PRNUM	INT	CHAN	1	Probe number
_EVNUM	INT	CHAN	0	Number of empirical value memory
_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 calculation

© Siemens AG, 2004. All rights reserved

Parameters for logging only

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

1) All parameters with dimensions, 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.
2) _FA is always a value in mm, even when the unit system set is inches.

2.2.2 Result parameters

Parameters	*Type*	*Validity*	*Meaning*
_OVR[ ]	REAL	CHAN	Result parameter – real number: Setpoint values, actual values, differences, offset values, etc.
_OVI[ ]	INT	CHAN	Result parameter – integer

2.3 Description of the most important defining parameters

2.3.1 Measurement variant: _MVAR

Function

The measuring variant of each individual cycle is defined in parameter **_MVAR**. _MVAR can be assigned certain positive integer values.

Please refer to the individual cycle descriptions!

The value in _MVAR is plausibility checked by the cycle. If it does not have a defined value, alarm 61307 is output: **"Incorrect measuring variant"**.
The cycle must be interrupted by an NC RESET.
_MVAR must be corrected.

2.3.2 Number of the measuring axis: _MA

Function

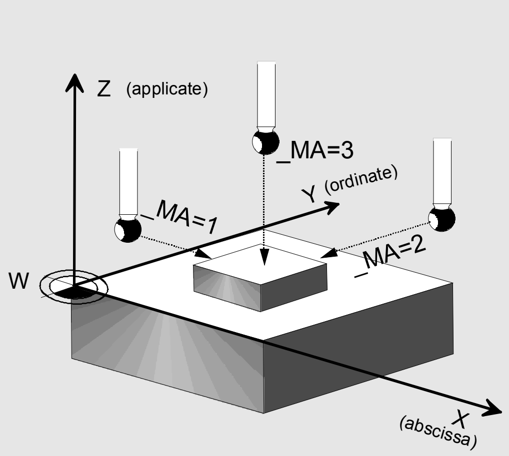
In some cycles or measuring variants, number 1, 2, or 3 must be specified in **_MA** for the measuring axis. This might by axis X, Y, or Z in the workpiece coordinate system depending on whether G17, G18, or G19 is active.

This always results in:

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

Example:
Workpiece measurement on milling machine with G17, various measuring directions

Axis definition, measuring axis _MA



With some measuring variants, for example, CYCLE998, positioning in another axis (which must be defined), called the offset axis, can be performed between measurements in the measuring axis. This is must be defined in parameter _MA with offset axis/measuring axis.
The higher digit codes the offset axis, the lower digit the measuring axis, the tens digit is 0.

© Siemens AG, 2004. All rights reserved

Example of _MA in CYCLE998:
_MA = 102
⇒ Offset axis: 1 (abscissa)
⇒ Measuring axis: 2 (ordinate)

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. Here it can be used as an alternative to _TNUM. However, a programmed _TNUM >0 always has priority.

Example:

- **Without tool management:**
  _TNUM = 12 tool T number 12
  is corrected
- **With tool management:**
  _TNUM = 0 _TNAME = "DRILL"
  → the tool called
  "DRILL" is corrected
  Or
  _TNUM = 13 _TNAME = "DRILL"
  → the tool with the internal
  T number 13 is corrected

In the case of replacement tools, the tool that was last used is corrected.

However, it is necessary that only one tool in a group be "active" at any one time. Otherwise, the internal tool number of the tool used must be determined and assigned to _TNUM via the system variable $P_TOOLNO during processing.

© Siemens AG, 2004. All rights reserved

2.3.4 Offset number: _KNUM

Parameters

With measuring variant _MVAR you can select whether **automatic tool offset** will be used or a **zero offset** will be corrected in a **workpiece measuring cycle**.

Parameter **_KNUM** contains the

- **tool offset memory number** (D number)
  or a
- code for the **zero offset** to be corrected.

Values of _KNUM: ≥0, integer
1. Specification _KNUM for tool offset, 7 digits:
_KNUM can accept values with up to 7 digits (for special MD settings, even 9 digit values, see Subsection 2.3.5).
_KNUM=0: no automatic tool offset

7 6 5 4 3 2 1

- Digits 3 2 1: D number
- Digit 4:
  0/1...automatic offset of the length effective in the measuring axis or set-up / sum offset (as of measuring cycles software version 6.3)
  2...automatic radius offset or set-up / sum offset (as of measuring cycles software version 6.3)
- Digit 5: Standard / inverted offset (sign changed)
  0...Standard
  1...Inverted (sign changed)
- Digit 6:
  0...Offset as per 4th digit
  1...Offset of L1
  2...Offset of L2
  3...Offset of L3
  4...Radius offset
- Digit 7 (As of measuring cycles software version 6.3):
  0...Offset in length / radius
  1...Correction in set-up / sum correction
  2...Offset in length / radius according to _TENV
  3...Correction in set-up / sum offset according to _TENV

Note: Tool environment _TENV -> see parameter _TENV

In the default setting, the D number has values between 0…9.
Depending on
MD 18102: MM_TYPE_OF_CUTTING_EDGE = 0 and
MD 18105: MM_MAX_CUTTING_EDGE_NO and a value of this MD of between 10...999, the last three digits are read as a D number.
If the value is ≥1000, _KNUM is evaluated as a 5-digit D number (unique D number, as in flat D number structure, see next Section).
Example of _KNUM=12003: D3 is correct, calculated as a radius offset, inverted (sign inverted).

© Siemens AG, 2004. All rights reserved

2. Specification _KNUM for zero offset:

- **_KNUM=0:** No automatic ZO correction
- **_KNUM=1... 99:** automatic ZO in settable frame / ZO G54...G57, G505...G599
- **_KNUM=1000:** automatic ZO in last channel-specific basic frame according to MD 28081: MM_NUM_BASE_FRAMES.
  The offset is calculated to ensure it has the right effect when G500 is activated. The corresponding basic frame must then also be active (relevant bit in $P_CHBFRMASK must be set).
- **_KNUM=1011...1026:** automatic ZO in 1st to 16th channel basic frame ($P_CHBFR[0]...$P_CHBFR[15])
- **_KNUM=1051...1066:** automatic ZO in 1st to 16th basic frame (NCU global) ($P_NCBFR[0]...$P_NCBFR[15])
- **_KNUM=2000:** automatic ZO in system frame (scratch system frame $P_SETFR)
- **_KNUM=9999:** automatic ZO in active frame:
  – active settable frame G54...G57, G505...G599 or
  – when G500 is active: last active basic frame according to $P_CHBFRMASK (highest set bit).
  Only when _KNUM=9999 is the changed frame immediately activated in the cycle, otherwise **by the user** by writing G500, G54...G5xy.

Notes:

- The remaining active frame chain must be retained.
- With NCU-global frames, correction for rotation is not possible.

The following must be set during installation (default setting):

- MD 28082: MM_SYSTEM_FRAME_MASK, Bit 0=1 and Bit 5=1 (system frames for scratching and cycles must be available)
- Additionally the MDs for the required basic frames

In the measuring cycles in AUTOMATIC mode the offset for the default setting is corrected additively with **fine offset** (MD 18600: MM_FRAME_FINE_TRANS=1).
Otherwise (when MD 18600=0), or in CYCLE961, or when _KNUM=2000, or when "measuring in JOG" is active, the offset is implemented in the **coarse offset**. Any existing fine offset is included in the calculation and then deleted.

When measuring workpieces with ZO (offset in CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE997) in AUTOMATIC mode, values can be written either to the coarse or fine offset:

© Siemens AG, 2004. All rights reserved

_CHBIT[21]:

0: ZO translation additive in FINE

1: ZO translation in COARSE, FINE=0
If the offset is written to the coarse offset
any existing fine offset in the
offset value is included and the
fine offset then deleted.

Note:
If _KNUM=2000 (scratch system frame $P_SETFR),
the offset value is always written to the coarse offset.

© Siemens AG, 2004. All rights reserved

2.3.5 Offset number _KNUM extended for tool offset: Maximum 9 digits

Parameters

For special tool offset structures (D number structures), parameter **_KNUM** can have up to nine digits.

The "**flat D number**" functionality is implemented as from NCK-SW 4.
This function is defined with MD 18102: MM_TYPE_OF_CUTTING_EDGE=1. This function allows up to 5-digit D numbers.

"**Unique D number**" is a second method of implementing a 9-digit _KNUM:
As from NCK-SW 5 and depending on
MD 18102: MM_TYPE_OF_CUTTING_EDGE = 0 and
MD 18105: MM_MAX_CUTTING_EDGE_NO ≥1000
D numbers have 5 digits and _KNUM has 9 digits.

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

The D number is contained in the five lowest digits of _KNUM. This is automatically recognized in the cycles by the MD settings. The remaining digits of _KNUM still have the same meaning but have been shifted two places along.

9 8 7 6 5 4 3 2 1

Digits 5 4 3 2 1: D number

Digit 6:
0/1...automatic offset of the lenght effective in the measuring axis or set-up / sum offset (as of measuring cycles software version 6.3)
2...Radius offset or set-up (as of measuring cycles software version 6.3)

Digit 7: Standard / inverted (sign changed)
0...Standard
1...Inverted (sign changed)

Digit 8:
0...Offset as per 6th digit
1...Offset of L1
2...Offset of L2
3...Offset of L3
4...Radius offset

Digit 9 (As of measuring cycles software version 6.3):
0...Offset in length / radius
1...Correction in set-up / sum offset
2...Offset in length / radius according to _TENV
3...Correction in set-up / sum offset according to _TENV

2.3.6 Correcting setup and additive offset in workpiece measurement: _DLNUM (from measuring cycles SW 6.3 onwards)

Parameters

Setup and additive offsets are assigned to the tool and a D number. Each D number can be assigned up to 6 setup and additive offsets using DL numbers in the program. If DL=0, no setup or additive offset is activated.
The existence or number of setup or additive offset is set in the machine data.

References: /FB1/, W1, "Tool Offset"

From measuring cycles SW 6.3 onwards, it is possible to correct a selected setup or additive offset in the measuring cycles when measuring workpieces with automatic tool offset using parameters _TNUM and _TNAME, _KNUM (D-number coded) and additional parameter **_DLNUM.**
_DLNUM is an integer. The value range is 0…6
This variable need only be defined by the user if the corresponding digit is also programmed in variable _KNUM.

Two **channel-oriented bits** are available for selecting additive and setup offsets (see Subsection 9.2.7).
The measuring cycles for workpiece measurement with automatic tool offset use channel bit _CHBIT[6] for selecting length and radius offsets in wear or geometry.
0: The offset value is added to the existing wear.
1: Geometry(new) = geometry (old) + wear (old) + offset value,
wear (new) = 0

_CHBIT[6] is also used for correcting setup and additive offsets:
0: The offset value is added to the existing additive offset.
1: Setup offset (new) = setup offset (old) + additive offset (old)
+ offset value,
additive offset (new) = 0

additionally **_CHBIT[8]**:
0: Correct additive/setup offset according to _CHBIT[6]
1: The offset value is added to the existing setup offset,
irrespective of _CHBIT[6]:

© Siemens AG, 2004. All rights reserved

2.3.7 Correcting the tool of a stored tool environment: _TENV (from measuring cycles SW 6.3 onwards)

Parameters

As from NCK SW 6.3, you can save the operating environment of a particular tool you are using. This is to allow you to correct the tool used to measure a workpiece taking account of the operating conditions (environment: e.g. plane, length assignment, …).
You then no longer have to specify the T, D, DL number in the offset explicitly. These are included in the stored tool environment. The name of a tool environment can have up to 32 characters.

References: /PGA/, "Programming Guide Job planning"

Parameter **_TENV** is used in measuring cycles for workpiece measurement with automatic tool offset to define the tool environment.
_TENV is type string [32]. _TENV is only considered if the corresponding digit is programmed in parameter _KNUM.

Note:
_TENV can only be used if function TOOLENV("NAME") has already been programmed by the user in the workpiece machining program. This sets up the tool environment.
The number tool environments that can be created in the SINUMERIK control is set in MD 18116: MM_NUM_TOOL_ENV.

A tool offset used in conjunction with a tool environment offers many possibilities. These will be shown in more detail using examples in the next Section.

© Siemens AG, 2004. All rights reserved

2.3.8 Example of automatic tool offset in workpiece measuring cycles

Examples of parameter settings for tool offset with and without stored tool environment

If the tool environment was stored during workpiece machining with TOOLENV("NAME"), it is possible to subsequently correct a tool under these stored conditions while measuring a workpiece.
First, the name of the tool environment _TENV="NAME" must be specified in the offset.
_TENV is evaluated if the corresponding position in _KNUM has value 2 or 3 (offset ... according to _TENV).
To correct tool T stored in tool environment "NAME", _TNUM=**0** must be set. Otherwise the programmed _TNUM / TNAME, D (contained in _KNUM), _DLNUM is corrected with the conditions of the specified tool environment "NAME". Further alternatives: See examples.
In "flat D number " structures _TNUM has no significance. Here, only _KNUM (for D) and _DLNUM are relevant.

Example 1: (without _TENV)
The wear of length 1 is to be corrected additively for tool T7 with D2. The tool environment is to be the environment currently active (= measuring environment).
Relevant data: _TNUM=7 _KNUM=0100002 _CHBIT[6]=0

Example 2: (without _TENV)
For tool T8 with D3, the wear of the length assigned to measuring axis _MA for this tool type and setting (G17, G18, or G19 etc.) is to be corrected additively.
The tool environment is to be the environment currently active (= measuring environment).
Relevant data: _TNUM=8 _KNUM=3 _CHBIT[6]=0 _MA=1

Example 3: (without _TENV)
For tool T5 with D2, the additive offset of DL=3, active in the length as defined for measuring axis _MA, is to be corrected additively.
The tool environment is to be the environment currently active (= measuring environment).
Relevant data: _TNUM=5 _KNUM=1001002 (or =1000002)
_DLNUM=3 _CHBIT[6]=0 _CHBIT[8]=0 _MA=1

© Siemens AG, 2004. All rights reserved

Example 4: (with _TENV)
The wear of length 1 or the tool and D number stored in tool environment "WZUMG1" is to be corrected additively (T and D not directly known).

Relevant data: _TNUM=0 _TENV= " WZUMG1"
_KNUM=2100000 _CHBIT[6]=0

Example 5: (with _TENV)
For tool T stored in tool environment "WZUMG2" but specific D number D2, the wear of the length assigned to measuring axis _MA for the stored tool type and stored setting (G17, G18, or G19) is to be corrected additively.

Relevant data: _TNUM=0 _TENV= " WZUMG2"
_KNUM=2001002 (or _KNUM=2000002)
CHBIT[6]=0 _MA=2

Example 6: (with _TENV)
Tool environment is to be the tool environment stored in "WZUMG3".
However, the following is to be corrected irrespective of T, D, DL stored in it.
For tool T6 with D2 the additive offset of DL=4 that is assigned to the length for tool type T6 and setting (G17, G18, or G19) stored in "WZUMG3" of the measuring axis is to be corrected additively.

Relevant data: _TNUM=6 _TENV=“WZUMG3“
_KNUM=3001002 (or _KNUM=3000002)
_DLNUM=4 _CHBIT[6]=0 _CHBIT[8]=0
_MA=1

© Siemens AG, 2004. All rights reserved

2.3.9 Variable measuring velocity: _VMS

Parameters

The measuring velocity 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 velocity must be selected to ensure safe deceleration within the probe deflecting path.

When _VMS = 0, 150 mm/min is the default value for the feedrate (when _FA=1). This feedrate value automatically increases to 300 mm/min if _FA>1 is selected.

If the basic system is in inches, 5.9055 inch/min or 11.811 inch/min takes effect.

2.3.10 Offset angle position: _CORA

Function

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

In CYCLE982 _CORA is also used to correct the spindle setting after measurement with reversal during milling measurement.

Parameters

Correction for mono probe:
The incorrect position (see above) can be corrected by means of parameter **_CORA**. _CORA is generally set to 90° or a multiple thereof.
If the direction of rotation is altered as a result of swiveling the milling head, _CORA must be preset to –360° (normally 0°).

© Siemens AG, 2004. All rights reserved

2.3.11 Tolerance parameters: _TZL, _TMV, _TUL, TLL, _TDIF, and _TSA

In Section 1.8 the correction strategy of measuring cycles is explained and a description of the effect of the parameters given.

Parameters

_TZL	REAL ≥0	Zero offset[1)2)]
_TMV	REAL >0	Mean value calculation with offset[1)]
_TUL/_TLL	REAL	Workpiece tolerance[1)]
_TDIF	REAL >0	Dimension difference check[1)2)]
_TSA	REAL >0	Safe area

1) For workpiece measurement with automatic tool offset only
2) Also for tool measurement

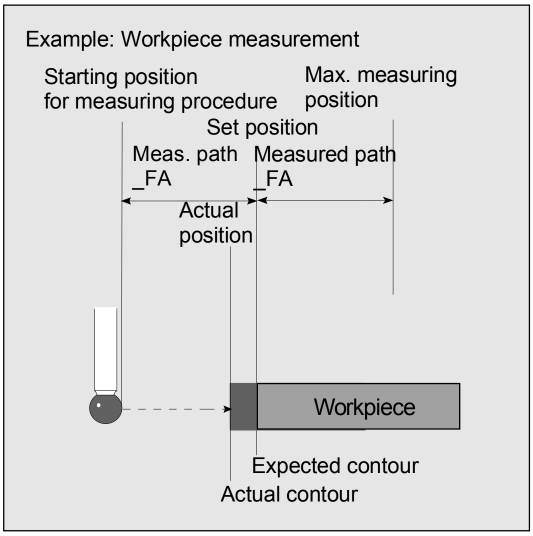
Value range
All of these parameters can have any value. However, only values increasing from _TZL to _TSA are meaningful (absolute values). Parameters _TUL/_TLL are specified in mm or inches depending on the active dimension system and are signed. All other parameters are programmed in the basic system.

Making the workpiece tolerance and setpoint symmetrical
If asymmetrical values are chosen for the tolerance parameters _TUL, _TLL (workpiece tolerance), the setpoint (_SETVAL) is corrected internally to place it in the center of a new, symmetrical tolerance band. This changed values appear in the result parameters: OVR[0] – setpoint, OVR[8] – upper tolerance limit, OVR[12] – lower tolerance limit.
The defining parameters themselves (_TUL, _TLL, _SETVAL) remain unchanged.

Example:
_TUL=0.0 _TLL=–0.004 _SETVAL=10
The result is:
OVR[8]=0.002 OVR[12]=–0.002 OVR[0]=9.998

© Siemens AG, 2004. All rights reserved

2.3.12 Measurement path: _FA



Parameters

Measurement path _FA defines the distance between the **starting position** and the expected switching position (**setpoint**) of the probe. _FA is data type REAL with values >0.
Values <0 can only be programmed in CYCLE971
Always specify _FA in **mm**.
The measuring cycles automatically generate a

Total meas. distance = 2 · _FA in mm.

The **maximum measuring** position is therefore _FA behind the set position.

See also Section 1.7 Measuring principle

Example:
The default setting is _FA=2.0.
As a result a total measurement distance of 4 mm is generated in the measuring cycle, starting 2 mm in front of and 2 mm behind the defined set position.

_FA is also used as a distance for traveling around workpieces or tool probes.

Caution
Even if inches is selected as the measuring system, measurement distance _FA is always specified in **mm**!

In that case, convert the measurement distance to mm:

_FA [mm] = _FA‘ [inch] · 25.4

Note
In previous measuring cycle versions, _FA was the name for "multiplication factor of measurement distance". Its task and function remain the same. _FA is now directly assigned mm as the unit of measurement.

© Siemens AG, 2004. All rights reserved

2.3.13 Probe type, probe number: _PRNUM

Function

- The data for the workpiece probes are stored in the data block (GUD6) in array **_WP[ ] – workpiece probe**.
- The data for the tool probes are stored in the data block (GUD6) machine-specifically in array **_TP[ ]** – **tool probe**.
- From measuring cycles SW 6.3, the data for the tool probes can be stored workpiece-specifically in data block (GUD6) in array **_TPW[ ] – tool probe in any workpiece coordinate system**.

Arrays are available for up to 99 probes. In the default setting there are arrays for three probes each.
_PRNUM states the number of the probe. This number is used as the index for the probe's array.
Array type _WP, _TP, or _TPW is selected in the cycle via the measuring variant and measuring task: workpiece or tool measurement.
Note:
Which **measuring input** (1 or 2) is used for workpiece measurement and which is used for tool measurement is defined in _CHBIT[0] and _CHBIT[1] (see Section 9.2).
The arrays are configured by the machine manufacturer during installation (see Chapter 9).

Parameters

Value of _PRNUM: >0, integer
_PRNUM can only have three digits in workpiece measurement. In that case the most significant digit is evaluated as the **probe type**.
The two least significant digits represent the probe number.

Digit			*Meaning*
3	**2**	**1**	
	–	–	Probe number (two digits)
0			Multi probe
1			Mono probe

© Siemens AG, 2004. All rights reserved

Example of workpiece measurement:

_PRNUM = 102
→ Probe type: Mono probe
→ Probe number: 2
→ Array index: _WP[1,n]

Example of tool measurement:

_PRNUM = 3
→ Probe number: 3
→ Array index: _TP[2,n]

2.3.14 Empirical value, mean value: _EVNUM

Function

The effect of empirical and mean values is described in Sections 1.8 and 1.9.

The empirical values and mean values are stored in data block (GUD5) in arrays

- **_EV[ ] empirical values** and
- **_MV[ ] mean values.**

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.

The number of existing empirical and mean values is entered in data block (GUD6) **_EVMVNUM[m,n]**.

- m: array dimension _EV[m]
- n: array dimension _MV[n]

The default setting provides 20 values each (array index _EV, _MV: 0...19).

Parameters

Values of _EVNUM:

= 0: without empirical value, without mean value memory

>0 to 9999: Empirical value memory number = mean value memory number

>9999: the higher 4 digits of _EVNUM are interpreted as the **mean value memory number**, the lower 4 digits as the **empirical value memory number**.

The array index for _EV and _MV is formed from the value in _EVNUM.

© Siemens AG, 2004. All rights reserved

Example:
_EVNUM = 11
→ EV memory: 11 → _EV[10]
→ MV memory: 11 → _MV[10]
_EVNUM = 90012
→ EV memory: 12 → _EV[11]
→ MV memory: 9 → _MV[8]

2.3.15 Multiple measurement at the same location: _NMSP

Parameters

Parameter **_NMSP** can be used to determine the number of measurements at the same location.
The measured values or distances-to-go Si (**I**=1…n) are averaged.
That results, for example, in distance-to-go D:

$$D = \frac{S_1 + S_2 + \ldots S_n}{n}$$;n: Number of measurements

2.3.16 Weighting factor for mean value calculation: _K

Function

The parameter for weighting factor **_K** can be applied to allow different weighting to be given to an individual measurement.

A detailed description is given in Section 1.8 "Measuring strategy and compensation value definition".

■

© Siemens AG, 2004. All rights reserved

Notes

© Siemens AG, 2004. All rights reserved

Measuring Cycle Auxiliary Programs

© Siemens AG, 2004. All rights reserved

3.1 Measuring cycle subroutines

3.1.1 Overview

Function

These measuring cycle subroutines are called directly by the cycles. With the exception of CYCLE100, CYCLE101, and CYCLE116, these subroutines cannot be executed by a direct call.

Programming

Cycle	*Function*	
CYCLE100	Activate logging	
CYCLE101	Deactivate logging	
CYCLE102	Measuring result display	
CYCLE103	Parameter setting in interactive mode	only up to measuring cycles – SW 4.5
CYCLE104	Internal subroutine: Measuring cycle interface	
CYCLE105	Internal subroutine: Log	
CYCLE106	Internal subroutine: Log	
CYCLE107	Output of measuring cycle messages	only up to measuring cycles – SW 6.2
CYCLE108	Output of measuring cycle alarms	only up to measuring cycles – SW 6.2
CYCLE109	Internal subroutine: Data transfer	
CYCLE110	Internal subroutine: Plausibility checks	
CYCLE111	Internal subroutine: Measuring functions	
CYCLE112	Internal subroutine: Measuring functions	
CYCLE113	Internal subroutine: Log	
CYCLE114	Internal subroutine: Load ZO memory, Load tool offset	
	Internal subroutine: Load tool offset	
CYCLE115	Internal subroutine: Load ZO memory	
CYCLE116	Calculation of the center point and radius on a circle	
CYCLE117	Internal subroutine: Prepositioning	
CYCLE118	Internal subroutine: Log	
CYCLE119	Internal subroutine: Arithmetic cycle for determining position in space	from meas. cycles – SW 6.3

© Siemens AG, 2004. All rights reserved

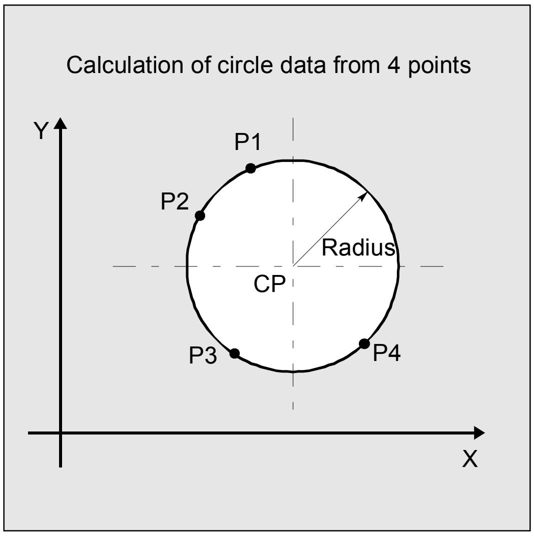
3.1.2 CYCLE116: Calculation of center point and radius of a circle

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.

An array of REAL variables of length 13 must be transferred as the parameter.



Programming

CYCLE116 (_DATE, _ALM)

Parameters

Input data

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

© Siemens AG, 2004. All rights reserved

Output data

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

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

This cycle is called as a subroutine by, for example, measuring cycle CYCLE979.

Example:

%_N_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 _DATE[10]=0 _DATE[11]=10 _DATE[12]=0 _ALM=0	
`M0`			
`STOPRE`			
`M30`			

© Siemens AG, 2004. All rights reserved

3.2 Measuring cycle user programs

Function

Measuring cycle user programs CYCLE198 and CYCLE199 are called in the measuring cycles and can be used to program necessary adjustments before or after a measurement (e.g. activate probe, position spindle).

3.2.1 CYCLE198: User program before measurement

Explanation

CYCLE198 is called at the beginning of each measuring cycle. It can be used to program actions necessary before starting a measurement (e.g. activate probe).
In the as-delivered state this cycle only contains one CASE statement that executes a jump to a label with subsequent M17 (end of subroutine) for each measuring cycle.

e.g.: **_M977**: ;before measurement in CYCLE**977**
M17 ;end of cycle

From this label all actions to be executed on each CYCLE977 call must be programmed.

3.2.2 CYCLE199: User program after measurement

Explanation

CYCLE199 is called in each measuring cycle when measurement is complete. It can be used to program actions necessary following completion of a measurement (e.g. deactivate probe).
The internal structure of the cycle is that same as that of CYCLE198, i.e. the program lines must be inserted between the label for a particular cycle and M17 (end of subroutine).

© Siemens AG, 2004. All rights reserved

3.3 Package structure of the 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!

Please refer to the notes on installation in Chapter 10.

© Siemens AG, 2004. All rights reserved

Subpackages

In many applications, not all the measuring cycles are used on one machine. Instead subpackages are used. The following overview shows which subpackages are advisable and executable. This will help you to save memory.

Basic package

Milling measuring cycles

Calibrate tool probe + measure tool	Calibrate workpiece probe + measure tool + write ZO or TC	
CYCLE971 CYCLE107[1] CYCLE108[1] CYCLE109 CYCLE110 CYCLE111 CYCLE198 CYCLE199	 CYCLE961 CYCLE976 CYCLE977 CYCLE978 CYCLE979 CYCLE998 CYCLE997[2]	CYCLE107[1] CYCLE108[1] CYCLE109 CYCLE110 CYCLE111 CYCLE112 CYCLE114 CYCLE115 CYCLE116 CYCLE119[2] CYCLE198 CYCLE199

Basic package

Turning measuring cycles

Calibrate tool probe + measure tool	Calibrate workpiece probe + measure tool + write ZO or TC	
CYCLE972 CYCLE982 CYCLE107[1] CYCLE108[1] CYCLE109 CYCLE110 CYCLE111 CYCLE198 CYCLE199	CYCLE973 CYCLE974 CYCLE994	CYCLE107[1] CYCLE108[1] CYCLE109 CYCLE110 CYCLE111 CYCLE114 CYCLE115 CYCLE117 CYCLE198 CYCLE199

Additional package
Measuring at milling machine in JOG mode

Semi-automatic calibration of tool probe + Measure tool + Calculation and setting of reference points

E_MS_CAL	E_MS_POC[2]	E_MT_CAL	CYC_JM
E_MS_CAN	E_MS_SPI[2]	E_MT_LEN	CYC_JMC
E_MS_HOL		E_MT_RAD	CYC_JMA[2]
E_MS_PIN		E_SP_NPV	

+

Additional package
Operator interface

Measurement result display selection

Activation: _CHBIT[10]=1

CYCLE102
CYCLE104

+

Additional package
Operator interface

Log in a file of the part program memory

Activation: CYCLE100

CYCLE100
CYCLE101
CYCLE105
CYCLE106
CYCLE113
CYCLE118

1) As from measuring cycle SW 6.2
2) Only up to measuring cycle SW 6.3

■

© Siemens AG, 2004. All rights reserved

Notes

© Siemens AG, 2004. All rights reserved

Measuring in JOG

© Siemens AG, 2004. All rights reserved

4.1 Overview function, procedure, prerequisites

Function

There are two types of measurement: **Workpiece measurement** and **tool measurement**. Measurement may be

- **Automatically** executed
  (cycles for automatic operation: refer to Chapter 5 for milling and Chapter 6 for turning) or
- **semi-automatic** in the JOG mode.

This chapter describes semi-automatic measurement for milling technology: **Measuring in JOG**.

Workpiece measurements may include:

- Calibrating the workpiece probe,
- Measuring the contour elements on a workpiece (edge, corner, hole, spigot, rectangle) and then aligning the workpiece by determining and setting a zero offset.

Tool measurements may include:

- Calibrating the tool probe,
- Determining the tool length or radius of milling tools, or tool length of drills and then setting the appropriate offset in the tool offset memory.

General sequence

The required function is selected via softkeys on the HMI in JOG mode. Input displays are shown. These have to be completed via screen forms. This describes the measurement tasks and selects the offset.
The user must put the tool or probe into a permissible starting position for the measurement task, e.g. using the traverse keys or handwheel (manual traverse).
Once you have pressed the "NC start" key in JOG mode, the remaining sequence is automatic.
To cancel measurement, press "RESET".

Notice
Be sure to select the correct channel!
The "Measure in JOG" function is **channel-specific**.

Calling CYCLE198, CYCLE199
CYCLE198 and CYCLE199 are called before and after a measurement, see Section 3.2.
From meas. cycles SW 6.3:
In addition, CYCLE199 is called on "Measure in JOG" after completion of complex measurements with several cycle calls, e.g. after "Center calculation from 3 holes/spigots with offset". CYCLE199 contains labels specially for the various complex measurements.

© Siemens AG, 2004. All rights reserved

Users can program their own special actions after the label in question.

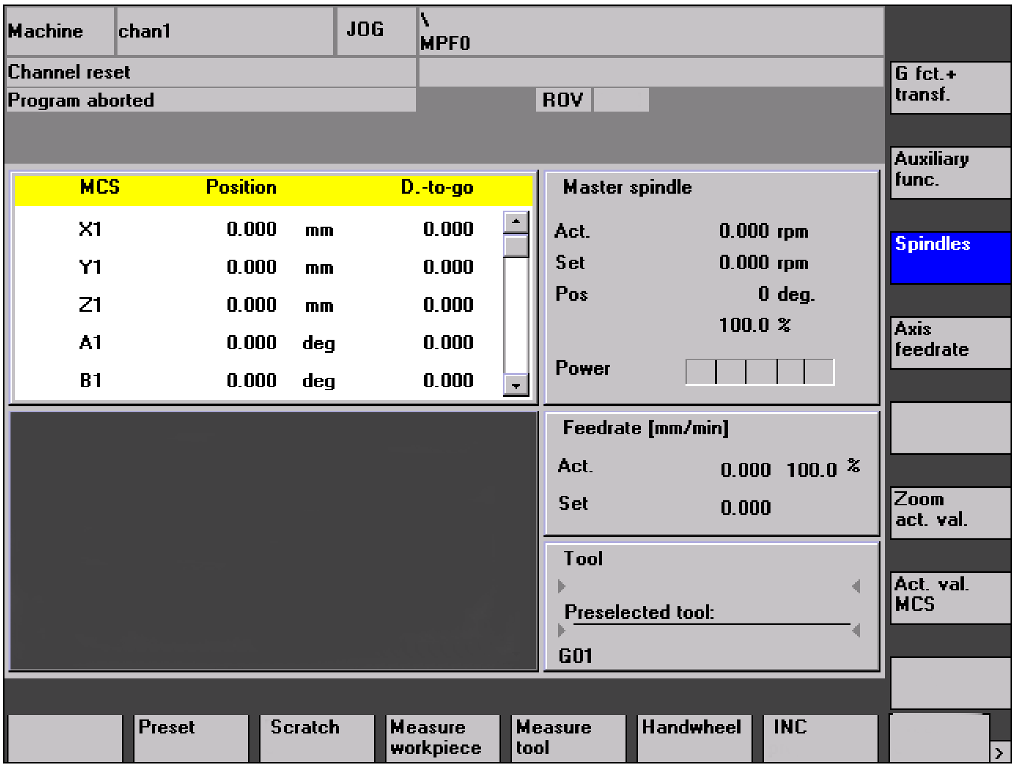
Entry for "Measure in JOG" and selection is performed via the softkeys

Measure workpiece (calculating and setting workpiece reference points or calibration of the workpiece probe)

Or

Measure tool (Measurement of milling or drilling tools or calibration of the tool probe)

in the basic display of JOG mode:



Prerequisites

The prerequisites for "Measuring in JOG" are detailed in Part 2 "Function description" (Chapter 8 ff.).

The following checklist is useful for determining whether the preconditions are fulfilled:

Machine

- All machine axes are designed in accordance with DIN 66217.
- All 3 geometry axes are present according to MD 20050 AXCONF_GEOAX_ASSIGN_TAB.
- Reference point approach was completed in all the required machine axes.

- A touch-trigger workpiece probe (3D) is provided for acquiring workpiece dimensions, and a touch-trigger tool probe for acquiring tool dimensions.

Control
810D, 840D, 840Di with HMI Advanced,
with necessary **option**:
"Inter-modal actions"
(ASUP's and synchronized actions in all modes)

General machine data for machine cycle runs
These machine data are described in Section 9.1.

Special machine data and other data for measuring in JOG
These special data and settings are described in Section 9.3.

Interrupt for starting the ASUP's
As from PLC-SW 6.1, interrupt number 9, previously 8, applies.
This interrupt must not be used by any other application during "measuring in JOG".

Data blocks and measuring cycles in general

- The data blocks:
  GUD5.DEF and
  GUD6.DEF
  in directory DEFINE of the software supplies has been loaded into the control and activated (directory "Definitions" in the file system).
- The measuring cycles in directory CYCLES of the software supplied have been loaded into the standard cycle directory of the control and a power-on has been performed.

Special files for measuring in JOG

- All files in directory JOG_MEAS\CYCLES of the software supplied have been loaded into the control via "Data in" and a power-on has been performed.
- Data block GUD7_MC.DEF in directory JOG_MEAS\DEFINE of the software supplied has been adapted and loaded into the control.
  Data block GUD7.DEF has been activated.

Detailed function description:
see Section 9.3 and information on upgrading Section 10.2 and in file SIEMENSD.TXT of the software supplied.

© Siemens AG, 2004. All rights reserved

4.2 Measuring workpieces (up to measuring cycles SW 6.2)

Introduction

From measuring cycles SW 6.3 onwards, the functional scope has been extended for workpiece measurement. This is described in Section 4.3. The following description only applies up to SW 6.2.

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. During the runtime of the measuring cycle, the ZO is active that is defined via a data block (GUD6) in the field and the work plane G17 … G19 is active that is set in the data block (GUD6) in another field. A field of the data block (GUD6) also defines which array is assigned to the probe in the spindle. The switching behavior parameters determined by probe calibration are stored in this array.
All the measuring points required for the measurement task are approached. Prepositioning can either be performed manually or in a program. Measurement is performed in the workpiece coordinate system (WCS).

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/system frame "set zero point" ($P_SETFRAME) 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 in the spindle, the tool offset for it has been selected and it has been calibrated.

© Siemens AG, 2004. All rights reserved

4.2.1 Sequence of operation and functions for workpiece measurement (up to measuring cycles SW 6.2)

Sequence

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 >
Spigots >
Probe calibration
<< Back

3. Selection in the screen form:

- Select the zero offset to which the specified setpoint position refers and for which the offset is to apply:
  – Off (just measure, no offset)
  – Settable zero offset G54, ...G57, ...
  – Basic frame/basis
  Up to measuring cycles SW 6.02.20, selection of "basis" puts the offset in the last channel-specific basic frame acc. to MD 28081: MM_NUM_BASE_FRAMES. The offset is calculated to have the right effect when G500 is activated. The corresponding basic frame then be active.
  From measuring cycles SW 6.02.20, it is possible to correct in the system frame "set zero" ($P_SETFRAME). This function must be activated via a setting data in data block GUD6:
  _JM_B[4] =1 : Selecting "Basis" means putting the offset into the system frame "set zero". G500 must be active during measurement.
  _JM_B[4] =0 : Selecting "basic frame" means offset into the last channel-specific basic frame acc. to MD 28081.
- 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", measurement runs with a preset measuring feed.
   The measuring probe is triggered. For example:
   During "measure edge" or "measure corner", the probe is automatically rapid-traverse retracted to its starting position for the measuring point. When a hole or spigot is measured, 4 points are automatically probed one after the other.

© Siemens AG, 2004. All rights reserved

The translation offset and also an offset for the rotation around the infeed axis is applied based on the measuring results and the specified setpoint position when determining the corner defined for the selected ZO. Selecting the basic frame always puts the offset in the last channel-specific basic frame according to
MD 28081: MM_NUM_BASE_FRAMES, if several are available.

Sequence for "measuring in JOG with active TRAORI"

The subsequent description refers to the following applications:

- Measuring in JOG, with active TRAORI and with the measuring probe "applied"[1)].
- Measuring in JOG, in the rotated (swiveled) planes

1. TRAORI is active
2. If a workpiece is measured and the measuring probe is not in the basic (initial) setting of the machine kinematics, then this must be communicated to the NCU as rotation in the active zero offset (ZO).
   Example 1: The workpiece cannot be clamped to the machine with the longitudinal and transverse sides. In this case, the workpiece coordinate system (WCS) must be rotated around Z through 90 degrees.
   Entry in the active ZO, rotation around Z.
   Example 2: The whole to be measured is located on a vertical side of the workpiece. In this case, the WCS must be rotated around X or Y through 90 degrees. Entry in the active ZO rotation around X or Y.
3. Aligning the workpiece coordinate system (WCS) in the tool direction in MDA:
   G0 C3=1 (valid for G17 plane).
4. With NC start, traversing motion is initiated and the tool aligns itself in the rotated WCS in the Z axis.
   The vertical orientation to the XY plane is kept.
5. The edge, corner,… is approached
   If the edge, corner … cannot be completely reached with the machine axes, the measuring probe can be advanced using the rotary axes.
   In this case, the WCS is not updated.
6. Start the measuring process!

Note

An "applied" measuring probe has an angular offset regarding the tool orientation in the feed axis Z assumed using the NC command C3=1.

4.2.2 Calibrating workpiece probe (up to measuring cycles SW 6.2)

Function

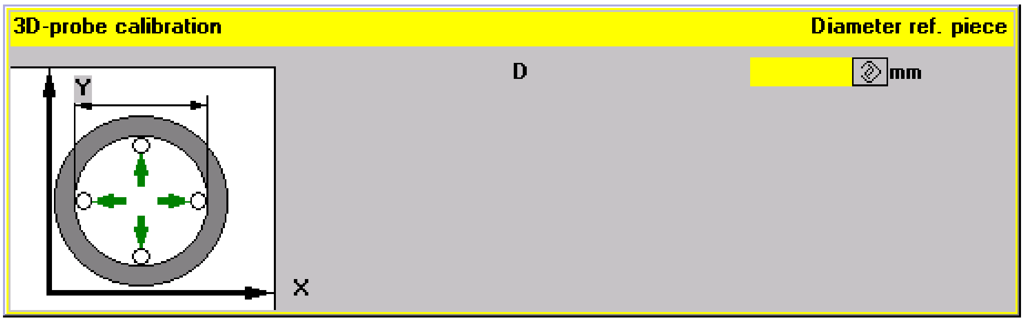
With milling machines and machining centers, the probe is usually loaded into the spindle from a tool magazine. This may result in errors when further measurements are taken on account of probe clamping tolerances in the spindle.

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".

© Siemens AG, 2004. All rights reserved

Calibrating the workpiece probe in any hole (radius)

This cycle permits probe calibration in any hole of a reference part meeting the geometrical accuracy and surface quality requirements, 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 data block (GUD6).



Operating sequence

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

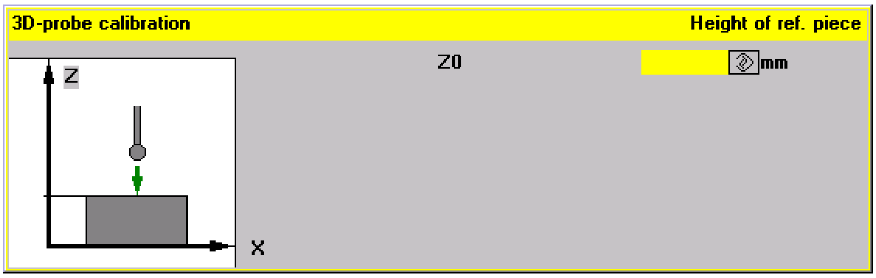
Enter diameter ∅ of the reference part (here: adjustment ring).

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.

© Siemens AG, 2004. All rights reserved

Calibrating a workpiece probe on any surface

With this measuring cycle you can calibrate the probe in the infeed axis on any surface, e.g. on the workpiece, to determine the length.



Operating sequence

Precondition

The measuring probe is located in the spindle. The precise radius of the probe ball must be entered in the tool offset block. The precise value of the height of the calibration surface of the active zero offset set with a date in block GUD6 must be known!

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 point of the calibration surface relative to the active zero offset set in the data block (GUD6) during measurement.

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

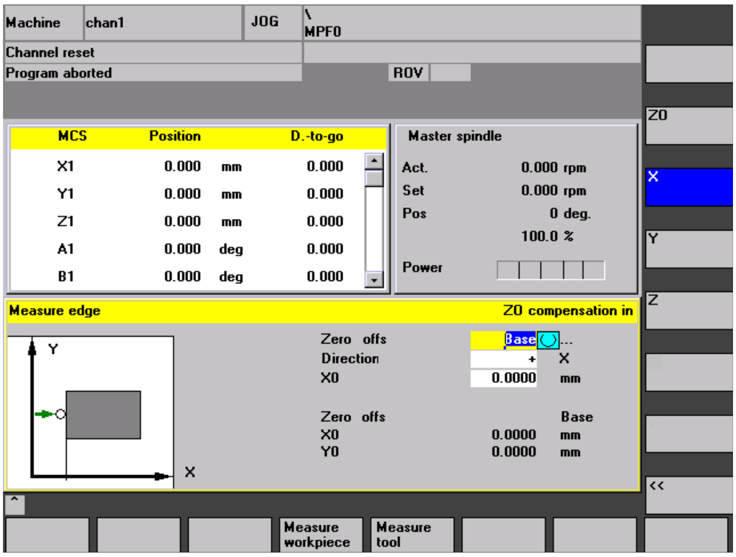
The calculated length of the probe is written to the tool offset data block.

© Siemens AG, 2004. All rights reserved

4.2.3 Measuring an edge (up to measuring cycles SW 6.2)

Function

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



Operating sequence

Precondition

The 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:
  – Off (just measure, no offset)
  – Basic frame/basis
  – 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!

© Siemens AG, 2004. All rights reserved

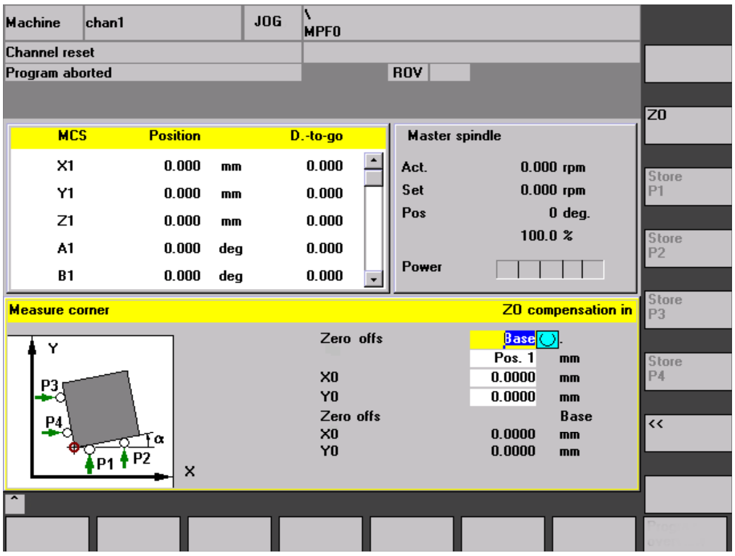
On "NC Start", the measuring operation is automatically performed with a measuring feedrate set via a data block (GUD).

- Probe triggered
- Automatic retraction to starting position in rapid traverse.
- The translation component of the selected ZO is applied based on the measuring results and the specified setpoint position. Selecting the basic frame always puts the offset in the last channel-specific basic frame according to MD 28081: MM_NUM_BASE_FRAMES, if several are available. The offset is applied in the coarse offset, and any fine offset is reset.

4.2.4 Measuring a corner (up to measuring cycles SW 6.2)

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.



Operating sequence

Precondition

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

Approach the workpiece

Position the probe at a selected corner of the workpiece.

© Siemens AG, 2004. All rights reserved

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:
  – Off (just measure, no offset)
  – Basic frame/basis
  – 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 a data block (GUD).

- Probe 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. When the basic frame is selected, the offset is always made in the last channel-specific basic frame according to MD 28081: MM_NUM_BASE_FRAMES, if several are available. The offset is applied in the coarse offset, and any fine offset is reset.

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

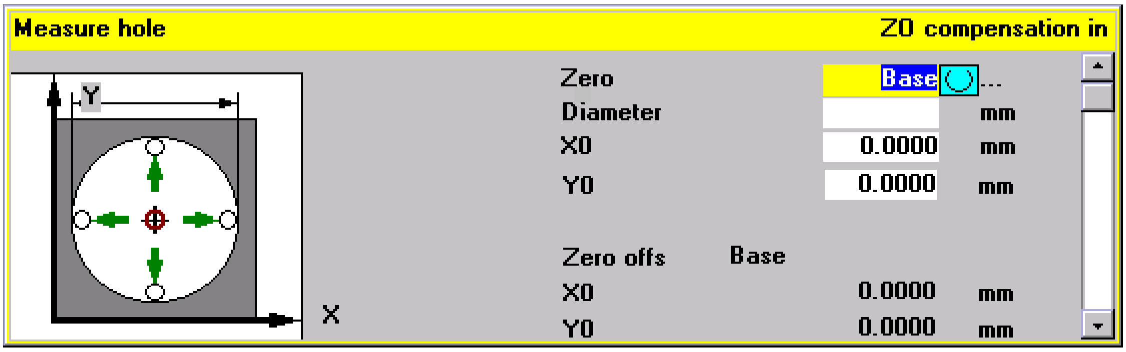
On a rectangular workpiece, three sampling points are sufficient for the calculation.

© Siemens AG, 2004. All rights reserved

4.2.5 Measuring a hole (up to measuring cycles SW 6.2)

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.



Operating sequence

Precondition

The 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:
  – Off (just measure, no offset)
  – Basic frame/basis
  – 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.

© Siemens AG, 2004. All rights reserved

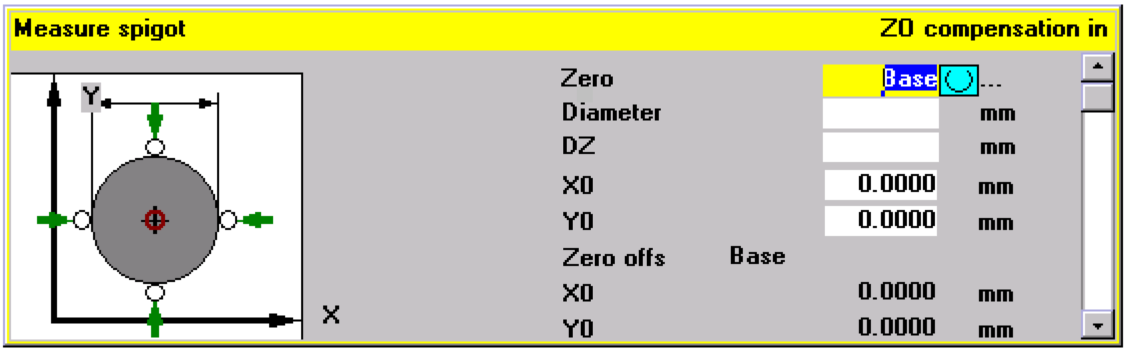
Set the feedrate override switch to the same value as for calibration!

Measurement is performed automatically as soon as you press "NC Start". One after the other, the probe samples 4 points on the inner surface of the hole. Once the measurement is complete, the translation offset is determined for the selected zero offset. Selecting the basic frame always puts the offset in the last channel-specific basic frame according to MD 28081: MM_NUM_BASE_FRAMES, if several are available. The offset is applied in the coarse offset, and any fine offset is reset.

4.2.6 Measuring a spigot (up to measuring cycles SW 6.2)

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.



Operating sequence

Precondition

The 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

© Siemens AG, 2004. All rights reserved

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:
  – Off (just measure, no offset)
  – Basic frame/basis
  – 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 depth**.

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 4 points on the outside of the spigot.

Once the measurement is complete, the translation offset is determined for the selected zero offset. Selecting the basic frame always puts the offset in the last channel-specific basic frame according to
MD 28081: MM_NUM_BASE_FRAMES, if several are available. The offset is applied in the coarse offset, and any fine offset is reset.

© Siemens AG, 2004. All rights reserved

4.3 Workpiece measurement (from measuring cycles SW 6.3)

4.3.1 Overview of function and sequence

Introduction

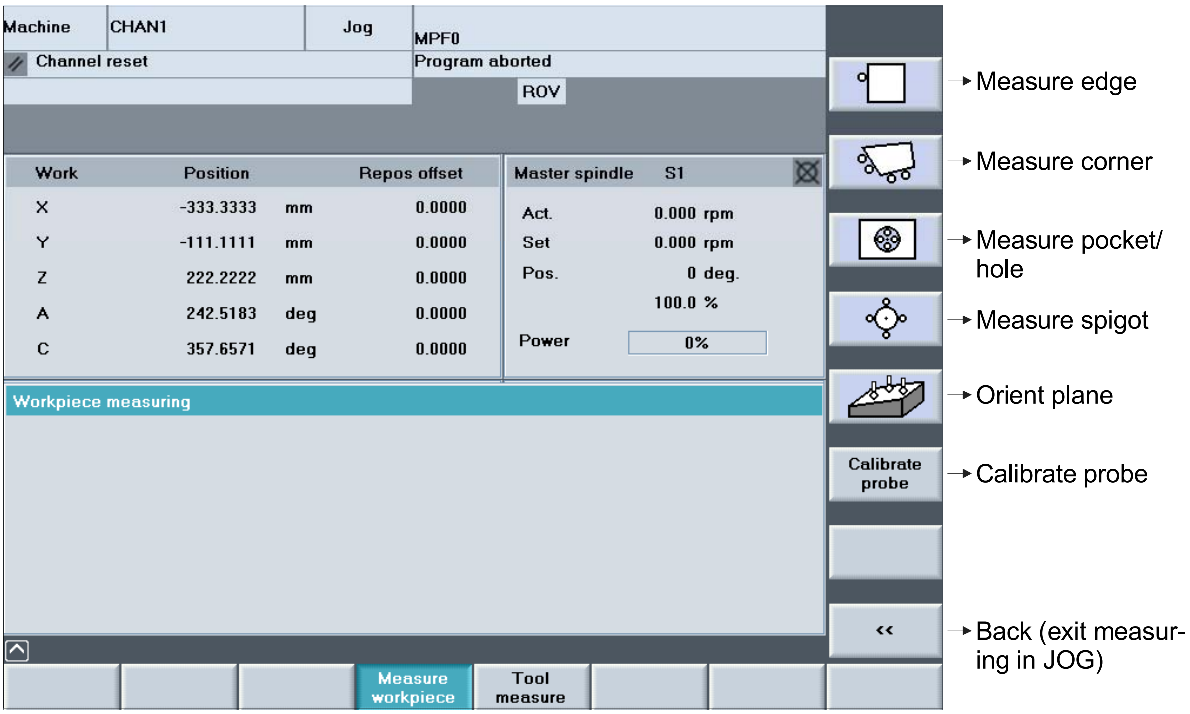
From measuring cycles SW 6.3 onwards, the functional scope has been extended for workpiece measurement. This is described in a separate chapter.

Function

The "measure workpiece" function permits set-up of a workpiece clamped onto a machine table. Reference points on the workpiece are measured by means of workpiece probes, and necessary ZO compensation is calculated and set.
For complete set-up, more than one call may be necessary (several axes, rotated clamping, see Section "Cascaded measurement").
The workpiece probe can be calibrated.

Sequence

1. The workpiece is clamped; the probe is in the spindle.
2. Pressing the "Measure workpiece" softkey displays the following softkey bar for selection:



© Siemens AG, 2004. All rights reserved

3. The measurement task is selected by pressing the relevant softkeys.
4. The relevant input displays are shown. The input screen forms must be filled out.
5. Select the zero offset (ZO) to which the specified setpoint position refers and for which the offset is to apply.
6. Input of setpoints
   - e.g. approximate diameter for hole/spigot
   - e.g. specifying the set positions in the meas. axis (at edge)
   - e.g. specifying the center point (for hole/spigot) or the corner point
7. Select axis and axis direction for edge/corner.
8. For "NC start", measurement is started and completed with a preset measurement feed. The first traversing motion is used to correct the mechanical slope of the measuring probe, determined when calibrating, and is automatically executed by the system!

Internal sequence

After pressing "NC start", an automatically generated ASUB **(/_N_SPF_DIR/_N_JM_MESS_SPF)** is started to call up a measuring cycle with parameters, supplied from entries made in screen forms. This measuring cycle generates the measurement paths and intermediate positions as a function of the specified setpoints.
While the measuring cycle is running, the following settings, defined in the datablock GUD6, in the field _JM_I[ ] are effective:

- Zero offset
- Working plane (G17...G19)
- the data field _WP[ ] assigned to the active measuring probe

The switching behavior parameters of the probe determined by calculating the probe are stored in this data field (data array) _WP[ ].

Zero offset

The translation offset and/or an offset for the rotation around the infeed axis in the selected ZO is applied based on the measuring results and the specified setpoint position, depending on the measurement task.
With the default setting (_JM_I[5]=0), the following offsets are possible:

- "Just measure"
  (no ZO correction, just display measurement results)
- "G54...G57, ..."
  (offset in the settable ZO)
- "Basis reference"
  (offset in the system frame "set zero point" $P_SETFRAME)

© Siemens AG, 2004. All rights reserved

In the extended correction selection (_JM_I[5]=1), the following options are available:

- Correction in any global basic frame of the frame chain active during measurement.
  Only possible if no rotation is measured:
  e.g. during "measure edge", "measure distance", "measure hole", and "measure spigot".
- Correction in any channel-specific basic frame of the frame chain active during measurement.

"Function interface of the measuring point softkeys (P1...P4)", in the PLC

Function

This functionality includes the transfer of the status "n. measuring point completed and measure value saved" at the PLC interface. This has the same significance as activating a corresponding, vertical measuring point softkey (P1 … P4) after a successful measurement in one of the workpiece measuring variants from measuring in JOG.
The function "reject n. measurement → n. measuring point inactive" can be controlled from the PLC interface – equivalent to the same operator sequence at the HMI when measuring in JOG.

Description of the PLC user interface

- Signal flow: PLC → measuring in JOG (HMI)

DB19.DB43 Bit0=1:	Switches softkey "1st measuring point" inactive --> reject measurement, The signal must be displayed on the HMI until DB19.DB43 bit4=0
DB19.DB43 bit0=1:	Switches softkey "2nd measuring point" inactive --> reject measurement, The signal must be displayed on the HMI until DB19.DB43 bit5=0
DB19.DB43 bit2=1:	Switches softkey "3rd measuring point" inactive --> reject measurement, The signal must be displayed on the HMI until DB19.DB43 bit6=0
DB19.DB43 bit3=1:	Switches softkey "4th measuring point" inactive --> reject measurement, The signal must be displayed on the HMI until DB19.DB43 bit7=0

- Signal flow: Measuring in JOG (HMI) → PLC

DB19.DB43 bit4=1:	If softkey "1st measuring point" active and can be seen on the screen, Measuring 1st measuring point completed and measured value saved
DB19.DB43 bit5=1:	If SK "2nd measuring point" active and can be seen on the screen, Measuring 2nd measuring point completed and measured value saved
DB19.DB43 bit6=1:	If softkey "3rd measuring point" active and can be seen on the screen, Measuring 3rd measuring point completed and measured value saved
DB19.DB43 bit7=1:	If softkey "4th measuring point" active and can be seen on the screen, Measuring 4th measuring point completed and measured value saved

© Siemens AG, 2004. All rights reserved

Example

The functionality shown can be used with a Siemens handheld unit (HHU) – when a measuring point has been successfully completed, an LED on the HHU keyboard is lit. The HHU key, assigned to this LED can then act from the sense as rejecting a measuring point.
The PLC user interface signals should be interlocked with the input and output signals of the HHU using the functionality of the PLc user program.
The use of an HHU should allow the process-related control of a measuring operation.

Sequence for "measuring in JOG with active TRAORI"

The subsequent description refers to the following applications:

- Measuring in JOG, with active TRAORI and applied measuring probe.
- Measuring in JOG, in the rotated (swiveled) planes

The sequence corresponds to that described in Subsection 4.2.1!

From measuring cycles SW 6.3 onwards, the descriptive points 2. and 3. can also be executed with the functionality of "swivel in JOG".

Note

If the "swiveling" or 5-axis transformation (TRAORI) are used to align the measuring probe, then these must also be appropriately set-up in the machine!
Please carefully observe the information provided by the machinery manufacturer!

Note

In general:
The translation offset is applied in the coarse offset. Any fine offset is reset.

© Siemens AG, 2004. All rights reserved

4.3.2 Calibrating workpiece probes (from measuring cycles SW 6.3)

Function

On milling machines and machining centers, the workpiece probe is inserted in the spindle. This can lead to measuring errors caused by the clamping tolerances of the probe in the spindle. The switching points (trigger points) of the measuring probe with reference to the spindle center must therefore be determined for each new probe inserted.
This is achieved with calibration by which the probe is calibrated in a hole or on a surface.

Precondition
The workpiece probe is the active tool in the spindle with activated tool offsets. The approximate length and the radius of the measuring probe sphere must be entered in the tool data.
An SPOS-capable spindle is required to calibrate the radius of the measuring probe sphere.

Note
The feedrate override should have the same value during calibration and measurement to reduce measurement inaccuracies.

Operation
After you have pressed the "probe calibration" softkey, a display image is shown, containing the following options:

- "Length" (calibrating the length of the probe at the surface)
- "Radius" (calibrating the radius of the measuring probe sphere in the hole)

The reference dimension for the diameter of the calibration hole and the height of the calibration surface to be specified can be pre-assigned using the following GUD parameters:

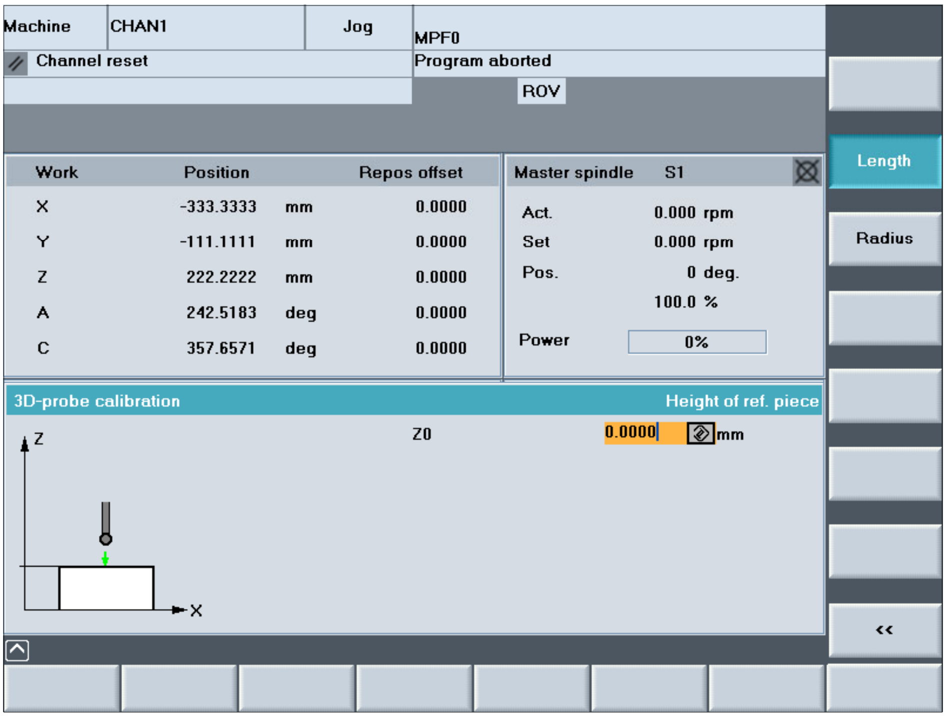
- E_MESS_CAL_D → diameter, calibration hole (calibration ring)
- E_MESS_CAL_L → calibration height in the feed axis (refer to WCS)

Calibrating probe length

By selecting "length", it is possible to calibrate the probe in the infeed axis on a suitable and precisely known surface, e.g. on the workpiece. The precise length 1 (L1) of the prove is also determined and entered in the tool offset memory.

Note the setting of _CBIT[14]:
_CBIT[14]=0: L1 referred to ball center
_CBIT[14]=1: L1 ref. to ball circumference

© Siemens AG, 2004. All rights reserved



Approaching the calibration surface

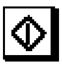
The probe must be positioned opposite to the calibration surface of the reference or workpiece.

Enter details in input form

Enter the known dimension (workpiece coordinate) of the calibration surface, referred to the WCS active when calibrating.

Note the settings in data block GUD6:

- Active zero offset during measurement: Variable _JM_I_[4]).
- Active working plane during measurement: Variable _JM_I_[3].

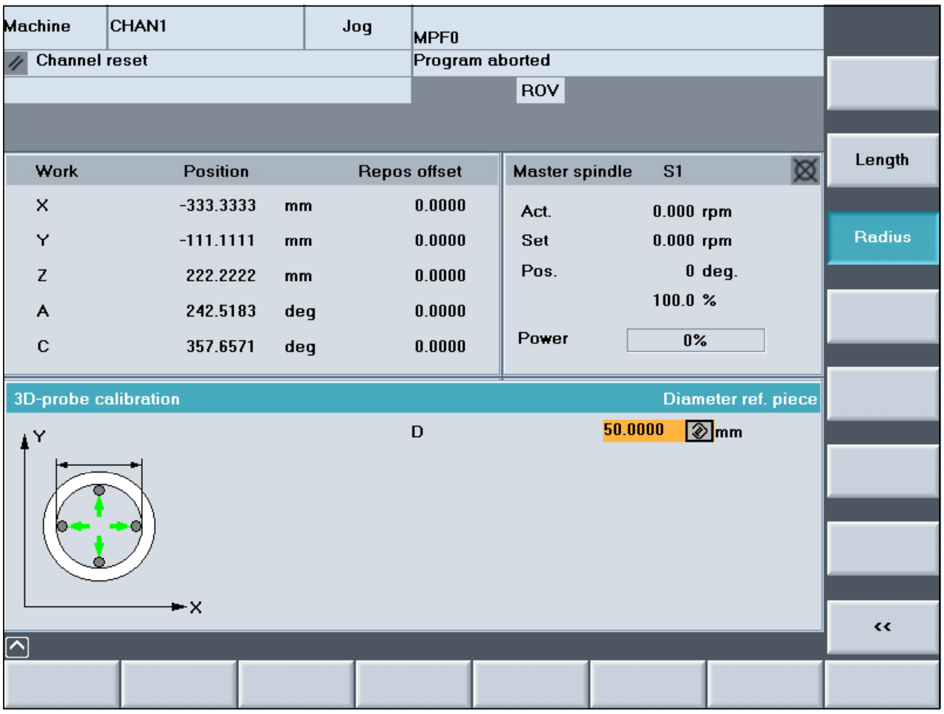


Calibration is performed automatically as soon as you press "NC Start". Trigger value and length offset L1 are stored.

Calibrating probe radius

With the "radius" selection, the probe can be calibrated in the axes of the working plane in a suitable hole (geometrical accuracy, low surface roughness) and precisely known diameter or in calibration ring.

© Siemens AG, 2004. All rights reserved



Approaching the calibration ring
Position the probe roughly in the hole center and at calibration depth.

Enter details in input form
Enter the known dimension of the calibration ring diameter.

Note the settings in data block GUD6:

- Active zero offset during measurement: Variable _JM_I_[4].
- Active working plane during measurement: Variable _JM_I_[3].

Calibration is performed automatically as soon as you press "NC Start".
The precise position of the hole center is determined first. After that, in both axes of the plane and in each of the two directions, the 4 switching points within the hole are determined and stored as trigger values in the probe's array.

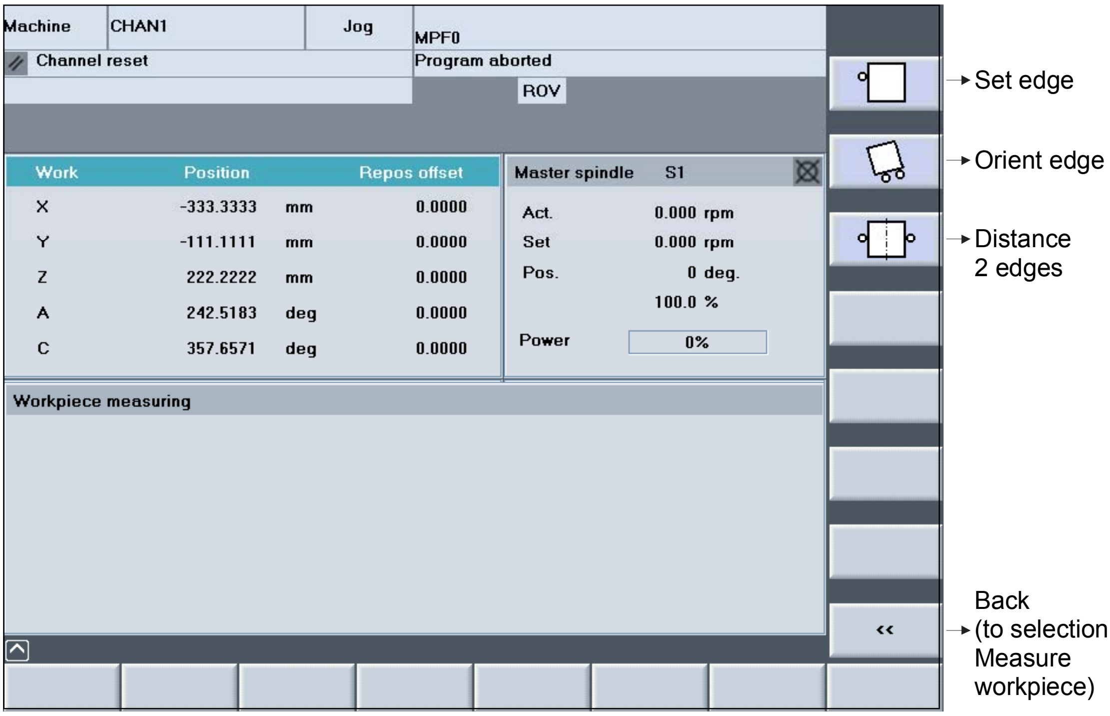
© Siemens AG, 2004. All rights reserved

4.3.3 Measuring the edge (from measuring cycles SW 6.3)

Function

After selecting "edge", a selection display is shown containing the following options:

- "Set edge"
- "Orient edge"
- "Distance 2 edges"



Precondition

The workpiece probe is the active tool with activated tool offsets in the spindle and is already calibrated (refer to Subsection 4.3.2).

Note:

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

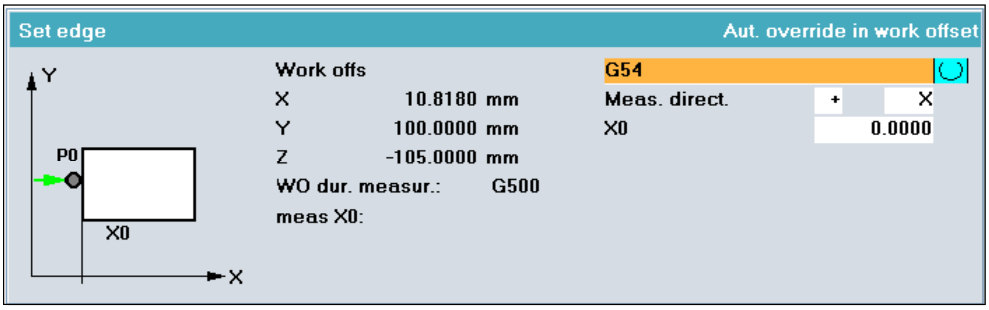
Setting the edge

This enable measurement of a reference point (paraxial edge) on a workpiece in one of the axes X, Y, or Z and setting as ZO (translation).

Approach the workpiece

Position the probe in front of the edge.

© Siemens AG, 2004. All rights reserved



Enter details in input form

- Select of the ZO
- Select an axis and measuring direction
- Enter the set position required for the reference point (edge) for the selected ZO.

Measurement is performed automatically with the set measuring feed as soon as you press "NC Start".
Measurement is followed by rapid-traverse retraction to the starting position. The offset is then automatically calculated. If the selected offset is the active ZO, then this is automatically activated.

Display and correction
After successful offset calculation, the frame components of the selected ZO are re-displayed along with the measured position of the edge in the measuring axis.
ZO correction is calculated in such a way that the measured edge assumes the required setpoint after activation of the selected ZO.

Notes

- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

Orienting the edge

By measuring two points on a straight workpiece edge, it is possible to determine the angle position of this edge with respect to a reference axis.
Orientation of the workpiece edge is possible by

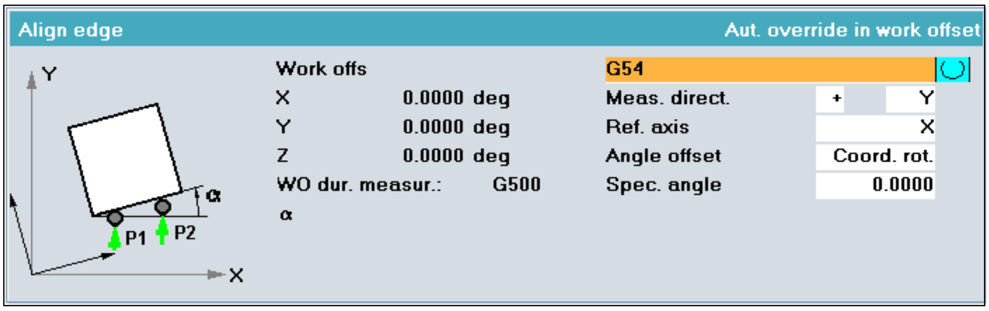
- "Coordinate rotation" or
- Rotating the workpiece using a rotary table (specifying the rotary axis).

2 measuring points are possible, P1 and P2.

© Siemens AG, 2004. All rights reserved

Approach the workpiece

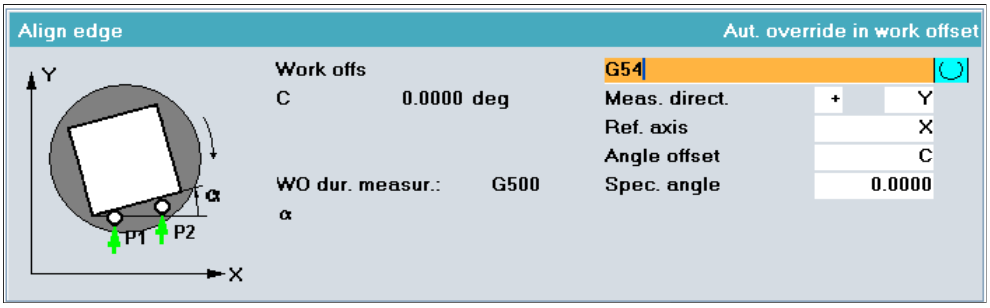
Position the probe in front of the edge, first and P1 and after completion of this measurement at P2.



Enter details in input form

- Select of the ZO
- Select axis X, Y, or Z and measurement direction
- Select reference axis for the angle
- Select the required angle offset: Select "coordinate rotation" or the name of the rotary axis
- When entering the "set angle", an orientation deviating from 0 degrees is possible.

Orienting the edge, rotating with rotary table:



Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start".
Measurement is followed by rapid-traverse retraction to the starting position.
After successful measurement, the measured value is stored internally and the "P1 stored" softkey that was previously switched inactive is activated.

After manual positioning at measuring point **P2** and with softkey "P1 stored" switched active, pressing "NC start" starts measurement of the 2nd measuring point in the selected measuring axis. After successful measurement, softkey "P2 stored" is activated, and the 2nd measured value is stored. A vertical softkey – "Calculate" is then displayed. After this softkey is pressed, then angle between the edge and the reference axis and the selected contour calculation is calculated.

© Siemens AG, 2004. All rights reserved

When the "coordinate rotation" angular offset (correction) is selected, then the offset is also activated if it is the active ZO.

When the "rotary axis" angular offset is selected (name of the rotary axis), a help function for the user is displayed to activate the offset (refer to Subsection 4.3.9).

Display and correction

After offset calculation has been completed, the frame components of the selected ZO are redisplayed along with the measured "alpha" angle.

In the case of ZO correction by "rotation of the workpiece coordinate system", the selected ZO is calculated such that the measured edge forms the required angle (set angle) with the selected reference axis after activation of the selected ZO.

In the case of ZO correction in a specified rotary axis, the difference in angle between the measured and desired angle is entered in the selected ZO (translation) of the specified rotary axis.
No check is made for correct assignment of the specified rotary axis!

Notes

- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

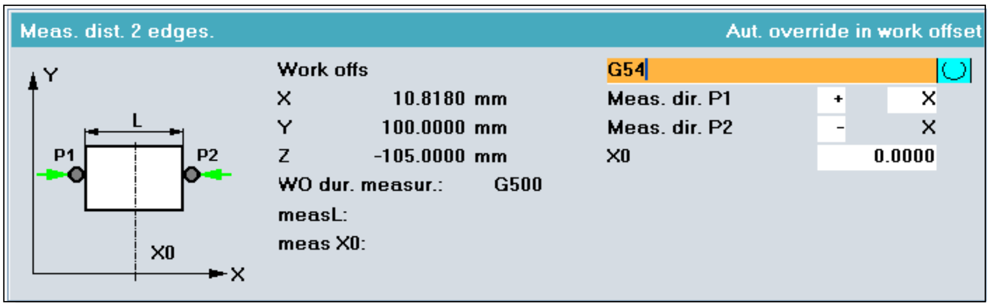
© Siemens AG, 2004. All rights reserved

Distance 2 edges

This means that the distance L of two edges parallel to an axis at a workpiece, e.g.: Slot, rib, or step, can be determined in one of the axes X, Y or Z and their center set as reference point in a ZO.
2 measuring points are possible, P1 and P2.

Approach the workpiece

Position the probe in front of the edge, first and P1 and after completion of this measurement at P2.



Enter details in input form

- Select of the ZO
- Select axis X, Y, or Z and measurement direction at P1
- Select the measuring direction at P2
- Enter the set position of the two edges for the selected ZO.

Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start". After successful measurement, the measured value is stored internally and the "P1 stored" softkey that was previously switched inactive is activated.
After manual positioning at measuring point **P2** and with softkey "P1 stored" switched active, pressing "NC start" starts measurement of the 2nd measuring point in the selected measuring axis. After successful measurement, softkey "P2 stored" is activated, and the 2nd measured value is stored.
A vertical "Calculate" softkey is then displayed. After this softkey is pressed, the distance and the distance center between the 2 measuring points in the selected measuring axis and the selected offset calculation are calculated. The offset is activated at the same time if it is the active ZO.

© Siemens AG, 2004. All rights reserved

Display and correction
After the offset calculation has been successfully completed, the frame components of the selected ZO are re-displayed so that the calculated center of the measured edge distance after activating the selected ZO assumes the required setpoint position (e.g. X0 in the corrected workpiece coordinate system.

Notes

- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

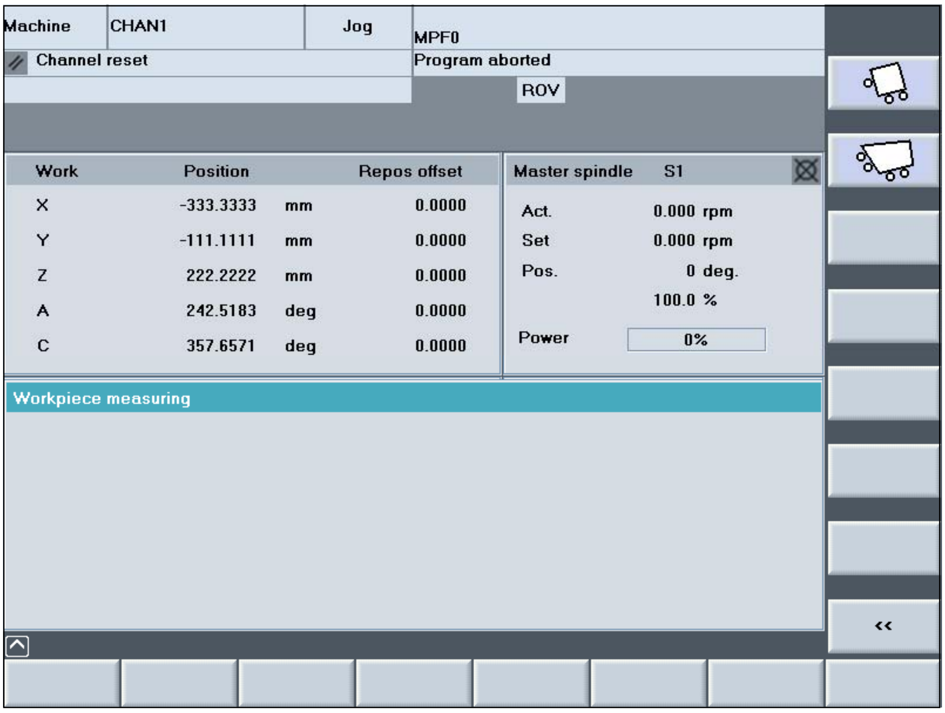
© Siemens AG, 2004. All rights reserved

4.3.4 Measuring a corner (from measuring cycles SW 6.3)

Function

After selecting "corner", a selection display is shown containing the following options:

- "right-angled corner"
- "any corner"



Precondition

The workpiece probe is the active tool with activated tool offsets in the spindle and is already calibrated (refer to Subsection 4.3.2).

Note:

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

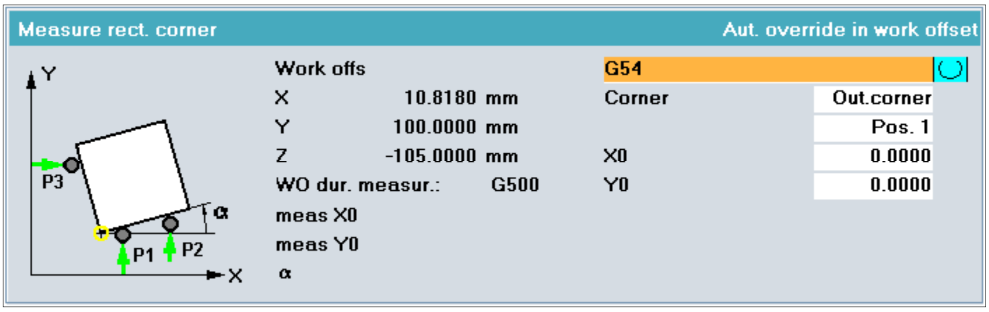
Right-angled corner

This permits measurement of a right-angled corner of a workpiece as the reference point in the axes of the working plane and setting it as the ZO (translation and rotation).
3 measuring points are required: P1, P2, and P3.
The 1st axis of the working plane functions as the reference axis (for G17: X axis). Edge P1, P2 functions as the reference edge on the workpiece. This edge is aligned parallel to the reference axis (G17: X axis).

© Siemens AG, 2004. All rights reserved

Approaching the workpiece

Position the probe on the corner, first at P1 and after completion of this measurement at P2, and after completion of this measurement at P3 - to measuring depth in each case.



Enter details in input form

- Select of the ZO
- Select outside or inside corner
- Select the position of the corner
- Enter the set position required for the reference point P0 (corner) for the selected ZO in both axes.

Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start". Measurement is performed in the axis and axis direction determined by the selected position. After successful measurement, the previously inactive softkey "P1 stored" is activated and the coordinates of the 1st measuring point P1 is stored internally.
After manual positioning in front of the 2nd measuring point **P2**, measuring is performed automatically at this measuring point on pressing "NC start". Proceed in the same way for measuring point **P3**.
If all of the measuring points have been successfully completed and all "Px saved" softkeys activated, then a vertical "calculate" softkey is displayed. After this softkey has been pressed, the corner coordinates P0 and the offset are calculated. The offset is activated at the same time if it is the active ZO.

Display and correction

After successful calculation and offset application, the coordinates of the corner point determined are displayed in the WCS active during measurement. Display of the translation frame components of the selected ZO is updated.
If you have selected "Just measure", the corner point determined and the angle of the reference axis will only be displayed.

© Siemens AG, 2004. All rights reserved

Notes

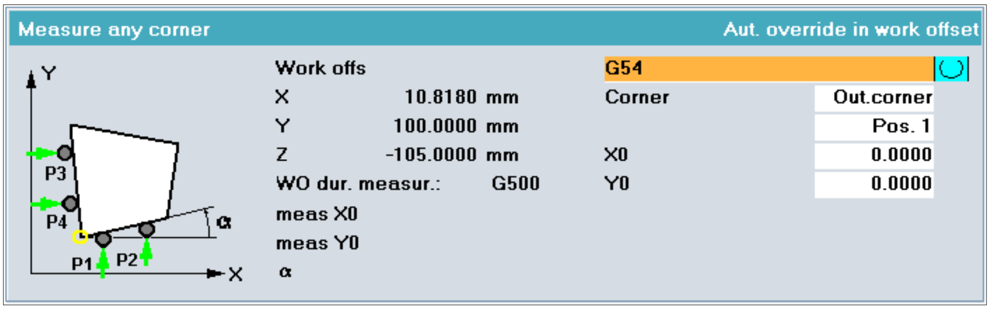
- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- A **support function** is available to the user when **activating the determined ZO correction** in the JOG mode. This is shown right after the measurement and is detailed in Subsection 4.3.9.

Any corner

This permits measurement of a corner of a workpiece as the reference point in the axes of the working plane and setting it as the ZO (translation and rotation).
The corner does not have to be right-angled.
4 measuring points are required: P1, P2, P3, and P4.
The 1st axis of the working plane functions as the reference axis (for G17: X axis). Edge P1, P2 functions as the reference edge on the workpiece. This edge is aligned parallel to the reference axis (G17: X axis).

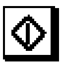
Approaching the workpiece

Position the probe on the corner, first at P1 and after completion of this measurement at P2, and after completion of this measurement at P3, etc.



Enter details in input form

- Select of the ZO
- Select outside or inside corner
- Select the position of the corner
- Enter the set position required for the reference point (corner) for the selected ZO in both axes.



Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start".
This is the same sequence as for "right-angled corner" but with 4 measuring points.

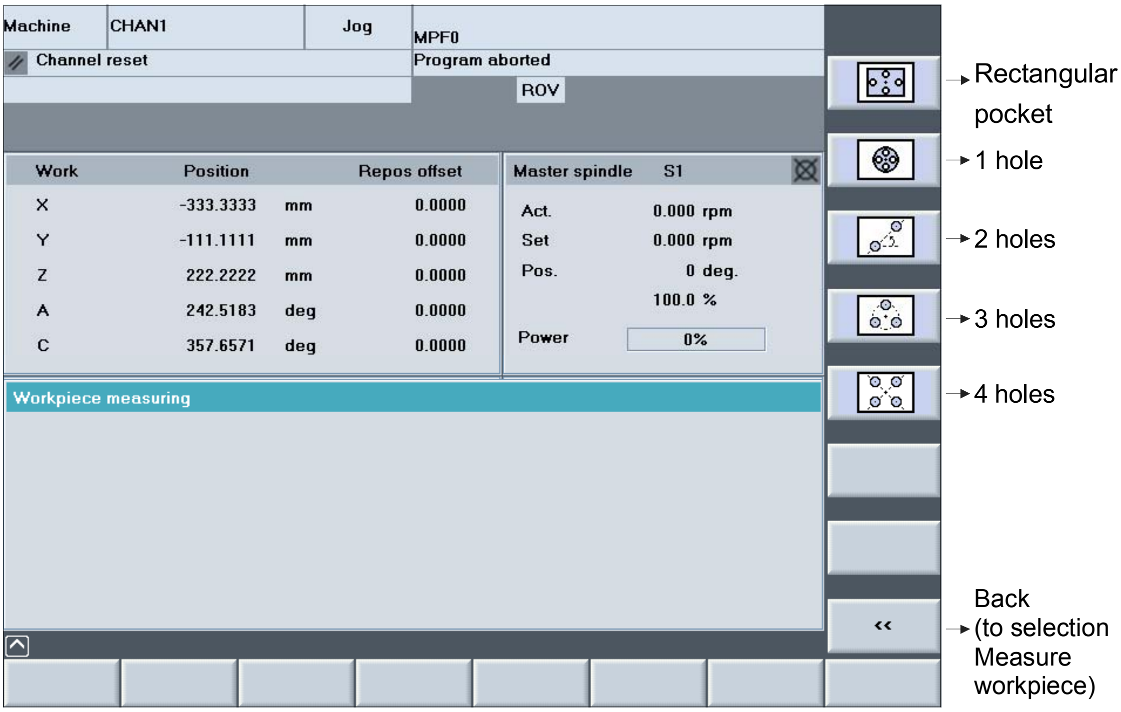
© Siemens AG, 2004. All rights reserved

4.3.5 Measuring a pocket, hole or spigot (from measuring cycles SW 6.3)

Function, pocket/hole

After selecting "pocket/hole", a selection display is shown containing the additional following options:

- "Rectangular pocket"
- "1 hole"
- "2 holes"
- "3 holes"
- "4 holes"

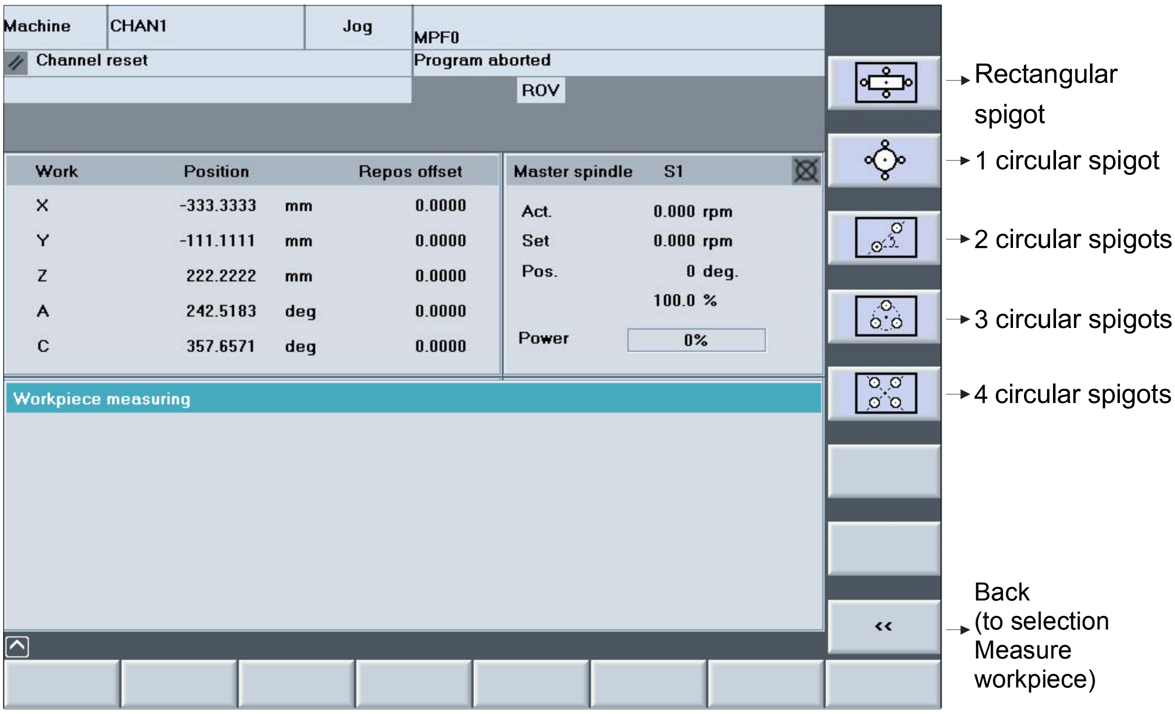


Function, spigot

After selecting "spigot", a selection display is shown containing the additional following options:

- "Rectangular spigot"
- "1 circular spigot"
- "2 circular spigots"
- "3 circular spigots"
- "4 circular spigots"

© Siemens AG, 2004. All rights reserved



Precondition

The workpiece probe is the active tool with activated tool offsets in the spindle and is already calibrated (refer to Subsection 4.3.2).

Note:

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

Rectangular pocket or 1 hole or 1 spigot

Depending on selection, the **center**

- of a paraxial rectangular pocket or
- a hole or
- a paraxial rectangular spigot or
- a circular spigot

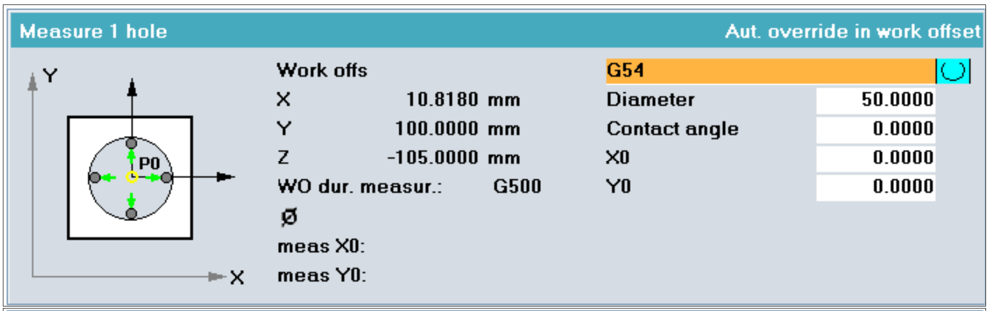
can be measured on the workpiece as the reference point and as the ZO (translation).

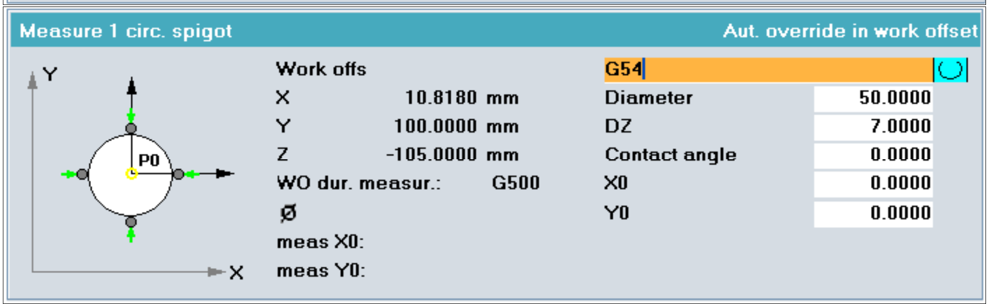
Measurement is performed in both axes of the working plane.

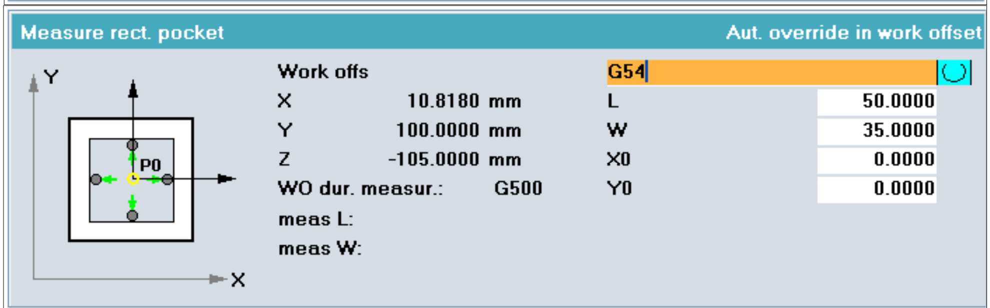
Approaching the workpiece

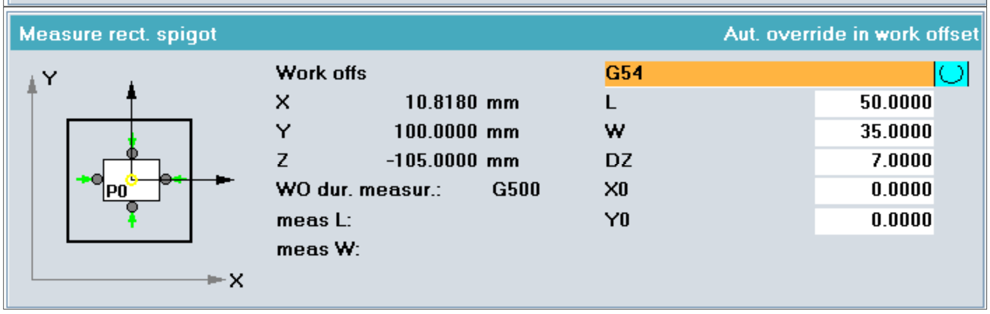
Position the probe roughly at measuring depth in the center of the pocket/hole, or in the case of spigots roughly in the center above the spigot.

© Siemens AG, 2004. All rights reserved









Enter details in input form

- Select of the ZO
- For rectangular pocket, rectangular spigot:
  Enter approximate length L (1st axis of the working plane) and width W (2nd axis of the working plane)
- For hole, circular spigot:
  Enter the approximate diameter
- For spigots only:
  Enter infeed value DZ
  (measuring depth as from starting position, value >0)
- For hole, circular spigot only:
  Enter the probing angle if measurement is not to be paraxial.
- Enter the set position required for the reference point P0 (center) for the selected ZO in both axes.

© Siemens AG, 2004. All rights reserved

Measurement is performed automatically as soon as you press "NC Start". The probe probes 4 points of the inside and outside wall in succession.
The offset is then automatically calculated. If the selected offset is the active ZO, then this is automatically activated.

Display and correction
After completion of calculation and offset, the diameter or width/length and the coordinates of the center P0 are displayed.
An offset is applied in the translation components of the selected ZO in both axes.

Notes

- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

2 holes or 2 circular spigots

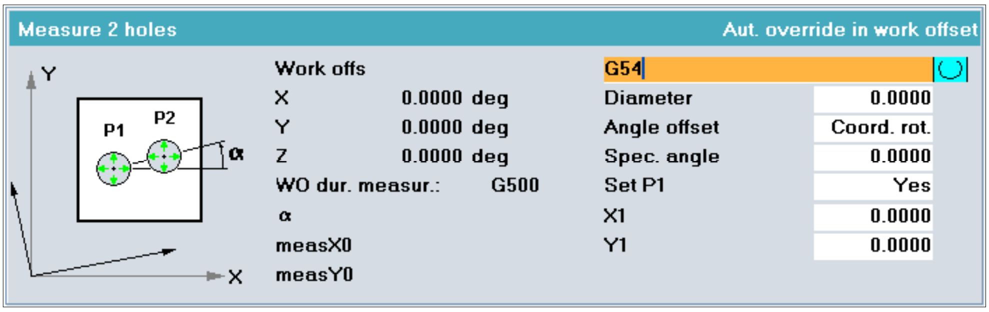
This permits calculation of the basic rotation (rotation in the working plane) of the clamped workpiece.
This enables orientation by

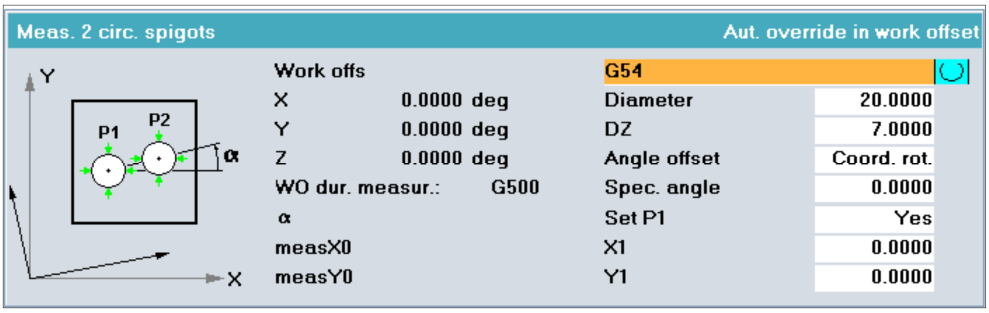
- "coordinate rotation"
- "or rotation of the workpiece with a rotary table (rotary axis).

For an angle offset by "coordinate rotation", it is possible to set the reference point for the calculated center of the 1st hole/spigot.
Measurement is performed in both axes of the working plane.

Approaching the workpiece
Position the probe roughly in the center of the 1st hole/spigot P1, after completion of this measurement in the rough center of the 2nd hole/spigot P2, for a hole to measuring depth in each case, for a spigot above the spigot.

© Siemens AG, 2004. All rights reserved





Enter details in input form

- Select of the ZO
- Enter the approximate diameter (hole/spigot). This must be selected so that it is applicable for all holes/spigots.
- For spigots only: Enter infeed value DZ (measuring depth as from starting position, value >0)
- Select the required angle offset: "Coordinate rotation" or axis name of the rotary axis
- Under the "set angle" input, an orientation deviating from 0 degrees with respect to the 1st axis of the working plane (e.g., G17: X axis) is possible.
- Only for offset type "coordinate rotation" and if "set P1" is selected:
  Enter the set position required for the reference point P1 (center of the 1st hole) for the selected ZO in both axes.

Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start". The probe probes 4 points of the inside and outside wall in succession. After successful measurement, the center is stored internally and the "P1 stored" softkey that was previously switched inactive is activated.

After manual positioning at point **P2** and with softkey "P1 stored" switched active, pressing "NC start" starts measurement of the 2nd hole/spigot.

© Siemens AG, 2004. All rights reserved

After successful measurement, softkey "P2 stored" is activated, and the 2nd center is stored. If all of the measuring points have been successfully completed and all "Px saved" softkeys activated, then a vertical "calculate" softkey is displayed. After this softkey is pressed, "alpha" is calculated.

When the "coordinate rotation" angular offset (correction) is selected, then the offset is also activated if it is the active ZO.

When the "rotary axis" angular offset is selected (name of the rotary axis), a help function for the user is displayed to activate the offset (refer to Subsection 4.3.9).

Display and correction
After offset calculation has been completed, the frame components of the selected ZO are redisplayed along with the measured "alpha" angle and the coordinates of the reference point P1.
ZO correction is effected in such a way that the angle determined is in the required set rotation and point P1 is in the set position in the corrected workpiece coordinate system after activation of the selected ZO.

Notes

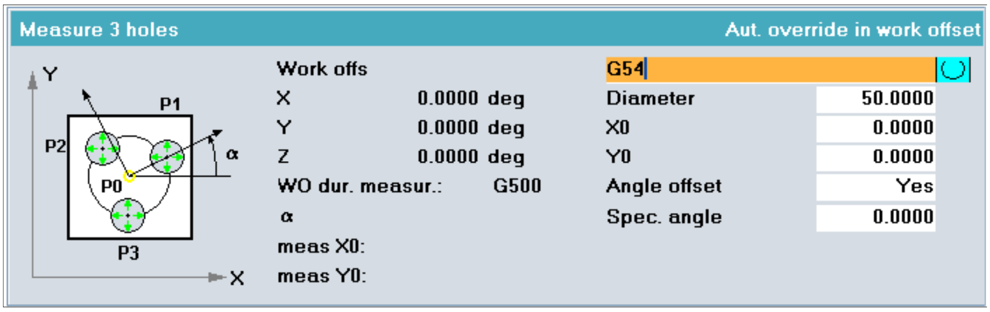
- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

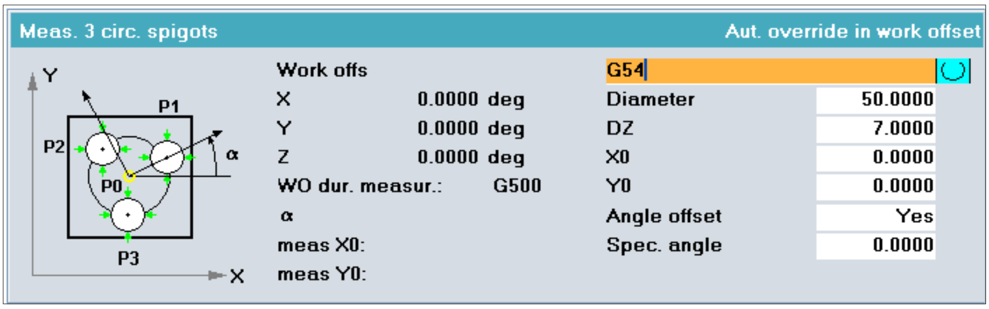
3 holes or 3 circular spigots

This permits calculation of the reference point P0 and the "alpha" rotation of the clamped workpiece.
This enables orientation by coordinate rotation and setting the center P0 of the partial circle on which the 3 holes/spigots are located as the reference point.
Measurement is performed in both axes of the working plane.

Approaching the workpiece
Position the probe roughly in the center of the 1st hole/spigot P1, after completion of this measurement in the rough center of the 2nd hole/spigot P2, after completion of this measurement in the rough center of the 3rd hole/spigot, for a hole to measuring depth in each case, for a spigot above the spigot.

© Siemens AG, 2004. All rights reserved





Enter details in input form

- Select of the ZO
- Enter the approximate diameter (hole/spigot). This must be selected so that it is applicable for all holes/spigots.
- For spigots only: Enter infeed value DZ (measuring depth as from starting position, value >0)
- Enter the set position required for the reference point P0 (center of the partial circle of the 3rd hole/spigot) for the selected ZO in both axes.
- If angle offset is selected:
  Under the "set angle" input, an orientation deviating from 0 degrees with respect to the 1st axis of the working plane (e.g., G17: X axis) is possible.

Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start". The probe probes 4 points of the inside and outside wall in succession. After successful measurement, the measured value is stored internally and the "P1 stored" softkey that was previously switched inactive is activated.
After manual positioning at the next center **P2** and with softkey "P1 stored" switched active, pressing "NC start" starts measurement for this hole/spigot.
The rest of the sequence is as for P1.
If all of the measuring points have been successfully completed and all "Px saved" softkeys activated, then a vertical "calculate" softkey is displayed. After this softkey is pressed, P0 and "alpha" are calculated.

© Siemens AG, 2004. All rights reserved

When the "coordinate rotation" angular offset (correction) is selected, then the offset is also activated if it is the active ZO.

When the "rotary axis" angular offset is selected (name of the rotary axis), a help function for the user is displayed to activate the offset (refer to Subsection 4.3.9).

Display and correction
After offset calculation has been completed, the frame components of the selected ZO are redisplayed along with the measured "alpha" angle and the coordinates of the reference point P0.
ZO correction is effected in such a way that the "alpha" angle determined and point P0 are at the desired setpoints (translation and rotation) in the offset workpiece coordinate system after activation of the selected ZO.

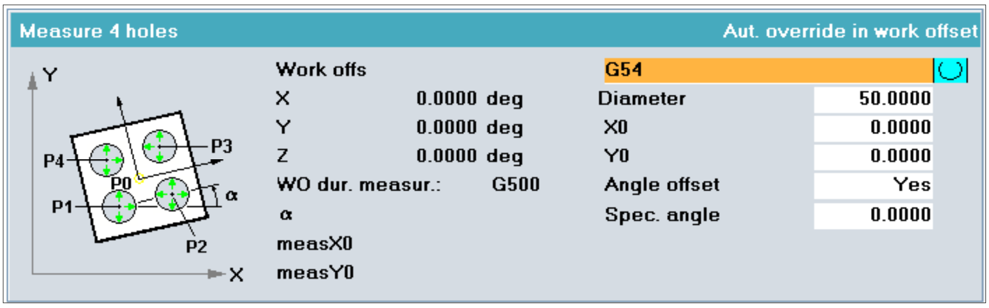
Notes
- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

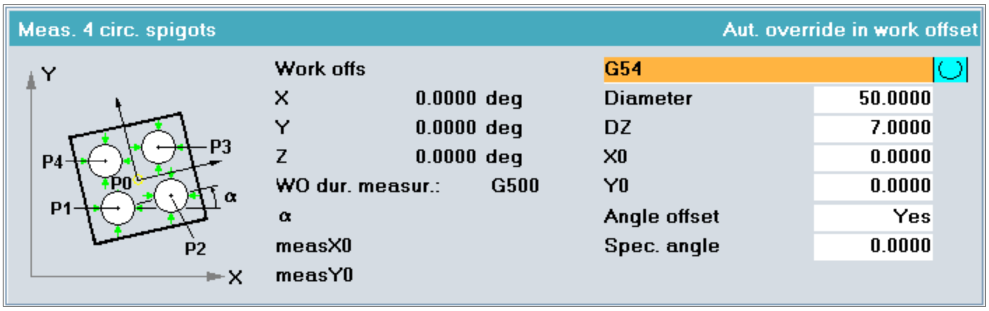
4 holes or 4 circular spigots

This permits calculation of the reference point P0 and the "alpha" rotation of the clamped workpiece.
This permits orientation by coordinate rotation and setting point P0 (intersection point of the 2 straight lines, whose hole centers are connected diagonally) as the reference point.
Measurement is performed in both axes of the working plane.

Approaching the workpiece
Position the probe roughly in the center of the 1st hole/spigot P1, after completion of this measurement in the rough center of the 2nd hole/spigot P2, etc., for a hole to measuring depth in each case, for a spigot above the spigot.

© Siemens AG, 2004. All rights reserved





Enter details in input form

- Select of the ZO
- Enter the approximate diameter (hole/spigot). This must be selected so that it is applicable for all holes/spigots.
- For spigots only: Enter infeed value (DZ) (measuring depth as from starting position, value >0)
- Enter the set position required for the reference point P0 (intersection point of the diagonals) for the selected ZO in both axes.
- If angle offset is selected:
  Under the "set angle" input, an orientation deviating from 0 degrees with respect to the 1st axis of the working plane (e.g., G17: X axis) is possible.

Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start". The probe probes 4 points of the inside and outside wall in succession. After successful measurement, the measured value is stored internally and the "P1 stored" softkey that was previously switched inactive is activated.
After manual positioning at the next center **P2** and with softkey "P1 stored" switched active, pressing "NC start" starts measurement for this hole/spigot.
The rest of the sequence is as for P1.
If all of the measuring points have been successfully completed and all "Px saved" softkeys activated, then a vertical "calculate" softkey is displayed. After this softkey is pressed, P0 and "alpha" are calculated.

© Siemens AG, 2004. All rights reserved

When the "coordinate rotation" angular offset (correction) is selected, then the offset is also activated if it is the active ZO.

When the "rotary axis" angular offset is selected (name of the rotary axis), a help function for the user is displayed to activate the offset (refer to Subsection 4.3.9).

Display and correction
After offset calculation has been completed, the frame components of the selected ZO are redisplayed along with the measured "alpha" angle and the coordinates of the reference point P0.
ZO correction is effected in such a way that the "alpha" angle determined and point P0 are at the desired setpoints (translation and rotation) in the offset workpiece coordinate system after activation of the selected ZO.

Notes

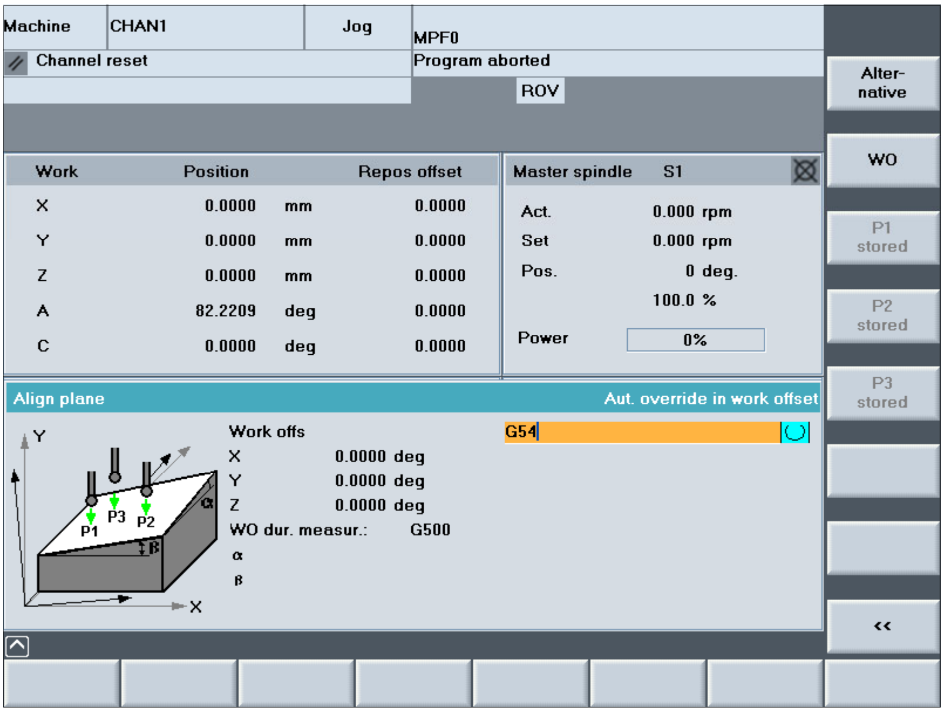
- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

© Siemens AG, 2004. All rights reserved

4.3.6 Orienting a plane (from measuring cycles SW 6.3)

Function

After selection of "orient plane", the following display is shown:



Precondition

The workpiece probe is the active tool with activated tool offsets in the spindle and is already calibrated (refer to Subsection 4.3.2).

Note:

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

Measuring a plane that is oblique in space

That permits measurement of a workpiece plane that is oblique in space and determination of "alpha" and "beta" rotation.
That enables orthogonal orientation of the infeed axis with respect to this plane by coordinate rotation.

Measurement is performed in the infeed axis (3rd axis) at 3 different points in the working plane.

© Siemens AG, 2004. All rights reserved

Approaching the workpiece
Position the probe above measuring point P1, after completion of this measurement above measuring point P2, after completion of this measurement above measuring point P3.

Enter details in input form
- Select of the ZO

Measurement is performed automatically at **P1** with the set measuring feed as soon as you press "NC Start". After successful measurement, the measured value is stored internally and the "P1 stored" softkey that was previously switched inactive is activated.
After manual positioning at the next measuring point **P2** and with softkey "P1 stored" switched active, pressing "NC start" starts measurement for this measuring point.
The rest of the sequence is as for P1.
If all of the measuring points have been successfully completed and all "Px saved" softkeys activated, then a vertical "calculate" softkey is displayed. After this softkey is pressed, "alpha" and "beta" are calculated.

A support function for the user is displayed to activate the offset (refer to Subsection 4.3.9).

Display and correction
After offset calculation has been completed, the frame components (rotation) of the selected ZO are redisplayed along with the measured "alpha" and "beta" angle.
ZO correction is performed in such a way that the plane with the calculated points P1 to P3 after activation of the selected ZO is parallel to the new working plane.

Notes
- The rejection, repetition, and end of measurement are described in Subsection 4.3.7.
- **Support** for the user is featured for **activation of the zero offset** determined in JOG mode and for necessary probe reorientation. This is shown right after the measurement and is detailed in Subsection 4.3.9.

© Siemens AG, 2004. All rights reserved

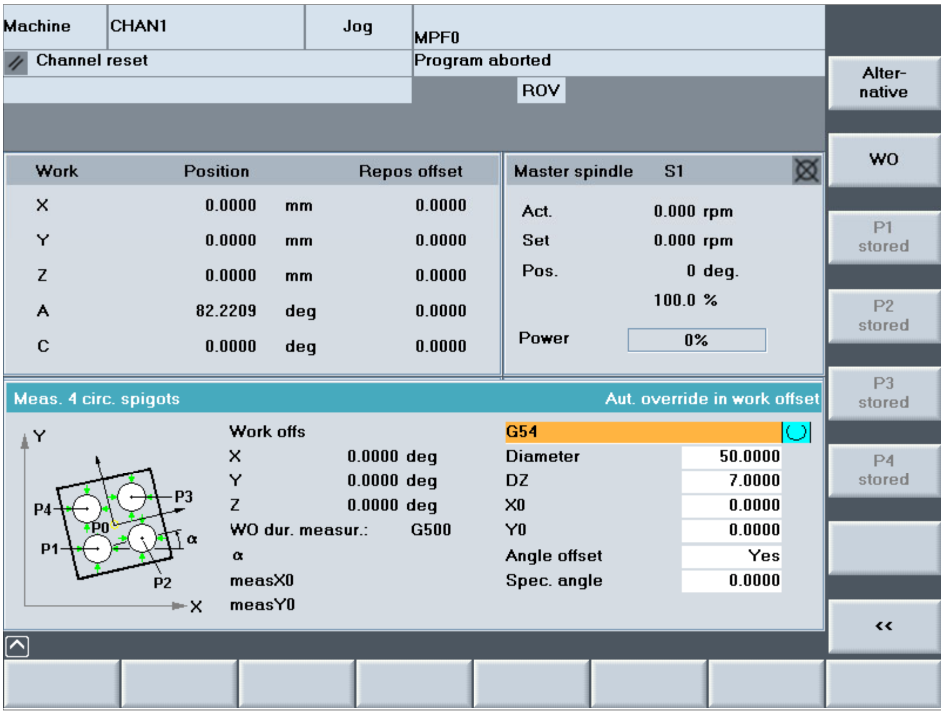
4.3.7 Rejecting, repeating, end of measurement (from measuring cycles SW 6.3)

Function

Rejection and repetition of measurements

The last measurement (Px) can be declared invalid any number of times by pressing the assigned softkey "Px stored". The softkey is then deactivated (grayed out).
Pressing "NC start" again repeats this measurement and the "Px stored" softkey is then reactivated (black text).

Example of vertical softkeys P1 to P4 - inactive:



End of measurements

If all "Px saved" softkeys are active, then a vertical "calculate" softkey is displayed.
When this softkey is pressed, the measuring points are finally confirmed/acknowledged and the translatory and rotary offset values are calculated. The result is immediately entered into the pre-selected "zero offset". If this is identical with "ZO when measuring", the offset values become immediately effective. This means that the measurements have been completed.

For offsets/corrections, that required e.g. a new alignment/ orientation of the measuring probe or the rotary table, the user is provided with a **support function** to activate the determined ZO offset in the JOG mode. This is displayed after pressing the "calculate" softkey and is described in detail in Subsection 4.3.9.

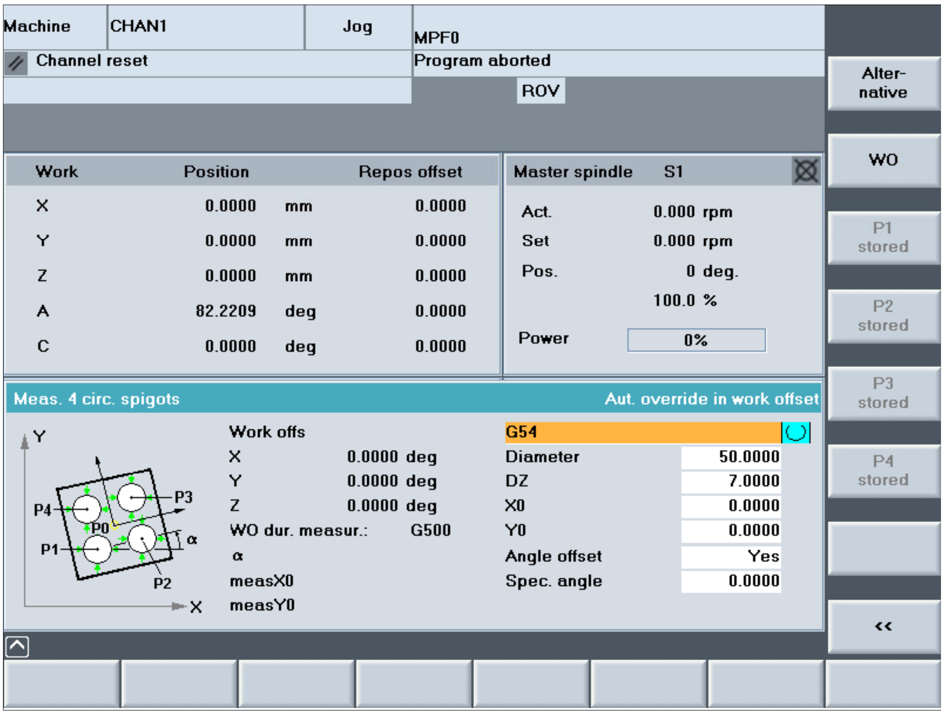
© Siemens AG, 2004. All rights reserved

The "Px stored" softkeys are then switched inactive and it is possible to start new measurement.
The number of softkeys Px is determined with the measurement task.

When the input screen form is exited, a measurement is also completed/interrupted.

Notes
A mode change is only possible after exiting the operator area "measuring in JOG".

Example for vertical softkeys P1 to P4 active,
the "calculate" softkey is displayed:



4.3.8 Information regarding cascaded measurements

Function

Often, it is not possible to completely set-up a workpiece using just one single measurement; instead, a series of measurements has to be made. This results in certain interdependencies regarding the selected measuring series.

Example:

- Orienting an oblique plane
- Orienting an edge, reference in X axis or
  Orienting an edge, reference in Y axis
- Setting a reference point in X, Y, Z

A zero offset can be applied after every measurement. The "measuring in JOG" function supports activation of another or changed ZO in JOG mode by displaying an additional activation screen form with queries after measurement.
If so selected, the new zero offsets are activated and the probe possibly also re-oriented.
The changed zero offset is offered as the preferred setting for the next measurement. This setting can therefore deviate from the basic setting _JM_I[4] in data block GUD6.

© Siemens AG, 2004. All rights reserved

4.3.9 Support of set-up in JOG - after measurement

Function

After an offset has been applied in the selected zero offset by the measuring function, this ZO is activated in JOG mode and the probe possibly oriented in the new WCS, e.g. for the following measurements.

For "measuring in JOG", an offset in the active ZO (the ZO that applies during measurement) is preset. This ZO is updated after measurement and is therefore active.
This is sufficient in cases where translation offset of the workpiece coordinate system and / or rotation about the infeed axis has occurred.

Positioning of the rotary axes for orienting the workpiece after measurement or orienting the probe because of rotation about the axes of the working plane has so far only been possible in MDA or AUTOMATIC mode.

To support set-up in JOG after measurement, an additional **activation screen form** is automatically shown **afterward**, if at least one of the following reasons applies:

- Offset in a ZO **not** active during measurement
- An offset causes rotation of the WCS about at least one **plane axis** following by a necessary reorientation of the probe
- An offset demands **rotary axis positioning** for workpiece alignment without affecting the WCS

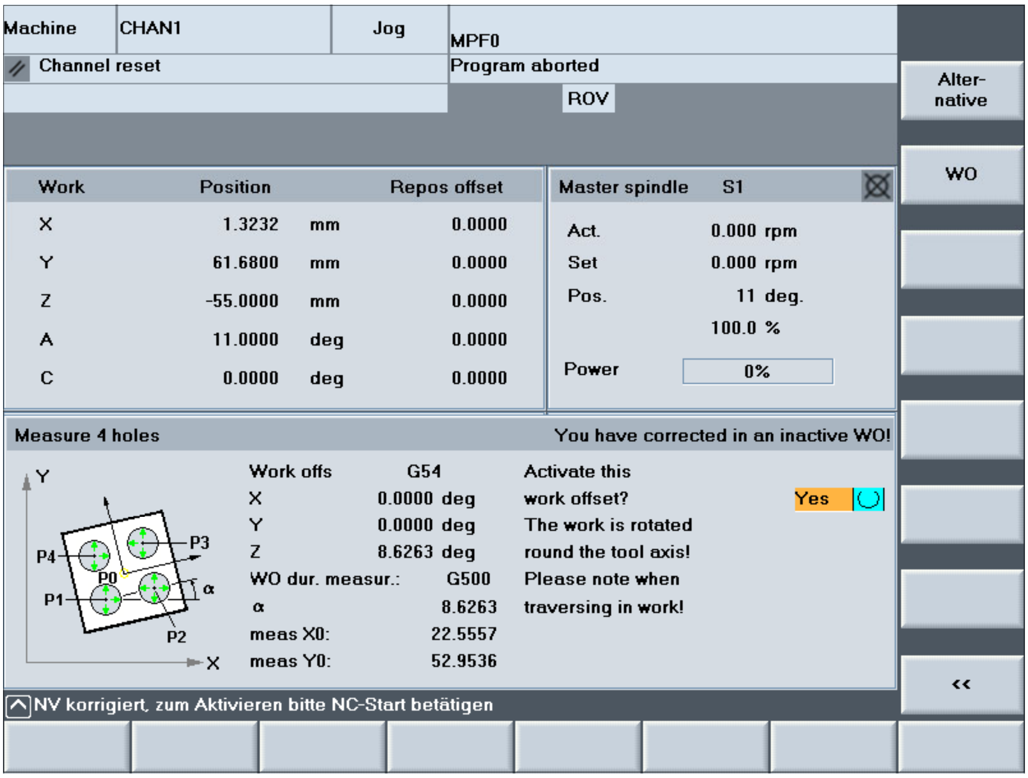
The system detects whether the measuring probe must be aligned/oriented using the function "swiveling" or the 5-axis transformation (TRAORI). The relevant screen form is displayed. If axial movements are required, the user is informed about this in the screen form. It is possible to decide whether pressing "NC start" will start the swivel cycle for orienting the probe or whether the selected rotary axis will move to orient the workpiece.

Notes
If the function "swiveling" or the 5-axis transformation (TRAORI) is used to align/orient the measuring probe, then this must also be set-up in the machine!
Please carefully observe the information provided by the machinery manufacturer!

© Siemens AG, 2004. All rights reserved

Example 1

4 holes were measured. An offset is applied in a ZO that was not active during measurement. Reorientation of the probe is not necessary. The following selection display appears with information:



Explanation

The display for the measurement function "measure 4 holes" is shown in the left part of the screen form. The display of the measurement result is also shown in the WCS that was active during measurement (G500) and display of the translation values of the offset ZO.

The operator is informed about the selection made in the toggle field (yes) with a text or in an upper image line:

"You have applied an offset in a non-active ZO!".

A message text also appears for the special case:

"The WCS is about the ...".

An additional text is output on the HMI "dialog line with user information and instructions. It provides instructions on action to take:

"Please press NC start to activate!"

After pressing "NC start", the ZO with the applied offset is activated. It is not necessary to traverse axes.

The previous measurement selection display is then shown automatically. Measurement can be performed again.

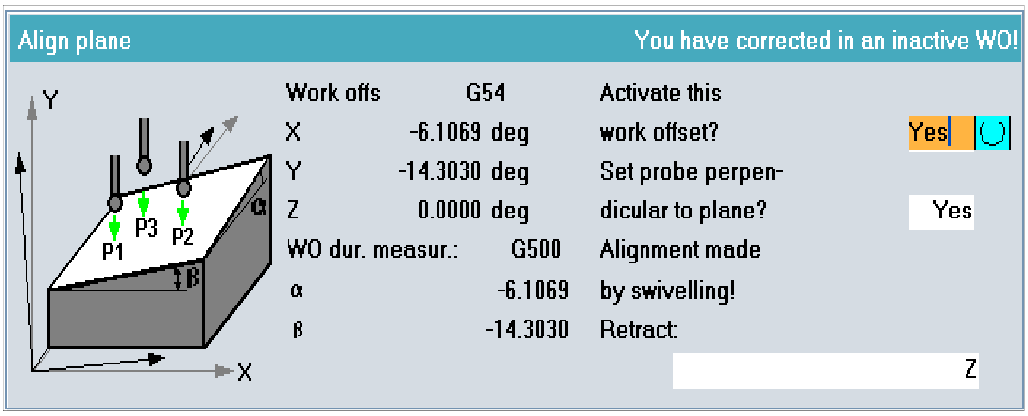
To avoid activating the ZO with the applied offset, exit the display by pressing the softkey "<<" (return).

© Siemens AG, 2004. All rights reserved

Example 2

An oblique plane was measured. An offset is applied in a ZO that was not active during measurement. "Swiveling" is active. This requires re-orientation of the probe with respect to the plane via the swivel cycle.

The following selection display appears with information:



Explanation

The display for the measurement function "orient plane" is shown in the left part of the screen form. The display of the measurement result is also shown in the WCS that was active during measurement and display of the translation values of the offset ZO.

The operator is informed about the selection made in the toggle field 1 (yes) with a text or in an upper image line:

"You have applied an offset in a non-active ZO!"

Because of the answer with "yes" in toggle field 2, the text in the upper display line is changed to:

"The new WCS has new orientation!"

With toggle field 3, retraction is supported.

If all screen forms have been filled out, a further text appears on a lower line of the display. It provides instructions on action to take:

"Please press NC start to activate!"

After pressing "NC start", execution is performed according to the entries that the user made in the activation screen form:

- Activating the corrected ZO or
- Retracting the measuring probe and re-aligning/orienting it using the swivel cycle.

The previous measurement selection screen is then re-displayed.

After the plane has been successfully aligned/oriented, the workpiece can continue to be gauged with measuring "edge", "holes", "spigots" etc. Refer to Subsection 4.3.8.

If the corrected ZO is not to be activated, exit the screen by pressing the softkey "<<" (return).

© Siemens AG, 2004. All rights reserved

4.4 Measuring tools

4.4.1 Overview of function and sequence

Function

The "tool measurement" function permits the following functions.

- Calibrating the tool probe
- Determining the tool length or radius of milling tools, or tool length of drills and then setting the appropriate offset in the tool offset memory.

The tools are measured on the machine.

General sequence

When you press softkey "Measure tool", the following selection and more appears on the vertical softkey bar:

Length
Diameter >
Calibrate probe >
<< back

1. The measuring variant is selected and the input screen form is filled with values.
2. Position the tool near the tool measuring probe with the axis traverse keys.
3. "NC start" starts measurement and the remaining sequence with offset entry runs automatically.

4.4.2 Calibrating the tool probe

Function

The function "Calibrate tool measuring probe" calculates the current distance (switching points) between the machine zero and the tool probe (calibration relative to machine) using a **calibration tool** and automatically writes them to a data storage area as trigger values.
The type 120 (end mill) can be defined as the tool type for the calibration tool. There is no special "calibration tool" type.

© Siemens AG, 2004. All rights reserved

Operating sequence

Precondition

- The tool probe is mounted in the machining space of the machine (usually on the machine table) and align with the machining axes (machine axes).
- All necessary data of the tool probe (form, dimensions, ...) are entered in the intended variables for "measuring in JOG" E_MESS_MT_... in data block GUD7 (see Section 9.3).
- The precise length 1 and the radius of the calibration tool must be stored in a tool offset data set (geometry).
- The calibration tool is in the spindle and is activated with the offset data set.

Approaching the tool measuring probe

Position the calibration tool roughly over the center of the measuring surface of the tool probe.

Select the function with softkey

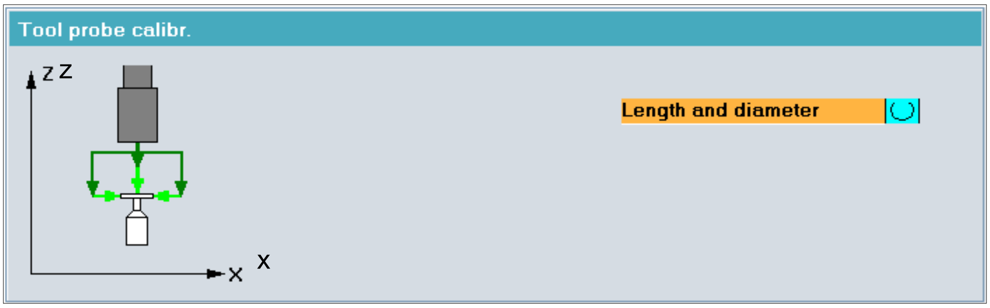
Measure tool | Continue with: | Calibrate probe >

In the input screen form, select the type of calibration by pressing the "alternative" softkey.

Calibrate length only:

Or

Calibrate length and diameter:



© Siemens AG, 2004. All rights reserved

On "NC start", calibration runs automatically with the measuring feed entered in _E_MESS_MT_FM in data block GUD7.
The current distance dimensions between the machine zero and tool probe (trigger points) are determined using the calibration tool and stored in a data area.
The tool probe is now prepared for measurement of tools.

4.4.3 Measuring milling and drilling tools

Function

This allows you to determine the tool length or radius of milling tools, or tool length of drills and then set the appropriate offset in the tool offset memory.

The tool offset mode for "measure in JOG" can be set via a variable in data block GUD6:

- _JM_B[0]=0: Offset in geometry component
- _JM_B[0]=1: Offset in wear component

For milling tools, cutting edges with a special shape, e.g. rounded cutting edges, can be taken into account with additional entries.

Measurement is performed with a motionless or rotating spindle:

- Radius measurement is performed with a rotation spindle.
- Length measurement is performed with a rotating spindle if the tool radius is greater than the upper radius of the too probe. Otherwise, measurement is with a motionless spindle.

Corresponding to the user specifications regarding the required measuring accuracy, it is possible that extremely small measurement feeds are applied. This is the reason that the tool gauging is automatically sub-divided into two measuring operations (corresponding to the default setting of E_MESS_... parameterization in GUD7_MC).

- The first measurement is always made from a starting position selected by the user with a high measurement velocity. This is used to detect the principle position of the tool to be gauged.
- The 2nd measurement is realized from an optimized position with a short measuring distance and a low measurement feed corresponding to the required measuring accuracy.

© Siemens AG, 2004. All rights reserved

The number of probing operations, the required/specified measurement accuracy as well as various limit values can be adapted to the real conditions in the GUD parameters associated with tool measurement (E_MESS_...) (refer to Subsection 9.3.2).
The mathematical interrelationship between the measuring accuracy and measurement feed can be taken from Subsection 5.2.1.

Preconditions

- The specific GUD parameters used to measure tools are adapted to the real user conditions.
- The reference points are approached.
- The tool probe is functional.
- The tool probe has been calibrated before measurement.
- The tool to be measured is located in the spindle and is activated.
- The tool offset data (length radius) are entered as approximate values and activated.

© Siemens AG, 2004. All rights reserved

Operating sequence

Approaching the tool probe

Position the active tool:

- Over the measuring surface of the tool probe (for length measurement) or
- from the side over the measuring probe when measuring the radius

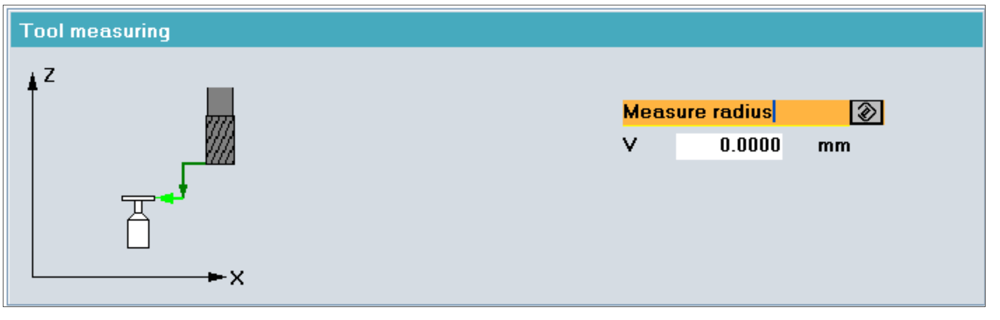
Select the function with softkey

Measure tool

Continue with selection:

Diameter

The system displays:



Or:

Length

The system displays:

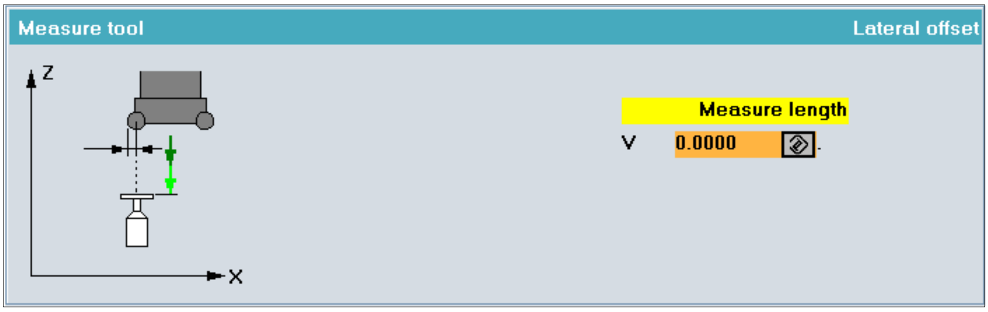
Length measurement of milling tools

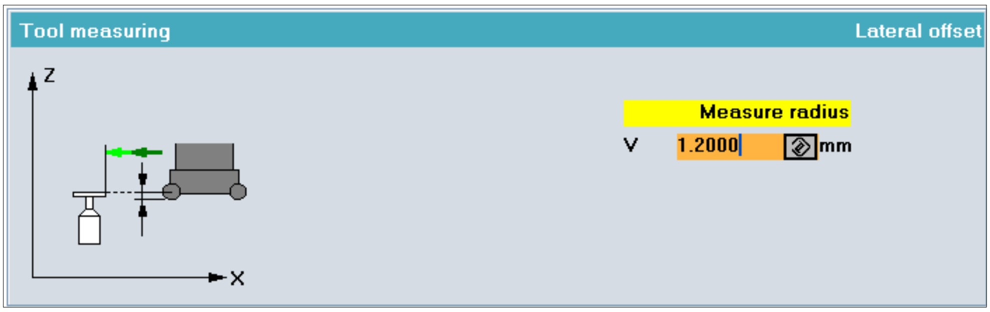
If the tool diameter (entered tool radius x 2) is greater than the entered upper diameter of the tool probe, the milling tool is placed in the center of the probe, offset by the tool radius and measured with rotating spindle. Otherwise, the tool is place in the center and measurement is performed with a motionless spindle.

© Siemens AG, 2004. All rights reserved

Measurement of milling tools with special cutting edge

If a cutting edge is rounded, for example, the tool must be measured with a probe offset. This offset (V) must be entered additionally for diameter determination.





Enter details in input form

Enter offset (V, positive value) when required.

Measurement is performed automatically as soon as you press "NC Start".

The tool offsets "radius" or "length 1" are calculated in and entered in the active tool offset data in accordance with the setting (in geometry or wear).

■

© Siemens AG, 2004. All rights reserved

Notes

© Siemens AG, 2004. All rights reserved

Measuring Cycles for Milling and Machining Centers

© Siemens AG, 2004. All rights reserved

5.1 General prerequisites

The measuring cycles below are intended for use on milling machines and machining centers.
Under certain conditions, workpiece measuring cycles CYCLE976, CYCLE977, and CYCLE978 can also be used on turning machines.
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.

Overview of measuring cycles

CYCLE961	Workpiece: Setup inside and outside corner
CYCLE971	Tool measurement for milling tools, calibrate tool probe
CYCLE976	Calibrate workpiece probe in a hole or on a surface
CYCLE977	Paraxial measurement of hole, shaft, groove, web or ZO calculation
CYCLE978	1-point measurement or ZO determination on surface
CYCLE979	Measurement of hole, shaft, groove, web, or ZO determination at an angle
CYCLE997	Measuring spheres or ZO determination (from measuring cycles SW 6.3)
CYCLE998	Angle measurement (ZO determination only)

Overview of the auxiliary programs required

CYCLE100	Log ON
CYCLE101	Log OFF
CYCLE102	Measurement result display selection
CYCLE103	Pre-assignment of input data
CYCLE104	Internal subroutine: Measuring cycle interface
CYCLE105	Generate log contents
CYCLE106	Log sequential controller
CYCLE107	Output of message texts (up to measuring cycles SW 6.2)
CYCLE108	Output of alarm messages (up to measuring cycles SW 6.2)
CYCLE109	Internal subroutine: Data transfer
CYCLE110	Internal subroutine: Plausibility checks
CYCLE111	Internal subroutine: Measuring functions
CYCLE112	Internal subroutine: Measuring functions
CYCLE113	Internal subroutine: Log
CYCLE114	Internal subroutine (tool offset)
CYCLE115	Internal subroutine (zero offset)
CYCLE116	Calculate circle center point
CYCLE118	Format real values
CYCLE119	Internal subroutine: Determine position in space (from measuring cycles SW 6.3)

The measuring cycle data are defined in the data blocks:

- GUD5.DEF
- GUD6.DEF

© Siemens AG, 2004. All rights reserved

Call and return conditions

The following general call and return conditions must be observed:

- The D offset must be activated with the data of the probe for workpiece measurement before the cycle is called. Tool type 1x0 or 710 (3D probe) is permitted. Dimension factors <> 1 may be active.
- The workpiece measuring cycles can also be used on turning machines if the following conditions are fulfilled:
  - The 3rd geometry axis exists.
  - Tool type of probe 5xy with cutting edge position 5 to 8
  - The tool length compensation is specific to turning machines (SD TOOL_LENGHT_TYPE=2)
  - If cutting edge position 5 or 7 is used, measurement is performed in the G17 plane
    If cutting edge position 6 or 8 is used, measurement is performed in the G19 plane
- Coordinate rotation is permitted for workpiece measuring cycles.
- Mirroring for the workpiece measuring cycles is permitted with the exception of calibration (condition: MD 10610=0).
- When using a multidirectional probe the best measurement results are achieved if, during calibration and measurement, the probe in the spindle is mechanically oriented to have one and the same point on the probe ball point, for example, in the + direction of the abscissa (+X with active G17) 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.
- Measurements must always be performed under the same conditions as applied when the probe was calibrated.

Measuring cycles from measuring cycles SW 6.2 and higher can only be used with NCK-SW 6.3 and higher.

© Siemens AG, 2004. All rights reserved

5.2 CYCLE971 tool: Measuring milling tools, drills

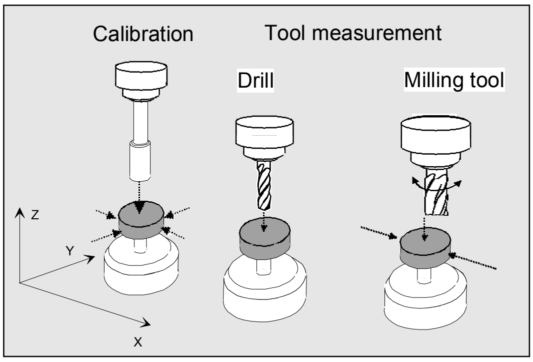
Programming

CYCLE971

Function

Measuring cycle CYCLE971 implements:

- Calibration of a tool probe
- Measurement of the tool length with motionless or rotating spindle for drills and milling tools
- Measure tool radius with motionless and rotating spindle for milling tools



Measuring variants

Measuring cycle CYCLE971 permits the following measuring variants which are specified via parameter _MVAR.

Value	*Meaning*
0	Calibrate tool probe (machine-related)
1	Measure tool with motionless spindle (length or radius, machine related)
2	Measure tool with rotating spindle (length or radius, machine related)
10000	Calibrate tool probes incrementally (machine-related)
10	Calibrate tool probes (workpiece-related)
11	Measure tool with motionless spindle (length or radius, workpiece related)
12	Measure tool with rotating spindle (length or radius, workpiece related)
10010	Calibrate tool probes incrementally (workpiece-related)

1) from meas. cycles SW 6.3

Result parameters

The measuring cycle CYCLE971 returns the following values in the data block GUD5 for the measuring variant **calibration**:

_OVR [8]	REAL	Trigger point in minus direction, actual value of 1st geometry axis
_OVR [10]	REAL	Trigger point in plus direction, actual value of 1st geometry axis
_OVR [12]	REAL	Trigger point in minus direction, actual value of 2nd geometry axis
_OVR [14]	REAL	Trigger point in plus direction, actual value of 2nd geometry axis
_OVR [16]	REAL	Trigger point in minus direction, actual value of 3rd geometry axis
_OVR [18]	REAL	Trigger point in plus direction, actual value of 3rd geometry axis
_OVR [9]	REAL	Trigger point in minus direction, difference of 1st geometry axis
_OVR [11]	REAL	Trigger point in plus direction, difference of 1st geometry axis

© Siemens AG, 2004. All rights reserved

_OVR [13]	REAL	Trigger point in minus direction, difference of 2nd geometry axis
_OVR [15]	REAL	Trigger point in plus direction, difference of 2nd geometry axis
_OVR [17]	REAL	Trigger point in minus direction, difference of 3rd geometry axis
_OVR [19]	REAL	Trigger point in plus direction, difference of 3rd geometry axis
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

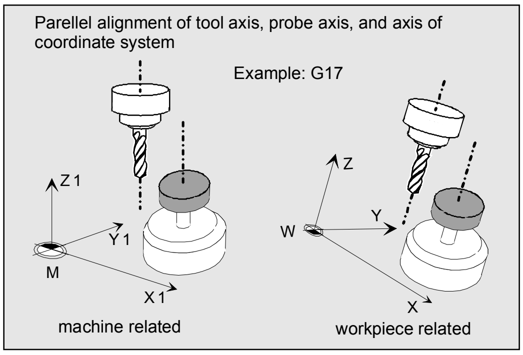
Measuring cycle CYCLE971 returns the following values
in the data block GUD5 for **tool measurement**:

_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 dimensional deviation
_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	D number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [7]	INTEGER	Number of empirical value memory
_OVI [8]	INTEGER	T number
_OVI [9]	INTEGER	Alarm number

© Siemens AG, 2004. All rights reserved

5.2.1 Measuring and compensation strategy

Measuring strategy



Prepositioning the tool
The tool must be aligned vertically with the probe before the measuring cycle is called.
Tool axis parallel to center line of probe.

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

Tool measurement with motionless spindle
Before the cycle call for measurement of **milling tools** the tool and spindle must be moved such that the selected cutting edge can be measured (length or radius).

The measurement feedrate is defined in _VMS.

Tool measurement 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. Length measurement of milling tools with rotating spindle might also be practical.

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 to prevent hard impacts when probing.
- Required measuring accuracy.

© Siemens AG, 2004. All rights reserved

When measuring with rotating tool the relation between measuring feedrate and speed must be taken into account. One cutting edge is taken into account. With multiple cutters the longest edge is used for the measuring result.

The following connections have to be taken into account:

$$n = \frac{S}{2 \cdot \pi \cdot r \cdot 0.001}$$

$$F = n \cdot \Delta$$

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

Example:

Given a peripheral speed of S = 90m/min, speeds of n = 2865 to 143 rpm will result for milling tools with a radius of r = 5 to 100 mm.

Given a specified measuring accuracy of Δ = 0.005 mm measuring feedrates of

F= 14 mm/min to F= 0.7 mm/min will result.

Compensation strategy

The tool measuring cycle is provided for various applications:

- **First-time measurement of a tool**
  (_CHBIT[3]=0):
  The tool offset values in geometry and wear are replaced.
  Compensation is written in the geometry component of length or radius. The wear component is reset.
- **Remeasurement of a tool**
  (_CHBIT[3]=1):
  The resulting difference is calculated into the wear component (radius or length).

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

© Siemens AG, 2004. All rights reserved

Compensation of length 1 or the tool radius only occurs in **tool measurement** if the measured difference lies in the tolerance band between _TZL and _TDIF!

Compensation of the tool probe trigger points _TP[ ] and. _TPW[ ] only occurs when the tool probe is **calibrated** if the measured difference lies in the tolerance band between _TZL and _TSA!

Compensation with correction table when measuring with rotating spindle

When measuring tools with a rotating spindle, the measuring precision can be compensated for by additional compensation values during measurement of the cutter radius or cutter length. These compensation values are stored in tables in the dependency peripheral speed / cutter radius. Users can also create their own compensation values in dedicated tables in data block GUD6.

This offset is activated with variable of data type INTEGER **_MT_COMP** >0.

_MT_COMP= 0: No compensation
_MT_COMP= 1: automatic compensation, i.e. internal compensation when TT130 is used (Heidenhain) or TS27R (Renishaw)
_MT_COMP= 2: compensation with user-defined compensation data, i.e. **even** if (Heidenhain) or Renishaw specified (alternate probe)

With the variables of data type INTEGER**_TP_CF** it is possible to activate the pre-prepared compensation tables of several tool probe models.

_TP_CF= 0: No status
_TP_CF= 1: TT130 (Heidenhain)
_TP_CF= 2: TS27R (Renishaw)

The user can enter his own compensation values in two arrays of data type REAL:

_MT_EC_R[6,5] for radius measurements and
_MT_EC_L[6,5] for length measurement.

© Siemens AG, 2004. All rights reserved

Structure of user arrays

_MT_EC_R _MT_EC_L [n,m]	m=0	m=1	m=2	m=3	m=4
n=0	0	1. Radius	2. Radius	3. Radius	4. Radius
n=1	1. Peripheral speed	Compensation value for 1st radius/ 1. Peripheral speed	Compensation value for 2nd radius/ 1. Peripheral speed	Compensation value for 3rd radius/ 1. Peripheral speed	Compensation value for 4th radius/ 1. Peripheral speed
n=2	2. Peripheral speed	Compensation value for 1st radius/ 2. Peripheral speed	Compensation value for 2nd radius/ 2. Peripheral speed	Compensation value for 3rd radius/ 2. Peripheral speed	Compensation value for 4th radius/ 2. Peripheral speed
n=3	3. Peripheral speed	Compensation value for 1st radius/ 3. Peripheral speed	Compensation value for 2nd radius/ 3. Peripheral speed	Compensation value for 3rd radius/ 3. Peripheral speed	Compensation value for 4th radius/ 3. Peripheral speed
n=4	4. Peripheral speed	Compensation value for 1st radius/ 4. Peripheral speed	Compensation value for 2nd radius/ 4. Peripheral speed	Compensation value for 3rd radius/ 4. Peripheral speed	Compensation value for 4th radius/ 4. Peripheral speed
n=5	5. Peripheral speed	Compensation value for 1st radius/ 5. Peripheral speed	Compensation value for 2nd radius/ 5. Peripheral speed	Compensation value for 3rd radius/ 5. Peripheral speed	Compensation value for 4th radius/ 5. Peripheral speed

Units:	mm or inch	for tool radius and compensation value
	m/min or ft/min	for peripheral speed

Function and notes:

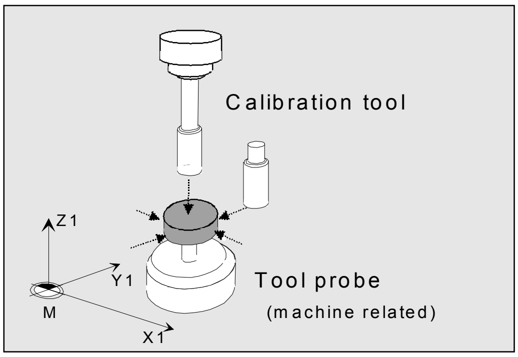
In the as-delivered state of the measuring cycles the default setting of the of the arrays is 0. The radii and peripheral speeds must be entered in ascending order.
Access to these arrays in automatic mode is only possible with **_MT_COMP = 2**. In tool measurement with rotating spindle a compensation value from these tables is calculated based on the tool radius of the tool to be measured. The value for the next lowest table peripheral speed and the next lowest table radius are always the values used. In radius measurement the corresponding compensation value in array **_MT_EC_R [n,m]** is subtracted from the measured tool radius. In length measurement the corresponding compensation value in array **_MT_EC_L [n,m]** is subtracted from the measured tool length.

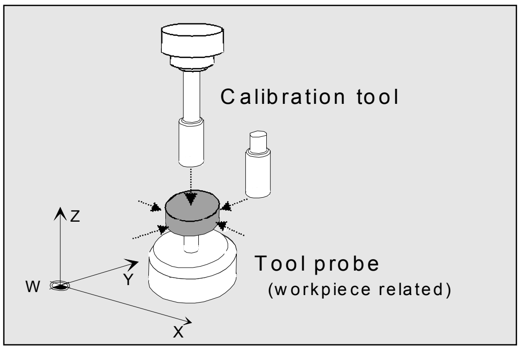
© Siemens AG, 2004. All rights reserved

5.2.2 Calibrating a tool probe

Calibration

The cycle uses the calibration tool to ascertain the current distance dimensions between machine zero (**machine-related** calibration) and workpiece zero (**workpiece-related** calibration) and the tool probe trigger points, and automatically loads them into the appropriate data area in data block GUD6.
Values are corrected without empirical and mean values.





Preconditions
The approximate coordinates of the tool probe must be entered before calibration starts in array _TP[_PRNUM-1, 0] to _TP[_PRNUM-1, 9] (machine related) or _TPW[_PRNUM-1, 0] to _TPW[_PRNUM-1, 9] (workpiece related).
The precise 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. Tool type 120 can be entered. There is no special "calibration tool" type.
Machining plane G17, G18, or G19 must be defined before the cycle is called.
All the necessary parameters have been assigned values.

Parameters

_MVAR	0	Calibrate tool probe (machine-related)
	10	Calibrate tool probes (workpiece-related)
	10000	Calibrate tool probes incrementally (machine-related)
	10010	Calibrate tool probes incrementally (workpiece-related)
_MA	1...3	Number of the measuring axis
	103, 203	Number of the offset and measuring axis
	102, 201	(not for _MVAR=10000 and _MVAR=10010)
_FA	>0	Measurement path
	<0	For **incremental** calibration (_MVAR=1000x0) the travel direction is also defined via _FA. _FA > 0: Travel direction + _FA > 0: Travel direction -

1) from meas. cycles SW 6.3

© Siemens AG, 2004. All rights reserved

_ID	REAL, ≥0	Offset The offset affects calibration of 3rd measuring axis if the calibration tool diameter is larger than the upper diameter of the probe. Here the tool is offset by the tool radius from the center of the probe, minus the value in _ID. The offset axis is also specified in _MA. Parameter _ID should usually set to 0.

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

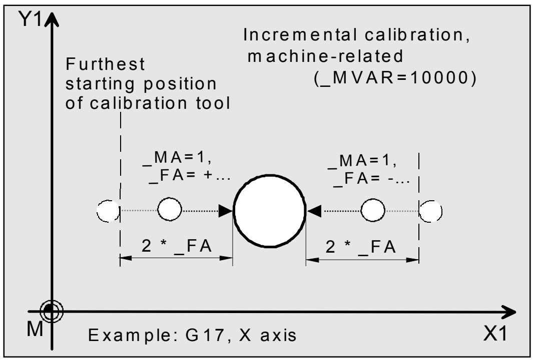
_TZL, _TSA not for incremental calibration!

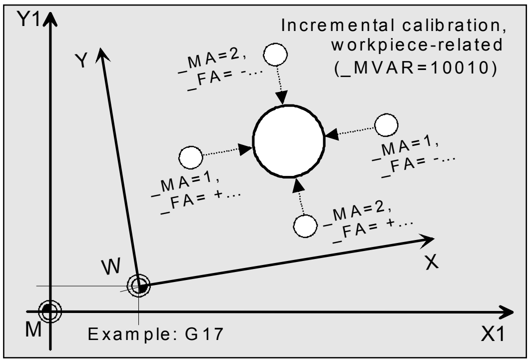
Sequence

Position before measuring cycle call

The calibration tool must be prepositioned as shown in the figure and for the selected variant. The tool must have reached a permissible starting position.

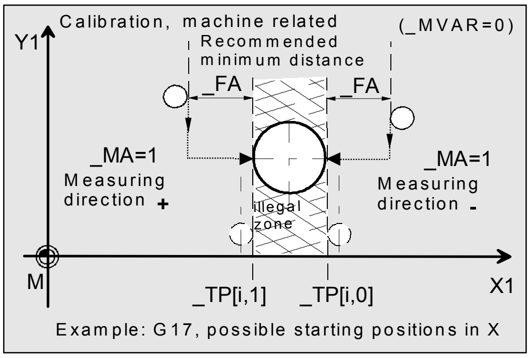
With **incremental calibration**, there is no generation of traversing movements before the actual measured block. The calibration tool must be positioned in front of the tool probe such that the calibration tool traverses to the tool probe when the measuring axis and a signed incremental measuring path up to the expected edge are entered in FA.



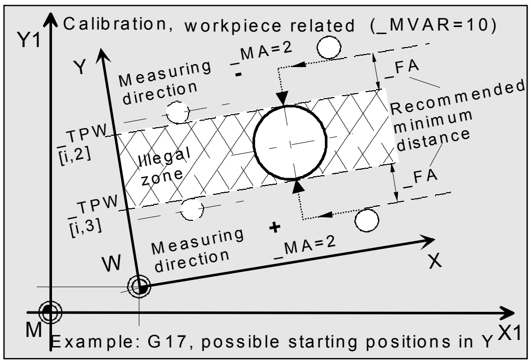


© Siemens AG, 2004. All rights reserved

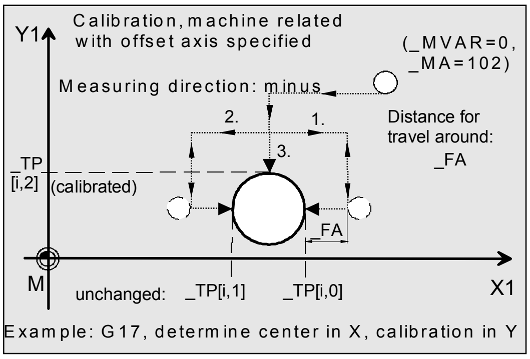
In normal **calibration** the measuring cycle calculates the approach path to the probe independently from the starting position and then generates the appropriate traverse blocks.



Note on calibrating in the 3rd measuring axis (_MA=3, _MA=103, _MA=203):
If the tool diameter (2x $TC_DP6) is smaller than the upper diameter of the probe (_TP[i,6]), the calibration tool is always positioned in the center of the probe.
If the tool diameter is larger, the calibration tool is offset by the tool radius toward the center onto the probe. The value of _ID is subtracted.
The axis in which the offset is applied (offset axis) is also specified in _MA (_MA=103 or MA=203).
If no offset axis is specified (_MA=3), the offset is applied in the abscissa, if necessary, (for G17: X axis).



Sequence on additional offset axis specification
Additionally specifying the offset axis in _MA (_MA= 102 or _MA= 201) first causes the exact center of the tool probe to be detected in the offset axis before calibration takes place in the measuring axis.
An entry in the array is only made for the measuring axis in the selected direction of measurement.



Position after end of measuring cycle
On completion of calibration, the calibration tool (radius) is _FA from the measuring surface.

© Siemens AG, 2004. All rights reserved

Programming example 1

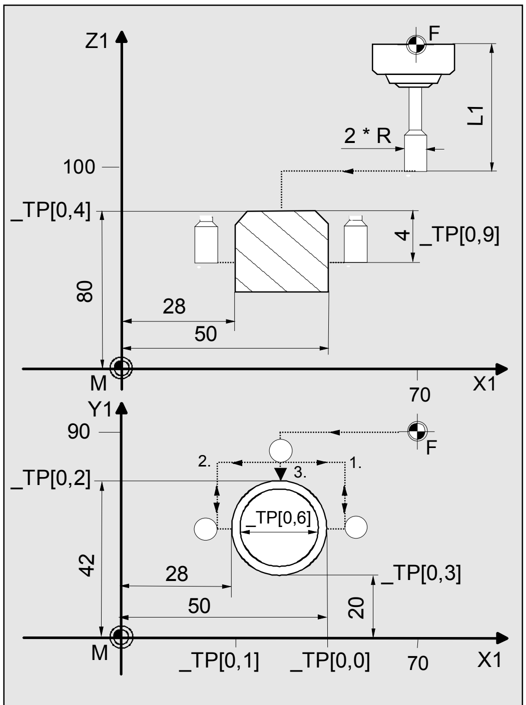
Complete calibration of tool probes (machine-related)

Values of the calibration tool T7 D1:

Tool type (DP1):	120
Length 1 - Geometry (DP3):	L1= 20.000
Radius - Geometry (DP6):	R = 5.000

Values of the tool probe 1 in data block GUD6, approximate values before calibration begins (machine-related):

_TP[0,0] = 50
_TP[0,1] = 28
_TP[0,2] = 42
_TP[0,3] = 20
_TP[0,4] = 80
_TP[0,6] = 20 (upper diameter)
_TP[0,9] = 4



Example: G17, procedure when _MA=102

%_N_CALIBRATE_MTT_MPF	
N05 G0 G17 G94 G90	;Machining plane, define feedrate type
N10 T7 D1	;Select calibration tool
N15 M6	;Change calibration tool and activate offset
N30 SUPA G0 Z100	;Position infeed axis over tool on hole center ;point
N35 SUPA X70 Y90	;Position in plane on tool on hole center ;point
N40 _TZL=0.005 _TSA=5 _VMS=0 _NMSP=1 _PRNUM=1 _FA=6 N41 _MVAR=0 _MA=102	;Parameters for calibration in the Y axis ;with previous calculation of the probe ;center in X. The array applied is ;tool probe 1: _TP[0,i]
N50 CYCLE971	;Calibration in minus Y direction
N55 SUPA Z100	;Traverse up in infeed axis in rapid traverse
N60 SUPA Y0 N65 _MA=2	;Move in plane to position from which ;calibration can start in the plus Y direction
N70 CYCLE971	;Calibration in plus Y direction (probe ;positioned on center in X)
N80 SUPA X70 Z100	;In X axis and Z axis, rapid traverse ;retraction of probe
N85 _MA=1	;Calibration in the X axis
N90 CYCLE971	;Calibration in minus X direction
N100 SUPA Z100	;In Z axis rapid traverse probe retraction

© Siemens AG, 2004. All rights reserved

N110 SUPA X10	;In X axis move to position from which ;calibration can start in the plus direction
N120 CYCLE971	;Calibration in plus X direction
N130 SUPA Z100	;Traverse up in infeed axis
N140 _MA=3	;Calibration in the Z axis on G17
N150 CYCLE971	;Calibration in minus Z direction
N160 M2	;End of program

Explanation

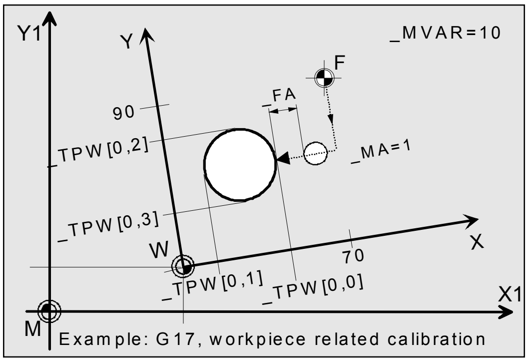
The new trigger values in -X, +X, -Y, +Y, and -Z are stored in the global data of tool probe 1 (_PRNUM=1) _TP[0,0...4] if they deviate by more than 0.005 mm (_TZL=0.005) from the old values. Deviations of up to 5 mm (_TSA=5) are permissible.

Programming example 2

Calibrating tool probe in minus X (workpiece-related) (from measuring cycles SW 6.3)

Values of the calibration tool T7 D1:

Tool type (DP1):	120
Length 1 - Geometry (DP3):	L1= 20.000
Radius - Geometry (DP6):	R = 5.000



Values of the settable ZO for G54:

Displacement: X=60, Y=15, Z= 30

Rotation through: X=0, Y=0, Z=18 degrees

Values of the tool probe 1 in data block GUD6, approximate values before calibration begins (workpiece-related):

_TPW[0,0] = 50
_TPW[0,1] = 28
_TPW[0,2] = 42
_TPW[0,3] = 20
_TPW[0,4] = 80
_TPW[0,9] = 4

© Siemens AG, 2004. All rights reserved

%_N_CALIBRATE_MTT_X_MPF	
N05 G0 G17 G94 G54	;Define machining plane, zero offset and ;feed type
N10 T7 D1	;Select calibration tool
N15 M6	;Change calibration tool and activate offset
N30 G0 Z100	;Position infeed axis over tool on hole center ;point
N35 X70 Y90	;Position in plane on tool on hole center ;point
N40 _TZL=0.005 _TSA=5 _VMS=0 _NMSP=1 _PRNUM=1 _FA=6 N41 _MVAR=10 _MA=1	;Parameters for calibration in the X axis ;The array of tool probe 1 is effective: ;_TPW[0,i]
N50 CYCLE971	;Calibration in minus X direction
N55 Z100	;Traverse up in infeed axis in rapid traverse
N60 M2	;End of program

Explanation

The calibration tool moves with its point from the starting position at N35 (X70, Y90, H100) in Y to the center of the probe Y31 ((_TPW[0,2] + (_TPW[0,3]) / 2 = (42+20) / 2=31)); then in the measuring axis X (_MA=1, G17) to position X61 (_TPW[0,0] + _FA + R = 50 + 6 + 5 = 61 Here it is lowered to position Z76 (_TPW[0,4] – _TPW[0,9] = 80 – 4 = 76). Then measuring (calibration) is performed in the minus X direction. At the end the calibration tool is again at position X61.

The new trigger values in minus X are stored in the data of tool probe 1 (_PRNUM=1) _TP[0,0] if they deviate by more than 0.005 mm (_TZL=0.005) from the old values. Deviations of up to 5 mm (_TSA=5) are permissible.

In block N55 the calibration tool is moved up to position Z100 and the program ends with block N60.

© Siemens AG, 2004. All rights reserved

5.2.3 Calibrating the tool probe automatically (from measuring cycles SW 6.3)

Automatic calibration

Measuring variants

_MVAR=100000 (machine-related)
_MVAR=100010 (workpiece-related)

are used to calibrate the tool probe automatically.
The cycle uses the calibration tool to determine the tool probe trigger points in **all axes** and loads them into the relevant data area of data block GUD6.
A measuring axis does not have to be specified in _MA. Otherwise, the same parameters apply as for calibration of an axis.
Values are corrected without empirical and mean values.

Preconditions
The approximate coordinates of the tool probe must be entered before calibration starts in array _TP[_PRNUM-1, 0] to _TP[_PRNUM-1, 9] (machine related) or _TPW[_PRNUM-1, 0] to _TPW[_PRNUM-1, 9] (workpiece related). These values must be so precise that the parameter values of _TSA and _FA are fulfilled.

The precise 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. Tool type 120 can be entered. There is no special "calibration tool" type.
Machining plane G17, G18, or G19 must be defined before the cycle is called.
All the necessary parameters have been assigned values.

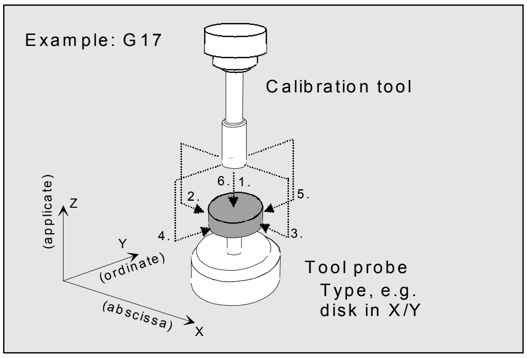
Parameters

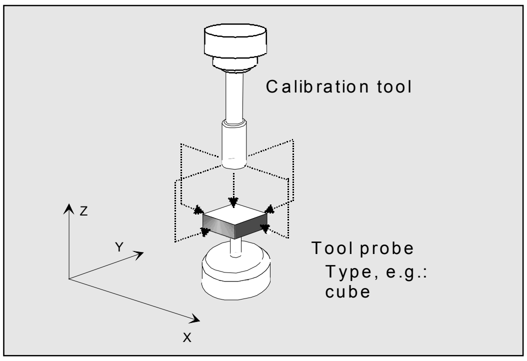
_MVAR	100000	Calibrate tool probe automatically (machine related)
	100010	Calibrate tool probe automatically (workpiece related)
_FA	>0	Measurement path

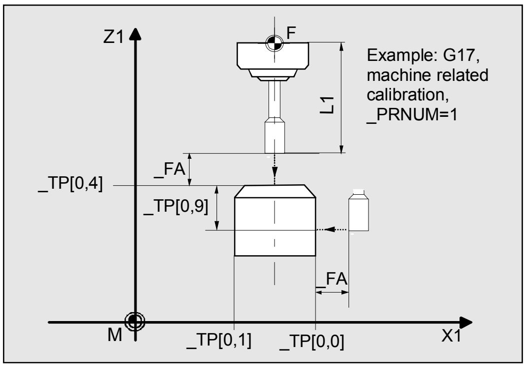
The following additional parameters are also valid
_VMS, _TZL, _TSA, _PRNUM, _ID and **_NMSP**. Set offset _ID=0 as standard
See Sections 2.2 and 2.3.

© Siemens AG, 2004. All rights reserved

Sequence







Position before measuring cycle call
The position before measuring cycle call can be anywhere, but:
The cycle must be able to position the 1st calibration point at distance _FA above the center of the probe without collision.
The cycle approaches this point in the axis sequence: applicate (tool axis) followed by axis of the plane.

All subsequent traversing movements are also performed by the measuring cycle with "automatic calibration" using the values entered in array _TP[ ] or TPW[ ] of the probe and the dimensions of the active calibration tool.

Calibration is performed in this sequence:

- –applicate, +abscissa, –abscissa, +ordinate, –ordinate;

then finally again in

- –applicate, but now in the located center.

For G17, for example, these are the axes:
–Z, +X, –X, +Y, –Y, –Z.
Calibration in the plus direction of the abscissa is performed after ascertaining the center of the probe in the ordinate. Additional movements are performed in the plane.
This sequence applies to
_TP[_PRNUM-1, 7]=133 or
_TPW[_PRNUM-1, 7]=133:
probe in Z axis can only be calibrated in minus direction, X, Y, in both directions.

Value _TP[k, 7] or _TPW[k, 7] =133 is the default value.
If some axes or axis directions on the probe cannot be approached the value must be changed.
Meaning:

Decimal point	units: 1. geometry axis (X)
	tens: 2. geometry axis (Y)
	hundreds: 3. geometry axis (Z)

Value 0: axis not possible
1: only minus direction possible
2: only plus direction possible
3: both directions possible

© Siemens AG, 2004. All rights reserved

Example _TP[k, 7]=123:

X in both directions,

Y only in plus direction,

Z can only be calibrated in minus direction.

It must always be possible to approach the tool axis (applicate, for example, Z axis for G17) in the minus direction. Otherwise automatic calibration is not possible.

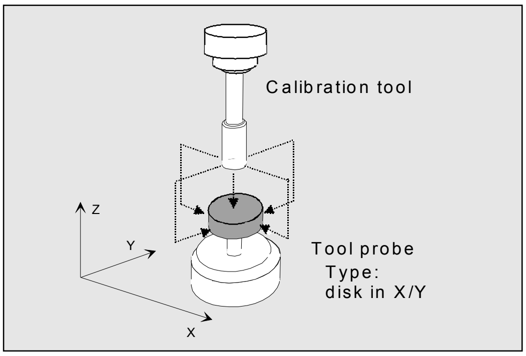
The sequence described above changes according to the value of _TP[k, 7] or _TPW[k, 7].

Position after end of measuring cycle

On successful completion of the calibration process, the calibration tool is positioned distance _FA above the center of the probe.

Programming example

Calibrate tool probe automatically, machine-related for G17



Values of the calibration tool T7 D1:

Tool type (DP1):	120
Length 1 - Geometry (DP3):	L1= 70.123
Radius - Geometry (DP6):	R = 5.000

Values of tool probe 1 in block GUD6 before calibration:

_TP[0,0] = 50 (minus X axis)
_TP[0,1] = 28 (plus X axis)
_TP[0,2] = 42 (minus Y axis)
_TP[0,3] = 20 (plus Y axis)
_TP[0,4] = 80 (minus Z axis)

_TP[0,6] = 21	(wheel diameter at upper edge)
_TP[0,7] = 133	(calibratable: minus Z axis in both directions in X and Y)
_TP[0,8] = 101	(wheel in X/Y)
_TP[0,9] = 4	(distance to upper edge, calibration depth)

%_N_AUTO_CALIBRATE_MPF	
N10 G17 G0 G90 G94	
N20 T7 D1	;Preselect calibration tool
N30 M6	;Change calibration tool and activate offset
N40 SUPA X39 Y31 Z100	;Take up start position

© Siemens AG, 2004. All rights reserved

N20 _MVAR=100000 _FA=6 _TSA=5 _TZL=0.001 _PRNUM=1 _VMS=0 _NMSP=1	;Parameters for calibration cycle
N30 CYCLE971	;Automatic calibration (complete)
N99 M2	

Explanation:
The tool probe is calibrated from the starting position in the –Z, +X, -X, +Y, -Y and again -Z axes. The values are entered in the _TP field and _OVR field if the results (values of the differences) are within the limits:
>_TZL, <_TSA.

© Siemens AG, 2004. All rights reserved

5.2.4 Measuring the tool

Function

The cycle determines the new tool length or the new tool radius and checks whether the difference can be corrected with an empirical value to the old tool length or radius within a defined tolerance range (upper limits: Safe area _TSA and dimensional deviation check _TDIF, lower limit: zero offset range _TZL, see Section 1.9).
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.

Precondition
- The tool probe must be calibrated.
- The tool geometry data (approximate values) 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 as way that collision-free approach to the probe is possible in the measuring cycle.

Special features of measurement with rotating spindle
- An additional compensation can be activated with variable _MT_COMP>0.
  (see Subsection 5.2.1 Measurement and compensation strategy)
- By default, the cycle-internal calculation of feed and speed is executed from the limit values defined in array _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.
  If probing is performed several times the speed is additionally reduced on the last probing operation. This speed reduction can be suppressed by setting channel-related bit _CHBIT[22].

© Siemens AG, 2004. All rights reserved

- The operator can deactivate the cycle-internal calculation via the measuring cycle bit _CBIT[12]=1 and specify his or her own values for feed and speed.
  Array _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] (speed/feedrate) 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 functions from array _CM[] are not active!
- 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]	REAL	Maximum permissible peripheral speed [m/min]/[feet/min] Default setting: 100 m/min.
_CM[1]	REAL	Maximum permissible speed for measuring with rotating spindle [rpm] (if it is exceeded, the speed is automatically reduced) Default setting: 1000 rpm
_CM[2]	REAL	Minimum feedrate for first probing operation [mm/min]/[inch/min] (prevents feeds from being too low with large tool radii) Default setting: 1mm/min.
_CM[3]	REAL	Required measuring accuracy [mm]/[inch] is effective with the last probing action Default setting: 0.005 mm
_CM[4]	REAL	Maximum feedrate for probing [mm/min]/[inch/min] Default setting: 20 mm/min.
_CM[5]	REAL	Direction of spindle rotation during measuring Default setting: 4 = M4
_CM[6]	REAL	Feed factor 1 0: Only one probing action at calculated feed (but at least value of _CM[2]) ≥1: 1. Probing with calculated feedrate (but at least value of _CM[2]) · feedrate factor 1 Default setting: 10
_CM[7]	REAL	Feed factor 2 0: 2. Probing with calculated feedrate (only active when _CM[6]>0) ≥1: 2. Probing action with calculated feed · Feed factor 2 3. Probing with calculated feed Feed factor 2 should be smaller than feed factor 1. Default setting: 0

Notice

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

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	1	Measure with motionless spindle, machine related
	2	Measure with rotating spindle, machine related
	11	Measure with motionless spindle, workpiece related[1)]
	12	Measure with rotating spindle, workpiece related[1)]
_MA		Number of the measuring axis
	1	Measure radius in abscissa direction (milling tool)
	2	Measure radius in ordinate direction (milling tool)
	3	Measure length at center point of the tool probe (drill or milling tool)
	103	Measure length, offset by radius in abscissa direction (milling tool)
	203	Measure length, offset by radius in ordinate direction (milling tool)
_ID	REAL, ≥ 0	Offset 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 and 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

The following additional parameters are also valid
_VMS, _TZL, _TDIF, _TSA, _FA, _PRNUM,
_EVNUM and **_NMSP**.
See Sections 2.2 and 2.3.
1) from meas. cycles SW 6.3

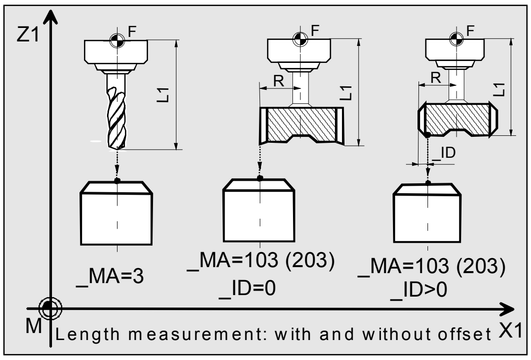
© Siemens AG, 2004. All rights reserved

Length measuring variants

Example: G17, machine related

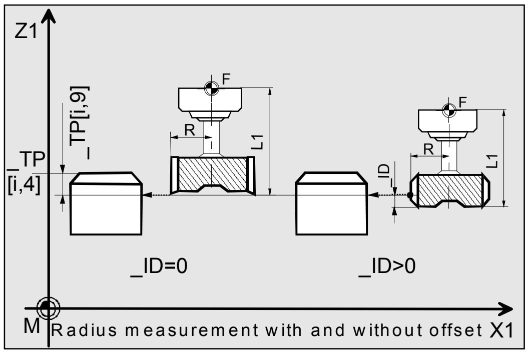
Note:

If the tool diameter (2x $TC_DP6) is smaller than the upper diameter of the probe (_TP[i,6]), the tool is always positioned in the center of the probe, if the tool diameter is larger, the tool is offset by the tool radius toward the center onto the probe. The value of _ID is subtracted. The axis in which the offset is applied (offset axis) is also specified in _MA (_MA=103 or MA=203). If no offset axis is specified (_MA=3), the offset is applied in the abscissa, if necessary, (for G17: X axis).



Radius measuring variants (milling tool)

Example: G17, machine related, _MA=1



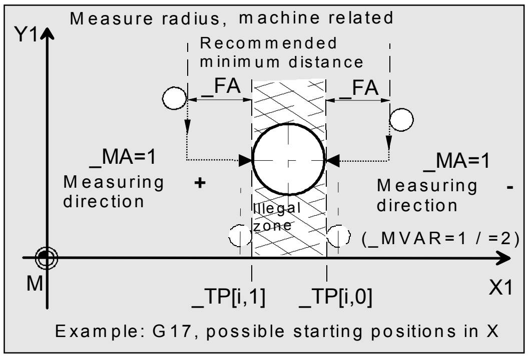
Sequence

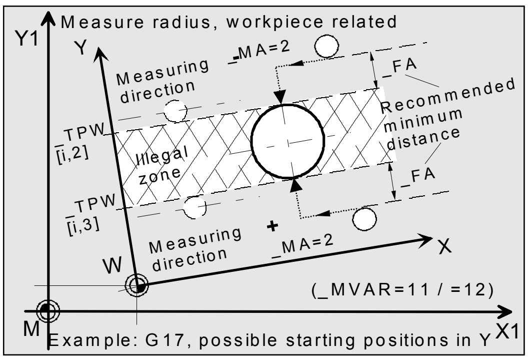
Position before measuring cycle call

Before cycle call a starting position must be taken up from which approach to the probe is possible without collision. The measuring cycle calculates the continued approach path and generates the necessary travel blocks.

Position after end of measuring cycle

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





© Siemens AG, 2004. All rights reserved

Programming example 1

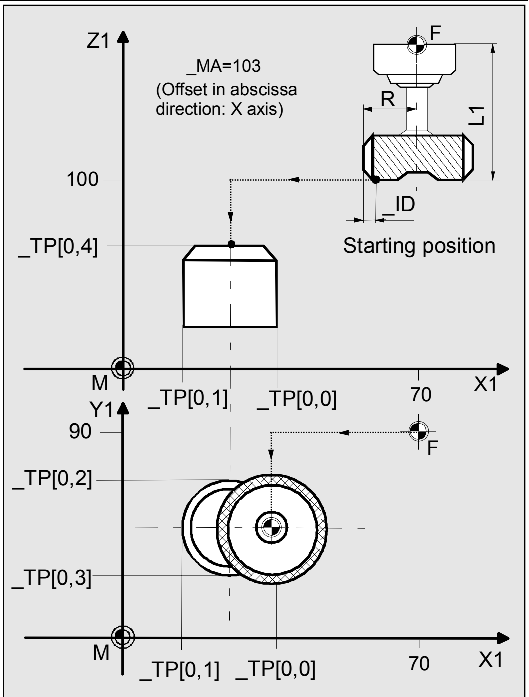
Measuring the length and radius or a milling tool (machine related)

Milling tool F3 with D1 is to be measured for the first time along length L1 and radius R (to determine geometry).
The length measurement is to be performed with motionless spindle. Radius measurement is to be performed with rotating spindle – in the X axis.
The tool has a specially shaped cutting edge and therefore requires an offset for measurement.

A measured value deviation of < 1.6 mm compared with the entered values is expected.
Values of tool T3, D1, before measurement:

Tool type (DP1):	123
Length 1 - Geometry (DP3):	L1= 70
Radius - Geometry (DP6):	R = 18
Length 1 - Wear (DP12):	0
Radius - Wear (DP15):	0

Tool probe 1 is used. It has already been calibrated under the same conditions.
Values: See programming example 1 "Calibration"



Example: G17, measuring tool radius with _ID>0

%_N_T3_MEAS_MPF	
N01 G17 G90 G94	
N05 T3 D1	;Selection of the tool to be measured
N10 M6	;Insert tool, offset active
N15 G0 SUPA Z100	;Position infeed axis with probe
N16 SUPA X70 Y90 SPOS=15	;Position X/Y, align cutting edge ;(if required)
N20 _CHBIT[3]=0 _CBIT[12]=0	;Offset of tool geometry, cycle calculation of ;feedrate and speed during measurement ;with rotating spindle
N30 _TZL=0.04 _TDIF=1.6 _TSA=2 _PRNUM=1 _VMS=0 _NMSP=1 _FA=3 _EVNUM=0	;Parameters for cycle
N31 _ID=2.2 _MVAR=1 _MA=103	;Offset in X axis for length measurement
N40 CYCLE971	;Measure length with motionless spindle
N50 SUPA X70	;Retract from probe in X
N70 _ID=2.4 _MA=1 _MVAR=2	;New offset for radius measurement
N80 CYCLE971	;Measure radius in minus X direction with ;rotating spindle
N90 SUPA Z100 M2	;Raise in Z, end of program

© Siemens AG, 2004. All rights reserved

Explanation

Length 1 (derived in block N40) and the radius (derived in block N80) of the active tool (T3, D1) are entered in the relevant geometry memory (_CHBIT[3]=3) if they

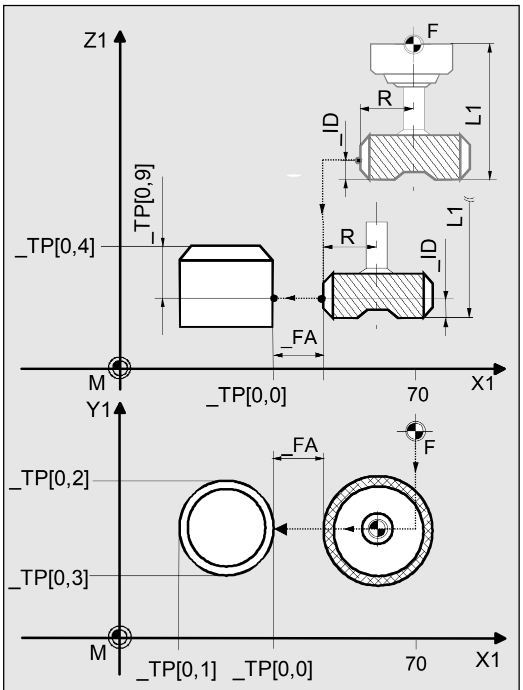
deviate by more than 0.04 mm (_TZL=0.04) and less than 1.6 mm (_TDIF=1.6)

from entered values L1, R.

If the differences are ≥ _TDIF or _TSA alarms are output.

Values are corrected without empirical values (_EVNUM=0).

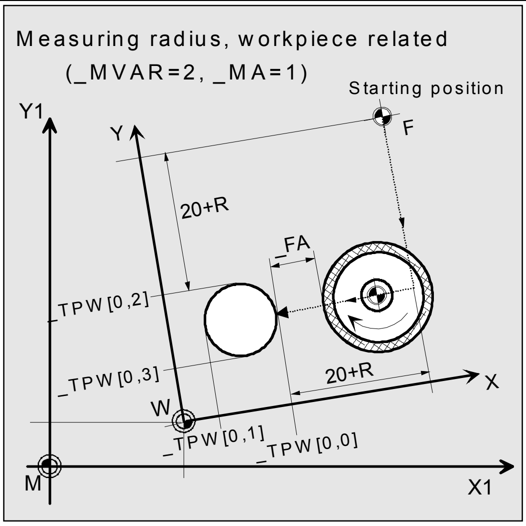
Wear values L1 and R of the tool are reset (_CHBIT[3]=0).



Example: G17, measuring tool radius with _ID>0

Programming example 2

Measuring the radius of a milling tool (workpiece-related) (from measuring cycles SW 6.3)



Milling tool T4 with D1 is to be remeasured in radius R (to ascertain wear).
Radius measurement is to be performed with rotating spindle – in the X axis.
A measured value deviation of < 0.6 mm compared with the entered values is expected.

Values of tool T4, D1, before measurement:

Tool type (DP1):	120
Length 1 - Geometry (DP3):	L1= 70
Radius - Geometry (DP6):	Rg = 18.0
Radius - Wear (DP15):	Rv= 0.024
	R = Rg + Rv

Values of the settable ZO for G54:
Displacement: X=60, Y=15, Z= 30
Rotation through: X=0, Y=0, Z=18 degrees

Tool probe 1 is used. It has already been calibrated under the same conditions (G17, G54, …).
Values: See programming example 2 "Calibration"

%_N_T4_MEASURE_MPF	
N01 _PRNUM=1	;Select tool probe 1
N02 G17 G54 G94 G90	;Plane, ZO, feedrate type, dimensioning
N05 T4 D1	;Selection of the tool to be measured
N10 M6	;Insert tool, offset active
N15 G0 Z=_TPW[_PRNUM-1,4]+20	;Position in infeed axis above the tool probe
N16 X=_TPW[_PRNUM-1,0]+$P_TOOLR+20 Y=_TPW[_PRNUM-1,2] +$P_TOOLR +20	;Position X/Y plane: tool edge 20 mm next to ;probe edge +X, +Y
N20 _CHBIT[3]=1 _CBIT[12]=0	;Offset in wear, internal cycle calculation ;of feedrate and speed during measurement ;with rotating spindle
N30 _TZL=0.04 _TDIF=0.6 _TSA=2 _VMS=0 _NMSP=1 _FA=3 _EVNUM=0	;Remaining parameters for cycle
N31 _ID=0 _MVAR=2 _MA=1	;Without offset
N40 CYCLE971	;Measurement with rotating spindle
N50 Z=_TPW[_PRNUM-1,4]+20	;Raise from probe in Z
N60 M2	

© Siemens AG, 2004. All rights reserved

Explanation

The tool moves in N40 (in cycle) with its point from the starting position in N16 in Y to the center of the probe (_TPW[0,2] + (_TPW[0,3]) / 2); then in the measuring axis X (_MA=1, G17) to position (_TPW[0,0] + _FA + R). Here it is lowered to the position in Z (_TPW[0,4] – _TPW[0,9]. Then measuring is performed in the minus X direction. At the end the tool (radius) is again positioned distance _FA in X in front of the probe. In block N50 the tool is raised 20 mm in Z above the probe. Then the program is ended (N60).

The difference in radius (derived in block N40) of the active tool (T4, D1) is subtracted from the wear and entered (_CHBIT[3]=1), if they deviate

by more than 0.04 mm (_TZL=0.04) and
less than 0.6 mm (_TDIF=0.6).

If the difference is ≥ _TDIF or _TSA, alarms are output.

Values are corrected without empirical values (_EVNUM=0).

© Siemens AG, 2004. All rights reserved

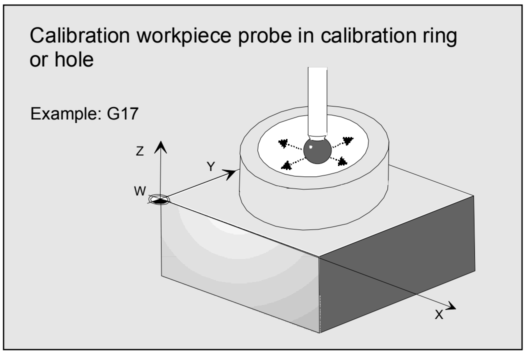
5.3 CYCLE976 calibrate workpiece probe

Programming

CYCLE976

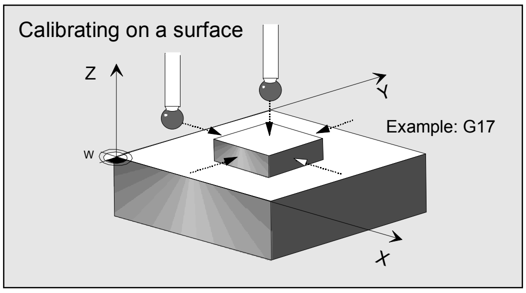
Function

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.

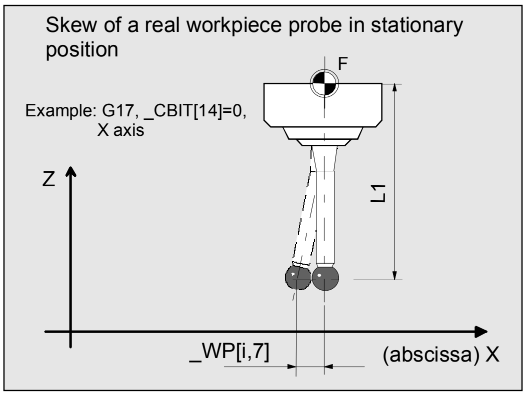


The probe trigger points must be determined in the axis directions that are dependent on:

- the probe ball diameter
- the mechanical design of the probe
- speed at which the probe hits an obstacle



With this calibration cycle it is possible to calibrate a workpiece probe **in a hole** (axes in the plane) or **on suitable surfaces** for a particular axis and direction.
Determining the skew of the workpiece probe
A real workpiece probe can deviate from its ideal vertical position even when not deflected. This positional deviation (skew) can be determined with measuring variants in this cycle and then entered in the intended array of the workpiece _WP[i, 7] for abscissa and _WP[i, 8] for ordinate (for detailed data: see Section 9.2).
These values are taken into account in precise probe positioning in subsequent measurements with a similarly calibrated workpiece probe.



Workpiece probe types that can be used

- Multidirectional probe (_PRNUM=xy)
- Monodirectional, bidirectional probe (_PRNUM=1xy)

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE976 permits the following **calibration** variants which are specified via parameter _MVAR.

- Calibration in hole (axes of the plane)

Digit						*Meaning*
6	**5**	**4**	**3**	**2**	**1**	
					1	Hole (for measurement in the plane), center of the hole known
					8	Hole (for measurement in the plane), center of the hole not known
				0		With any data in the plane (workpiece-related)
			0			Without including probe ball in calculation
			1			Including probe ball in calculation (for measurement in plane)
		0				4 axis directions
		1				1 axis direction (also specify measuring axis and axis direction)
		2				2 axis directions (also specify measuring axis)
	0					Without determining skew of probe
	1					Determining skew of probe
0						Paraxial calibration (in the plane)
1						Calibration at an angle (in the plane)

Note:
When _MVAR=xx1x0x calibration is only performed in one direction. It is not possible to determine skew or calculate probe ball.

- Calibration on surface

Digit						*Meaning*
6	**5**	**4**	**3**	**2**	**1**	
					0	Calibration on surface (workpiece-related)
	1	0	0	0	0	Calibration on a surface in the applicate with calculation of probe length (workpiece-related, only permissible when _MA=3)

© Siemens AG, 2004. All rights reserved

Result parameters

Measuring cycle CYCLE976 returns the following values in data block GUD5 for **calibration**:

_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 (skew of probe)
_OVR [21]	REAL	Positional deviation ordinate (skew of probe)
_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	Probe number
_OVI [9]	INTEGER	Alarm number

1) For calibration variant with unknown
Hole center point

© Siemens AG, 2004. All rights reserved

5.3.1 Calibrating a workpiece probe in a hole of known hole center point

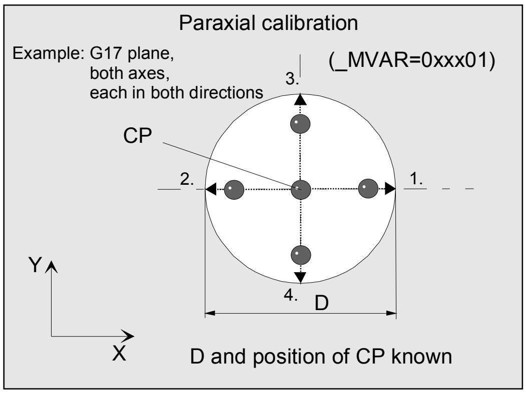
Function

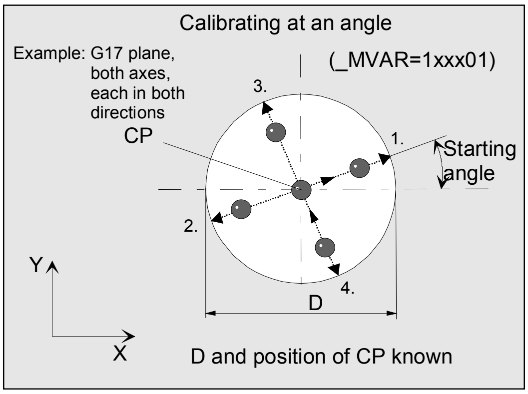
With measuring cycle and measuring variant
_MVAR= xxxx01
it is possible to calibrate the probe in the axes of the plane (G17, G18, or G19) in a calibration ring. A hole that is perpendicular to the selected plane and with the same quality requirements regarding geometrical accuracy and surface roughness can also be used.

The center point (CP) of the hole and its diameter (D) must be known for this calibration variant!

The calculated trigger points are automatically loaded in the relevant data area _WP[ ] of block 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 performed either **paraxially** with the axes of the active workpiece coordinate system **or at an angle** to these axes.
The number of axes and axis directions can be selected in _MVAR. If fewer than four axis directions are selected (_MVAR= xx1xx01, xx2xx01), additional information must be supplied in _MA and possibly in _MD.





Parameters

_MVAR	xxxx01	Calibration variant
_SETVAL	REAL, >0	Calibration setpoint = diameter of hole
_MA	1, 2	Meas. axis, only for _MVAR= xx1xx01, = xx2xx01 (only 1 axis or only 1 axis direction)
_MD	0 positive axis direction 1 negative axis direction	Meas. axis, only for _MVAR= xx1x01 (calibrate one axis direction only)
_PRNUM	>0	Probe number
_STA1	REAL	Starting angle, only for _MVAR= 1xxx01 (calibration performed at this angle)

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

_CORA only relevant for monodirectional probe

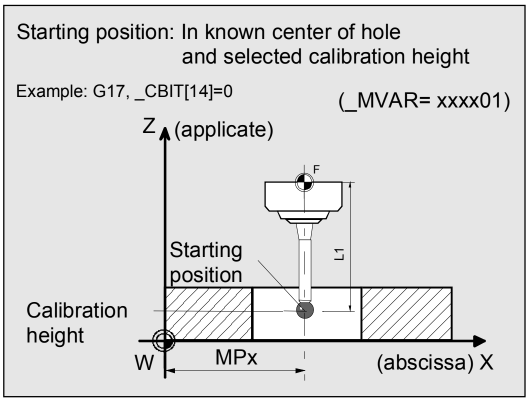
Precondition

The probe must be called with tool length offset. Tool type, preferably: 710.

Notice

The first time calibration is performed the default setting in the array of probe _WP[ ] is still "0". For that reason _TSA> probe ball radius must be programmed to avoid alarm "Safe area violated".

Sequence



Position before measuring cycle call

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

Axis sequence, axis direction sequence

Paraxial, two axis directions:
Calibration starts in the positive axis direction. If _MVAR=xx0xx1 (all four directions), calibration starts in the abscissa. This is followed by the ordinate.

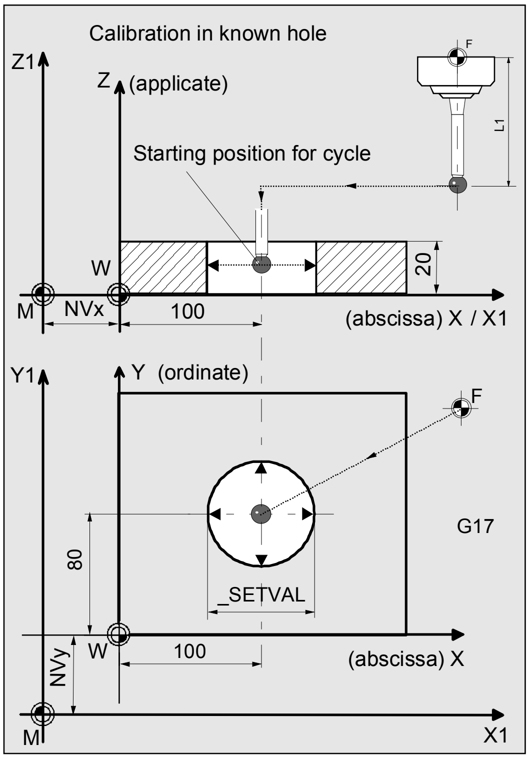
At an angle:
The axes travel in combination acc. to starting angle _STA1 plus steps of 90 degrees. Otherwise the same principle as for "paraxial" applies.

Position after end of measuring cycle

When calibration is complete the probe is again positioned at calibration depth in the center of the hole.

© Siemens AG, 2004. All rights reserved

Programming example



Calibrating a workpiece probe in the X-Y plane, known hole center point

Workpiece probe 3, used as tool T9, D1, is to be recalibrated in a known hole with

MPx=100,000, MPy=80,000, D=110,246 mm

in axes X and Y in both axis directions with G17 and paraxially (to redetermine trigger values _WP[i,1] to _WP[i,4].

The skew (positional deviation _WP[i,7], _WP[i,8]) and precise ball diameter _WP[i,0] of the probe is also to be ascertained.

The radius of the probe ball and length 1 must be entered in the tool offset memory under T9 D1, before the cycle is called.

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) must refer to the center of the probe ball: _CBIT[14]=0.

Careful when positioning! Radius R in length (L1) is ignored. But the desired calibration height can be entered directly.

Zero offset, with settable ZO G54:

NVx, NVy, ...

Arrays for workpiece probe 3: _WP[2, ...]
(already contains approximate values)

%_N_CALIBRATE_IN_X_Y_MPF	
N10 G54 G90 G17 T9 D1	;ZO selection, select probe as tool and ;plane
N20 M6	;Insert probe and activate tool offset
N30 G0 X100.000 Y80.000	:Position probe at center of on hole center ;point
N40 Z10	;Position probe in hole at calibration height
N50 _CBIT[14]=0	;Length 1 relative to probe ball center

© Siemens AG, 2004. All rights reserved

N60 _TSA=1 _PRNUM=3 _VMS=0 _NMSP=1 _FA=1 _TZL=0 N61 _MVAR=010101 _SETVAL=110.246	;Assign parameters for calibration cycle: ;Calibrate probe 3 in ;4 axis directions with calculation of skew ;and active diameter of probe ball
N70 CYCLE976	;Measuring cycle call, calibrate paraxially
N80 Z40	;Position probe above workpiece
N100 M2	;End of program

Explanation

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], _WP[2,8], the active probe ball diameter in _WP[2,0].

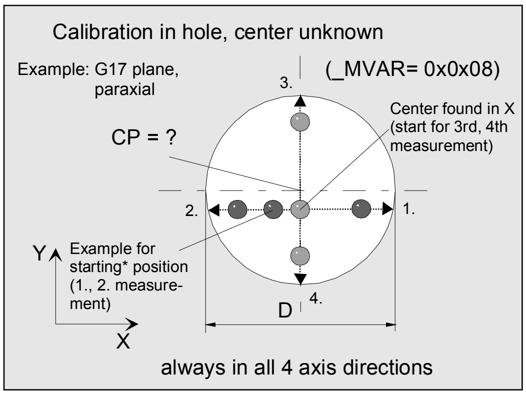
5.3.2 Calibrating a workpiece probe in a hole of unknown hole center point

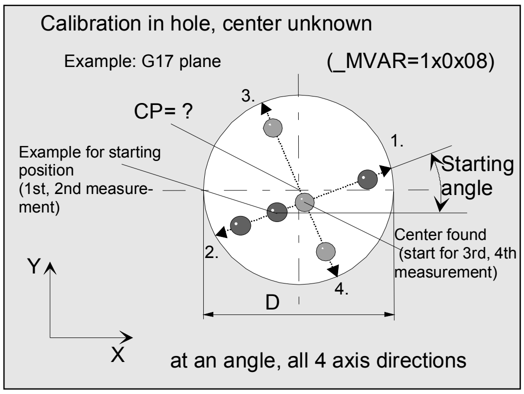
Function

With measuring cycle and measuring variant

_MVAR= xx0x08

it is possible to calibrate the probe in the axes of the plane (G17, G18, or G19) in a calibration ring. A hole that is perpendicular to the selected plane and with the same quality requirements regarding geometrical accuracy and surface roughness can also be used.
The center (CP) of the hole is **unknown** in the precise position. But diameter (D) is known.





In this measuring variant first the hole center and then the positional deviation (skew) of the probe is calculated. Then the trigger points in all 4 axis directions on the plane are calculated.
In addition to the values in array _WP[ ], the measuring cycle also provides the determined hole center point in result array _OVR[6], _OVR[7].
Calibration can be performed paraxially or at an angle to the active workpiece coordinate system. All 4 axis directions are always calibrated.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	xx0x08	Calibration in hole, center unknown
_SETVAL	REAL, >0	Calibration setpoint = diameter of hole
_PRNUM	>0	Probe number
_STA1	REAL	Starting angle, only for _MVAR= 1xxx08 (calibration performed at this angle)

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

_CORA only relevant for monodirectional probe

Precondition

- The probe must be called with tool length offset.
- Tool type, preferably: 710.
- The exact diameter of the hole is known.
- The spindle must be SPOS-capable.
- **Probe in spindle can be positioned 0...360 degrees (all-round coverage).**

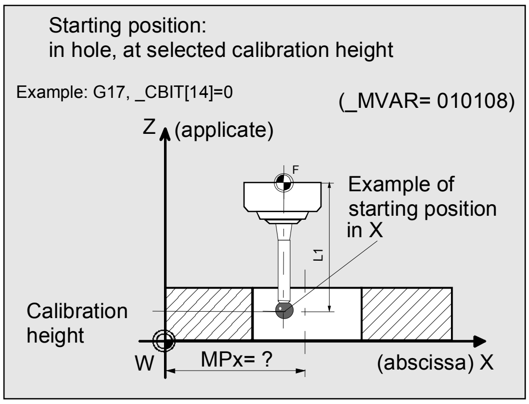
Notice

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

Sequence

Position before measuring cycle call

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



Axis sequence, axis direction sequence

- Paraxial:
  Calibration always starts in the positive axis direction, first in the abscissa, then in the ordinate.
- At an angle:
  The axes always travel in combination acc. to starting angle _STA1, _STA1+180 degrees, _STA1+90 degrees, and _STA1+270 degrees.

Otherwise the same principle as for "paraxial" applies.

© Siemens AG, 2004. All rights reserved

2 measuring runs, one with spindle reversal
The cycle performs two measurement operations to determine the skew of the probe and the center point of the hole.

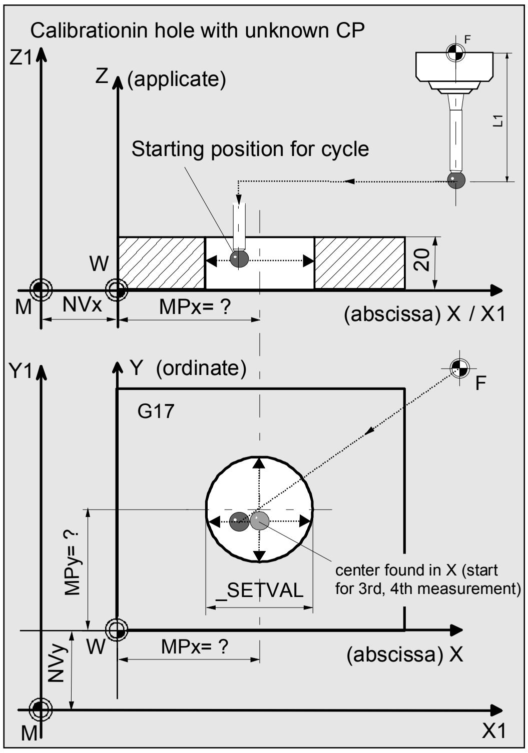
1. Spindle positioned 180 degrees from initial position with SPOS and all axis directions traversed.
2. Spindle positioned at initial position and all axis directions traversed again.

Position after end of measuring cycle
When calibration is complete the probe is again positioned at calibration height in the center of the hole.

Note
Repeating calibration using the determined hole center is advisable if the starting position at the beginning is severely eccentric or measuring accuracy demands are high.

© Siemens AG, 2004. All rights reserved

Programming example



Calibrating a workpiece probe in the X-Y plane, unknown hole center point

Workpiece probe 2, used as tool T10, D1, and not precisely known center point (CP) is to be calibrated in axes X and Y in both axis directions with G17 and paraxially (to redetermine trigger values _WP[i,1] to _WP[i,4].
The skew (positional deviation _WP[i,7], _WP[i,8]) and precise ball diameter _WP[i,0] of the probe is also to be ascertained.

The radius of the probe ball and length 1 must be entered in the tool offset memory under T10, D1, before the cycle is called.

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) must refer to the center of the probe ball: _CBIT[14]=0.
Careful when positioning! Radius R in length (L1) is ignored. But the desired calibration height can be entered directly.

Zero offset, with settable ZO G54:
NVx, NVy, ...

Arrays for workpiece probe 2: _WP[1, ...]
(already contains approximate values)

%_N_CALIBRATE2_IN_X_Y_MPF	
N10 G54 G90 G17 T10 D1	;ZO selection, select probe as tool and ;plane
N20 M6	;Insert probe and activate tool offset
N30 G0 X100 Y80	;Position probe in hole
N40 Z10	;Position probe in hole at calibration height
N50 _CBIT[14]=0	;Length 1 relative to probe ball center

© Siemens AG, 2004. All rights reserved

N60 _TSA=1 _PRNUM=2 _VMS=0 _NMSP=1 _TZL=0 **N61 _MVAR=010108 _SETVAL=110.246** **_FA=_SETVAL/2**	;Assign parameters for calibration cycle: ;calibrate probe 2 in 4 axis directions with ;calculation of skew and calculation of ;diameter of probe ball
N70 CYCLE976	;Measuring cycle call, calibrate paraxially
N80 Z40	;Position probe above workpiece
N100 M2	;End of program

Explanation

The hole center is determined twice, the spindle with the probe being rotated through 180° between each measurement if a multi probe is used, in order to record any positional deviation (skew) of the measuring probe. Triggering is then determined in all 4 axis directions.

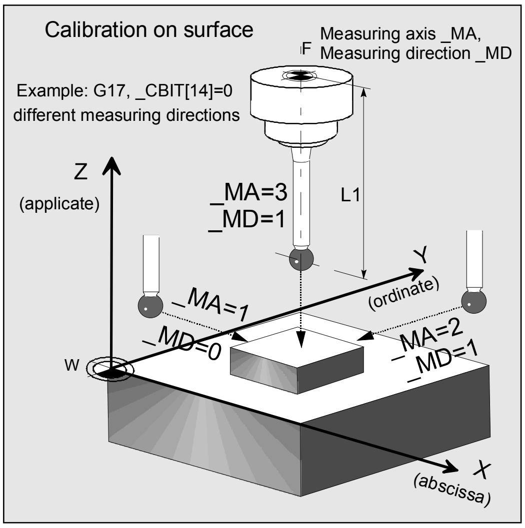
The new trigger values in -X, +X, -Y and +Y are stored in the global data of probe 2_WP[1,1...4], the positional deviation in the X and Y direction in _WP[1,7], _WP[1,8], the active probe ball diameter in _WP[1,0]. The calculated hole center is entered in OVR[6], OVR[7].

© Siemens AG, 2004. All rights reserved

5.3.3 Calibration of a workpiece probe on a surface

Function

With this measuring cycle and measuring variant
_MVAR=0
a workpiece probe can be calibrated in one axis and one direction on a known surface with sufficiently good surface roughness and which is perpendicular to the measuring axis . This can be done on a workpiece, for example.
The trigger point of the relevant axis and axis direction is calculated and entered in the workpiece probe array _WP[i,1] to _WP[i,5] provided.



Precondition
The probe must be called as a tool with a tool length offset.
Tool type, preferably: 710

When using the cycle on a turning machine:
set type and _CBIT[14]=0.

Parameters

_MVAR	0	Calibration variant: Calibration on surface
_SETVAL	REAL	Calibration setpoint (position of surface)
_MA	1, 2, or 3	Measuring axis
_MD	0 positive axis direction 1 negative axis direction	Measuring direction
_PRNUM	INT, >0	Probe number

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

_CORA only relevant for monodirectional probe.

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

© Siemens AG, 2004. All rights reserved

Sequence

Position before measuring cycle call

The probe must be positioned facing the calibration surface. Recommended distance: >_FA.

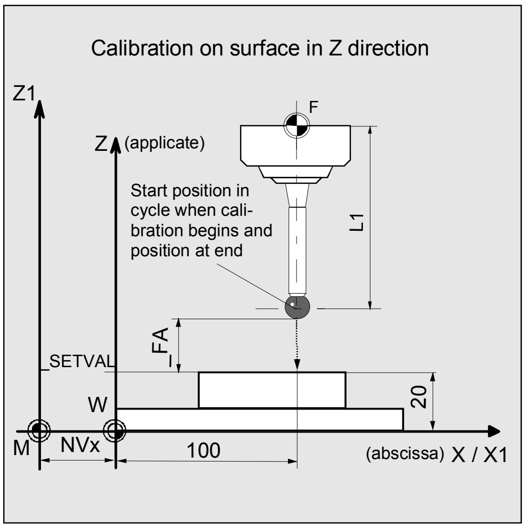
Position after end of measuring cycle

When calibration is complete the probe (ball radius) is distance _FA from the calibration surface if _MA=3, if _MA=1 or _MA=2 it is at the starting position.

Programming example

Calibrating a workpiece probe on the workpiece.

Workpiece probe 1 is to be calibrated in the Z axis on the surface at position Z=20,000 mm of a clamped workpiece. Determine trigger value in minus direction _WP[0.5].



Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ...

The workpiece probe is to be inserted as tool T9 with offset D1.
The radius of the probe ball and length 1 must be entered in the tool offset memory under T9 D1, before the cycle is called.

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) must refer to the center of the probe ball: _CBIT[14]=0.
Careful when positioning! Radius R in length (L1) is ignored. But the desired calibration height can be entered directly when calibrating in the abscissa or ordinate (_MA=1, =2).
Arrays for workpiece probe 1: _WP[0, ...]

© Siemens AG, 2004. All rights reserved

%_N_CALIBRATE_IN_Z_MPF	
N10 G54 G90 G17 T9 D1	;ZO selection, select probe as tool and ;plane
N20 M6	;Insert probe and activate tool offset
N30 G0 X100 Y80	;Position probe above surface
N40 Z55	;Lower probe, distance > (_FA + R)
N50 _CBIT[14]=0	;Length 1 relative to probe ball center
N60 _TSA=4 _TZL=0 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2 **N61 _MVAR=0 _SETVAL=20 _MA=3 _MD=1**	;Define parameter for calibration cycle, ;calibrate probe 1 in minus Z axis
N70 CYCLE976	;Measuring cycle call
N80 Z55	;Position probe above workpiece
N100 M2	;End of program

Explanation

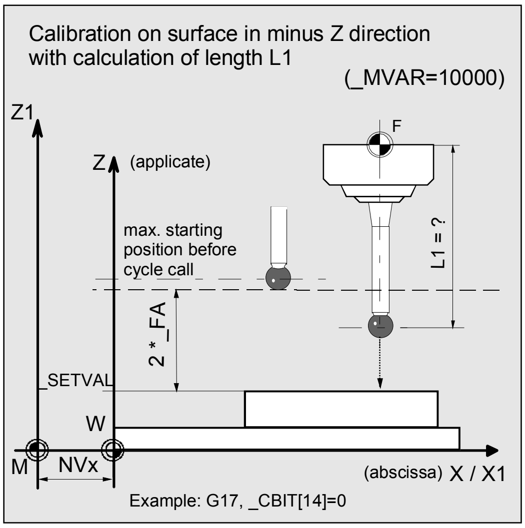
The new trigger value in the minus Z direction is entered in the global data of workpiece probe 1 in _WP[0,5].

© Siemens AG, 2004. All rights reserved

5.3.4 Calibrating a workpiece probe in the applicate determining probe length

Function

With this measuring cycle and measuring variant
_MVAR=10000
the workpiece probe can be calibrated in the tool axis (applicate) on a known surface with sufficiently good surface roughness and which is perpendicular to the measuring axis.
This can be done on a workpiece, for example.
The trigger point of the relevant axis and axis direction is calculated and entered in the workpiece probe array _WP[i,5] provided.
At the same time length 1 (L1) of the probe is calculated according to the setting of _CBIT[14]:
_CBIT[14]=0: L1 referred to ball center
_CBIT[14]=1: L1 ref. to ball circumference
and entered in the tool offset memory.



Precondition
The probe must be called as a tool with a tool length offset.
Tool type, preferably: 710

When using the cycle on a turning machine:
set type and _CBIT[14]=0.

Notice
If you want to position with the tool in the program, the approximate probe length should be known and entered in the tool offset memory. Otherwise position the probe with JOG in front of the calibration surface.
The exact ball radius must be known and entered.
The first time calibration is performed the default setting in the array of the probe is still "0". Therefore:
_TSA> probe ball radius must be programmed to avoid alarm "Safe area violated".

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	10000	Calibration in applicate with length calculation
_SETVAL	REAL	Calibration setpoint (position of surface)
_MA	3	Measuring axis, only tool axis (applicate) possible
_MD	0 positive axis direction 1 negative axis direction	Measuring direction
_PRNUM	>0	Probe number

The following additional parameters are also valid **_VMS, _CORA, _TSA, _FA** and **_NMSP**.

_CORA only relevant for monodirectional probe. _TSA is only evaluated with reference to the trigger value, not with reference to the tool length.

See Sections 2.2 and 2.3.

Sequence

Position before measuring cycle call

The probe must be positioned facing the calibration surface such that it is deflected within **the max. measuring path of 2** · **_FA [mm]** on cycle start.

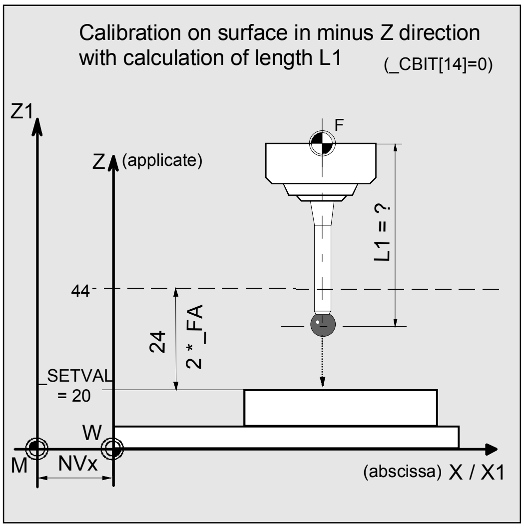
Position after end of measuring cycle

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

Programming example

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

Workpiece probe 1 is to be calibrated in the Z axis on the surface at position Z=20,000 mm of a clamped workpiece. Determine trigger value in minus direction _WP[0,5] and length 1 (L1).



Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ...

The workpiece probe is inserted as tool T9 with offset D1.
The radius of the probe ball must be entered in the tool offset memory under T9, D1 before the cycle is called.

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= ?
Radius (DP6):	R = 3.000

© Siemens AG, 2004. All rights reserved

Length 1 (L1) must refer to the center of the probe ball: _CBIT[14]=0.
Careful when positioning! Radius R in length (L1) is ignored.
Equally, L1 is only an approximate value or even unknown or zero.

Arrays for workpiece probe 1: _WP[0, ...]

%_N_CALIBRATE_Z_L_MPF	
N10 G54 G90 G17 T9 D1	;ZO selection, select probe as tool and ;plane
N20 M6	;Insert probe and activate tool offset
; ...	;The probe is prepositioned in front of the ;calibration surface within distance 2 · _FA
N50 _CBIT[14]=0	;Length 1 relative to probe ball center
N60 _TSA=25 _TZL=0 _PRNUM=1 _VMS=0 _NMSP=1 _FA=12 **N61 _MVAR=10000 _SETVAL=20 _MA=3 _MD=1**	;Define parameter for calibration cycle, ;calibrate probe 1 in minus Z axis ;with determination of length 1
N70 CYCLE976	;Measuring cycle call ;determine calibration in minus Z and L1
N100 M2	;End of program

Explanation

On cycle call the probe travels in the minus Z direction max. 24 mm (_FA=12) at measuring feedrate 300 mm/min (_VMS=0, _FA>1). If the probe is triggered within this measuring path of 24 mm, length 1 (geometry) is calculated and entered in tool offset memory T9, D1, D3.
The trigger value of probe 1 in the minus Z direction is used as the ball radius of the probe from T9, D1, DP6 and then entered in _WP[0,5] – if _CBIT[14]=0.
If _CBIT[14]=1, this value is =0.

© Siemens AG, 2004. All rights reserved

5.4 CYCLE977 workpiece: Measure hole/shaft/groove/web/rectangle parallel to axes

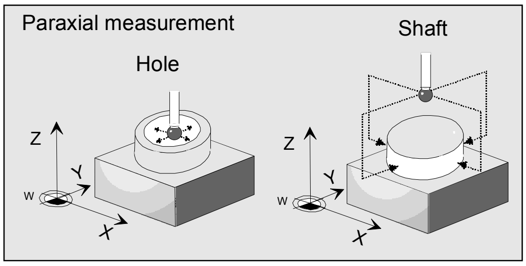
Programming

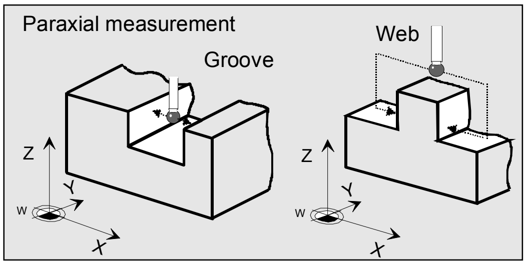
CYCLE977

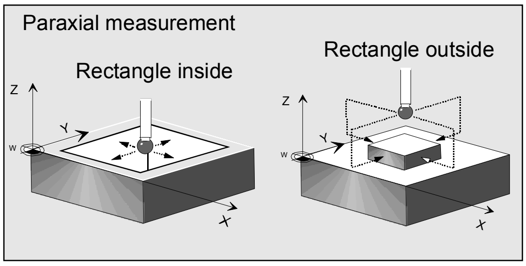
Function

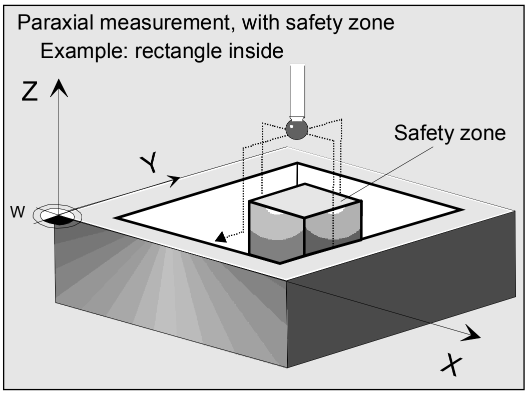
With this measuring cycle you can measure the dimensions of the following contour elements on a workpiece using different measuring variants:

- Hole
- Shaft
- Groove
- Web
- Inside rectangle
- Outside rectangle









Measurement is performed paraxially to the workpiece coordinate system.

In some measuring variants defined **safety zones** are taken into account during the measuring operation.

CYCLE977 can

- measure the contour elements

and additionally either

- perform an automatic **tool offset**

for a specified tool based on the differences in diameter or width, or

- correct a **zero offset** (ZO)

based on the differences between the center positions.

Workpiece probe types that can be used

- Multidirectional probe (_PRNUM=xy)
- Monodirectional, bidirectional probe (_PRNUM=1xy)

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE977 permits the following measuring variants that are specified with parameter _MVAR:

Value	*Measuring variant*
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
101	ZO calculation in hole with ZO compensation
102	ZO calculation on shaft with ZO compensation
103	ZO calculation in groove with ZO compensation
104	ZO calculation on web with ZO compensation
105	ZO determination in inside rectangle with ZO correction
106	ZO determination in outside rectangle with ZO correction
1001	Measure hole traveling around a safety zone and tool offset
1002	Measure shaft taking account of a safety zone and tool offset
1003	Measure groove traveling around a safety zone and tool offset
1004	Measure web taking account of a safety zone and tool offset
1005	Measure inside rectangle with safety zone and tool offset
1006	Measure outside rectangle with safety zone and tool offset
1101	ZO determination, hole traveling around a safety zone with ZO correction
1102	ZO determination, shaft taking account of a safety zone with ZO correction
1103	ZO determination, groove traveling around a safety zone with ZO correction
1104	ZO determination, web taking account of a safety zone with ZO correction
1105	ZO determination, inside rectangle with safety zone with ZO correction
1106	ZO determination, outside rectangle with safety zone with ZO correction

© Siemens AG, 2004. All rights reserved

Result parameters

Depending on the measuring variant _MVAR=xxx1 to _MVAR=xxx4, measuring cycle CYCLE977 supplies the following values as results in data block GUD5:
(**not** with rectangle measurement, see next table):

_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
_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	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
_OVI [13]1)	INTEGER	DL number (from measuring cycles SW 6.3)

1) for workpiece measurement with tool offset only
2) For ZO correction only

© Siemens AG, 2004. All rights reserved

Measuring cycle CYCLE977 returns the following values depending on measuring variant

rectangle measurement (_MVAR= xxx5, =xxx6)

in data block GUD5:

_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
_OVI [0]	INTEGER	D number or ZO number
_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][1)]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number
_OVI [11][2)]	INTEGER	Status offset request
_OVI [13][1)]	INTEGER	DL number (from measuring cycles SW 6.3)

1) for workpiece measurement with tool offset only

2) For ZO correction only

© Siemens AG, 2004. All rights reserved

5.4.1 Measuring contour elements

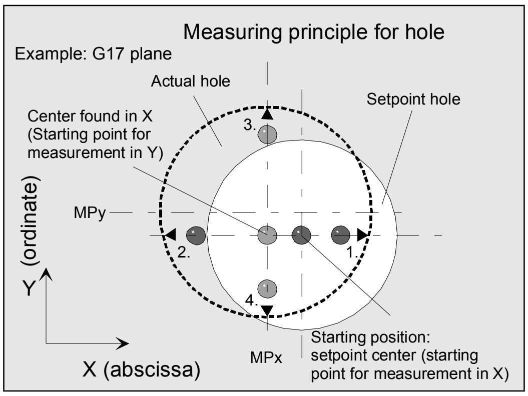
Function

Using this measuring cycle and various _MVAR measuring variants the following contour elements can be measured:

_MVAR = xxx1 – hole
_MVAR = xxx2 – shaft
_MVAR = xxx3 – groove
_MVAR = xxx4 – web
_MVAR = xxx5 – rectangle, inside
_MVAR = xxx6 – rectangle, outside

If no tool offset or no ZO correction is to be applied, set
_KNUM = 0
Detailed information on the parameters:
Refer to Section 2.3.

Measuring principle for hole or shaft
Two points each are measured in the abscissa and ordinate. The actual position of the center point (CP) in relation to workpiece zero is calculated from these four measured values. The actual diameter is calculated from the two points in the ordinate.
The center of the abscissa is calculated from the two points in the abscissa. Then the probe is positioned on this calculated center and the two points on the ordinate measured. The hole and shaft center points are now known and the results entered in array _OVR[ ].
The positive direction of an axis is measured first.

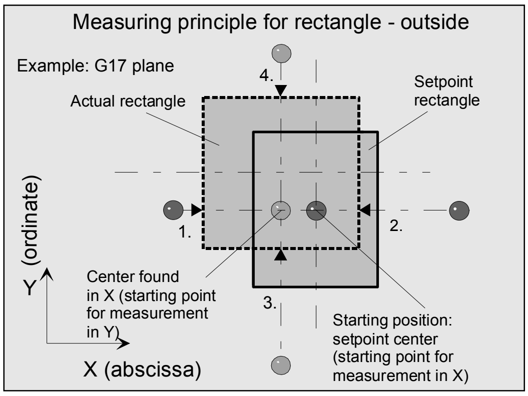


Measuring principle for groove or web
The groove or web lies parallel to the axes of the workpiece coordinate system.
2 measuring points are measured with specified measuring axis _MA.
The actual value of the groove width and web width and the actual position of the groove center and web center in relation to workpiece zero are calculated from the two measured values. The results are entered in array _OVR[ ].
The positive direction of the axis is measured first.

© Siemens AG, 2004. All rights reserved

Measuring principle for inside and outside rectangle

The rectangle lies parallel to the axes of the workpiece coordinate system.
The measuring cycle ascertains 2 measuring points in both axes and determines the actual rectangle center and both actual values of the rectangle lengths. The procedure is the same as for hole and shaft.
The results are entered in array _OVR[ ].
The positive direction of an axis is measured first.

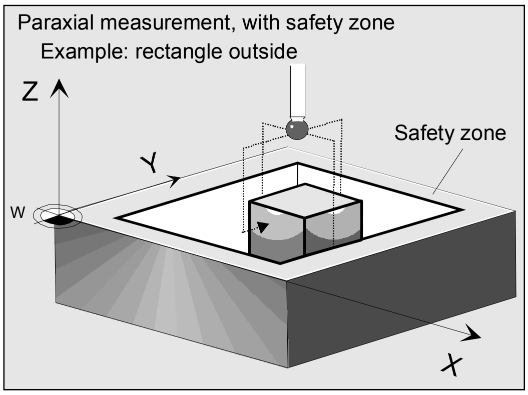


Procedure for specifying a safety zone

If required,

_MVAR = 1xxx

can take account of a safety zone (_SZA, _SZO) in the travel movement. The safety zone refers to the centerpoint or center line of the hole, shaft, groove, web, rectangle. The starting point in the height is always above the hole, shaft, groove, web, or rectangle.



Options for hole and shaft diameter, groove or web width, and tool compensation.

- An empirical value from data block GUD5 can be included with the correct sign.
- A mean value can be derived from several workpieces, measurement calls.

Precondition

The probe must be called as a tool with a tool length offset.
Tool type, preferably: 710
when using the cycle on a turning machine:
set type and _CBIT[14]=0.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	xxx1 xxx2 xxx3 xxx4 xxx5 xxx6	Measure hole Measure shaft Measure groove Measure web Measure rectangle, inside Measure rectangle, outside
_SETVAL	REAL, >0	Setpoint (acc. to drawing) (only for hole/shaft/groove/web)
_SETV[0] **_SETV[1]**	REAL, >0	Setpoint value rectangle length (in the abscissa) Setpoint value rectangle length (in the ordinate) (only when measuring a rectangle)
_ID	REAL	Incremental infeed in applicate, direction before sign (for shaft, web or rectangle measurement and for hole/groove/shaft/web measurement traveling around and considering a safety zone)
_SZA	REAL, >0	• Diameter or width of protection zone (inside for hole/groove, outside for shaft/web) • Length of safety zone in abscissa (only when measuring a rectangle)
_SZO	REAL, >0	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	0: Without automatic tool offset, without ZO determination

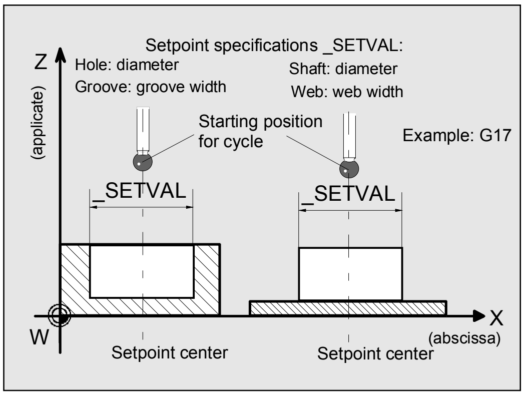
The following additional parameters are also valid **_VMS, _CORA, _TSA, _FA, _PRNUM, _EVNUM, _NMSP**.
The following also applies for measuring variants with tool compensation (even when KNUM=0):
_TZL, _TMV, _TUL, _TLL, _TDIF,

See Sections 2.2 and 2.3.

With _TSA, the diameter or width is monitored for "tool compensation", the center for "ZO determination".
_CORA only relevant for monodirectional probe.

© Siemens AG, 2004. All rights reserved

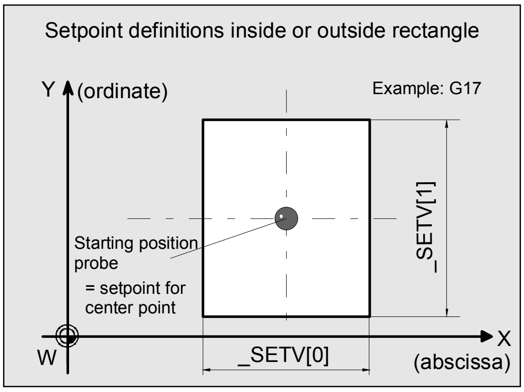
Sequence



Specification of setpoints

The setpoint for diameter or width is specified in
_SETVAL and
The setpoint for lengths of rectangle in
_SETV[0], _SETV[1].

The position of the probe in the abscissa, ordinate at the beginning of a cycle is evaluated for the setpoint of the center point of a hole, shaft, or rectangle, or for the center of a groove, web. This value is also entered in result array
_OVR[1], _OVR[2] (for hole, shaft, groove, web),
_OVR[2], _OVR[3] (for rectangle).



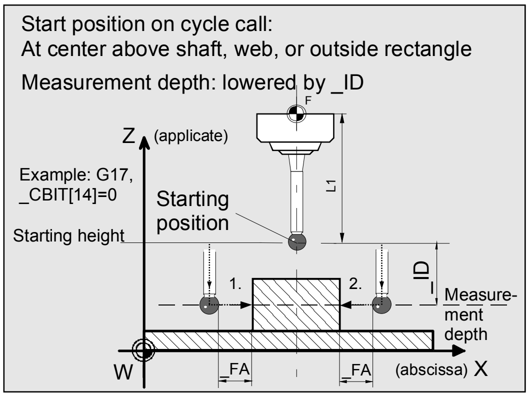
Specification of measuring axis:

The measuring axis in _MA only has to be specified for web or groove:
_MA=1: measurement in abscissa
_MA=2: measurement in ordinate.

For the remaining contour elements measurement is always performed in both axes of the plane and in both directions.

Position before measuring cycle call for shaft, web, rectangle – outside

_MVAR	*Pre-positioning*	
	in the plane	*in applicate*
2/102	Shaft center point	Above shaft
4/104	Web center, meas. axis	Above web
6/106	Rectangle center point	Above rect.



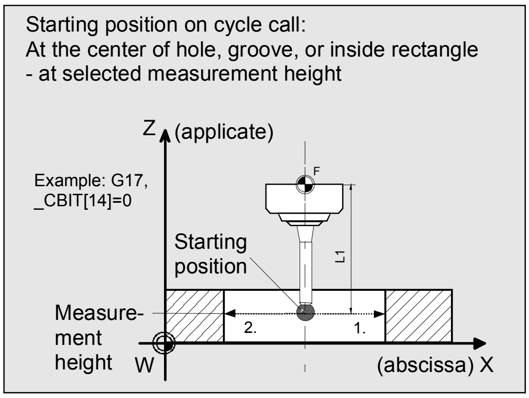
The probe must be positioned at the center point in the plane and the probe ball positioned above the upper edge such that when infeed of value _ID (sign) is applied, measurement depth is reached.

© Siemens AG, 2004. All rights reserved

Position before cycle call for hole, groove, rectangle – inside

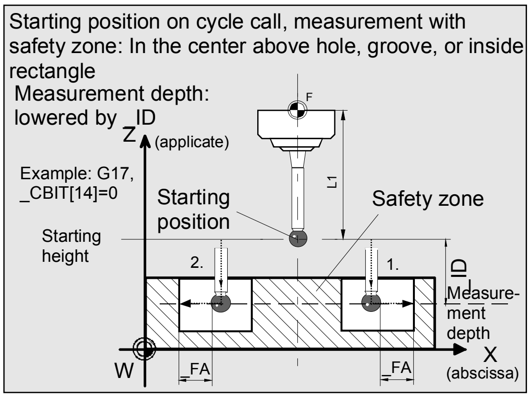
The probe must be positioned at the center point in the plane. The probe ball must be positioned at measurement height inside the hole/groove/rectangle.

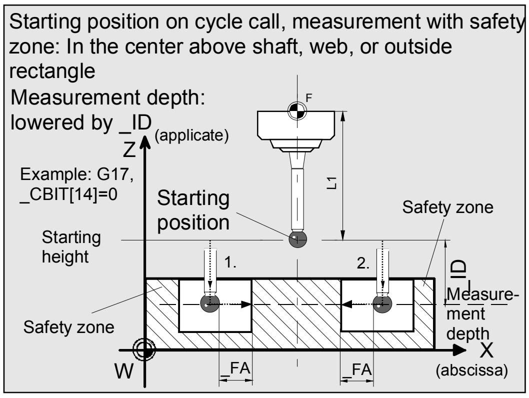
_MVAR	Pre-positioning in the plane	in applicate
1/101	Hole center point	At meas. height
3/103	Groove center, meas. ax.	At meas. height
5/105	Rectangle center point	At meas. height



Position before measuring cycle call when measuring with safety zone

_MVAR	Pre-positioning in the plane	in applicate
1001/1101	Hole center point	Above hole
1003/1103	Groove center, meas. ax.	Above groove
1005/1105	Rectangle center point	Above rect.
1002/1102	Shaft center point	Above shaft
1004/1104	Web center, meas. axis	Above web
1006/1006	Rectangle center point	Above rect.

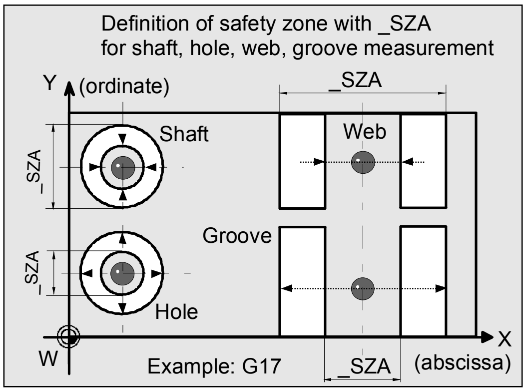




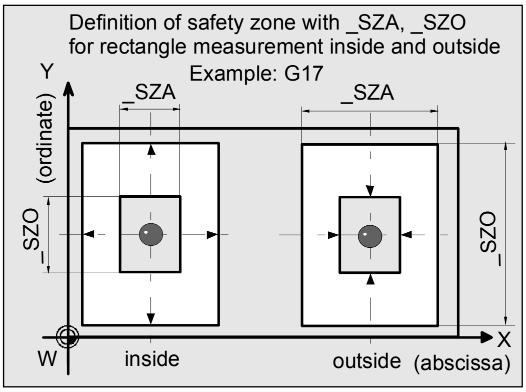
Note:

If the value selected for _FA is so large that the safety zone is violated the distance is automatically reduced in the cycle. However, there must be sufficient room for the probe ball.

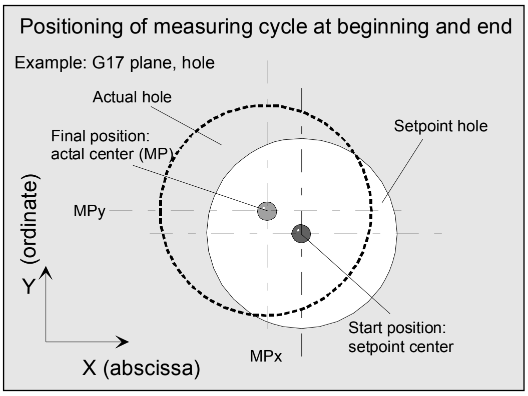
Specification of safety zone:
The safety zone (diameter or width) for shaft, hole, web, groove is defined in _SZA.



For a rectangle, the safety zone (length) is defined with _SZA in the abscissa and with _SZO in the ordinate.



Position after end of measuring cycle
When measurement is complete the probe is positioned above the **calculated** center point or center at starting position height.



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.

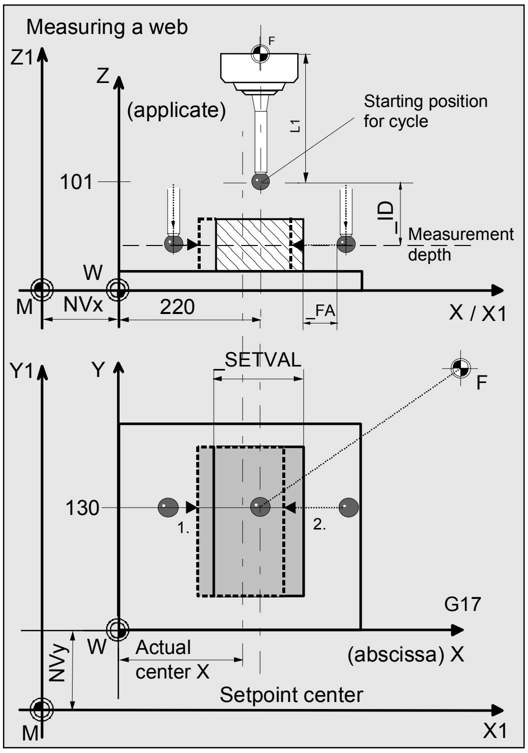
The range of positions of the center or diameter, or groove, web width, rectangle length, must be within the value specified in _FA for all workpieces to be measured.
Otherwise there is danger of collision or the measurement cannot be performed!

© Siemens AG, 2004. All rights reserved

Programming example

Measuring a web – paraxial

In the G17 plane, a web is to be measured with desired width 132 mm.
The assumed center is X=220.
The maximum possible deviation of the center is taken as 2 mm, the width 1 mm. To obtain a minimum measuring path of 1 mm, the measuring path is programmed as _FA=2+1+1=4 mm (max. measuring path _FA=8 mm).
However a measured deviation of the web center of >1.2 mm is not permissible.



Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ...

Workpiece probe 1, used as tool **T9, D1**, is to be used.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) must refer to the center of the probe ball (_CBIT[14]), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

%_N_WEB_MEASURE_MPF	
N10 G54 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 X220 Y130	;Position probe in X/Y plane cent web center ;in X and measuring position Y
N40 Z101	;Position Z axis above web

N60 _TSA=1.2 _PRNUM=1 _VMS=0 _NMSP=1 _FA=4 **N61 _MVAR=104 _SETVAL=132 _MA=1 _ID=-40 _KNUM=0**	;Set parameters for measuring cycle call ;Without ZO or tool compensation, ;Observe negative sign in _ID! ;Probe lowered in Z axis! ;Measuring variant with ZO determination ;selected (MVAR=X1xxx), because web ;center to be monitored (with _TSA), ;but _KNUM=0: without execution ;of ZO determination and ZO correction
N70 CYCLE977	;Call measuring cycle
N80 G0 Z160	;Traverse up Z axis
N100 M2	;End of program

Explanation

The measuring results of web width, web center in X, and associated differences are entered in result array _OVR[ ]. If the difference in the position of the web center is >1.2mm (_TSA=1.2) an alarm is output. Only cancellation with NC RESET is then possible.
The setpoint of the web center is the position of the probe in the workpiece coordinate system at the beginning of the cycle in the X axis.

© Siemens AG, 2004. All rights reserved

5.4.2 Measurement and tool offset

Function

With this measuring cycle and measuring variants

_MVAR = x0xx

a hole, shaft, groove, a web, or a rectangle can be measured paraxially.
Automatic tool offset is also possible. This tool is specified in **_TNUM** and **_TNAME**. The D number and type of offset are specified in coded form in variable **_KNUM**.

From measuring cycles SW 6.3, extended tool offset is available. With this function a tool from a particular stored tool environment **_TENV**, and additive, setup offsets can be corrected by specifying the DL number in **_DLNUM**.

For more detailed information about parameters see Section 2.3.

If the dimensions of a tool are corrected to this extent, the next workpiece can be manufactured with lower tolerances.

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	Measure groove by contouring a protection zone with tool offset
	1004	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

© Siemens AG, 2004. All rights reserved

_SETVAL	REAL, >0	Setpoint (acc. to drawing) (only for hole/shaft/groove/web)
_SETV[0] **_SETV[1]**	REAL, >0	Setpoint value rectangle length (in the abscissa) Setpoint value rectangle length (in the ordinate) (only when measuring a rectangle)
_ID	REAL	Incremental infeed of applicate with sign (only for measuring shaft, web, or rectangle, and for measuring hole/groove/shaft/web traveling around or taking account of a safety zone)
_SZA	REAL, >0	• Diameter/width of the protection zone (inside for hole/groove, outside for shaft/web) • Length of the safety zone in the abscissa (only for measuring rectangle)
_SZO	REAL, >0	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, >0	0: without automatic tool offset >0: with automatic tool offset (individual values: See Section 2.3, parameter _KNUM)
_TNUM	INT, ≥0	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)
_DLNUM	INT, ≥0	DL number for additive, setup offset (from measuring cycles SW 6.3)
_TENV	STRING[32]	Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)

The 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.

_CORA only relevant for monodirectional probe. With _TSA, the diameter or width is monitored for "tool compensation". The remaining parameters must also be assigned if _KNUM=0 because they refer to the workpiece.

Programming example

Measuring a hole – paraxially with tool offset

The diameter of a hole in a workpiece is to be measured in the G17 plane and the radius of a tool corrected accordingly.

Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ...

The probe is to be workpiece probe 1, inserted as tool **T9, D1**.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]

© Siemens AG, 2004. All rights reserved

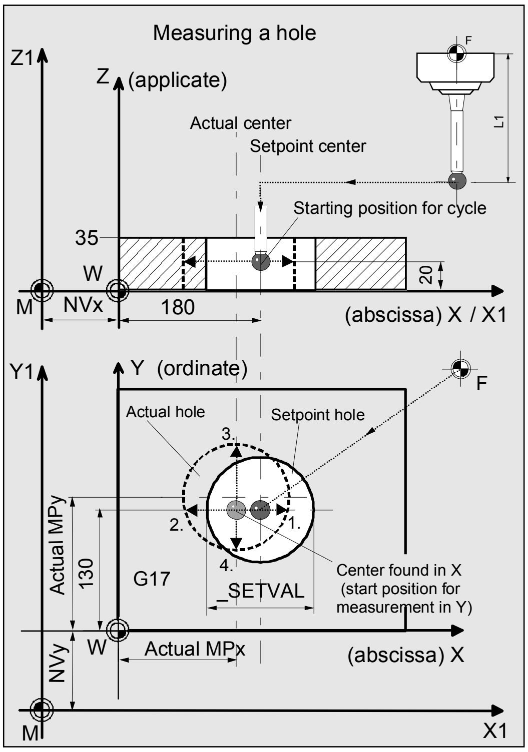
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

The hole was machined with milling tool **T20, D1** in the same environment as for measuring (G17, …) with circular milling.
The radius of this tool is to be corrected for wear, according to the measuring result for the hole diameter difference (actual value – set value). This tool offset will therefore affect the production of the next workpieces or possible remachining.

The offset must take the empirical value in memory _EV[9] into consideration. Mean value calculation _MV[9] and inclusion in calculation are also to be used.



A maximum deviation of the diameter from the setpoint of 1 mm is expected.

%_N_DRILL_MEASURE_MPF	
N10 G54 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 X180 Y130	;Position probe in X/Y plane to ;hole center point
N40 Z20	;Position Z axis to measuring depth
N50 _CHBIT[4]=1	;Include average value
N60 _TUL=0.03 _TLL=-0.03 _EVNUM=10 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1 N61 MVAR=1 _SETVAL=132 _TNUM=20 _KNUM=2001	;Set parameters for measuring cycle call, ;Probe 1 (multidirectional), ;Measure hole, setpoint diameter ;132 mm, compensation in radius of T20, D1
N70 CYCLE977	;Call measuring cycle
N560 G0 Z160	;Retract Z axis from hole
N570 M2	;End of program

© Siemens AG, 2004. All rights reserved

Explanation

The difference calculated from the actual and setpoint diameter is compensated for 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.
- Cancel with NC RESET on 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 (_TUL/_TLL) is exceeded, the radius in T20 D1 is compensated 100% by this difference/2.
- 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/2.
- If it is less than 0.02 mm (_TMV), the mean value is calculated from the mean value in mean value memory _MV[9] and inclusion of weighting factor _K=3 (only for _CHBIT[4]=1! with mean value memory).
  - 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[9].
  - 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].

The results are entered in result array _OVR[ ]. The wear of the radius of T20, D1 is included if a change is necessary.

© Siemens AG, 2004. All rights reserved

5.4.3 Measurement and ZO determination

Function

With this measuring cycle and measuring variants

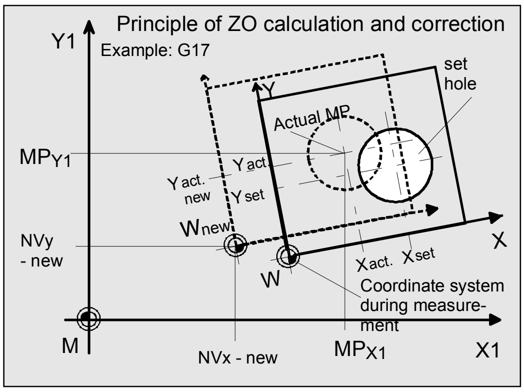
_MVAR=x1xx

a hole, shaft, groove, a web, or a rectangle can be measured paraxially. The zero offset (ZO) of the associated workpiece can also be determined and corrected. A possible rotation of the workpiece is kept without changing it.

The angular position cannot be determined with this cycle.

Compensation of the ZO is executed in such a way that the actual center (position of center on the machine, e.g.: MP_{X1}, MP_{Y1}) includes the desired setpoint position in the workpiece coordinate system when the compensated ZO (frame) is applied.

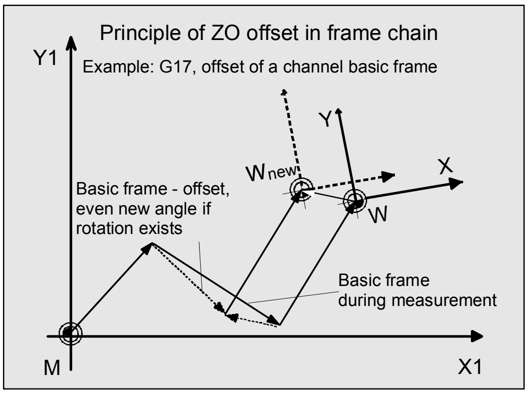
Mirroring can be active in a frame of the frame sequence. Dimension factors must never be active.



The ZO to be corrected is specified in coded form with variable **_KNUM** >0.

The ZO can be specified and corrected by various methods, e.g. in various settable frames, in various basic frames, system frames, fine offset, or coarse offset, etc.

For detailed information on specifying _KNUM for the zero offset: Refer to Section 2.3.



ZO determination in a hole, on a shaft, or rectangle

ZO correction of a workpiece is applied in the abscissa and ordinate using the actual value/setpoint difference of the position of the center point.

ZO determination in a groove or on a web

ZO correction of a workpiece is applied in measuring axis _MA (abscissa and ordinate) using the actual value/setpoint difference of the position of the center.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	101	ZO calculation in hole with ZO compensation
	102	ZO determination on a shaft with ZO correction
	103	ZO determination in a groove with ZO correction
	104	ZO determination on a web with ZO correction
	105	ZO determination in inside rectangle with ZO correction
	106	ZO determination in outside rectangle with ZO correction
	1101	ZO determination in hole traveling around a safety zone with ZO correction
	1102	ZO determination of shaft taking account of a safety zone with ZO correction
	1103	ZO determination in groove traveling around a safety zone, ZO correction
	1104	ZO determination on web taking account of a safety zone with ZO correction
	1105	ZO determination, inside rectangle with safety zone with ZO correction
	1106	ZO determination, outside rectangle with safety zone with ZO correction
_SETVAL	REAL, >0	Setpoint (acc. to drawing) (only for hole/shaft/groove/web)
_SETV[0] **_SETV[1]**	REAL, >0	Setpoint value rectangle length (in the abscissa) Setpoint value rectangle length (in the ordinate) (only when measuring a rectangle)
_ID	REAL	Incremental infeed of applicate with sign (only for measuring shaft, web, or rectangle, and for measuring hole/groove/shaft/web traveling around or taking account of a safety zone)
_SZA	REAL, >0	• Diameter/width of the protection zone (inside for hole/groove, outside for shaft/web) • Length of the safety zone in the abscissa (only for measuring rectangle)
_SZO	REAL, >0	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, >0	0: No automatic ZO correction >0: With automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)

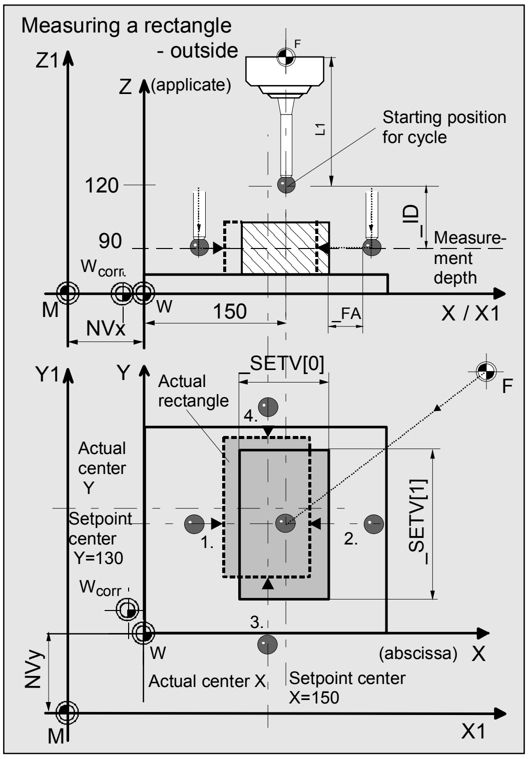
The following additional parameters are also valid **VMS, _CORA, _TSA, _FA, _PRNUM,** and **_NMSP**. See Sections 2.2 and 2.3.

_CORA only relevant for monodirectional probe. With _TSA the center is monitored with ZO determination.

© Siemens AG, 2004. All rights reserved

Programming example

ZO determination on a rectangle with CYCLE977



In the G17 plane, an outside rectangle is to be measured with the desired lengths in X=100.000 and in Y=200.00 mm.
The settable ZO G54 is to be corrected in such a way that the center of the rectangle is at X=150,000 and Y=170,000 mm.
Measurement is also performed at G54. After measurement is complete, the changed ZO is activated.

The maximum possible deviation of the center is taken to be 2 mm; the maximum possible deviation in the lengths, 3 mm. To obtain a minimum measuring path of 1 mm, the measuring path is programmed as _FA=2+3+1=6 mm (max. total measuring path _FA=12 mm).
A measured deviation of the center of the rectangle from the setpoint of >1.8 mm is however not permitted in either axis.
Height of rectangle and measuring height in Z: see Fig.

Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ... (values when measuring)

The probe is to be workpiece probe 1, inserted as tool **T9, D1**.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

© Siemens AG, 2004. All rights reserved

%_N_ZO_RECTANGLE_MPF	
N10 G54 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 X150 Y170	;Position probe in X/Y plane to ;rectangle center (setpoint position)
N40 Z120	;Position Z axis above rectangle
N60 _KNUM=1 _TSA=1.8 _PRNUM=1 _VMS=0 _NMSP=1 _FA=6 **N61 _MVAR=106 _SETV[0]=100 _SETV[1]=200 _ID=-30**	;Set parameters for measuring cycle call, ;Measuring height lowered by 30 mm in Z
N70 CYCLE977	;Call measuring cycle
N80 G54	;Repeat call of ZO G54 ;This activates the altered ZO offset!
N90 G0 Z160	;Traverse up Z axis
N100 M2	;End of program

Explanation

Automatic compensation is performed in G54 – translation in axes X and Y by the calculated difference between actual value and setpoint of the rectangle centerpoint, if it is less than 1.8 mm (_TSA=1.8) in both axes. Otherwise alarm "Safe area violated" is output and program execution cannot be continued.
If the values are inside the tolerance, the setpoint and actual values for center point and length of rectangle in the abscissa and ordinate as well as the differences are entered in result array OVR[]. The zero offset (ZO) for G54 is entered in the data management ($P_UIFR[1]) and is activated by programming G54 again in block N80.

© Siemens AG, 2004. All rights reserved

5.5 CYCLE978 workpiece: Measuring a surface parallel to the axis

Programming

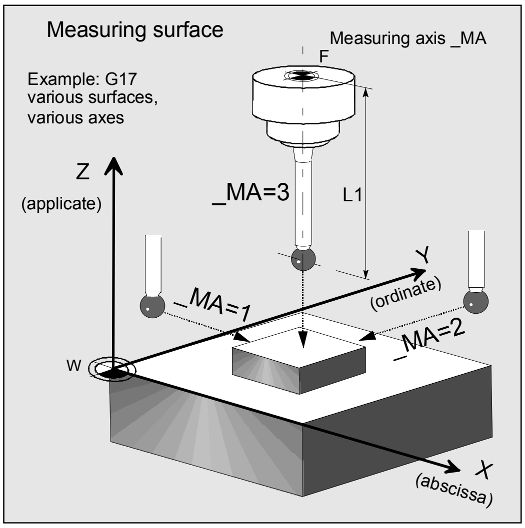
CYCLE978

Function

This measuring cycle determines the position of a **paraxial surface** in the workpiece coordinate system. This is done with 1-point measurement
If differences are ascertained, different measuring variants will also allow correction with

- automatic **tool compensation** for a specified tool, or
- **zero offset** (ZO).

Special measuring variants allow you to perform the measurement as a **differential measurement**. The special procedure for this measurement permits use of an **uncalibrated** multidirectional probe.



Workpiece measuring probe types that can be used

- Multidirectional probe (_PRNUM=xy)
- Monodirectional, bidirectional probe (_PRNUM=1xy)

Notice
A monodirectional or bi-directional probe must be calibrated! These probes cannot be used for differential measurements!

Preconditions for differential measurement

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

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

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE978 permits the following measuring variants that are specified with parameter _MVAR.

Value	*Measuring variant*
0	Measure surface and tool offset
100	ZO determination on surface and ZO correction
1000	Measure surface with differential measurement and tool offset
1100	ZO determination on surface with differential measurement and ZO correction

Result parameters

Depending on the measuring variant, CYCLE978 makes the following values available as results in the block GUD5:

_OVR [0]	REAL	Setpoint value 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	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
_OVI [13]1)	INTEGER	DL number (from measuring cycles SW 6.3)

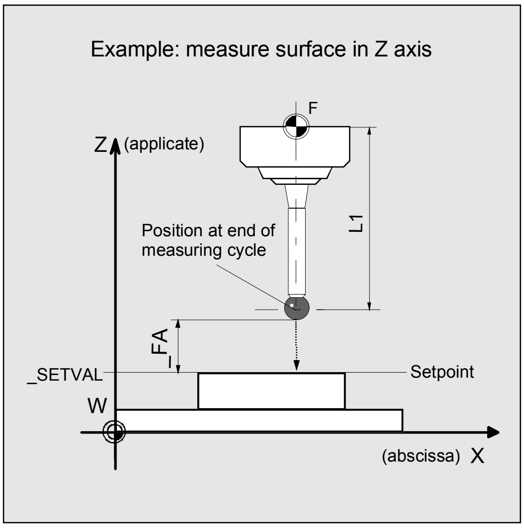
1) For 1-point measurement with automatic tool offset only
2) For ZO correction only

© Siemens AG, 2004. All rights reserved

5.5.1 Measuring the surface

Sequence

Position before measuring cycle call
The probe is positioned in relation to the surface to be measured in such a way that during traversal of the specified measuring axis _MA in the direction of the setpoint _SETVAL, the intended measuring point on the surface will be reached.
Recommended distance from surface: >_FA.



The absolute value of the positional deviation from the setpoint must not be greater than the measuring path _FA. Otherwise, no measurement will be performed.

Position after end of measuring cycle
After the end of the measuring cycle, the probe (ball circumference) is at a distance _FA from the measuring surface.

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.

Precondition
The probe must be called as a tool with a tool length offset.
Tool type, preferably: 710
when using the cycle on a turning machine:
set type and _CBIT[14]=0.

Special procedure for differential measurement

The measuring point is measured twice during differential measurement.

1. with spindle rotated through 180 degrees compared with the position at the beginning of the cycle (rotation of the probe by 180 degrees).
2. with the spindle position that applied at the beginning of the cycle

This is how the position deviation of the probe in the measuring axis is determined.
The tool radius of the probe + R or – R is defined as the trigger point defined for the axis direction.

© Siemens AG, 2004. All rights reserved

The values are entered and stored in the global user data (GUD6) in the array for the associated workpiece probe _WP[ ].
A **multidirectional** probe does not have to be calibrated at the beginning of the cycle for measuring variants
_MVAR= 1000 or _MVAR=1100.

These measuring variants with differential measurement are only useful with the measuring axes _MA=1 or _MA=2.

Notice

In the case of great measurement accuracy demands, differential measurement is not recommended!

5.5.2 Measurement and ZO determination

Function

With this measuring cycle and measuring variants
_MVAR=100, _MVAR=1100
it is possible to determine the position of a surface that is parallel with the axis in the workpiece coordinate system.

The zero offset (ZO) of the associated workpiece can also be determined and corrected.
The offset is corrected in such a way that the real position of the surface (actual value) adopts the required set angle (_SETVAL) in the workpiece coordinate system when the corrected ZO (frame) is used.

Mirroring can be active in a frame of the frame sequence. Dimension factors must never be active.

The ZO to be corrected is specified in coded form with variable **_KNUM** >0.
The ZO can be specified and corrected by various methods, e.g. in various settable frames, in various basic frames, system frames, fine offset, or coarse offset, etc.
If _KNUM=0, there is no ZO correction.

For detailed information on specifying _KNUM for the zero offset: Refer to Section 2.3.

© Siemens AG, 2004. All rights reserved

An empirical value stored in data block GUD5 in array _EV[ ] can be included in calculation of the result after measurement is completed.
This is activated in _EVNUM (see Section 2.3).

Parameters

_MVAR	100	ZO determination on surface and ZO correction
	1100	ZO determination on surface with differential measurement and ZO correction
_SETVAL	REAL	Setpoint with respect to workpiece zero
_MA	1...3	Number of the measuring axis
_KNUM	0, >0	0: without automatic ZO correction >0: with automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)

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

_CORA only relevant for monodirectional probe.

Programming example

ZO calculation at a workpiece with CYCLE978

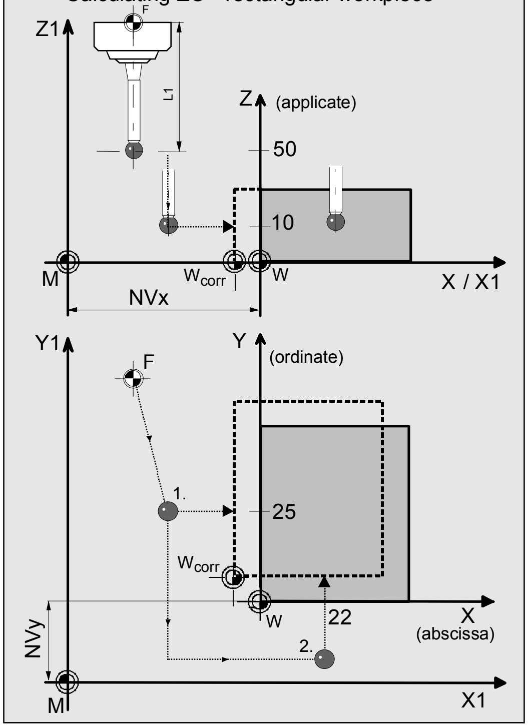
A rectangular workpiece is clamped in the G17 plane. The ZO in axes X and Y is to be checked. Any deviation from the active values should be automatically corrected in settable ZO G54. The corrected ZO should also be activated so that machining of the workpiece can start directly after.

The empirical value entered in array _EV[9] (data block GUD5) for the X axis and _EV[10] for the Y axis are to be included in the measuring results.

The permissible deviation is 3 mm from the setpoint value is assumed. To obtain a minimum measurement path of 1 mm to the surface, the measurement path is programmed with _FA=3+1=4.
(max. total measurement path=8 mm).

The value of the positional deviation should not be monitored. Therefore _TSA > _FA is set.
Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ... (values when measuring)

Calculating ZO - rectangular workpiece



© Siemens AG, 2004. All rights reserved

Workpiece probe 1, used as tool **T9, D1**, is to be used.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

%_N_ZO_DETERMINING_1_MPF	
N10 G54 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 G90 X-20 Y25	;Position probe in X/Y plane in front of ;measuring surface
N40 Z10	;Position probe at measuring height
N60_TSA=6 _PRNUM=1 _VMS=0 _NMSP=1 _FA=4 **N61 _MVAR=100 _SETVAL=0 _MA=1 _KNUM=1** **_EVNUM=10**	;Set parameters for measuring cycle call
N70 CYCLE978	;Measuring cycle for ZO determination in
N80 G0 X-20	;Retract in X axis
N90 Y-20	;Position in Y axis
N100 X22	;Position in X axis
N110 _EVNUM=11 _MA=2	;Set parameters for measuring cycle call
N120 CYCLE978	;ZO determination in Y axis
N130 G54	;Repeat call of ZO G54 ;This activates the changes!
N140 G0 Y-20	;Retract in Y axis
N150 Z50	;Retract in Z axis
N160 X-40 Y80	;Retract in X/Y
N200 M2	;End of program

Explanation

Automatic compensation is performed in G54 – translation of axes X and Y by the calculated difference between actual value and setpoint.
The setpoints and actual values as well as the differences are entered in result array OVR[ ]. At the end of the program the values for the Y axis (ordinate) are in the result array as these were the last to be measured.
The zero offset (ZO) for G54 is entered in the data management ($P_UIFR[1]) and is activated by programming G54 again in block N130.

© Siemens AG, 2004. All rights reserved

5.5.3 Measuring and tool offset

Function

With this measuring cycle and measuring variants
_MVAR=0, _MVAR=1000
it is possible to determine the dimension (position) of a surface that is parallel with the axis in the workpiece coordinate system.

Automatic tool offset is also possible. This tool is specified in **_TNUM** and **_TNAME**. The D number and type of offset are specified in coded form in variable **_KNUM**.

From measuring cycles SW 6.3, extended tool offset is available. With this function a tool from a particular stored tool environment **_TENV**, and additive, setup offsets can be corrected by specifying the DL number in **_DLNUM**.

Detailed information on the parameters:
Refer to Section 2.3.

Empirical values and mean values
An empirical value stored in data block GUD5 in array _EV[ ] can be included in calculation of the result after measurement is completed.
Optionally, averaging is performed over a number of parts (array _MV[ ]) and the tolerance bands are checked.
Both are activated in _EVNUM (see Section 2.3).

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	0	Measure surface and tool compensation
	1000	Measure surface with differential measurement and tool offset
_SETVAL	REAL	Setpoint (acc. to drawing)
_MA	1...3	Number of the measuring axis
_KNUM	0, >0	0: without automatic tool offset >0: with automatic tool offset
_TNUM	INT, ≥0	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)
_DLNUM	INT, ≥0	DL number for additive, setup offset (from measuring cycles SW 6.3)
_TENV	STRING[32]	Name of tool environment for automatic tool compensation (from measuring cycles SW 6.3)

The 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.

_CORA only relevant for monodirectional probe.
The parameters must also be assigned if _KNUM=0 because they refer to the workpiece.

Programming example

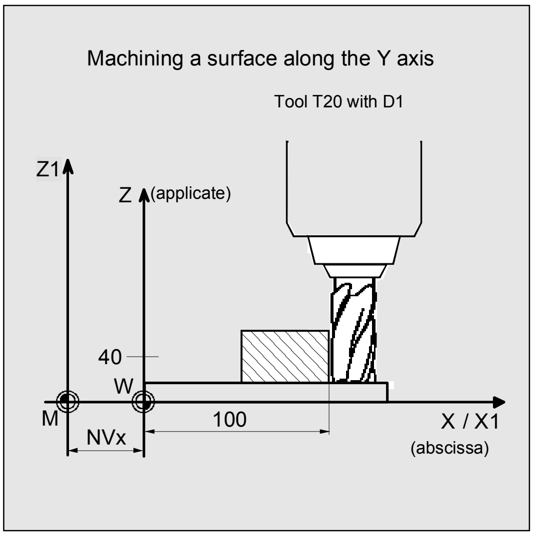
1-point measurement in X axis with tool compensation

A surface parallel with the Y axis has been machined with milling tool T20, D1 on a set-up workpiece.
This surface should be a distance of exactly 100,000 mm in the X axis from the defined workpiece zero and be measured.

If the absolute value of the difference determined is >0.01, the radius of this tool is to be automatically offset in the wear.

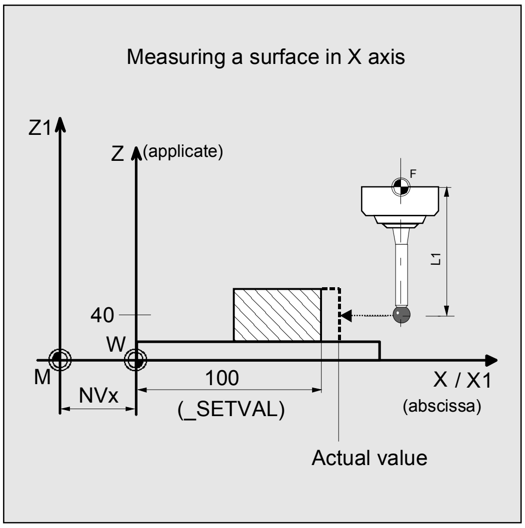
1 mm is assumed to be the maximum permissible deviation of the position of the surface.
To obtain a minimum measuring path of 1 mm, the measuring path is programmed as _FA= 1+1=2 mm (max. measuring path=4 mm).

The offset must take the empirical value in memory _EV[19] into consideration. Mean value calculation _MV[19] and inclusion in calculation are also to be used.



© Siemens AG, 2004. All rights reserved

This tool offset will therefore affect the production of the next workpieces or possible remachining.



Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, ...

The probe is to be workpiece probe 1, inserted as tool **T9, D1**.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

%_N_ONE_POINT_MEASURE_MPF	
N10 G54 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 G90 X120 Y150	;Position probe in X/Y plane in front of ;measuring surface
N40 Z40	;Position probe at measuring height
N50 _CHBIT[4]=1	;with mean value calculation
N60 _TUL=0.03 _TLL=-0.03 _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 **N61 _MVAR=0 _SETVAL=100 _MA=1 _KNUM=2001**	;Set parameters for measuring cycle call
N70 CYCLE978	;Measuring cycle for single-point ;Measurement in X axis
N80 G0 Z160	;Traverse up Z axis
N100 M2	;End of program

© Siemens AG, 2004. All rights reserved

Explanation

The difference calculated from the actual and setpoint value (position of surface) 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 cannot be 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 from the mean value in mean value memory _MV[19] and inclusion of weighting factor _K=3 (only for _CHBIT[4]=1! with mean value memory).
  - 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.

The results are entered in result array _OVR[ ]. The wear of the radius of T20, D1 is included if a change is necessary.

© Siemens AG, 2004. All rights reserved

5.6 CYCLE979 workpiece: Measure hole/shaft/groove/rib at an angle

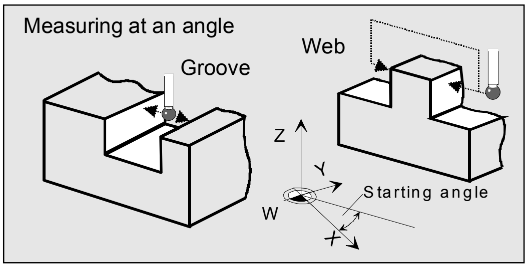
Programming

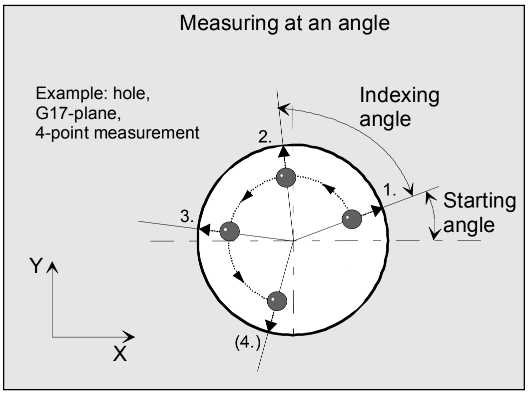
CYCLE979

Function

With this measuring cycle you can measure the dimensions of the following contour elements on a workpiece using different measuring variants:

- Hole
- Shaft
- Groove
- Web





Measurement is performed at a specified starting angle to the abscissa of the workpiece coordinate system.
For hole, shaft, additional measurements are performed at an indexing angle, added to the previous angle.
This allows you to measure **circle segments** of a workpiece contour whose center points lie outside the machine.

CYCLE979 can

- measure the contour elements

and additionally either

- perform an automatic **tool offset**

for a specified tool based on the differences in diameter or width, or

- correct a **zero offset** (ZO)

based on the differences between the center positions.

Workpiece probe types that can be used

- Multidirectional probe (_PRNUM=xy)
- Monodirectional, bidirectional probe (_PRNUM=1xy)

When measuring contour elements **hole**, **shaft**, **3** or **4 point measurement** can be used. This is also specified in _PRNUM (in this cycle only):
_PRNUM=0zxy → 3-point measurement
_PRNUM=1zxy → 4-point measurement

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE979 permits the following measuring variants that are specified with parameter **_MVAR**.

Value	Measuring variant
1	Measure hole with tool offset
2	Measure shaft with tool offset
3	Measure groove with tool offset
4	Measure web with tool offset
101	ZO calculation in hole with ZO compensation
102	ZO calculation on shaft with ZO compensation
103	ZO calculation in groove with ZO compensation
104	ZO calculation on web with ZO compensation

Result parameters

Depending on the measuring variant, CYCLE979 makes the following values available as results in the block GUD5:

_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 of hole, shaft or width of groove, web
_OVR [12][1)]	REAL	Lower tolerance limit for diameter of hole, shaft or width of 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 dimensional deviation
_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	Probe number
_OVI [6][1)]	INTEGER	Mean value memory number
_OVI [7][1)]	INTEGER	Empirical value memory number

© Siemens AG, 2004. All rights reserved

_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number
_OVI [11][2)]	INTEGER	Status offset request
_OVI [13][1)]	INTEGER	DL number (from measuring cycles SW 6.3)

1) For workpiece measurement with tool offset only
2) For ZO correction only

5.6.1 Measuring a hole, shaft, groove, web

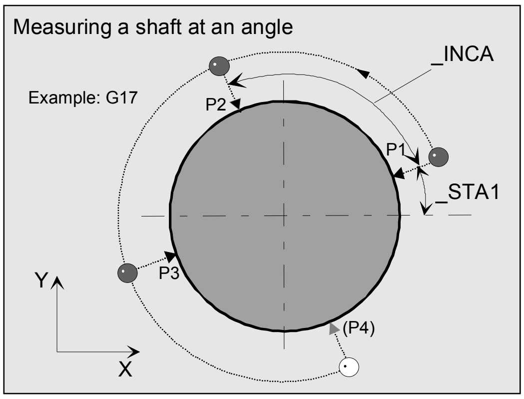
Function

Using this measuring cycle and various _MVAR measuring variants the following contour elements can be measured at an angle:

_MVAR=x01 → hole
_MVAR=x01 → shaft
_MVAR=x01 → groove
_MVAR=x01 → web

If no tool offset or ZO correction is to be applied, set **KNUM=0**.
Detailed information on the parameters:
Refer to Section 2.3.

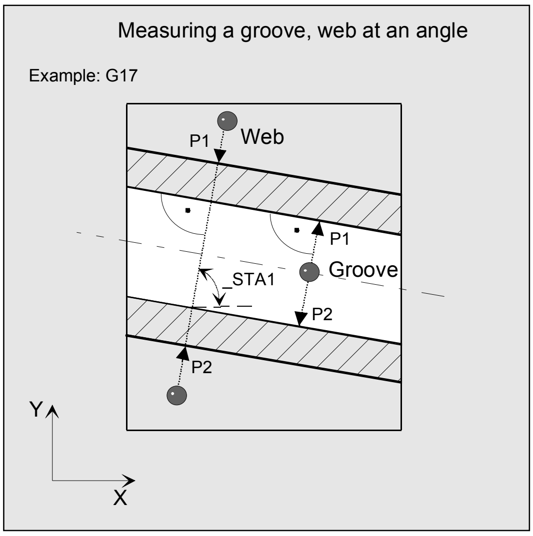
Measuring principle for hole or shaft
Inside the hole or outside the hole when traveling around a shaft. the measuring cycle measures points P1, P2, P3, or also P4.
The position of the points is determined by starting angle _STA1, indexing angles _INCA, the diameter and the set center point.
These four 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.



Measurement is performed in the radial direction:
- toward the set center point in the case of shaft,
- away from the set center point in the case of a hole.

The sum of the starting angle plus all incremental angles must not exceed 360 degrees.

© Siemens AG, 2004. All rights reserved



Measuring principle for groove or web
The measuring cycle measures points P1 and P2 inside the groove and outside the web.
The actual value of the groove width and web width and the position of the groove center and web center in relation to workpiece zero are calculated from the measured values.

Options for hole and shaft diameter, groove or web width, and tool compensation

- An empirical value from data block GUD5 can be included with the correct sign.
- A mean value can be derived from several workpieces, measurement calls.

Precondition
The probe must be called as a tool with a tool length offset.

Tool type, preferably: 710
when using the cycle on a turning machine:
set type and _CBIT[14]=0.

The probe must have been calibrated with "Determine active probe ball diameter". Calibration with an additional "Determine position deviation" of the workpiece probe improves the measuring precision.

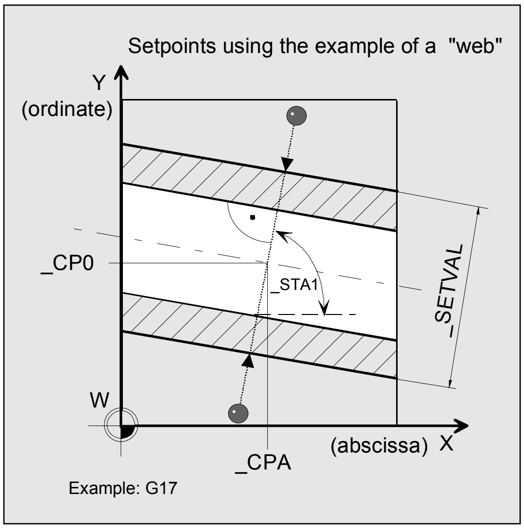
Notice
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane, orientation of the spindle in the plane and measuring velocity are the same for both measurement and calibration. Deviations can cause additional measuring errors.

© Siemens AG, 2004. All rights reserved

Sequence

Specification of setpoints

The setpoint for diameter or width is specified in **_SETVAL**.



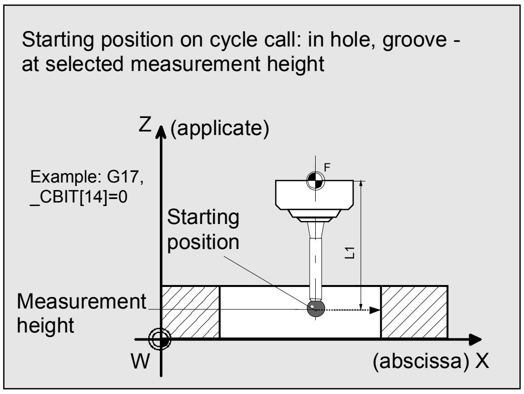
The setpoint for the center point of the hole, shaft, or for the measured center of the groove, web, is specified by

_CPA for the abscissa and
_CPO for the ordinate.

Measuring axes

Measuring axes are not specified. As a rule, both axes of the plane are included in the measurements, depending on the angle.

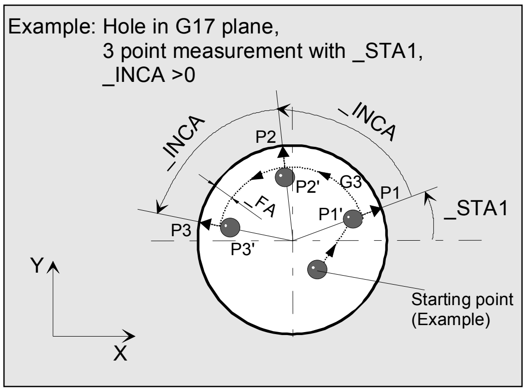
Position before measuring cycle call



In all measuring variants the probe must be positioned at the required **measuring height** in the applicate (tool axis) close to the first measuring point P1.
It must be possible to approach the first measuring point P1 via intermediate point P1' from this position without collision using linear interpolation.

Recommended distance from contour: >_FA.

Procedure for hole, shaft



The intermediate positions of the measuring points are approached along a circular path (G2, G3). The distance between the probe ball (ball circumference) and the hole or shaft is _FA. The travel direction G2 or G3 is derived from the sign of _INCA: G3 is angle is positive.

The velocity along the circular path is programmed with **_RF**.

Procedure for groove

The probe is in the groove and approaches both measuring points one after the other in the selected measuring height along an oblique straight line as defined by angle _STA1 and which travels through CPA, CPO.

© Siemens AG, 2004. All rights reserved

Procedure for web

In the measuring variants for a **web**

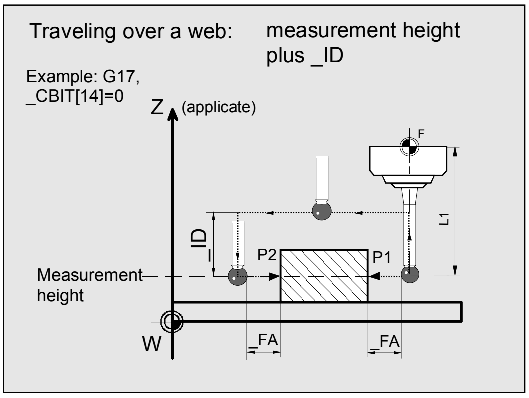
_MVAR=4, _MVAR=104

additional information for traveling over the web with **_ID** is required.

_ID specifies the distance (with sign) from the measuring height.

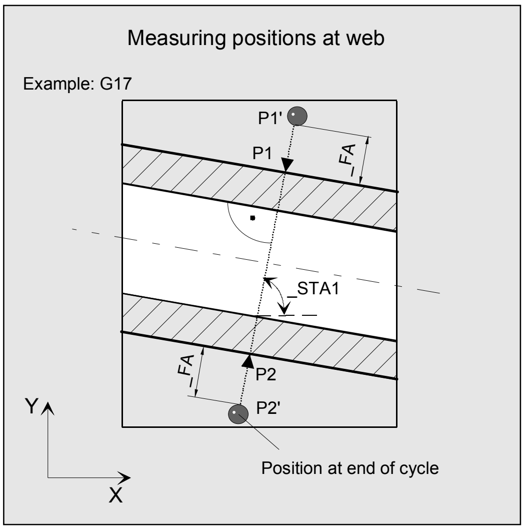
Caution

If _CBIT[14]=0, length 1 (L1) of the probe refers to the ball center. Radius R is then not taken into account in the length and must be included in _ID!



Measuring point P2 is approached via P2' along an oblique straight line according to angle _STA1 and which runts through _CPA, _CPO.

P1', P2' are both distance _FA (path) from the contour.



Position at end of measuring cycle

At the end of the measuring cycle, the probe (ball circumference) is distance _FA (path) from the last measuring point (setpoint) at measuring height.

Notice

The range of positions of the center or diameter, or groove, web width, must be within the value specified in _FA for all workpieces to be measured. Otherwise there is danger of collision or the measurement cannot be performed!

© Siemens AG, 2004. All rights reserved

5.6.2 Measuring and tool offset

Function

With this measuring cycle and measuring variants
_MVAR=1...4
contour elements hole, shaft, groove, web, can be measured at an angle.

Automatic tool offset is also possible. This tool is specified in **_TNUM** and **_TNAME**. The D number and type of offset are specified in coded form in variable **_KNUM**.

From measuring cycles SW 6.3, extended tool offset is available. With this function a tool from a particular stored tool environment **_TENV**, and additive, setup offsets can be corrected by specifying the DL number in **_DLNUM**.

Detailed information on the parameters:
Refer to Section 2.3.

Empirical values and mean values
An empirical value stored in data block GUD5 in array _EV[ ] can be included in calculation of the result after measurement is completed.
Optionally, averaging is performed over a number of parts (array _MV[ ]) and the tolerance bands are checked.
Both are activated in _EVNUM (see Section 2.3).

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	1 2 3 4	Measure hole with tool offset Measure shaft with tool offset Measure groove with tool offset Measure web with tool offset
_SETVAL	REAL, >0	Setpoint diameter, width (acc. to drawing)
_CPA	REAL	Center point of abscissa (with reference to workpiece zero)
_CPO	REAL	Center point of ordinate (with reference to workpiece zero)
_STA1	-360 to +360 degrees	Start angle
_ID	REAL	Incremental raise of the applicate with sign (only measure for rib, raise to pass)
_INCA	-360 to +360 degrees	Indexing angle (only for measuring hole or shaft) Useful values for three-point measurement: -120 ... + 120 degrees Useful values for four-point measurement: -90 ... + 90 degrees
_RF	REAL, >0	Feedrate for circular interpolation (mm/min) (only for measuring hole/shaft)
_KNUM	0, >0	0: without automatic tool offset >0: with automatic tool offset (individual values: See Section 2.3, parameter _KNUM)
_TNUM	INT, ≥0	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)
_DLNUM	INT, ≥0	DL number for additive/setup offset (> meas. cycle SW 6.3)
_TENV	STRING[32]	Name of tool environment for automatic tool compensation (from measuring cycles SW 6.3)

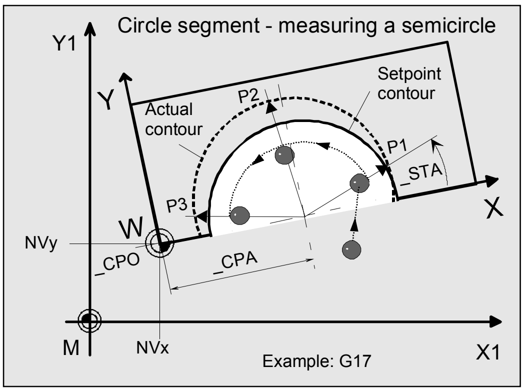
The following additional parameters are also valid
_VMS, _CORA, _TZL, _TMV, _TUL, _TLL,
_TDIF, _TSA, _FA, _PRNUM, _EVNUM,
_NMSP, and **_K**.
See Sections 2.2 and 2.3.

_CORA only relevant for monodirectional probe.
The remaining parameters must also be assigned if _KNUM=0 because they refer to the workpiece.
With _TSA, the diameter or width difference is monitored.

© Siemens AG, 2004. All rights reserved

Programming example

Measuring a hole with CYCLE979



The trueness of a circular segment in plane G17 (semi-circle, contour element "hole") is to be checked. Machining was performed with milling tool T20, D1.

If the amount deviates by <0.01 mm from
  the set diameter _SETVAL= 130 mm
the tool radius of this tool is to be automatically corrected in terms of wear. The maximum permissible deviation is taken as max. 1 mm. To obtain a minimum measurement path of 1 mm to the contour, the measurement path is programmed with
  _FA = 1+1=2 mm (max. total measurement path=4 mm).
The center point of the circular segment (setpoint) is
  X=180 mm, Y=0 mm (_CPA, _CPO).
Measurement is to be performed with three-point measurement at a measuring height of Z= 20 mm at initial angle 15° and following angles 80°.
Traversing between points is carried out with a circular feed of _RF= 900 mm/min.

The offset must take the empirical value in memory _EV[19] into consideration. Mean value calculation _MV[19] and inclusion in calculation are also to be used.

This tool offset will therefore affect the production of the next workpieces or possible remachining.

Clamping for workpiece:
Zero offset, with settable ZO G54:
  NVx, NVy, ...

The probe is to be workpiece probe 1,
inserted as tool **T9, D1**.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

© Siemens AG, 2004. All rights reserved

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

%_N_DRILL_SEGMENT_MPF	
N10 G54 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 X210 Y-20	;Position probe in X/Y plane close to P1
N40 Z20	;Position probe at measuring height
N50 _CHBIT[4]=1	;With mean value calculation
N60 _TUL=0.03 _TLL=-0.03 _EVNUM=20 _K=3 _TZL=0.01 _TMV=0.02 _TDIF=0.06 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2 N61 _MVAR=1 _SETVAL=130 _STA1=15 _INCA=80 _RF=900 _TNUM=20 _KNUM=2001 _CPA=180 _CPO=0	;Set parameters for measuring cycle call ;Three-point measurement with probe 1
N70 CYCLE979	;Call measuring cycle for hole ;measurement in X/Y
N80 G0 Z160	;Traverse up Z axis
N100 M2	;End of program

Explanation

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 cannot be 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.

© Siemens AG, 2004. All rights reserved

- If it is less than 0.02 mm (_TMV), the mean value is calculated from the mean value in mean value memory _MV[19] and inclusion of weighting factor _K=3 (only for _CHBIT[4]=1! with mean value memory).
  - If the mean value obtained is >0.01 (_TZL), the radius value in parameter T20 D1 is divided by 2 and corrected – and the mean value in _MV[19] is deleted.
  - If the mean value is < 0.01 (_TZL), the radius value in T20 D1 is not corrected, but if the mean value memory is active (_CHBIT[4]=1) it is saved in the mean value memory _MV[19].

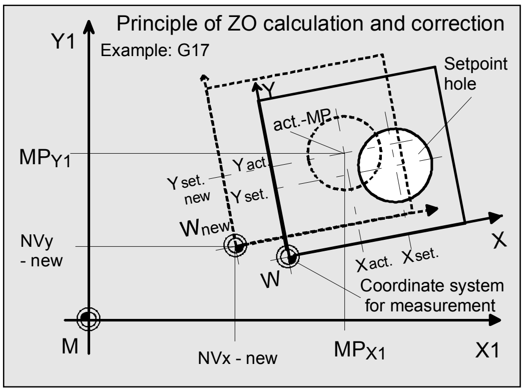
The results are entered in result array _OVR[ ].

© Siemens AG, 2004. All rights reserved

5.6.3 Measurement and ZO determination

Function

With this measuring cycle and measuring variants
_MVAR=10x
a hole, shaft, groove, or web can be measured at an angle. The zero offset (ZO) of the associated workpiece can also be determined and corrected. A possible rotation of the workpiece is kept without changing it.
The angular position cannot be determined with this cycle.
Compensation of the ZO is executed in such a way that the actual center (position of center on the machine, e.g.: MP_{X1}, MP_{Y1}) includes the desired setpoint position in the workpiece coordinate system when the compensated ZO (frame) is applied.
Mirroring can be active in a frame of the frame sequence.
Dimension factors must never be active.



The ZO to be corrected is specified in coded form with variable **_KNUM** >0.
The ZO can be specified and corrected by various methods, e.g. in various settable frames, in various basic frames, system frames, fine offset, or coarse offset, etc.

For detailed information on specifying _KNUM for the zero offset: Refer to Section 2.3.

The following applies to all measuring variants with ZO determination in CYCLE979:
The difference between the setpoint (_CPA and _CPO) and the actual value of the **center point** derived by the cycle determines the ZO correction (offset).
This value is monitored here with _TSA.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	101 102 103 104	ZO calculation in a hole with ZO compensation ZO calculation on a shaft with ZO compensation ZO calculation in a groove with ZO compensation ZO calculation on a web with ZO compensation
_SETVAL	REAL, >0	Setpoint diameter, width (acc. to drawing)
_CPA	REAL	Center point of abscissa (with reference to workpiece zero)
_CPO	REAL	Center point of ordinate (with reference to workpiece zero)
_STA1	-360 to +360 degrees	Start angle
_ID	REAL	Incremental raise of the applicate with sign (only measure for rib, raise to pass)
_INCA	-360 to +360 degrees	Indexing angle (only for measuring hole or shaft) Useful values for three-point measurement: -120 ... + 120 degrees Useful values for four-point measurement: -90 ... + 90 degrees
_RF	REAL, >0	Feedrate for circular interpolation (mm/min) (only for measuring hole/shaft)
_KNUM	0, >0	0: without automatic ZO correction >0: with automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)

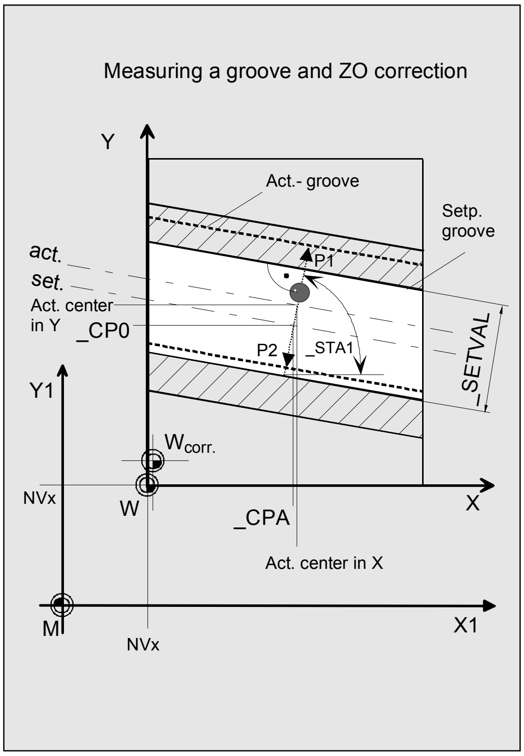
The following additional parameters are also valid
_VMS, _CORA, _TSA, _FA, _PRNUM and **_NMSP**
See Sections 2.2 and 2.3.
_CORA only relevant for monodirectional probe.
With _TSA, the center point difference is monitored.

© Siemens AG, 2004. All rights reserved

Programming example

Measuring a groove and determining the ZO with CYCLE979

The groove width on a workpiece is to be measured in plane G17 and measuring height Z=40 mm. The groove lies at an angle of 70° in its width from the X axis (_STA1).



The resulting center of the slot in the measured path should lie in the corrected workpiece coordinate system at

X=150 mm, Y=130 mm (_CPA, _CPO).

Any deviation from the selected ZO must be compensated for automatically in G55 by means of additive ZO.
Measurement is also performed with G55.

The maximum conceivable deviation of the groove center is taken as 1 mm. The measuring path is therefore specified as _FA= 2 mm (max. measuring path = 4 mm) and ensures that there is still a minimum measuring path of 1 mm up to the edge of the groove.

A deviation of < 0.8 mm of the center is permitted. This should be monitored with _TSA.

Clamping for workpiece:
Zero offset, with settable ZO G55:
NVx, NVy, ... (values before the correction [offset])

%_N_OFFSET_GROOVE_MPF	
N10 G55 G17 G90 T9 D1	;ZO, select tool as probe ...
N20 M6	;Insert probe, ;Activate tool offset
N30 G0 X150 Y130	;Position probe in X/Y plane ;in setpoint center
N40 Z40	;Position probe at measuring height
N60 _TSA=0.8 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2 N61 _MVAR=103 _SETVAL=130 _CPA=150 _CPO=130 _STA1=70 _KNUM=2	;Set parameters for measuring cycle call
N70 CYCLE979	;Call measuring cycle for ZO ;determination in X/Y plane
N80 G0 Z160	;Traverse up Z axis
N90 G55	;Repeat call of zero offset G55 ;This activates the changes!
N100 M2	;End of program

© Siemens AG, 2004. All rights reserved

Explanation

Automatic compensation is performed in G55, offset in X and Y by the calculated difference between the actual value and set position of the groove center point, should it be less than 1 mm (_TSA) in both axes. Otherwise alarm "Safe area violated" is output and program execution cannot be continued. It must be interrupted with NC RESET.
The corrected ZO G55 is activated in block N90.

The results are entered in result array _OVR[ ].

© Siemens AG, 2004. All rights reserved

5.7 CYCLE998 workpiece: Angle measurement and ZO determination

Programming

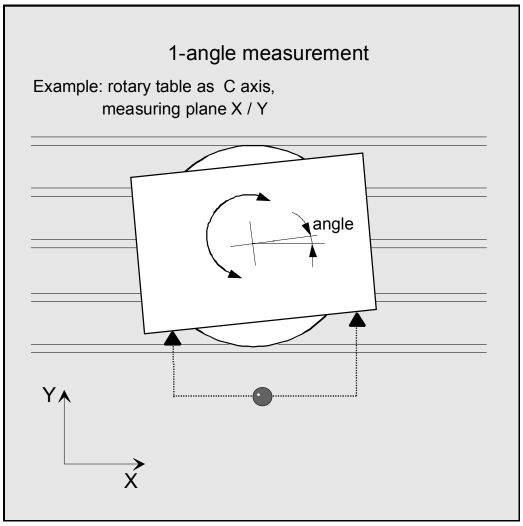
CYCLE998

Function

This measuring cycle enables you to determine the angular position of surfaces of a workpiece. This can be used to close the workpiece clamping and correct the ZO as regards **angular position**.

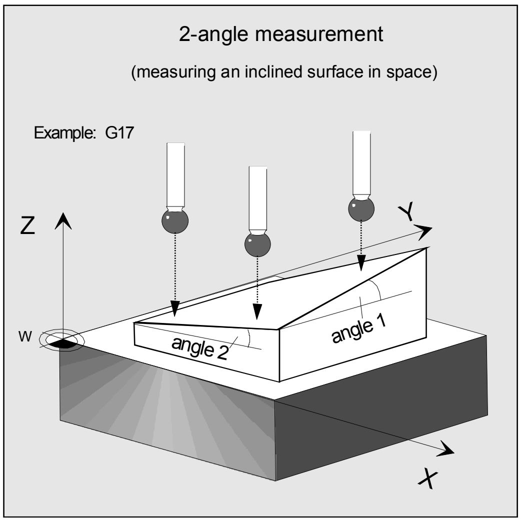
With 1-angle measurement:

- When a workpiece is clamped rotated in the plane: The angular offset is applied in the rotation component of the geometry axis that is perpendicular to the measurement plane.
- If a workpiece is on a rotary table: The angular offset is applied additively in the translation component of the rotary axis (table axis).



With 2-angle measurement:

- If a workpiece has a plane that is inclined in space: The angular offsets are applied in the rotation part of the geometry axes.



The angular position is corrected, taking account of set angles in the specified frame (ZO).

Note:
In this cycle, only the **rotation** components of the frame are determined and corrected (except for rotary table). To complete correction of the ZO (frame), a further measuring cycle is required to determine the translation component (e.g. CYCLE977 or CYCLE978).

An empirical value _EV[ ] stored in the GUD5 data block can be included in the measurement result with the correct sign. This is activated in _EVNUM (see Section 2.3).

A special measuring variant permits **differential measurement** with the axes of the plane. The special procedure for this measurement permits use of an **uncalibrated** multidirectional probe.

© Siemens AG, 2004. All rights reserved

Workpiece probe types that can be used

- Multidirectional probe (_PRNUM=xy)
- Monodirectional, bidirectional probe (_PRNUM=1xy)

Notice

A monodirectional or bi-directional probe must be calibrated! These probes cannot be used to take differential measurements!

Preconditions for differential measurement

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

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

Maximum measurement angle

The cycle is capable of measuring a maximum angle of -45 ...+45 degrees.

Measuring variants

Measuring cycle CYCLE998 permits the following measuring variants that are specified with parameter _MVAR.

Value	*Measuring variant*
105	Angle measurement and ZO determination, positioning at an angle from measuring point to measuring point
1105	Angle measurement with differential measurement and ZO determination, positioning at an angle from measuring point to measuring point
100105	Angle measurement and ZO determination, paraxial positioning from measuring point to measuring point in the offset axis
106	2-angle measurement and ZO determination, positioning at an angle from measuring point to measuring point in height
100106	2-angle measurement and ZO determination, paraxial positioning from measuring point to measuring point in the height

© Siemens AG, 2004. All rights reserved

Result parameters

CYCLE998 makes the following values available as results in the GUD5 data block:

_OVR [0]	REAL	Setpoint angle or 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 or 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 or 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	Probe number
_OVI [7]	INTEGER	Empirical value memory number
_OVI [9]	INTEGER	Alarm number
_OVI [11][1)]	INTEGER	Status offset request

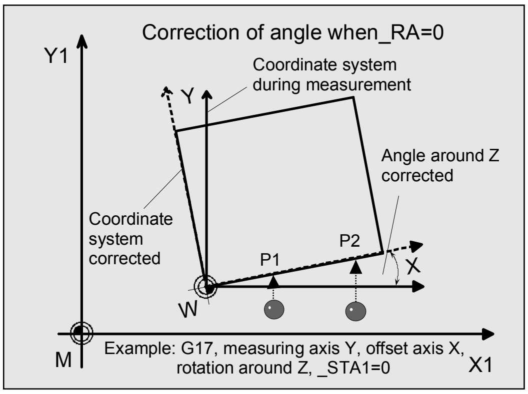
1) For measuring variant _MVAR=x00106 only

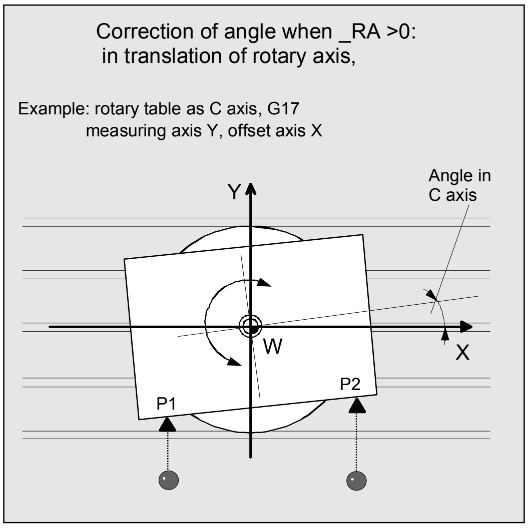
© Siemens AG, 2004. All rights reserved

5.7.1 1-angle measurement

Function

With this measuring cycle and measuring variants
_MVAR=x0x105
it is possible to determine the angular position of a surface in the plane of the workpiece coordinate system.
This can be used to determine and correct the rotation component in the zero offset (ZO, frame) of the workpiece in the plane.
The rotation is corrected in such a way that the real position of the surface (actual value) adopts the required set angle (_STA1) in the workpiece coordinate system when the corrected ZO (frame) is used.





The ZO to be corrected is specified in coded form with variable **_KNUM** >0.
The ZO can be specified and corrected by various methods, e.g. in various settable frames, in various basic frames, system frames.
If _KNUM=0, there is no ZO correction.

For detailed information on specifying _KNUM for the zero offset: Refer to Section 2.3.

In addition to _KNUM, another item of data is required to determine the type of angular offset in parameter **_RA**:

- _RA=0: Correction of the rotation around the 3rd axis, that is not contained in _MA (neither measuring nor offset axis)
- _RA>0: Channel axis number of the rotary table. The angular correction is made in the translatory part of the channel axis _RA.

© Siemens AG, 2004. All rights reserved

Parameters

For 1-angle measurement:

_MVAR	105	Angle measurement and ZO determination, positioning at an angle from measuring point to measuring point
	1105	Angle measurement with differential measurement and ZO determination, positioning at an angle from measuring point to measuring point
	100105	Angle measurement and ZO determination, paraxial positioning from measuring point to measuring point in the offset axis
_SETVAL	REAL, >0	Setpoint (axis position) at measuring point 1 in the meas. axis (for _MVAR=105 and _MVAR=1105 only)
_STA1	REAL, -45 to +45 deg.	Setpoint angle
_MA	201	Measuring axis: 1 (abscissa), offset axis: 2 (ordinate)
	102	Measuring axis: 2 (ordinate), offset axis: 1 (abscissa)
	301	Measuring axis: 1 (abscissa), offset axis: 3 (applicate)
	302	Measuring axis: 2 (ordinate), offset axis: 1 (applicate)
	203	Measuring axis: 3 (applicate), offset axis: 2 (ordinate)
	103	Measuring axis: 3 (applicate), offset axis: 1 (abscissa)
_ID	REAL, >0	Distance between measuring points P1 and P2 in offset axis
_RA	0	Offset of rotation about axis that is not contained in _MA
	>0	Channel axis number of the rotary table The angle offset is applied in the translation component of the channel axis number (rotary axis).
_MD	INT, 0 or1	0: Positive measuring direction 1: Negative measuring direction (only for variants with paraxial intermediate positioning _MVAR=10x10x)
_KNUM	0, >0	0: without automatic ZO correction >0: with automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)

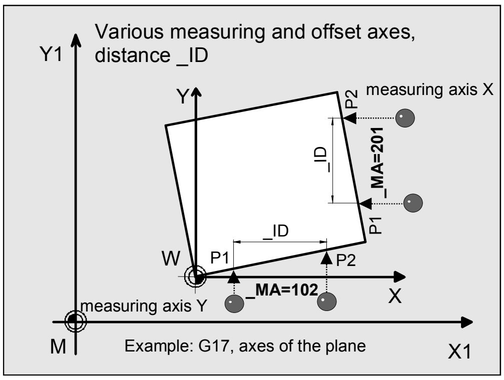
The following additional parameters are also valid **_VMS, _CORA, _TSA, _FA, _PRNUM, _EVNUM** and **_NMSP**.
See Sections 2.2 and 2.3.

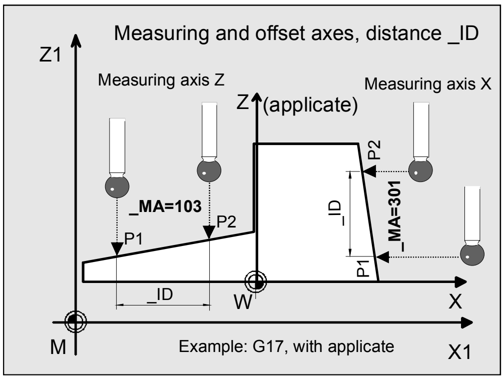
_CORA only relevant for monodirectional probe.
With _TSA, the difference of the angle is monitored and this value is additionally traversed to _STA1 with intermediate positioning at an angle. _TSA has the dimension unit degrees in this case!

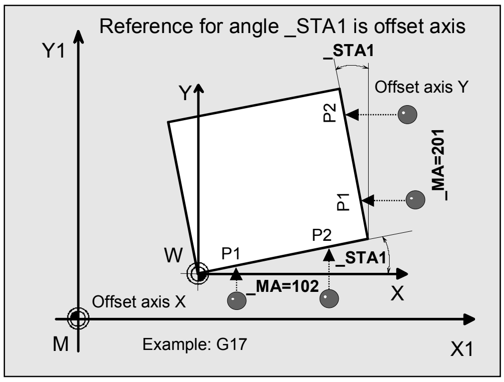
Notice
Precise angle definition requires a minimum surface finish at least at the measuring points. The distances between the measuring points must be selected as large as possible.

© Siemens AG, 2004. All rights reserved

Sequence







Measurement axis _MA

In this cycle, not only the measuring axis but also the offset axis are specified in **_MA**. The offset axis is the 2nd axis of the measuring plane. Intermediate positioning to the measuring point is performed in this axis for paraxial positioning; for positioning at an angle it is performed in both axes. It is also possible to specify the applicate as the measurement or offset axis.

Distance measuring point 1 to measuring point 2 in the offset axis: _ID

Parameter **_ID** is used to define the distance between P1 and P2 in the offset axis. Only positive values are permissible for _ID. P1 must therefore be selected in the offset axis before the cycle begins.

_STA1 set angle

The setting in _MA makes all 3 measurement planes possible. The set angle **_STA1** therefore refers to the positive direction of the offset axis and is negative in the clockwise direction, positive in the counterclockwise direction.

The set angle _STA1 specifies the required angle between the edge and the positive direction of the offset axis. In the case of _STA1=0, the edge is aligned paraxially with regards to the offset axis after correction.

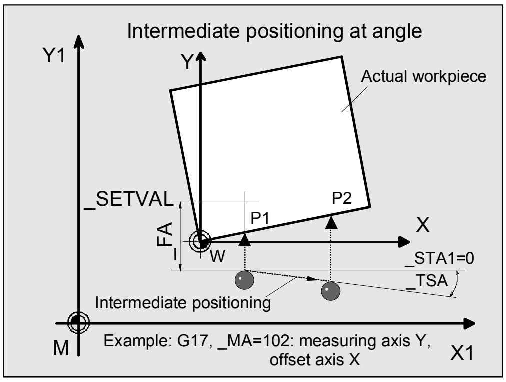
With measuring variants "Positioning at an angle“ (_MVAR=00x105), _STA1 is also used for positioning. The positioning angle is formed together with **_TSA**. _STA1 should therefore deviate only a little from the measured angle.

© Siemens AG, 2004. All rights reserved

Procedure with MVAR=00x105: Intermediate positioning at an angle

Position before measuring cycle call

The probe is positioned with respect to the surface to be measured in such a way that during traversal of the measuring axis _MA specified in the direction of the setpoint **_SETVAL**, **measuring point 1** on the surface will be reached.
Recommended distance from _SETVAL: >_FA.
The measuring operation then starts with the measuring feed at distance **_FA** in front of _SETVAL.



The absolute value of the positional deviation from the setpoint must not be greater than the measuring path _FA. Otherwise, no measurement will be performed.

In the other two axes, the positions are retained for the measurement in measuring point 1 at the beginning of a cycle.

Intermediate positioning at an angle

The starting point for measurement 2 is approached at an angle. The angle comprises _STA1 and _TSA. _TSA contains the value for a permissible angle deviation and leads away from the setpoint.

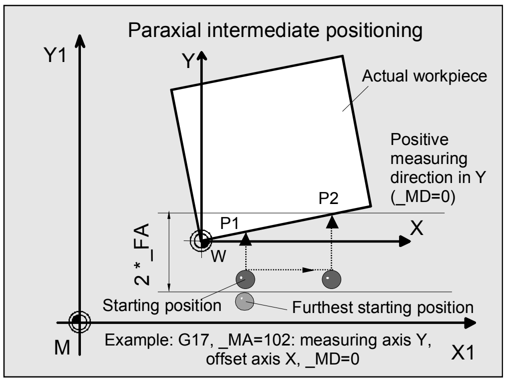
Procedure with MVAR=10x105: paraxial intermediate positioning

Position before measuring cycle call

The probe is positioned with respect to the surface to be measured in such a way that during traversal in the specified measuring axis _MA and direction of the measurement in **_MD** both measuring points on the surface within the

total measuring path: **2 · _FA** in mm

will be reached.
Otherwise no measurement or complete measurement will result.
In the other two axes, the positions are retained for the measurement in measuring point 1 at the beginning of a cycle.



The starting point for measurement 2 is approached **paraxially** in the offset axis. Measuring point 2 is also approached with the measuring axis in direction _MD.

© Siemens AG, 2004. All rights reserved

Position after end of measuring cycle
After the end of measurement, the probe is at the last measuring point at distance _FA from the measuring surface.

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.

Precondition
The probe must be called as a tool with a tool length offset.

Tool type, preferably: 710
when using the cycle on a turning machine:
set type and _CBIT[14]=0.

Special procedure for differential measurement

Measuring point 1 is measured twice during differential measurement.

1. with spindle rotated through 180 degrees compared with the position at the beginning of the cycle (rotation of the probe by 180 degrees).
2. with the spindle position that applied at the beginning of the cycle

The tool radius of the probe + R or - R is defined as the trigger point defined for the axis direction.
The values are entered and stored in the global user data (GUD6) in the array for the associated workpiece probe _WP[ ].

A **multidirectional** probe does not have to be calibrated at the beginning of the cycle for measuring variant **_MVAR=1105**.
This measuring variant with differential measurement is only useful with the measuring axes _MA=x01 or _MA=x02.

Notice

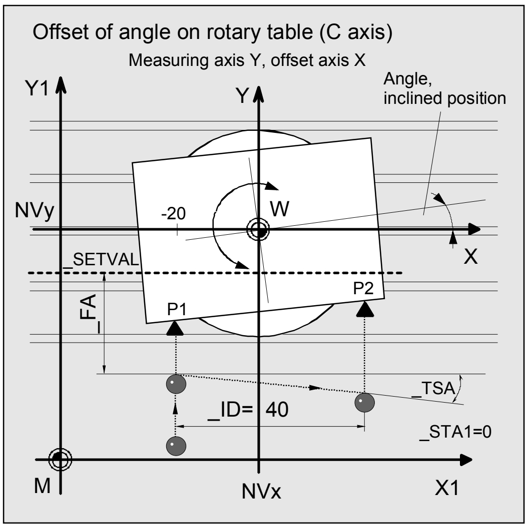
In the case of great measurement accuracy demands, differential measurement is not recommended!

© Siemens AG, 2004. All rights reserved

Programming example

1-angle measurement with CYCLE998

A rectangular workpiece (60 x 40 mm) is clamped in the G17 plane on a rotary table. The intention is to orient it with its edges running parallel with axes X and Y.
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 _TSA=5°. The measuring path is programmed as _FA=8 mm (max. total measurement path = 16 mm). The measuring points should be 40 mm apart. Intermediate positioning is to be at an angle.
The rotary table is the 4th axis in the channel (C axis).



Clamping for workpiece:
Zero offset, with settable ZO G54:
NVx, NVy, NVz, NVc

Workpiece probe 1, inserted as tool **T9, D1**, is to be used as probe.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration. Careful when positioning! Radius R in length (L1) is ignored.

%_N_ANGLEMEAS_MPF	
N10 G54 G17 G90 T9 D1	;Select T No. probe
N20 M6	;Insert probe as tool, activate offset
N30 G0 C0	;Position rotary table at 0°
N40 X-20 Y-40	;Position probe in X/Y plane opposite ;measuring point
N50 Z40	;Z axis at measurement height
N60 _PRNUM=1 _VMS=0 _NMSP=1 _EVNUM=0 **N61 _MVAR=105 _SETVAL=-18 _MA=102 _ID=40 _RA=4 _KNUM=1 _STA1=0 _TSA=5 _FA=8**	;Set parameters for measuring cycle call

© Siemens AG, 2004. All rights reserved

N70 CYCLE998	;Measuring cycle for angle measurement
N80 G0 Z160	;Traverse up Z axis
N90 G54 C0	;Repeat call of ZO G54 ;This activates the changes! ;Position rotary table at 0° ;(edge is now setup).
N100 M2	;End of program

Explanation

Measurement is performed in the Y direction, offset in the X direction, intermediate positioning at an angle.
The cycle determines the measuring direction from the actual position in the Y direction and _SETVAL.
Automatic correction is performed in G54, offset in the C axis (4th channel axis) with the calculated angle (_STA1=0).
In block N90, the corrected ZO G54 is activated and the C axis is moved from position zero to position zero; the ZO difference is eliminated. The workpiece is then paraxial.
The results are entered in result array _OVR[ ].

© Siemens AG, 2004. All rights reserved

5.7.2 2-angle measurement

Function

Measuring variants

_MVAR=106 and _MVAR=100106

can calculate and correct the angular position of a plane oblique in space on a workpiece by measuring three 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)

Parameters

For 2-angle measurement:

_MVAR	106	2-angle measurement and ZO determination, Intermediate positioning at an angle
	100106	2-angle measurement and ZO determination, Intermediate positioning parallel to contour
_SETVAL	REAL, >0	Setpoint (axis position): Expected position on workpiece surface in measuring point P1 in the applicate (for _MVAR=106 only)
_STA1	REAL	Setpoint angle about 1st axis of the plane
_INCA	REAL	Setpoint angle about 2nd axis of the plane
_MD	0, 1	0: Positive measuring direction 1: Negative measuring direction (for _MVAR=100106 only)
_ID	REAL, >0	Distance between measuring points P1 and P2 in the 1st axis of the plane (abscissa)
_SETV[0]	REAL, >0	Distance between measuring points P1 and P3 in the 2nd axis of the plane (ordinate)
_KNUM	0, >0	0: without automatic ZO correction >0: with automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)

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

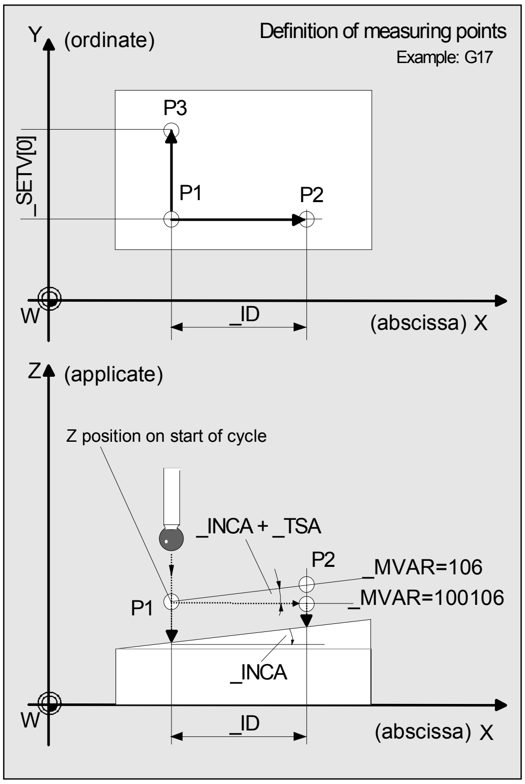
_CORA only relevant for monodirectional probe.
With _TSA, the difference of the angle is monitored and this value is additionally traversed to _STA1 with intermediate positioning at an angle. _TSA has the dimension unit degrees in this case!

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.

© Siemens AG, 2004. All rights reserved

Procedure for 2-angle measurement



Position before measuring cycle call
Before the cycle is called, the probe must be positioned over the 1st measuring point (P1) in the plane and at the appropriate depth in the applicate. The measuring axis is always the applicate.
Measuring point P1 must be selected in the plane such that _ID and _SETV[0] result in positive values.

Procedure for variant "intermediate positioning at an angle" (MVAR=106):
After completion of the measurement in P1 the probe is positioned at P2 in the abscissa and applicate (X and Z in G17) taking angle _INCA and maximum deviation in _TSA into account. After the measurement has been performed in P2, repositioning to P1 is performed by the same path. Then the probe is positioned from P1 to P3 in the ordinate and applicate (Y and Z in G17) taking account of angle _STA1 and maximum deviation in _TSA and then measured.

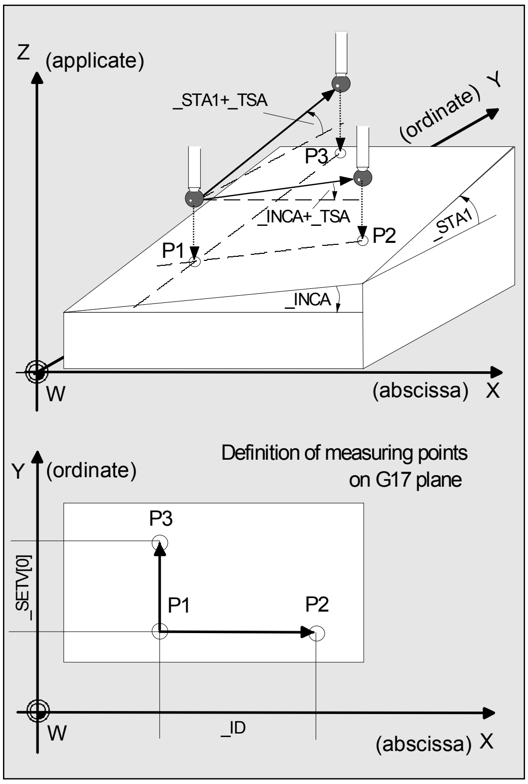
Procedure for variant "intermediate positioning parallel to axis" (MVAR=100106):
Positioning from P1 to P2 is performed in the abscissa, from P1 to P3 in the ordinate.
It must also be possible to reach P2 and P3 from starting position P1 in the applicate (in Z in G17) without collision.

Position after end of measuring cycle
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).

© Siemens AG, 2004. All rights reserved

Programming example 2

2-angle measurement with CYCLE998
(determining an oblique plane in space)



The task is to check the angular position of a machined oblique surface on a workpiece.
The result is taken from the result parameters _OVR[ ] for evaluation.
A measuring point 1(P1) must be selected where P2 in the ordinate (Y axis in G17) has the same value as P1 and the abscissa value (_ID) is positive. P3 must still have the same value in the abscissa (X axis in G17) as P1. The ordinate value (_SETV[0]) must be positive.
Positioning in the applicate must be performed as far as possible 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), maximum deviation _TSA= 5 degrees.

Workpiece probe 1, used as tool **T9, D1**, is to be used as probe.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

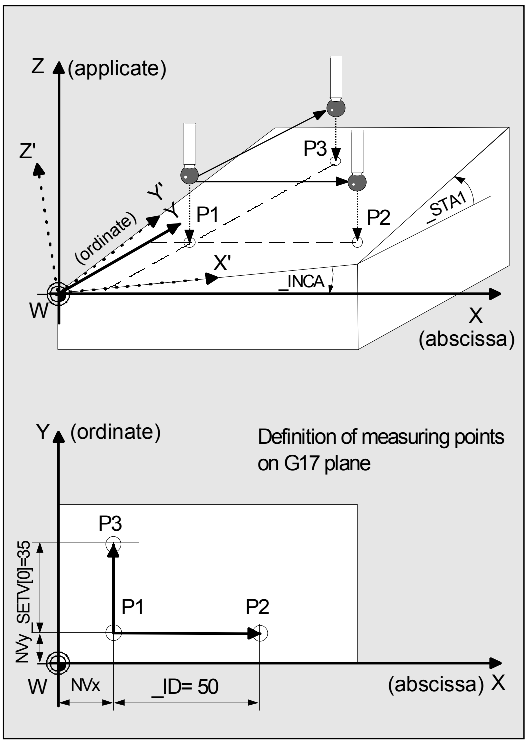
%_N_INCLINEDMEAS_MPF	
N10 G54 G17 G90 T9 D1	;Select T No. probe
N20 M6	;Activate offsets
N30 X70 Y-10	;Position probe in X/Y plane above ;measuring point
N40 Z40	;Position Z axis at measuring point level and ;select tool offset
N60 _MVAR=106 _SETV[0]=30 _ID=40 _KNUM=0 _RA=0 _STA1=8 _INCA=12 _TSA=5 _PRNUM=1 _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	;Traverse up Z axis
N540 M30	;End of program

© Siemens AG, 2004. All rights reserved

Explanation

Both measured angles are entered in result field _OVR[ ]. A ZO correction is not applied (_KNUM=0).

Programming example 3



Orientation of an oblique workpiece surface for remachining using CYCLE800

Initial state

- The workpiece is clamped on the swivel table (swiveling workpiece holder) and aligned roughly paraxially to the machine axes.
- The swivel table is in its home position.
- The probe is in inserted as T9 and positioned in JOG 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 ZO G56 at which the 2-angle measurement is to be performed and the G17 machining plane is defined as X0 Y0 Z20.

Task

Remachining will be performed with G57 active. The workpiece should be aligned so that for G17, the tool is located vertically on the previously inclined surface, the workpiece zero (G57) is the lefthand corner and the workpiece edges run in parallel to axes X and Y (G57). CYCLE978 should be used to set the 3 edges. To determine the angle, CYCLE998 (1 and 2 angle measurement).

%_N_PLANE_SETUP_MPF	
N500 G56 G17 G90	;Select ZO and machining plane
N505 T9 D1	;Select probe
N506 M6	;Activate tool compensation for probe
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 ZO memory G56 to G57
N530 G1 F500 X20 Y25	;Approach of the 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 correction 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 determination on surface, setpoint 0, ;meas. axis Z', safe area 10 mm, ;meas. path 20 mm before and after expected ;switching position, ZO correction in G57
N620 CYCLE978	;ZO determination on surface in Z' axis for ;**setting the zero in Z'**
N625 G57	;Activate the changed zero offset
N630 X20 Y-20	;Place in plane before the front edge
N640 Z-5	;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	;Position at measuring height before the front ;edge
N700 _MVAR=100 _MA=2 _SETVAL=0 _FA=10	;ZO determination on surface, meas. in ;Y' direction, measurement path 10 mm in ;front of to 10 mm behind expected edge
N710 CYCLE978	;ZO determination on surface with meas. ;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	

© Siemens AG, 2004. All rights reserved

N740 Y25	;Place in front of the left edge
N750 _MA=1	;Measure in +X'
N760 CYCLE978	;ZO determination on surface, measurement ;in ;+X' direction, and ZO correction in G57 ;memory. Measurement path 10 mm in front ;of up to 10 mm behind expected edge to ;**set 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.
Cycle CYCLE800 is not part of the "measuring cycle package" but of the "standard cycles".

Explanation

- 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.
- 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.

© Siemens AG, 2004. All rights reserved

5.8 CYCLE961 workpiece: Setting up an internal and external corner

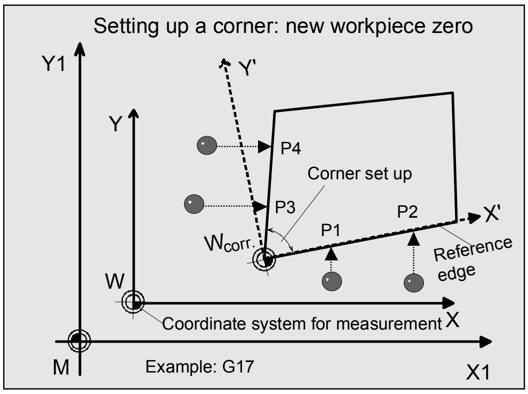
5.8.1 Function overview CYCLE961

Programming

CYCLE961

Function

The cycle can measure the **position of an internal or external corner** of a workpiece in the selected plane with different measuring variants The position of this corner can also be used as the workpiece zero in a defined zero offset (ZO).
In certain measuring variants an additional offset can be defined.
The measurements are performed with different specified values depending on the measuring variant used:



Specification of distances and angles

- The workpiece is a rectangle:
  3-point measurement
- Unknown workpiece geometry:
  4-point measurement

Specification of 4 points

- Unknown workpiece geometry:
  4-point measurement

Compensation of the zero offset

The ZO correction is applied in the **coarse offset**.
If a fine offset is available (MD18600: MM_FRAME_FINE_TRANS=1), it is reset.
If _KNUM=0, there is no zero offset (ZO).
When _KNUM 0, the corresponding ZO 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 (rotation about Z for G17) is offset in such a way that the workpiece coordinate system lies in the plane parallel to the reference edge.

Workpiece probe type that can be used
Multidirectional probe (_PRNUM=xy)

© Siemens AG, 2004. All rights reserved

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.

Precondition
The probe must be called as a tool with a tool length offset.
Tool type, preferably: 710
when using the cycle on a turning machine:
set type and _CBIT[14]=0.

Measuring variants

Measuring cycle CYCLE961 permits the following measuring variants that are specified with parameter _MVAR.

Value	*Measuring variant*
105	Setting-up an internal corner of a square/rectangle, specifying the angle and distances
106	Setting-up an external corner of a square/rectangle, specifying the angle and distances
107	Set up internal corner, specify angle and distances
108	Set up external corner, specify angle and distances
117	Set up internal corner, specify 4 points
118	Set up external corner, specify 4 points

© Siemens AG, 2004. All rights reserved

5.8.2 Setting up a corner with definition of distances and angles

Function

With this measuring cycle and measuring variants

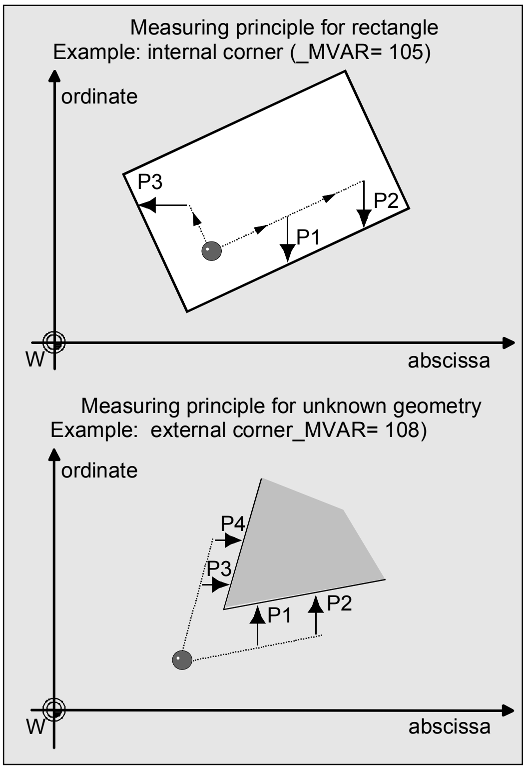
_MVAR=105, _MVAR=106

it is possible to measure and set up the internal and external corners of a **rectangle,**

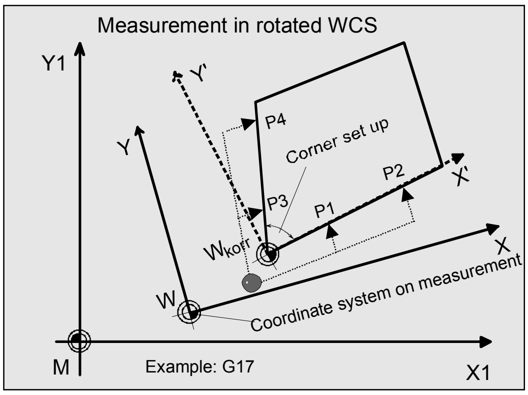
with measuring variants

_MVAR=107, _MVAR=108

it is possible to measure and set up the internal and external corners of an **unknown workpiece geometry**.



The cycle approaches either 3 (for a rectangle) or 4 measuring points (if 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 (rectangle) the corner to be calculated can be offset.
The result, the position of the corner, is stored as an absolute value in the specified ZO (offset and rotation) and in the result parameters _OVR[ ].



The measuring points are derived from the specified angle and distances. Measurement is performed paraxially to the existing workpiece coordinate system (WCS).

Notes:
When **Set up internal corner** the cycle only traverses in the plane at measuring height.
With **Set up external corner**, the corner can either be **passed over** the shortest path (lift in applicate) or **traveled around** in the plane.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	105	Set up internal corner of a rectangle (geometry known, 3 measuring points)
	106	Set up external corner of a rectangle (geometry known, 3 measuring points)
	107	Set up internal corner (geometry unknown, 4 measuring points)
	108	Set up external corner (geometry unknown, 4 measuring points)
_FA	REAL	Measuring path, only included if _FA larger than internally calculated
_KNUM	0, >0	0: without automatic ZO correction >0: with automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)
_STA1	REAL	Approximate angle between positive direction of abscissa to reference edge of workpiece **in MCS**[1)] (precision: <10 degrees): • Negative value in clockwise direction • Positive value in counterclockwise direction
_INCA	REAL	Approximate angle between reference edge and 2nd edge of workpiece (precision: <10 degrees): • Negative value in clockwise direction • Positive value in counterclockwise direction
_ID	REAL	Incremental retraction in applicate when measuring external corner, used to pass over the corner special **_ID=0:** The corner is traveled around – not passed over.
_SETV[0]	REAL, >0	Distance between starting point and measuring point 2 _STA1 (P1 is at _SETV[0] / 2)
_SETV[1]	REAL, >0	Distance between starting point and measuring point 4 in direction STA1+ INCA (P3 is at _SETV[1] / 2)

For measuring variants 105 and 106 only (rectangle):

_SETV[2]	REAL	Offset of zero offset WCS (corrected) in abscissa
_SETV[3]	REAL	Offset of zero offset WCS (corrected) in ordinated
_SETV[4]	REAL, possible values: 1, 2, 3, 4	Selection of offset: 1: Measured corner entered as zero point 2: Measured corner is entered as zero point offset by _SETV[3]. 3: Measured corner is entered in both axes offset by _SETV[2] (abscissa) and _SETV[3] (ordinate). 4: Measured corner is entered as zero point offset by _SETV[3].

1) Transformation deactivated, otherwise basic coordinate system

The following additional parameters are also valid
_VMS, _PRNUM, and **_NMSP**.
See Sections 2.2 and 2.3.

© Siemens AG, 2004. All rights reserved

Result parameters

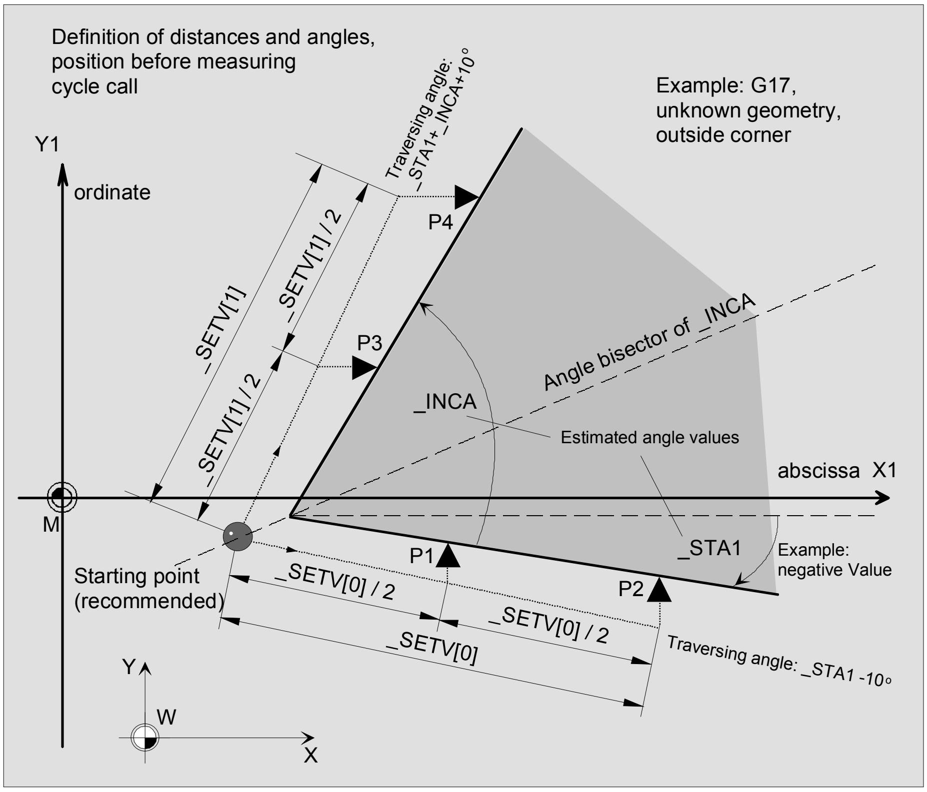
Measuring cycle CYCLE961 places the following values in data block GUD5 are results:

_OVR [4]	REAL	Angle to abscissa axis) in the workpiece coordinate system (WCS)
_OVR [5]	REAL	Actual value for corner point in abscissa in WCS
_OVR [6]	REAL	Actual value for corner point in abscissa in WCS
_OVR [20]	REAL	Angle to abscissa axis in the workpiece coordinate system (WCS)1)
_OVR [21]	REAL	Actual value for corner point in abscissa in MCS1)
_OVR [22]	REAL	Actual value for corner point in abscissa in MCS1)
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

1) Transformation deactivated, otherwise basic coordinate system

Sequence

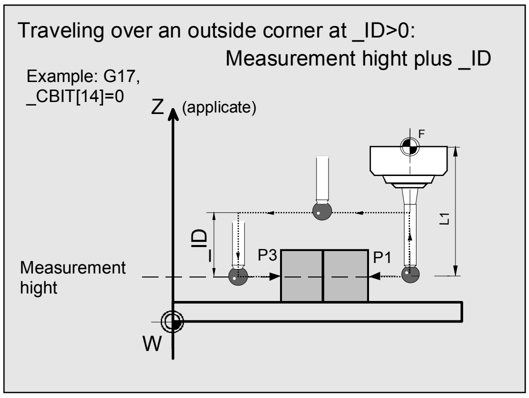
Defining distances and angles



© Siemens AG, 2004. All rights reserved

Position before measuring cycle call

The probe is positioned at measuring depth opposite the corner to be measured. It must be possible to approach the measuring points from here without collision. The measuring points are derived from the programmed distance between the starting point and
_SETVAL[0] (measuring point 2), or
_SETVAL[1] (meas. point 4) in direction of angle _STA1, _INCA.
The starting points for measuring point 1 and 3 are each located at half the distance.
Measurement is performed paraxially with the existing **workpiece coordinate system**.
_STA1 refers to the machine coordinate system.
The measuring cycle generates the required traversing blocks and performs the measurements at the measuring points.
During travel an additional tolerance angle of 10 degrees is added to the programmed angles in the cycle.
First measuring point **P 2**, then P 1, P 3, and depending on parameterization, P 4 is approached.



Traversing between P 1 and P 3 on outside edge:

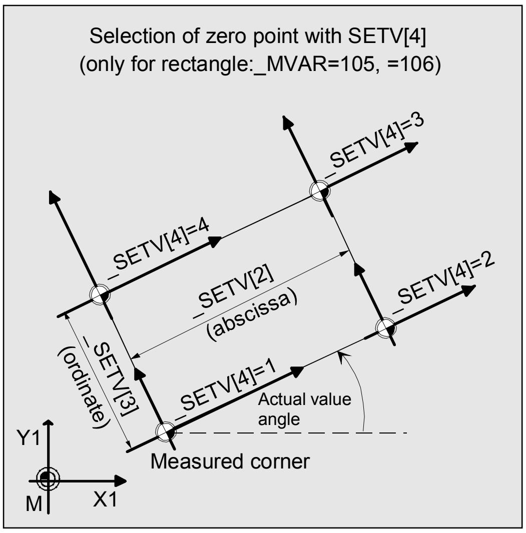
_ID=0: The corner is traveled around
_ID>0: At P 1, the probe is lifted in the applicate after measurement of _ID and approached via corner P.

Position after end of measuring cycle

The probe is again positioned at the starting point (at measuring depth opposite the measured corner).

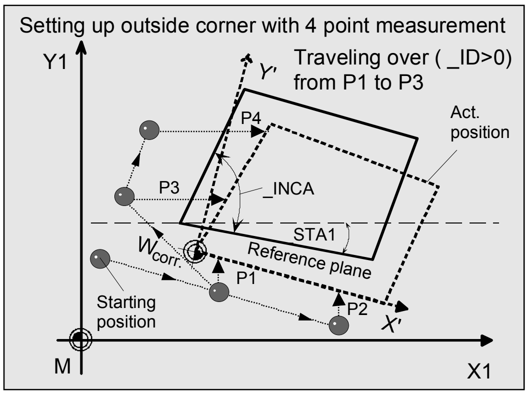
Selection of offset with _SETV[4]

With measuring variants MVAR=105, =106 (rectangle) the measured corner can be selected offset as workpiece zero.
The offset is specified in _SETV[2] (abscissa) and _SETV[3] (ordinate).
_SETV[4] can assume values 1 to 4.



© Siemens AG, 2004. All rights reserved

Programming example



The coordinates of the external corner of a workpiece with unknown geometry are to be determined.
Zero offset G55 is to be corrected in such a way that this corner is workpiece zero for G55.
The reference edge lies approximately at _STA1=-35 and the 2nd edge approximately at _INCA= 80 degrees in addition. The distance to measuring points 2 and 4 is 100 mm.
The corner is to be passed over from P1 to P3 at distance _ID= 30 mm above measuring height.
The starting point opposite the corner that is to be set up is reached before the measuring cycle is called.

Workpiece probe 1, inserted as tool **T9, D1**, is to be used as probe.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

%_N_CORNER_SETUP_MPF	
N10 G500 G17 G90 T9 D1	;Select probe, offset active
N20 _PRNUM=1 _VMS=0 _NMSP=1 **N21 _MVAR=108 _FA=20 _KNUM=2 _STA1=-35** **_INCA=80 _ID=30 _SETV[0]=100 _SETV[1]=100**	;The probe is at the starting position, ;e.g. by traversing in JOG ;Parameterize CYCLE961
N30 CYCLE961	
N40 G55	;Call corrected ZO G55
N100 M2	

© Siemens AG, 2004. All rights reserved

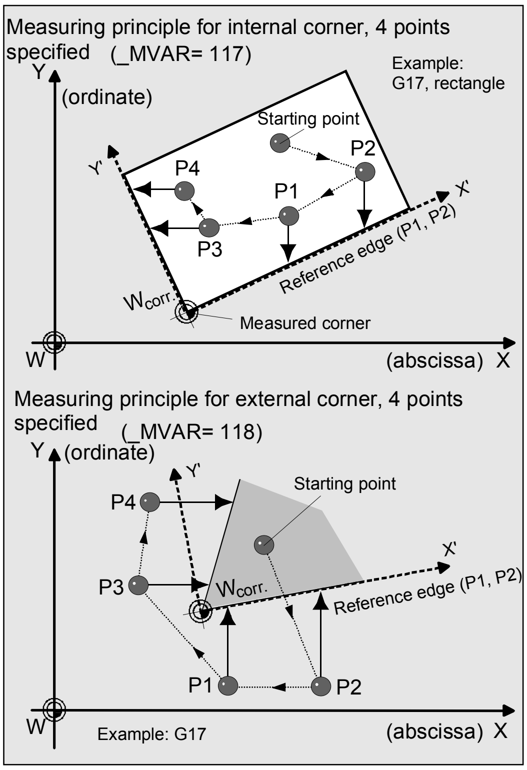
5.8.3 Setting up a corner with 4 points

Programming

CYCLE961

Function

With this measuring cycle and measuring variants
_MVAR=117, _MVAR=118
it is possible to measure and set up the internal and external corners of an **unknown workpiece geometry**.



One after the other, points **P2**, P1, P3, P4 are traversed in the cycle with positioning feedrate at positioning height. At each of these points the probe is lowered to measuring depth and then traversed at measuring feedrate parallel to the axis against the workpiece corner.
The cycle uses the relative positions of points P1 to P4 to determine the approach directions and the measuring axis. The cycle calculates the corner point and the angle between the reference edge and the positive abscissa axis of the current plane from the results of measurement.

The angle is calculated by measuring **P2 and P1** (reference edge).
The position of corner, corner point coordinates, and rotation are stored in result parameter _OVR[ ].

If _KNUM>0, absolute correction to the coarse offset in the specified ZO (translation and rotation) is performed.

The measuring points are derived from the specified 4 points. Measurement is performed paraxially to the existing workpiece coordinate system (WCS).

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	117	Set up internal corner, specify 4 points
	118	Set up external corner, specify 4 points
_FA	REAL	Measurement path
_KNUM	0, >0	0: without automatic ZO correction >0: with automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)
_ID	REAL	Incremental infeed in applicate for measuring depth
_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

The following additional parameters are also valid
_VMS, _PRNUM, and **_NMSP**.

See Sections 2.2 and 2.3.

Result parameters

Measuring cycle CYCLE961 places the following
values in data block GUD5 are results:

_OVR [4]	REAL	Angle to abscissa axis) in the workpiece coordinate system (WCS)
_OVR [5]	REAL	Actual value for corner point in abscissa in WCS
_OVR [6]	REAL	Actual value for corner point in abscissa in WCS
_OVR [20]	REAL	Angle to abscissa axis in the workpiece coordinate system (WCS)[1)]
_OVR [21]	REAL	Actual value for corner point in abscissa in MCS[1)]
_OVR [22]	REAL	Actual value for corner point in abscissa in MCS[1)]
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

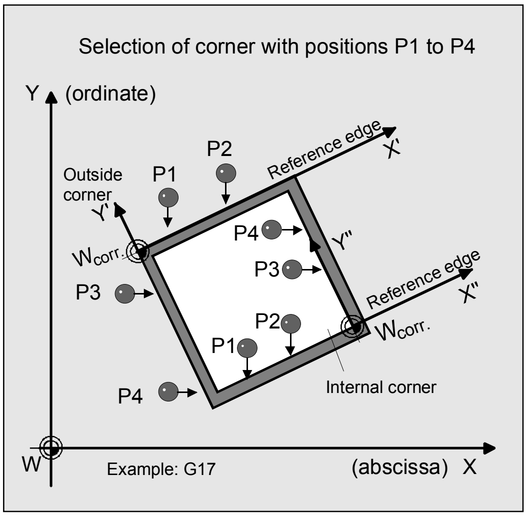
1) Transformation deactivated, otherwise basic coordinate system

© Siemens AG, 2004. All rights reserved

Sequence

Defining the 4 points

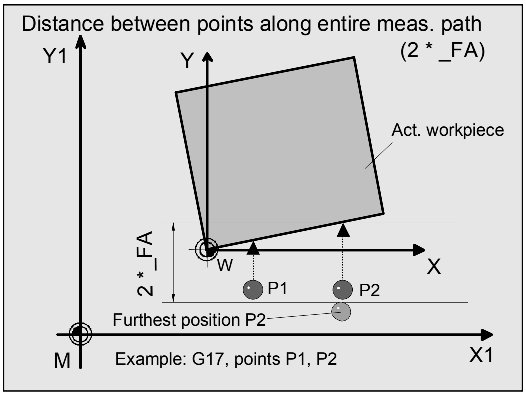
The position of points P1 and P2 in relation to each other determines the direction of the abscissa axis (X axis in G17) of the **new** coordinate system. A negative offset between P1 and P2 in the abscissa (X axis in G17) results in an additional rotation about 180°!
The position of the corner is selected with all 4 points. So for a rectangle, for example, different corners can be selected as the zero point depending on whether measuring variant internal or external corner is applied.



The individual points or _FA must be selected such that the contour is reached within a

total measuring path: 2 · _FA in mm.

Otherwise, no measurement will be performed.

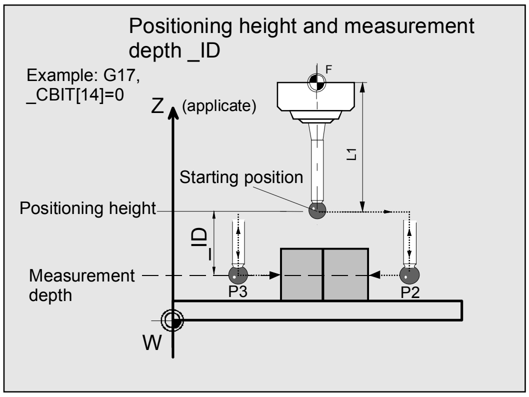


A minimum total measurement path of

2 · 20 mm is generated internally.

Position before measuring cycle call

The probe is above the workpiece at positioning height. It must be possible to reach all points without collision.
The measuring cycle generates the traversing blocks and performs the measurements at the measuring points from points P1 to P4. The measuring depth is derived from the positioning height lowered by the value in **_ID** (negative sign).
After measurement at one point the probe is again raised to positioning height and traversed to the next point and then lowered again to measuring depth.



Point **P2** is approached first, followed by P1, P3, and P4.

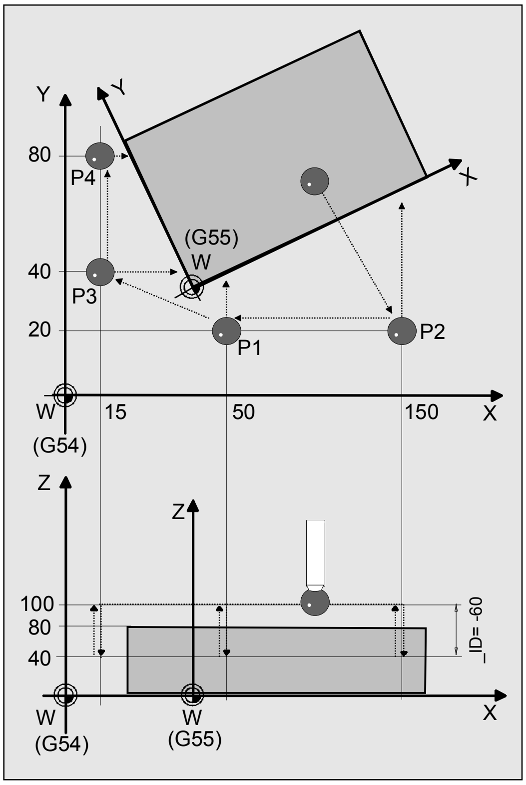
Position after end of measuring cycle

The probe is at the positioning height at point P4.

© Siemens AG, 2004. All rights reserved

Programming example

The coordinates of the corner of a workpiece are to be determined by outside measurement. ZO G55 must be corrected in such a way that the corner point is workpiece zero when G55 is selected.



Measurement is performed in plane G17 with active G54. The values in mm of the coordinates of points P1…P4 from which the workpiece can be approached parallel to the axis are:

P1.x=50 P1.y=20
P2.x=150 P2.y=20
P3.x=15 P3.y=40
P4.x=15 P4.y=80.

The probe is to be positioned at a height of 100 mm.
The measuring depth is 60 mm lower.
The workpiece corner is expected to be at a distance less than 200 mm at each point (_FA=100[mm]).

Workpiece probe 1, used as tool **T9, D1**, is to be used.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration. Careful when positioning! Radius R in length (L1) is ignored.

%_N_CORNER_SETUP_1_MPF	
N10 G54 G17 G90 T9 D1	;Select ZO, plane, probe, …
N20 G0 Z100	;Position probe at measuring height
N30 X100 Y70	;Position probe in X/Y plane above ;workpiece
N50 _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=-60** **N51 _VMS=0 _NMSP=1 _PRNUM=1 _FA=100 _KNUM=2**	;Measuring variant external corner ;Coordinates from P1 to P4 ;Measurement path 100 mm to expected ;edge (max. measurement path=200 mm)
N60 CYCLE961	;Cycle call

© Siemens AG, 2004. All rights reserved

N70 G55	;Call corrected ZO G55
N80 G0 X0 Y0	;Position probe in X/Y plane above ;corner (new zero point)
N100 M2	;End of program

© Siemens AG, 2004. All rights reserved

5.9 CYCLE997 workpiece: Measuring spheres and ZO determination (from measuring cycles SW 6.3)

5.9.1 Function overview CYCLE997

Programming

CYCLE997

Function

With measuring cycle CYCLE997, different measuring variants can be used to measure

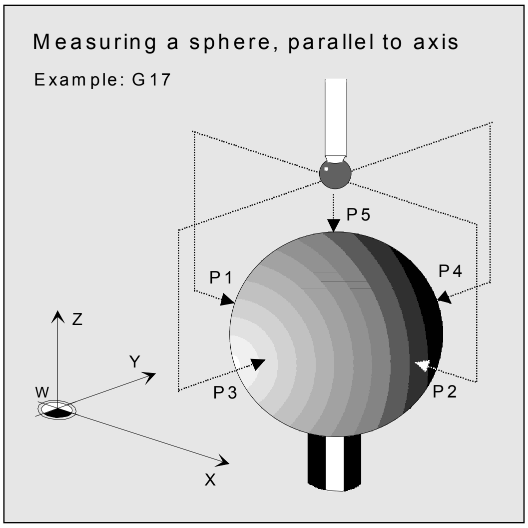
- a sphere or
- 3 equal sized spheres, fixed to a common base (workpiece).

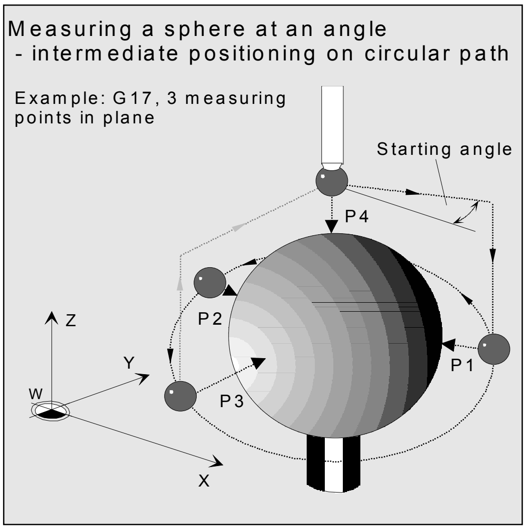
Measurement is performed **paraxially** with the existing workpiece coordinate system (WCS) **or at an angle** in the plane.

The center point (position of sphere) is derived from 4 or 5 measuring points around the circumference with a known diameter. With an additional measurement the diameter can also be determined.
Intermediate positioning for measuring points P1 to P3 and P4 (determining circle in plane) is performed with measuring variant "at an angle" on a circular path, otherwise parallel with the axis.
Positioning in the infeed axis and between the spheres is always a linear movement.

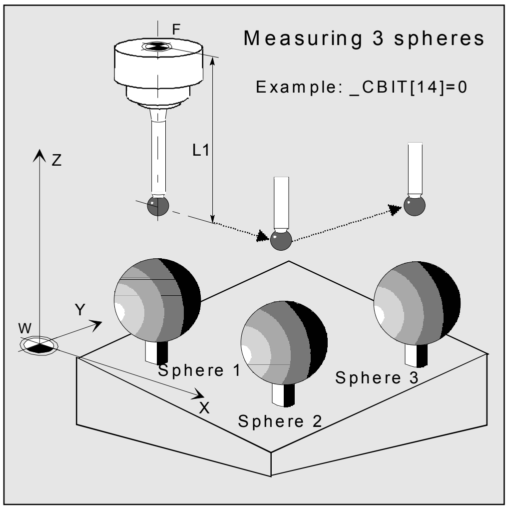
CYCLE997 can measure the sphere and in addition automatically correct a **zero offset (ZO)** on the basis of the position of the center of the sphere.

With "Measure 3 spheres" the angles in space of the sphere grouping can also be determined.
The target of the ZO correction can be selected with _KNUM.





© Siemens AG, 2004. All rights reserved



Workpiece measuring probe type that can be used

Multidirectional probe (_PRNUM=xy)
Measurement is performed in all three coordinate axes.

Different lengths can be specified for the probe in _CBIT[14]:

- _CBIT[14]=0: L1 referred to probe ball center
- _CBIT[14]=1: L1 is the total length

In measurement the same setting as for calibration of the workpiece probe must be used.

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.

Precondition

The probe must be called as a tool with a tool length offset.
Tool type, preferably: 710

For "Measuring at an angle" (_MVAR=xx1109) the probe must have been calibrated with "Determine active probe ball diameter".
Calibration with an additional "Determine position deviation" improves the measuring precision.

In ZO (frame) the approximate values for the position of the spheres in offset and rotation are entered and activated.
Only small deviations are expected.
The sphere diameter must be much larger than the probe ball diameter.

Important

The user must select measuring points for the particular measuring variant such that during measurement or intermediate positioning a collision with a sphere fixture or other obstacle is ruled out.

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE997 permits the following measuring variants which are specified via parameter _MVAR.

Value							*Measuring variant*
7	**6**	**5**	**4**	**3**	**2**	**1**	
				1		9	Measuring a sphere and ZO determination
					0		No measurement repetition
					1		With measurement repetition (with derived values)
			0				Measurement paraxial (to axes of the WCS)
			1				Measurement at an angle (intermediate positioning on circular path)
		0					Measure 1 sphere
		1					Measure 3 spheres
	0		1				3 circular measuring points (for "Measuring at an angle" only)
	1		1				4 circular measuring points (for "Measuring at an angle" only)
0							Without diameter determination (ball diameter known)
1							With diameter determination

Parameters

_SETVAL	REAL	Setpoint sphere diameter
_SETV[0]	REAL	Setpoint center abscissa – 1st sphere
_SETV[1]	REAL	Setpoint center ordinate – 1st sphere
_SETV[2]	REAL	Setpoint center applicate – 1st sphere
_SETV[3]	REAL	Setpoint center abscissa – 2nd sphere 1)
_SETV[4]	REAL	Setpoint center ordinate – 2nd sphere 1)
_SETV[5]	REAL	Setpoint center applicate – 2nd sphere 1)
_SETV[6]	REAL	Setpoint center abscissa – 3rd sphere 1)
_SETV[7]	REAL	Setpoint center ordinate – 3rd sphere 1)
_SETV[8]	REAL	Setpoint center applicate – 3rd sphere 1)
_RF	REAL	Velocity for intermediate paths on circular path (G2 or G3) (for _MVAR=xx11x9, – "Measuring at an angle" only)
_KNUM	INT 0, >0	0: without automatic ZO correction >0: With automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)
_STA1	REAL	Starting angle (for _MVAR=xx11x9, – "Measuring at an angle" only)
_INCA	REAL	Incremental angle (for _MVAR=xx11x9, – "Measuring at an angle" only)
_TNVL	REAL	Limit value for distortion of triangle (sum of deviations) ZO is only corrected if the calculated distortion is below this limit value. (for _MVAR=x1x1x9 – "Measuring 3 spheres" and _KNUM>0 only)

1) For measuring variants _MVAR=x1x1x9, measuring 3 spheres, only

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

© Siemens AG, 2004. All rights reserved

Result parameters

The measuring cycle CYCLE997 supplies the following values as results in the GUD5 block:

_OVR [0]	REAL	Setpoint sphere diameter, 1st sphere
_OVR [1]	REAL	Setpoint center point coordinate abscissa 1st sphere
_OVR [2]	REAL	Setpoint center point coordinate ordinate 1st sphere
_OVR [3]	REAL	Setpoint center point coordinate applicate 1st sphere
_OVR [4]	REAL	Actual value sphere diameter 1st sphere
_OVR [5]	REAL	Actual value center point coordinate abscissa 1st sphere
_OVR [6]	REAL	Actual value center point coordinate ordinate 1st sphere
_OVR [7]	REAL	Actual value center point coordinate applicate 1st sphere
_OVR [8]	REAL	Difference sphere diameter 1st sphere
_OVR [9]	REAL	Different center point coordinate abscissa 1st sphere
_OVR [10]	REAL	Difference center point coordinate ordinate 1st sphere
_OVR [11]	REAL	Difference center point coordinate applicate 1st sphere
_OVR [12]	REAL	Actual value sphere diameter 2nd sphere 1)
_OVR [13]	REAL	Actual value center point coordinate abscissa 2nd sphere 1)
_OVR [14]	REAL	Actual value center point coordinate ordinate 2nd sphere 1)
_OVR [15]	REAL	Actual value center point coordinate applicate 2nd sphere 1)
_OVR [16]	REAL	Difference sphere diameter 2nd sphere 1)
_OVR [17]	REAL	Difference center point coordinate abscissa 2nd sphere 1)
_OVR [18]	REAL	Difference center point coordinate ordinate 2nd sphere 1)
_OVR [19]	REAL	Difference center point coordinate applicate 2nd sphere 1)
_OVR [20]	REAL	Actual value sphere diameter 3rd sphere 1)
_OVR [21]	REAL	Actual value center point coordinate abscissa 3rd sphere 1)
_OVR [22]	REAL	Actual value center point coordinate ordinate 3rd sphere 1)
_OVR [23]	REAL	Actual value center point coordinate applicate 3rd sphere 1)
_OVR [24]	REAL	Difference sphere diameter 3rd sphere 1)
_OVR [25]	REAL	Difference center point coordinate abscissa 3rd sphere 1)
_OVR [26]	REAL	Difference center point coordinate ordinate 3rd sphere 1)
_OVR [27]	REAL	Difference center point coordinate applicate 3rd sphere 1)
_OVR [28]	REAL	Safe area
_OVI [0]	INTEGER	ZO number
_OVI [2]	INTEGER	Measuring cycle number = 997
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number
_OVI [11]	INTEGER	Status offset request
_OVI [12]	INTEGER	Additional error information on alarm output, internal measurement evaluation

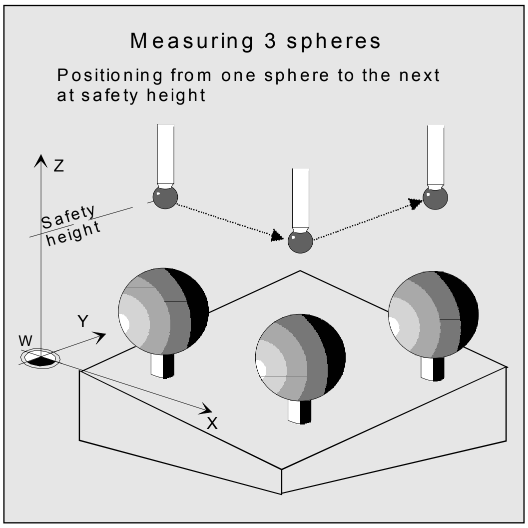
1) For measuring variants _MVAR=x1x1x9, measure 3 spheres, only

© Siemens AG, 2004. All rights reserved

5.9.2 Measurement and ZO determination

Measurement and calculation strategy

At the beginning of the cycle the probe must be in the infeed axis at safety height. It must be possible to reach all spheres from here without collision.
The cycle starts with measurement of the 1st sphere. Active G17 to G19 defines the plane with abscissa, ordinate. The applicate is the infeed axis.
4 or 3 measuring points are approached at the height of the center point setpoint of the applicate. The actual center point of the circle in the plane is calculated internally from these measured values (center of sphere in plane). For measurement "at an angle" the auxiliary cycle CYCLE116 is used for calculation.
The last measuring point is located exactly above the calculated sphere center in the plane and is approached using the applicate as the measuring axis.
The actual **sphere center point** in abscissa, ordinate, applicate is calculated from all these measuring points.



If measuring variant "Measuring 3 spheres" is selected with _MVAR, these spheres are then measured in the order sphere 2, sphere 3 in the same way.

Selecting the measuring variant
With variant "Measure at angle" (_MVAR=0x1109) fast calculation of the sphere position is possible if the sphere diameter is known (low number of measuring points and few intermediate positioning actions)
"Paraxial measurement" (_MVAR=0x0109) always requires 5 measuring points with more intermediate positioning actions.
In both types of measurement it is possible to **repeat measurement** with the located sphere center point (_MVAR=xxx119). Repeating measurement improves the measuring result.
It is also possible to calculate the **sphere diameter** (_MVAR=10xx1x9). In this case an additional measurement is taken parallel to the axis in the plus direction of the abscissa at the height of the sphere center calculated in the first measurement. Calculation of the sphere diameter and measurement repetition can be combined (_MVAR=10xx119). Here the diameter is calculated after each position measurement.

© Siemens AG, 2004. All rights reserved

Safe area

All setpoint/actual differences are checked for compliance with the safe area (parameter _TSA).
If this value is exceeded, alarm

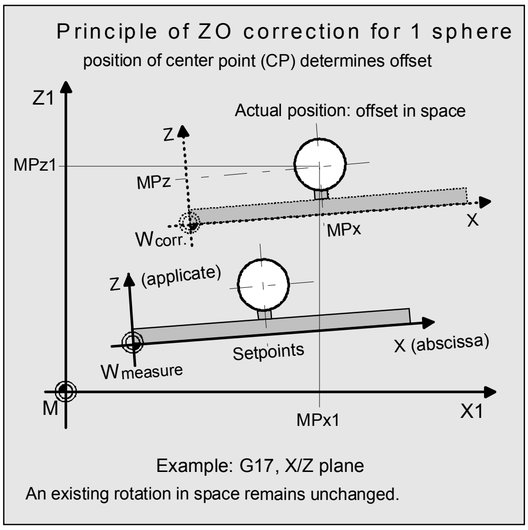
"61303 Safe area violated"

is output and an NCE RESET is necessary.
Measurement is then canceled.
If _CBIT[0]=1 the measurement is first repeated.

ZO correction when measurement of one sphere only (_MVAR=x0x1x9):

The actual value/setpoint differences of the center point coordinates are included the calculation of the translatory component of the ZO. The offset acts such that the calculated sphere center point in the offset ZO includes the specified setpoint position (workpiece coordinates, 3 axes).
The offset is applied to the ZO number as defined in _KNUM. When KNUM=0, there is no offset. No scaling factors must be active in the ZOs / frames.
Settings in _CHBIT[21]:
You can set whether a ZO offset should be FINE or COARSE in the translation component.

- _CHBIT[21]=0:
  Offset is additive in FINE (if FINE is available as set in the MD, otherwise in COARSE).
- _CHBIT[21]=1:
  Offset is COARSE, FINE is included in calculation and then reset (if FINE is available as set in the MD).



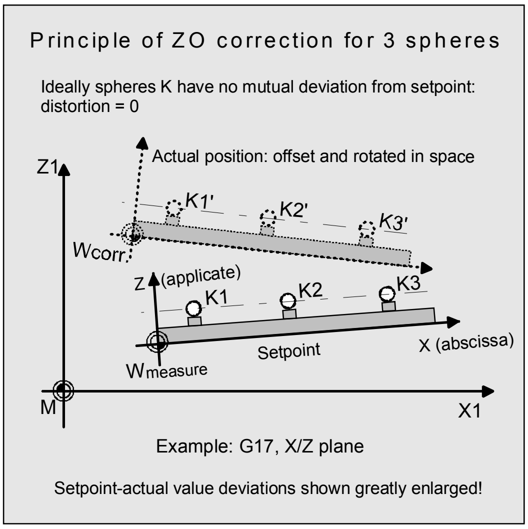
ZO correction when measuring 3 spheres (_MVAR=x1x109):

Compensation of the entire active frame with its translational and rotary components is performed after 3 spheres have been measured with cycle CYCLE119 (see Subsection 5.9.4 "CYCLE119…"). No mirroring or scaling factor may be active.

The offset acts such that the triangle formed by the 3 sphere center points includes the specified setpoint center positions (workpiece coordinates).

The sum of the deviations of the spheres in relation to each other (distorsion) must lie within the value in **_TNVL**. Otherwise no offset is performed and an alarm is output.

Offset compensation is always performed in COARSE (as described for _CHBIT[21]=1).



Note:

In this measuring variant (Measure 3 spheres) compensation in an NCU-global basic frame is not possible (_KNUM=1051 to 066). This frame has no rotation component.

Sequence

Position before measuring cycle call

Before measuring cycle CYCLE997 is called the probe must be positioned at safety height above the set sphere center point (setpoints in _SETV[...]) of the 1st sphere.

General

The measuring cycle generates the travel movements for approaching the measuring points itself and executes the measurements according to the selected measuring variant.

Note

The value selected for parameter **_FA** should be so large that all spheres can be reached within total measurement path 2 · _FA.

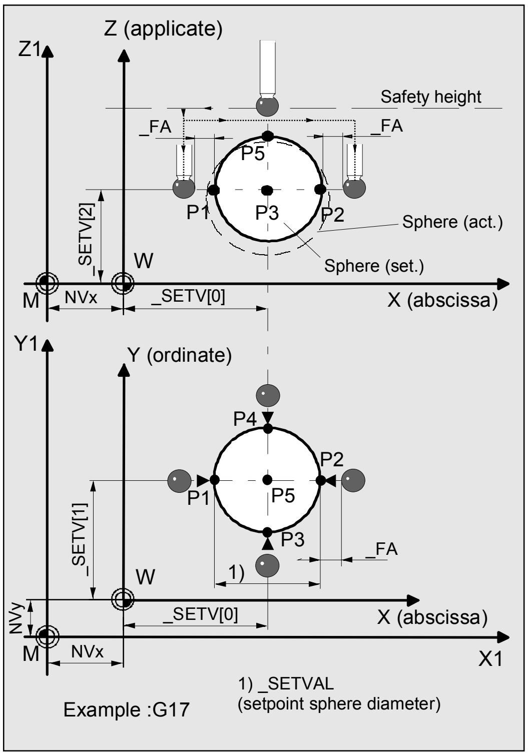
Otherwise, no measurement will be performed or they will be incomplete.

© Siemens AG, 2004. All rights reserved

Further procedure

Measuring variant "Paraxial measurement" (_MVAR=x01x9):

All intermediate positioning actions and measurement movements are paraxial with the active workpiece coordinate system. The measuring points are approached at distance _FA from the sphere lateral surface (setpoint sphere diameter). Measuring point P1 is approached first. After positioning in the abscissa, ordinate, the applicate is lowered to the height of the center point setpoint and the 1st measurement is taken Then P2 to P4 are approached and measured parallel to the axis.



P2 is approached via positioning of the applicate at distance _FA above the sphere (setpoint diameter) and lowering to measuring height again (setpoint center point of applicate). P3 and P4 are approached in the same way.

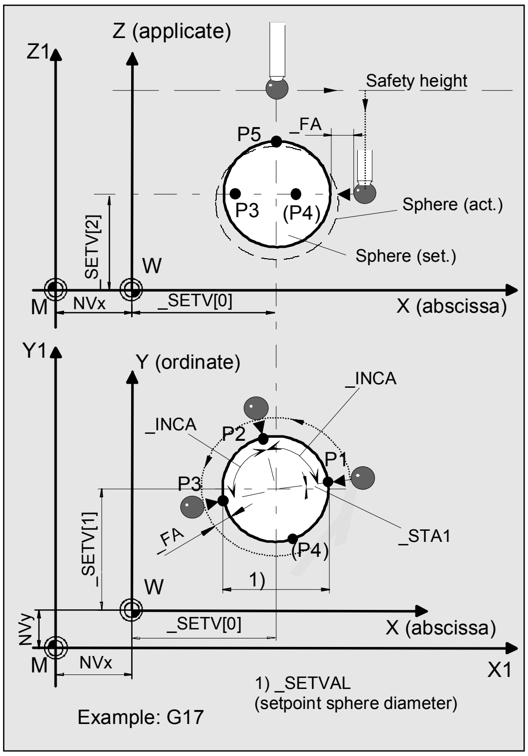
P3 and P4 lie at the center derived from P1 and P2 (actual value center point of abscissa).
The applicate is positioned from P4 to a distance _FA above the sphere and then approached in the abscissa, ordinate of the calculated actual value point (P5). The last measurement is taken: in the minus direction of the applicate.

After this measurement, the applicate is positioned to safety height (height as beginning of the cycle). When 3 spheres are measured, the abscissa, ordinate are positioned toward the **set center point of the next sphere**. Procedure continues as above.

No sphere fixture or other obstacle must be located in this entire traversing range. It may be necessary to select this measuring variant with intermediate positioning on a circular path (_MVAR=xx1109). The position and number of measuring points is then variable.

© Siemens AG, 2004. All rights reserved

Further procedure
Measuring variant "Measure at an angle" (_MVAR=xx11x9):
With _STA1 (starting angle) the angle position of P1 is defined, with _INCA, the incremental angle after P2 and then after P3. If the measuring variant is selected with 4 measuring points on a circle (_MVAR=1x1109), _INCA is also valid from P3 to P4.



The measuring points are approached at distance _FA from the sphere lateral surface (setpoint sphere diameter). Measuring point P1 is approached first. After joint positioning of abscissa and ordinate, the applicate is lowered to the height of the center point setpoint of the applicate and the 1st measurement is taken radially in the direction center point setpoint abscissa/ordinate.

Then P2 to P3 and P4 are measured on a **circular path** with feedrate _RF and measured in the same way as P1.

The applicate is positioned from P4 or P3 to a distance _FA above the sphere and then approached in the abscissa and ordinate of the calculated actual value point (P5). The last measurement is taken: in the minus direction of the applicate.

After this measurement, the applicate is positioned to safety height (height as beginning of the cycle). When 3 spheres are measured the abscissa and ordinate are positioned simultaneously **toward measuring point P1 of the next sphere** and continued as described above.

No sphere fixture or other obstacle must be located in this entire traversing range.

The sum of the starting angle _STA1 and all incremental angles _INCA may not exceed 360 degrees.

Position after end of measuring cycle
At the end of the cycle the probe is located above the first calculated actual center point of the 3rd or only sphere at safety height (height same as at beginning of cycle).

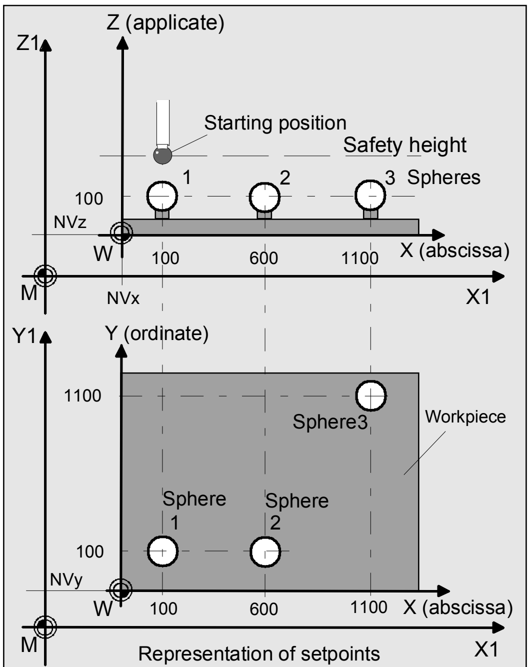
© Siemens AG, 2004. All rights reserved

5.9.3 Programming example CYCLE997

Programming example

Determining positional deviations in space

Three spheres each with a diameter of 50 mm are measured. Sphere center points 1 to 3 are expected for (X,Y,Z)=(100, 100, 100), (600, 100, 100) and (1100, 1100, 100).



Representation of setpoints

In its actual position the workpiece with 3 spheres is offset in space and clamped rotated. The deviations are measured and the ZO corrected.

ZO correction of the active frame is to be performed in accordance with the measured values. The exact diameter of the sphere is known.

A measured sphere lateral surface with a maximum deviation of ± 5 mm is expected (--> _FA=5).

Fixture of workpiece (ZO) with G54:
NVx, NVy, NVz

Workpiece probe 1, inserted as tool **T20, D1**, is to be used as probe.

The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T20, D1 in the tool offset memory:

Tool type (DP1):	710
Length 1- Geometry (DP3):	L1= 50.000
Radius (DP6):	R = 3.000

Length 1 (L1) refers to the center of the probe ball (_CBIT[14]=0), as for calibration.
Careful when positioning! Radius R in length (L1) is ignored.

© Siemens AG, 2004. All rights reserved

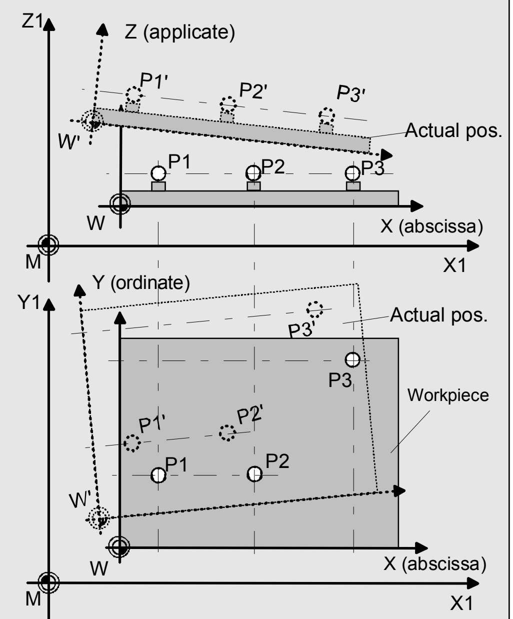
%_N_ DETERMINE_KS _MPF	
N10 G17 G54	;X-Y plane, active ZO
N20 T20 D1	;Select and activate ;probe with tool offset D1 (M6)
N30 G0 G90 Z200	;Approach position Z at safety height
N40 X100 Y100	;Approach position X, Y of 1st sphere
N50 _SETVAL=50 _SETV[0]=SET(100,100,100, 600, 100, 100,1100, 1100, 100)	;Setpoint parameters for measuring cycle ;call
N60 _MVAR=010109 _KNUM=9999 _TNVL=1.2	;Measure 3 spheres parallel to axis ;Offset in active frame, ;The offset is applied only if calculated ;distortion is less than 1.2 mm
N70 _VMS=200 _NMSP=1 _FA=5 _PRNUM=1	;Measuring velocity 200 mm /min ;Measurement at same measuring point ;Measurement path 5 mm in front of to 5 mm ;behind setpoint position (sphere lateral ;surface) ;Probe array _WP[0,0….9])
N100 CYCLE997	;Call measuring cycle
...	
N200 M2	;End of program

5.9.4 CYCLE119: Arithmetic cycle for calculating position in space

Function

This auxiliary cycle calculates the deviations in position and angle to the active frame from 3 defined setpoint positions in space (reference triangle) and 3 actual positions, and corrects a selected frame if necessary.

CYCLE119 is called as a subroutine by measuring cycle CYCLE997.
To allow this cycle to be used universally, its data are transferred via parameters.



3 linked points P1 to P3 (triangle) are offset in space and arranged rotated: P1', P2', P3'

© Siemens AG, 2004. All rights reserved

Programming

CYCLE119(_SETPOINT,_MEASPOINT,_ALARM,_RES,
_REFRAME,_COR,_RESLIM)

Parameters

Input data	*Data type*	*Meaning*
_SETPOINT[3,3]	REAL	Field for 3 setpoint positions in the sequence 1st, 2nd, 3rd, geometry axis (X,Y,Z) These points are the reference triangle.
_MEASPOINT[3,3]	REAL	Field for 3 measured positions in the sequence 1., 2., 3. Geometry axis (X,Y,Z). This is the real position in space of the described triangle.
_COR	INT	0: No compensation 1...99: ZO correction in G54...G57, G505..G599 1000: ZO correction of last active channel basic frame acc. to MD 28081 1011 to 1026: ZO correction in channel basic frame 2000: ZO correction in scratch system frame $P_SETFR 9999: ZO correction in active frame, settable frame G54 to ...G57, G505...G599 or in case of G500 in last active basic frame as per $P_CHBFRMASK
_RESLIM	REAL	Limit value for distortion (only relevant if _COR >0) If _RES is below this limit value, ZO is offset, otherwise an alarm is output.

Output data	*Data type*	*Meaning*
The results of calculation are stored in these transfer parameters		
_ALARM	INT	Cycle alarm number for feedback (transfer value must be 0 on cycle call).
_RES	REAL	Result of calculation < 0: No frame was calculated An alarm (_ALARM > 0) is returned. >=0: Calculation was successful. The size of the value is a measure of the distortion of the triangle, for example, due to inaccurate measurements. It is the sum of the deviations of the individual points in mm.
_REFRAME	FRAME	Result frame, difference from active frame If this result frame is linked to the active frame, the measured triangle position is given the desired setpoint position (workpiece coordinates).

Notes for compensation:
The frame to be corrected must not contain any mirroring or scaling factors. If no channel basic frame exists for G500, a cycle alarm (_ALARM>0) is output.
The new frame data with renewed programming of the G command of the associated settable frame is activated (G500, G54 to …) outside this cycle by the user.

© Siemens AG, 2004. All rights reserved

Programming example Application of CYCLE119

%_N_ Check _MPF	
;Calculate new frame according to transferred points and correct in active frame ;Apply (_COR=9999) if distortion is _RES < 1.2 mm:	
DEF REAL _SETPOINT[3,3],_MEASPOINT[3,3]	
DEF REAL _RES, _RESLIMIT	
DEF INT _ALARM	
DEF FRAME _REFRAME	
N10 G17 G54 T1 D1	
N20 _SETPOINT[0,0]=SET(10,0,0)	;Setpoint coordinates 1st point (X1,Y1,Z1)
N30 _SETPOINT[1,0]=SET(0,20,0)	;Setpoint coordinates 2nd point (X2,Y2,Z2)
N40 _SETPOINT[2,0]=SET(0,0,30)	;Setpoint coordinates 3rd point (X3,Y3,Z3)
;Program section for determining actual workpiece coordinates of the 3 points:	
...	
;Assignment of derived values:	
N100 _MEASPOINT[0,0]=SET(11,0,0)	;Actual value coordinates 1st point (X1,Y1,Z1)
N110 _MEASPOINT[1,0]=SET(1,20,0)	;Actual value coordinates 2nd point (X2,Y2,Z2)
N120 _MEASPOINT[2,0]=SET(1,0,30)	;Actual value coordinates 3rd point (X3,Y3,Z3)
;Calculation with compensation in G54:	
N200 CYCLE119(_SETPOINT,_MEASPOINT, _ALARM,_RES,_REFRAME,9999,1.2)	
IF (_ALARM==0) GOTOF _OKAY	
MSG ("Error: " <<_ALARM)	
M0	;Alarm occurred
GOTOF _END	
_OKAY: G54	;Activate corrected frame (ZO)
N400 G0 X... Y... Z...	;Traverse in corrected frame
...	
N500 _END: M2	

■

© Siemens AG, 2004. All rights reserved

Measuring Cycles for Turning Machines

© Siemens AG, 2004. All rights reserved

6.1 General prerequisites

The measuring cycles below are intended for use on turning machines.
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.

Overview of measuring cycles

(CYCLE972)[1)]	Calibrate tool probe, measure turning tools
CYCLE973	Calibrate workpiece probe in the reference groove or on surface
CYCLE974	1-point measurement with automatic tool offset or ZO determination
CYCLE982	Calibrate tool probe, measure turning and milling tools
CYCLE994	2-point measurement on diameter with automatic tool offset

Overview of the auxiliary programs required

CYCLE100	Logging on
CYCLE101	Logging off
CYCLE102	Measurement result display selection
CYCLE103	Preassignment of input data
CYCLE104	Internal subroutine: Measuring cycle interface
CYCLE105	Generate log contents: Log
CYCLE106	Logging the sequential controller Log
CYCLE107	Output of message texts (up to measuring cycles SW 6.2)
CYCLE108	Output of alarm messages (up to measuring cycles SW 6.2)
CYCLE109	Internal subroutine: Data transfer
CYCLE110	Internal subroutine: Plausibility checks
CYCLE111	Internal subroutine: Measuring functions
CYCLE113	Read system date and time Log
CYCLE114	Internal subroutine (tool offset)
CYCLE115	Internal subroutine (zero offset)
CYCLE117	Internal subroutine: Measuring functions
CYCLE118	Format real values: Log

1) CYCLE982 is the preferred cycle to use. CYCLE972 does not support graphic measuring cycles.

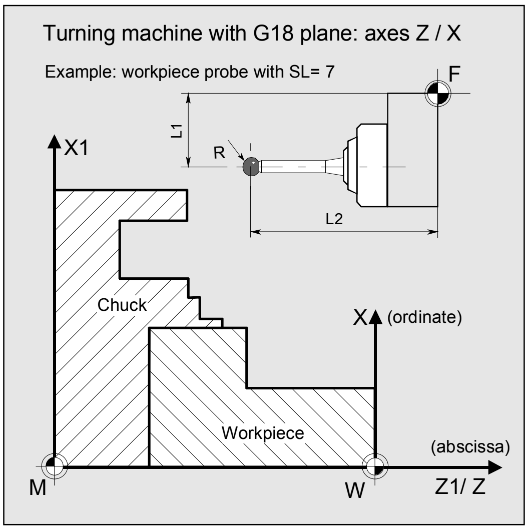
The measuring cycle data are defined in the data blocks:
- **GUD5.DEF**
- **GUD6.DEF**

The measuring cycles with measuring cycles SW 6.2 can only be used with NCK-SW 6.3 and higher.

© Siemens AG, 2004. All rights reserved

Call and return conditions

- D compensation with the data of the calibration tool or the workpiece probe or the tool to be measured must be activated in accordance with the measuring variant before a measuring cycle is called.
- The permissible tool type for the workpiece probe is type 5xy with cutting edge positions SL 5 to 8. Lengths refer to the center of the probe ball.
- No scaling factors <>1 must be active in the frames. Mirroring is permissible in the workpiece measuring cycles except for calibration (condition: MD 10610: MIRROR_REF_AX =0).
- The G functions active before the measuring cycle call are reactivated at the end of the cycle.



Level definition

The measuring cycles work internally with the abscissa and ordinate of the current plane G17 to G19.
The default setting for turning machines is **G18**.

Notes on the spindle

Spindle commands in the measuring cycles always refer to the active **master spindle** of the control. When using measuring cycles on machines with several spindles, the spindle concerned before the cycle call must be defined as the master spindle.

References: /PG/ "Programming Guide Fundamentals"

© Siemens AG, 2004. All rights reserved

6.2 CYCLE982, CYCLE972 Tool: Measuring turning tools

6.2.1 Function overview

Programming

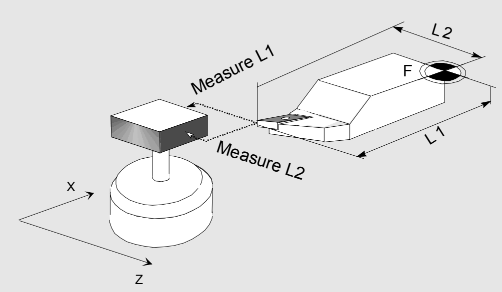
CYCLE982
CYCLE972

Function

Cycles CYCLE982, CYCLE972 implement the

- **Calibration of a tool probe**
  and the
- **Measuring turning tools**

(machine-related, probe arrays _TP[ ]).



Tool lengths L1, L2 of turning tools with cutting edge positions SL = 1 to 8 are measured.

It is only possible to measure tools with one calibrated tool probe.

Notes
CYCLE982 is the **preferred** cycle to use.
CYCLE972 does not provide graphics measuring cycle support.
Measuring cycle CYCLE982 provides extended measuring and calibration features over CYCLE972.

These variations are described in Section 6.3.

Measuring variants

Measuring cycles CYCLE982, CYCLE972 permit the following measuring variants that are specified with parameter _MVAR.

Value	*Meaning*
0	Calibrate tool probe (machine-related)
1	Measure tool (machine-related)

© Siemens AG, 2004. All rights reserved

Result parameters

Measuring cycles CYCLE982, CYCLE972 return the following values in data block GUD5 for measuring 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	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

Measuring cycles CYCLE982, CYCLE972 return the following values in data block GUD5 for measuring variant **tool measurement**:

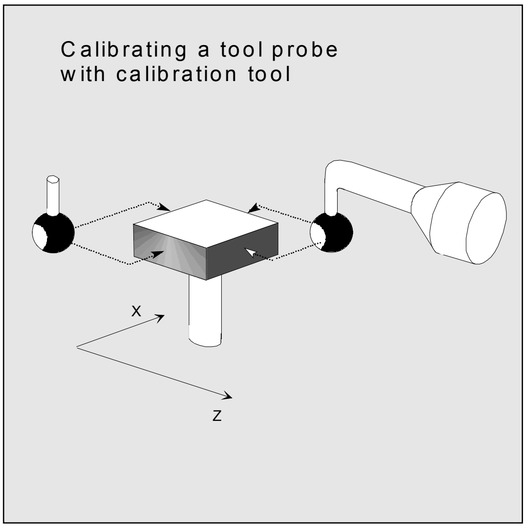
_OVR [8]	REAL	Actual value length L1
_OVR [9]	REAL	Difference length L1
_OVR [10]	REAL	Actual value length L2
_OVR [11]	REAL	Difference length L2
_OVR [27]	REAL	Zero offset area
_OVR [28]	REAL	Safe area
_OVR [29]	REAL	Permissible dimension difference
_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	D number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [7]	INTEGER	Empirical value memory number
_OVI [8]	INTEGER	Tool number
_OVI [9]	INTEGER	Alarm number

© Siemens AG, 2004. All rights reserved

6.2.2 Calibrating tool probes (machine-related)

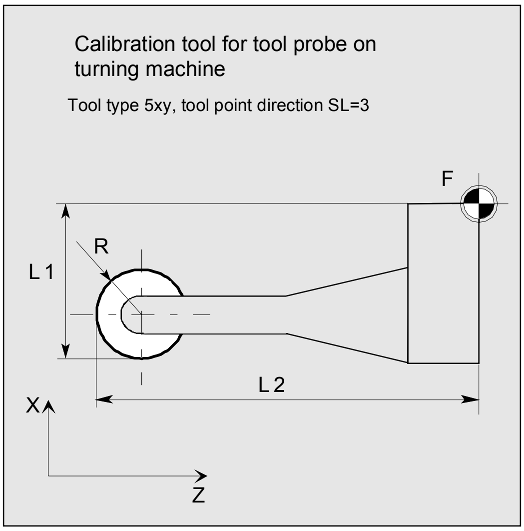
Calibration

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 data block GUD6 (_TP [ ]fields).
Values are corrected without empirical and mean values.

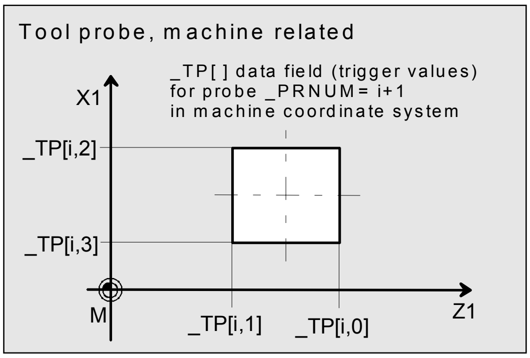


Note
If no special calibration tool is available, a turning tool with cutting edge position SL=3 can be used instead for calibration of 2 sides of the probe (see Subsection 6.2.3).

Preconditions
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 measuring cycle is called. A turning tool must be specified as the tool type (type 5xy). The cutting edge position must be SL= 3.



The lateral surfaces of the probe cube must be aligned parallel to the machine axes Z1, X1 (abscissa and ordinate).
The approximate coordinates of the tool probe PRNUM with respect to the machine zero must be entered in array
_TP[_PRNUM-1,0] to _TP[_PRNUM-1,3]
before calibration starts.
These values are used for automatic approach of the probe with the calibration tool and their absolute value must not deviate from the actual value by more than the value in parameter _TSA. It is also necessary for the probe to be reached within the
total measurement path 2 · _FA
will be reached.



© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	0	Calibrate tool probe (machine-related)
_MA	1, 2	Measuring axis
_PRNUM	INT	Probe number

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

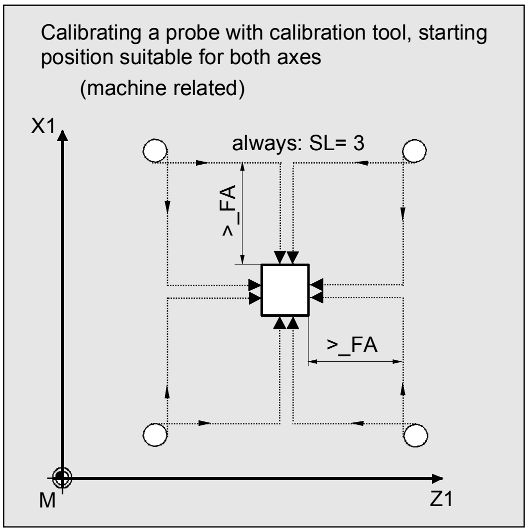
Sequence

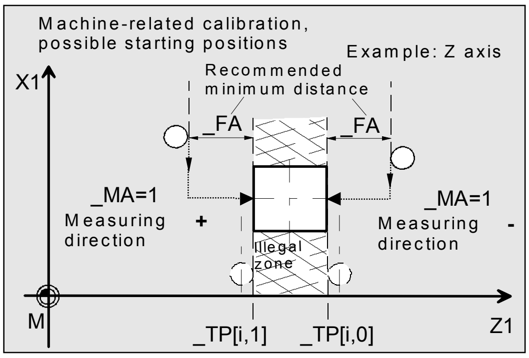
Position before measuring cycle call

The calibration tool must be prepositioned as shown in the figure.
The measuring cycle calculates the center of the probe and the approach paths automatically and generates the necessary traverse blocks.

Position after end of measuring cycle

On completion of calibration, the calibration tool is _FA from the measuring surface.





© Siemens AG, 2004. All rights reserved

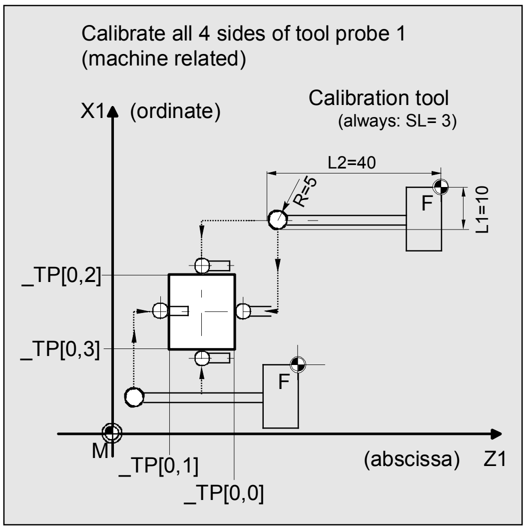
Programming example

Calibrating tool probes (machine-related)

Tool probe 1 is stationary but provides a switching signal. The calibration tool is inserted in the turret as tool T7.

Values of the calibration tool T7 D1:

Tool type (DP1):	500
Cutting edge position (DP2)	3
Length 1 - Geometry (DP3):	L1 = 10
Length 2 - Geometry (DP4):	L2 = 40
Radius - Geometry (DP6):	R = 5



This radius must be taken into account in selecting the starting position for calibration of _TP[0,1], _TP [0,3] (increase distance from probe by 2 · R).

Values of tool probe 1 in data block GUD6 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 measurement path of 1 mm, the measurement path is programmed with _FA = 1+5= 6 mm (max. total measurement path = 12 mm).

%_N_CALIBRATE_MTT_MPF	
N05 G94 G90 DIAMOF	
N10 T7 D1	;Calibration tool
N15 G0 SUPA Z300 X240	;Starting position in minus X direction, ;Traversal with ZO deactivated
N20 _TZL=0.001 _PRNUM=1 _VMS=0 _NMSP=1 **N21 _MVAR=0 _MA=2 _TSA=5 _FA=6**	;Parameters for calibration cycle
N30 CYCLE982	;Calibration in minus X direction
N35 G0 SUPA Z60	;Approach new starting position
N38 _MA=1	;Select another measuring axis
N40 CYCLE982	;Calibration in minus Z direction
N45 G0 SUPA X20	;Approach new starting position
N48 _MA=2	
N50 CYCLE982	;Calibration in plus X direction
N55 G0 SUPA Z0	;Approach new starting position
N58 _MA=1	
N60 CYCLE982	;Calibration in plus Z direction
N65 G0 SUPA X240	;Approach change position in each axis
N70 SUPA Z300	
N99 M2	

© Siemens AG, 2004. All rights reserved

6.2.3 Determining dimensions of calibration tool

Function

If no special calibration tool is available, a turning tool with cutting edge position SL=3 can be used instead for calibration of 2 sides of the probe (_TP[i,0], _TP[i,2]).
With the following procedure it is possible to determine the dimensions as the calibration tool.
Example: X axis, probe PRNUM=1 (_TP[0,2])

1. Approximate probe data in the data block GUD6: Parameters _TP[0,0]..._TP[0,3]
2. Measure the turning tool at the presetting location.
3. Enter all tool data in the tool offset (incl. e.g.: L1=60.000) and use the tool in the revolver.
4. Machine a test part (turn to X dimension), e.g.:
   Set diameter: 200.000 mm
   Actual diameter: 200.100 mm.
5. Adjust tool offset (L1=59.950).
6. Finish-turn the test part again, e.g.:
   Set diameter: 195.000 mm
   Actual diameter: 195.000 mm,
   The setpoint must be equal to the actual value, then:
7. Calibrate tool probe in the X axis (see example program in Subsection 6.2.2).
8. Measure tool (see following Section). The aim is to determine value L1=59.950 (see Item 5.).

Another tool can then be measured and used as the calibration tool. Calibrate the probe; the subsequent tool measurement must result in the same tool length.

© Siemens AG, 2004. All rights reserved

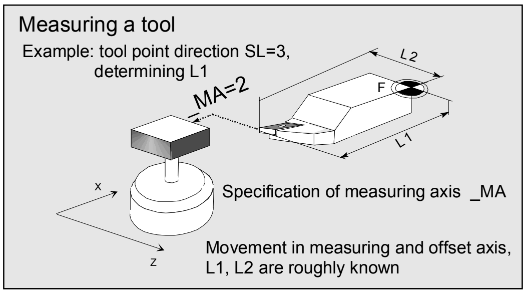
6.2.4 Measuring a turning tool (machine-related)

Function

The cycle determines the new tool length (L1 or L2) and checks whether the difference from the old tool length can be corrected within a defined tolerance range:

Upper limits: Safe area _TSA and
Dimensional difference check _TDIF,
Lower limit: Zero offset range _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.



Compensation strategy

The tool measuring cycle is provided for various applications:

- **First-time measurement of a tool** (_CHBIT[3]=0):
  The tool offset values in geometry and wear are replaced.
  The offset is applied in the geometry component of the length.
  The wear component is deleted.
- **Remeasurement of a tool** (_CHBIT[3]=1):
  The resulting difference is calculated into the wear component (length).

Empirical values may optionally be included. The mean value is not calculated.

Precondition

The tool probe must be calibrated.
The approximate tool dimensions must be entered in the tool offset data:
Tool type 5xy, cutting edge position, tool nose radius, length 1, length 2.
The tool to be measured must be active with its tool offset values when the cycle is called.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	1	Measure tool (machine-related)
_MA	1, 2	Measuring axis

The following additional parameters are also valid **_VMS, _TZL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM**, and **_NMSP**.

See Sections 2.2 and 2.3.

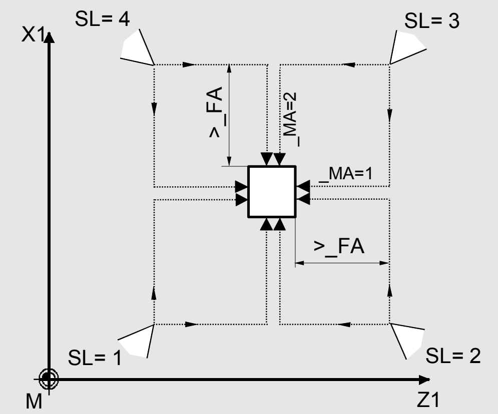
Sequence

Position before measuring cycle call
Before the cycle is called, the tool must be moved to the tool tip starting position, as shown in the figure.

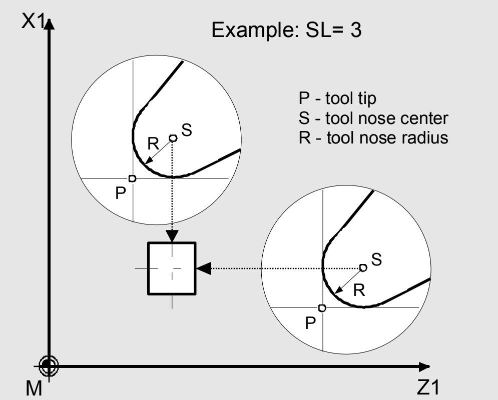
The measuring cycle calculates the center of the probe and the associated approach paths automatically. The necessary traverse blocks are generated.
The tool nose center (S) is positioned at the center of the probe.

Position after end of measuring cycle
On completion of the cycle, the tool nose is positioned facing the measuring surface and _FA from it.

Measuring turning tool with different tool tip directions and starting positions suitable for both axes.

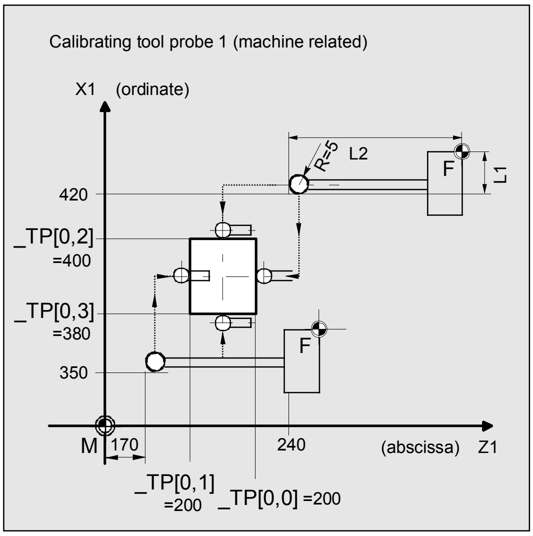


Length measurement turning tool:
offset around tool nose radius



Programming example

Calibrating the tool probe with subsequent measurement of turning tool (machine-related)



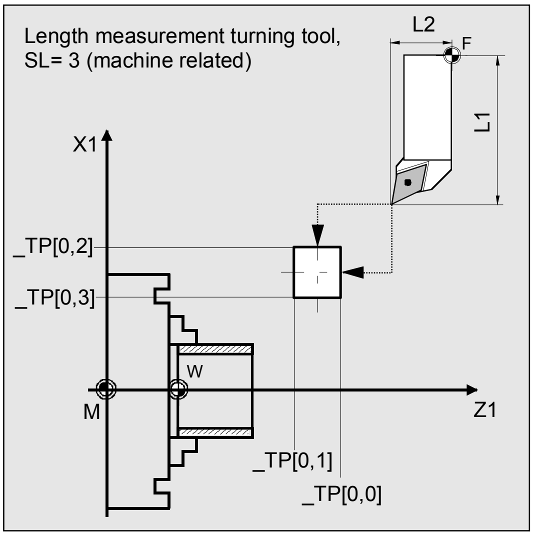
Calibration tool T7, D1 is to be used to calibrate all 4 sides of probe 1.
After that, **turning tool T3, D1** is to be remeasured in both lengths L1 and L2 (wear calculation).

The dimensions of the calibration tool T7 are in lengths L1, L2 and the radius **R=5.0 mm** are known precisely and entered in offset field D1.
The cutting edge position is SL=3.

The default values of probe 1 as applied in data block GUD6 with a tolerance of approx. 1 mm:

_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.



The lengths for the tool to be measured T3, D1 are known, remeasurement in wear:

Tool type (DP1):	500
Cutting edge position (DP2):	3
Length 1 – Geometry (DP3):	L1=100.654
Length 2 – Geometry (DP4):	L2=60.321
Radius (DP6):	R=2.000
Length 1 – Wear (DP12):	0
Length 2 – Wear (DP13):	0

© Siemens AG, 2004. All rights reserved

%_N_T3_MEAS_MPF	
;Calibration:	
N10 G0 G18 G94 G90 DIAMOF	
N20 T7 D1	;Call calibration tool
N30 SUPA Z240 X420	;Starting position for calibration
N40 _TZL=0.001 _PRNUM=1 _VMS=0 _NMSP=1 **N50 _MVAR=0 _FA=1 _TSA=1 _MA=2**	;Parameter definition
N60 CYCLE982	;Calibration in minus X direction
N70 G0 SUPA Z240	;New starting position
N80 _MA=1	;Set other measuring axis (Z)
N90 CYCLE982	;Calibration in minus Z direction
N100 G0 SUPA X350	;New starting position
N110_MA=2	;Set other measuring axis (X)
N120 CYCLE982	;Calibration in plus X direction
N130 G0 SUPA Z170	;New starting position
N140 _MA=1	;Set other measuring axis (Z)
N150 CYCLE982	;Calibration in plus Z direction
N160 G0 SUPA X350	;Go to tool change position in each axis
N170 SUPA Z520	;Traverse
N180 SUPA X420	
;Measurement:	
N200 T3 D1	;Selection of the tool to be measured
N210 G0 SUPA Z240 X420	;Starting position for measurement
N220 _MVAR=1 _MA=2 _TDIF=0.8	;Change of parameter definition for ;measurement, otherwise calibration
N230 _CHBIT[3]=1	;Offset in wear (remeasuring)
N240 CYCLE982	;Tool measurement in minus X direction (L1)
N250 G0 SUPA Z240	;New starting position
N260 _MA=1	;Set other measuring axis (Z)
N270 CYCLE982	;Tool measurement in minus Z direction (L2)
N280 G0 SUPA X420	;Retraction axis by axis
N290 SUPA Z520	
N300 M2	

Explanation

N10 to N180, calibrate:
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).

© Siemens AG, 2004. All rights reserved

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.

Explanation

N200 to N300, measure:
The probe is completely calibrated.
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 "tip" 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=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.

© Siemens AG, 2004. All rights reserved

Recommended parameters

The following parameters are recommended so that this programming example runs reliably:

- **Calibration:**
  - _TZL = 0.001 Zero offset range
  - _TSA = 1 Safe area
  - _FA=1 Measurement path
- **First-time measurement of a tool:**
  - _TZL = 0.001 Zero offset range
  - _TDIF=3 Dimension difference check
  - _TSA=3 Safe area
  - _FA=3 Measurement path
- **Remeasure the tool:**
  - _TZL = 0.001 Zero offset range
  - _TDIF=0.3 Dimensional difference check
  - _TSA=1 Safe area
  - _FA=1 Measurement path

© Siemens AG, 2004. All rights reserved

6.3 CYCLE982 tool: Measure turning and milling tools

6.3.1 Function overview

Programming

CYCLE982

Function

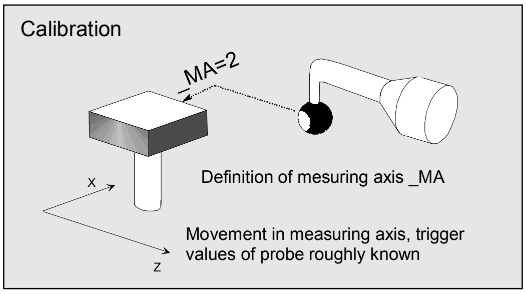
Cycle CYCLE982 permits

- calibration of a tool probe,
- measurement of tool lengths L1 and L2 for turning tools with cutting edge positions 1 to 8,
- the tool lengths for milling tools and drills on turning machines,
- the radius for milling tools.

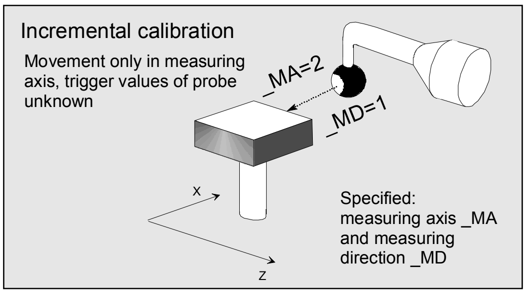
Measurement of milling cutters/drills requires NCK software version SW 5 or higher.

The following measurement and calibration tasks are supported by CYCLE982:

- **Calibration** as the preparation for measurement / automatic measurement
  The 4 switching positions of the probe are roughly known and entered in the array of the associated tool probe. 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.

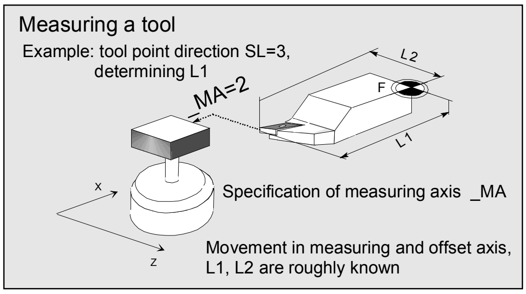


- **Incremental calibration** as preparation for incremental measurement
  The switching positions of the probe are not known. The calibration tool must have been positioned in front of the probe manually (in JOG mode) before the cycle is called.
  It is only possible to determine the switching position 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.

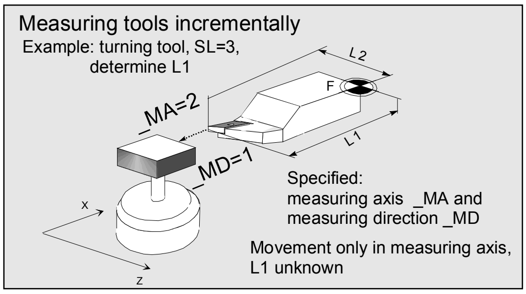


© Siemens AG, 2004. All rights reserved

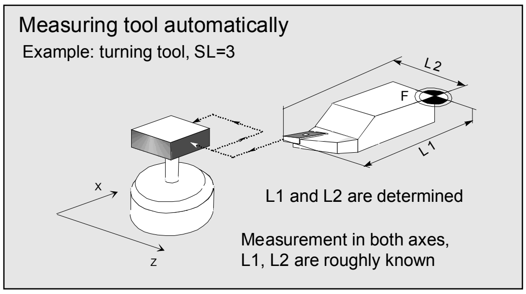
- **Measuring**
  Only measured values that are in the measurement axis _MA can be calculated.
  The geometry of the tool to be measured is roughly known and entered in the tool offset.
  Positioning of the tool with respect to the calibrated probe is performed in the cycle. The geometry must be determined precisely or wear (initial measurement or remeasurement of a tool).



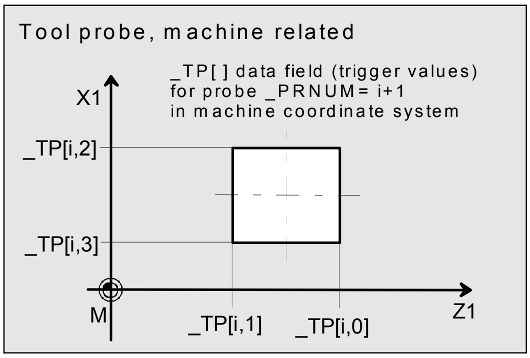
- **Incremental measurement**
  The geometry of the tool to be measured is not known. The tool must have been positioned in front of the probe manually before the cycle is called.
  The geometry is to be determined exactly. Only one measured value that is in the measurement axis _MA can be calculated. The cycle approaches the probe in the measuring axis in the specified measuring direction _MD.



- **Automatic measurement**
  All values that can be determined are determined automatically according to the active tool type.
  The geometry of the tool to be measured is roughly known and entered in the tool offset.
  Positioning of the tool with respect to the calibrated probe is performed in the cycle. The geometry must be determined precisely or wear (initial measurement or remeasurement of a tool).

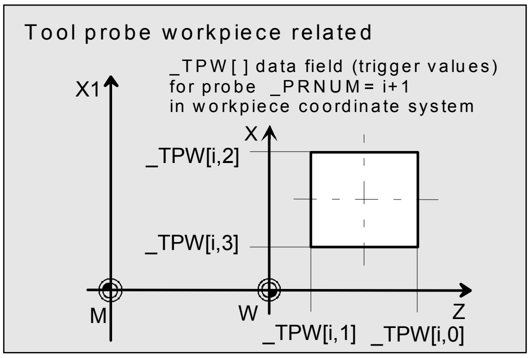


- **Machine-related measurement, calibration**
  The switching positions of the tool probe refer to the machine zero.
  The array for the tool probe _PRNUM is used:
  _TP[PRNUM-1,...].

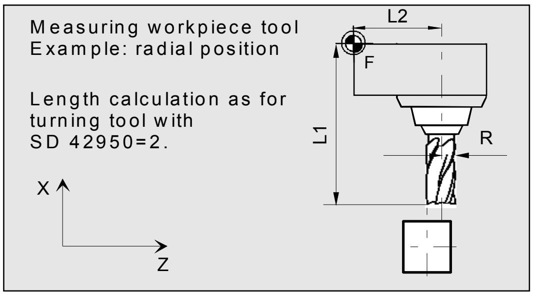


© Siemens AG, 2004. All rights reserved

- **Machine-related measurement, calibration**
  The switching positions of the tool probe refer to the workpiece zero. The array for the tool probe _PRNUM is used: **_TPW**[PRNUM-1,...].



Special aspects with milling tools
The tool length correction is specific to the turning machine (SD 42950: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 _CHBIT[20]=1). 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 calculating the mean value over both measured values.



Not all functions are available. Certain functions require a certain SW software version of the measuring cycles and NCK. This information is given with each function.

Note for turning machines with a Y axis
Before CYCLE982 is called, the Y axis (applicate in G18) must be put in a position corresponding to the center of the probing surface of the tool probe in this axis.
The Y axis is not positioned in the cycle itself.

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE982 permits the following measuring variants that are specified with parameter _MVAR.

Digit						*Meaning*
6	**5**	**4**	**3**	**2**	**1**	
					0	**Calibrate tool probe** with calibration tool
					1	**Measure turning and milling tool/drill**, Measurement axis in _MA (is specified for • Turning tools: Cutting edge position 1...8, • Milling tools: Points 3 to 5 in _MVAR)
					2	**Automatic measurement** (determine both lengths, for milling cutter, the radius tool.) The following is specified: • Turning tools: Of edge positions 1...8, • Milling tools: Points 3 to 5 in _MVAR)
				0		Relative to machine
				1		Relative to the workpiece (from meas. cycles SW 6.3)
						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 0			1 2	Correct length only (for measuring only) or Measure milling cutter automatically
		1			1	Correct radius only (for measuring only)
		2			1	Correct length and radius (only for measurement, not for incremental measurement)
		3			2	Measure upper tool edge automatically: Correct length and radius, travel round measuring cube opposite starting position side (for automatic measurement only, e.g. groove mill)
		4			2	Measure upper tool edge automatically: 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, e.g. groove mill)
						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 1					0 1	**Incremental calibration** or **Incremental measurement** (limited variants, no automatic measurement)

© Siemens AG, 2004. All rights reserved

- The following measuring variants are not possible for incremental measurement:
  1xxxx2; 102xx1; 112xx1
- The following measuring variants are permitted if _CHBIT[20]=1 (suppression of the starting angle position with _STA1) on a milling tool:
  xxx0x1 (with x: 0 or 1, no other values)
- A measuring variant can also be impermissible if it cannot be performed with the specified measuring axis _MA, e.g. determining the milling cutter radius. However, with this position of the milling cutter it is not in the measuring axis.

Result parameters

The measuring cycle CYCLE982 returns the following values in the data blockGUD5 for the measuring 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	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [9]	INTEGER	Alarm number

The measuring cycle CYCLE982 provides the following values in data block GUD5 as results in **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 dimensional deviation

© Siemens AG, 2004. All rights reserved

_OVR [30]	REAL	Empirical value
_OVI [0]	INTEGER	D number
_OVI [2]	INTEGER	Measuring cycle number
_OVI [3]	INTEGER	Measuring variant
_OVI [5]	INTEGER	Probe number
_OVI [7]	INTEGER	Empirical value memory
_OVI [8]	INTEGER	T number
_OVI [9]	INTEGER	Alarm number

Note on tool types

During measurement or calibration, the tool type (tool parameter DP1 in the tool offset data) of the active tool is evaluated.

- Type 5xy: Turning tool or calibrating tool
- Type 1xy: Milling tool
- Type 2xy: Drill

Use of tool types 711 to 799 is also possible. These are treated as a milling tool (type 1xy).

From measuring cycles SW 6.3, drill (Typ 2xy) also with SD 42950: TOOL_LENGTH_TYPE=0 can be gauged (refer to Subsection 6.3.8). Otherwise this is only possible for drills and milling tools with SD 42950: TOOL_LENGTH_TYPE=2.

© Siemens AG, 2004. All rights reserved

6.3.2 Calibrating a tool probe

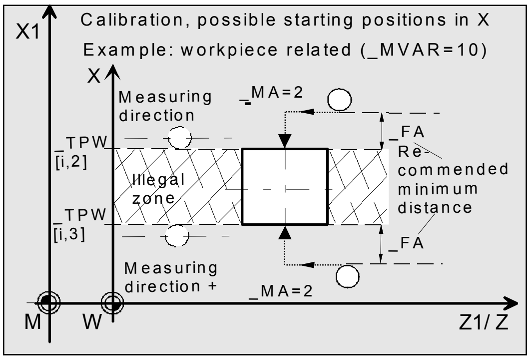
Function

Calibrating tool probes - machine-related

Measuring variant **_MVAR=0** permits calibration of a tool probe relative to the calibration tool relative to the machine.
This variant is already described in detail in Subsection 6.2.2.

Calibrating a tool probe relative to the workpiece (from measuring cycles SW 6.3)

Measuring variant **_MVAR=10** permits calibration of a tool probe relative to the calibration tool relative to the workpiece.
The switching positions of the tool probe refer to the workpiece zero. The array for the tool probe _PRNUM is used: _TPW[PRNUM-1,...].



Transformations can be activated in workpiece-related measurement, calibration.

The requirements and procedures are as for machine-related calibration (see Subsection 6.2.2).

Parameters

_MVAR	0 10	Calibrate tool probe (machine-related) (see Subsection 6.2.2) Calibrate tool probe (workpiece-related) (> meas. cycle SW 6.3)
_MA	1, 2	Measuring axis
_PRNUM	INT	Probe number

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

© Siemens AG, 2004. All rights reserved

Programming example

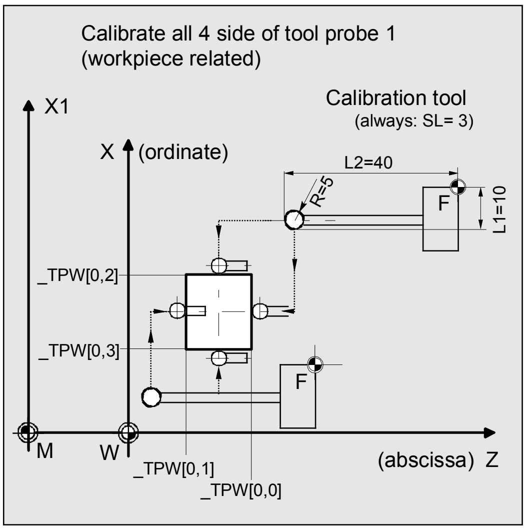
Calibrating a tool probe (relative to workpiece)

Tool probe 1 is in the machining area and is oriented parallel to the axis of the workpiece coordinate system.

The calibration tool is inserted in the turret as tool T7.

Values of the calibration tool T7 D1:

Tool type (DP1):	500
Cutting edge position (DP2)	3
Length 1 - Geometry (DP3):	L1 = 10
Length 2 - Geometry (DP4):	L2 = 40
Radius - Geometry (DP6):	R = 5



This radius must be taken into account in selecting the starting position for calibration of _TPW[0,1], _TPW[0,3] (increase distance from probe by 2 · R).

Values of the settable ZO G54:
Displacement: X=0, Z=60.000 mm, no rotation

Values of tool probe 1 in data block GUD6 which were determined manually to 5 mm accuracy beforehand (relative to the workpiece zero):

_TPW[0,0] = 50
_TPW[0,1] = 20
_TPW[0,2] = 70
_TPW[0,3] = 40

To obtain a minimum measurement path of 1 mm, the measurement path is programmed with
_FA = 1+5= 6 mm
(max. total measurement path = 12 mm).

%_N_CALIBRATE_MTT_WCS_MPF	
N05 G54 G94 G90 DIAMOF	
N10 T7 D1	;Calibration tool
N15 G0 Z100 X120	;Starting position in minus X direction, ;Traversal with ZO deactivated
N20 _TZL=0.001 _PRNUM=1 _VMS=0 _NMSP=1 **N21 _MVAR=10 _MA=2 _TSA=5 _FA=6**	;Parameters for calibration cycle
N30 CYCLE982	;Calibration in minus X direction
N35 G0 Z80	;Approach new starting position
N38 _MA=1	;Select another measuring axis
N40 CYCLE982	;Calibration in minus Z direction
N45 G0 X10	;Approach new starting position
N48 _MA=2	

© Siemens AG, 2004. All rights reserved

N50 CYCLE982	;Calibration in plus X direction
N55 G0 Z-10	;Approach new starting position
N58 _MA=1	
N60 CYCLE982	;Calibration in plus Z direction
N65 G0 X10	;Approach start position in each axis
N70 Z100	
N80 X120	
N100 M2	;End of program

Explanation

The calibration tool moves out of the starting position of N5 (X120, Z100) in Z to the center of the probe with its "tool tip". An offset is applied to compensate for the calibration tool radius. This places the radius center point in the center of the probe. The tool tip position is shown: Z30 ((_TPW[0,0] + (_TPW[0,1]) / 2 -R = (50+20) / 2 -5=30)). This is followed by traversal in measuring axis X (_MA=2, G18) to position X76 (_TPW[0,2] + _FA = 70 + 6 = 76). This is where actual calibration (like measurement) starts in the minus X direction. At the end the calibration tool is again at position X76.

The new trigger values in minus X are stored in the data of tool probe 1 (_PRNUM=1) _TP[0,2] if they deviate by more than 0.001 mm (_TZL=0.001) from the old values. Deviations of up to 5 mm (_TSA=5) are permissible.
After that, the sides in the minus Z direction, plus X direction, and plus Z direction are approached, calibrated, and the values entered in array _TPW[0,...].

© Siemens AG, 2004. All rights reserved

6.3.3 Measuring the tool

Function

This cycle and its various measuring variants are for measuring:

_MVAR=1: Turning tools (machine-related)
This variant is described in detail in Subsection 6.2.4.
_MVAR=11: Turning tools (workpiece-related)
_MVAR=xxx01: Milling tools, drills (machine-related)
_MVAR=xxx11: Milling tools, drills (workpiece-related)
Workpiece-related or machine-related measurement require an appropriately calibrated tool probe (see Subsection 6.3.2 or 6.2.2).

These measuring variants can only determine offset values that are in the measurement axis _MA.

The cycle determines the new tool length (L1 or L2), for milling tools the radius too, and checks whether the difference from the old tool length can be corrected within a defined tolerance range:

Upper limits: Safe area _TSA and
Dimensional difference check _TDIF,
Lower limit: Zero offset range _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.

Compensation strategy

The tool measuring cycle is provided for various applications:

- **Initial measurement of a tool** (_CHBIT[3]=0):
  The tool offset values in geometry and wear are replaced.
  The offset is applied in the geometry component of the length.
  The wear component is deleted.
- **Re-measurement of a tool** (_CHBIT[3]=1):
  The resulting difference is calculated into the wear component (radius or length).

Empirical values may optionally be included. The mean value is not calculated.

© Siemens AG, 2004. All rights reserved

If _CHBIT[20]=1, positioning of the milling spindle at the value of _STA1 can be suppressed.
That is possible with the following milling cutter measuring variants:
_MVAR=xxx001 (with x: 0 or 1, no other values).

Precondition

The tool probe must be calibrated.
The approximate tool dimensions must be entered in the tool offset data:
Tool type, cutting edge position on turning tools, radius, length 1, length 2.
The tool to be measured must be active with its tool offset values when the cycle is called.

For a **milling cutter**, the 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.

The **drill** is also SD 42950: TOOL_LENGTH_TYPE = 0 possible (see Subsection 6.3.8).

Parameters

_MVAR	1 or xxx01 11 or xxx11	Measure tool (machine-related) Measure tool (workpiece-related) (> meas. cycle SW 6.3) More precise parameterization for milling tools is entered in the 3rd to 5th digits of _MVAR.
_MA	1, 2	Measuring axis
_STA1	REAL	For milling tools: Start angle
_CORA	REAL	For milling tools: Offset angle position after reversal (for measurement with reversal only _MVAR=xx1x1)

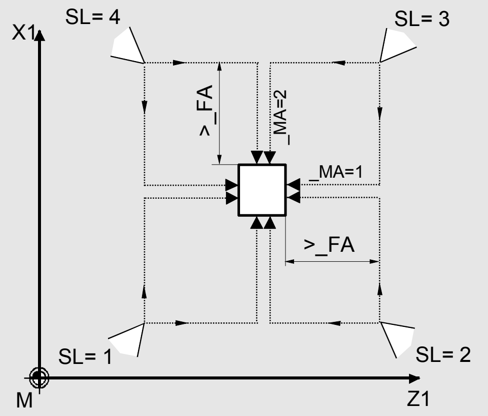
The following additional parameters are also valid
_VMS, _TZL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, and **_NMSP**.
See Sections 2.2 and 2.3.

© Siemens AG, 2004. All rights reserved

Sequence

Position before measuring cycle call
Before the cycle is called, the tool must be moved to the starting position, as shown in the diagram for turning tools. The measuring cycle then calculates the approach position automatically. This position determines the direction of measurement in the measuring axis _MA.
For milling tools, the measuring point on the tool is determined by entered lengths 1 and 2 (please note: SD 42950: TOOL_LENGTH_TYPE).
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). This starting position must ensure collision-free approach.

Measuring turning tool with different tool tip directions and starting positions suitable for both axes.



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.

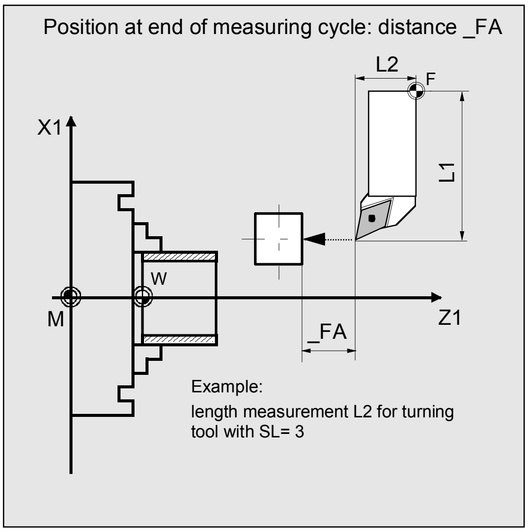
© Siemens AG, 2004. All rights reserved

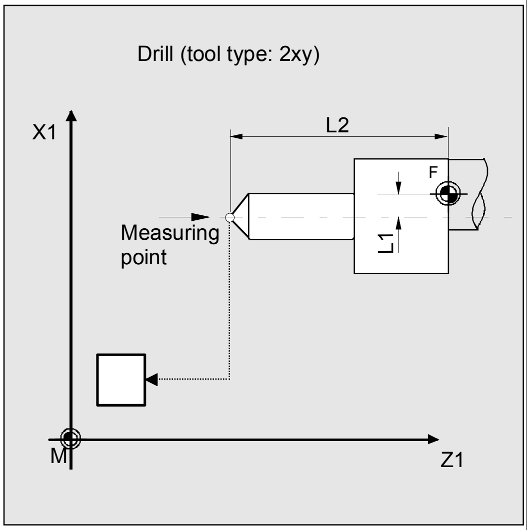
Measurement with reversal causes a second measurement at each measuring point P with a spindle rotation through 180 degrees from the starting angle. The offset angle entered in _CORA is summated to these 180 degrees. That enables selection of a certain 2nd milling cutting edge that is offset from the 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 measuring variants are possible for a milling cutter without taking the starting angle _STA1 into account (see Subsection 6.3.7).

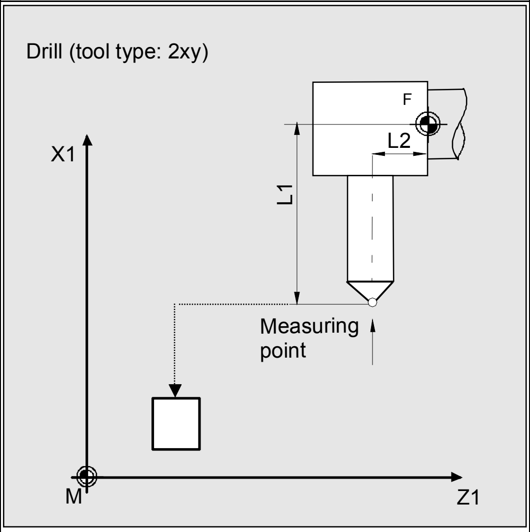
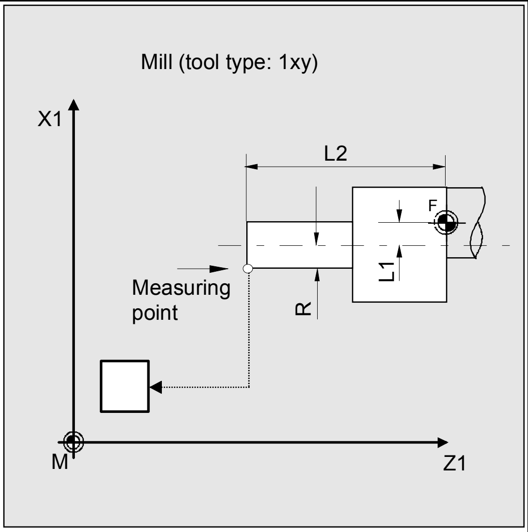
Note on measuring with rotating spindle
If selection of a certain miller cutting edge is not possible, it is possible to measure with a rotating spindle. The user must then program the direction of rotation, speed, and feedrate very carefully before calling up CYCLE982 to prevent damage to the probe. A low speed and feedrate must be selected.

Position after end of measuring cycle
When the cycle is complete, the tool nose is _FA from the measuring surface and facing it.

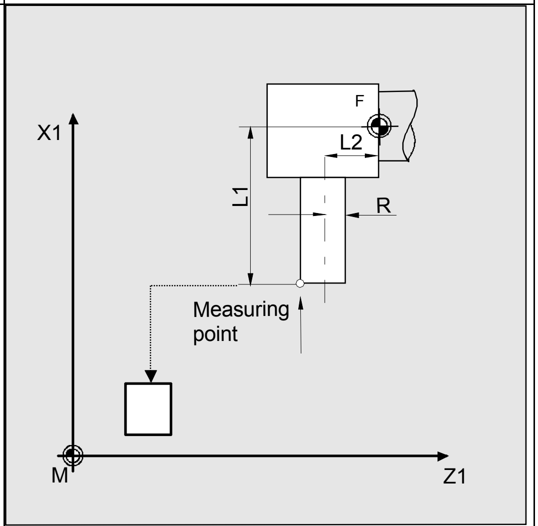
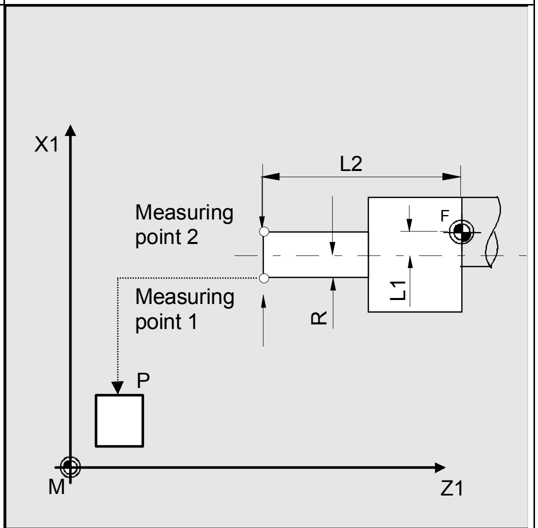
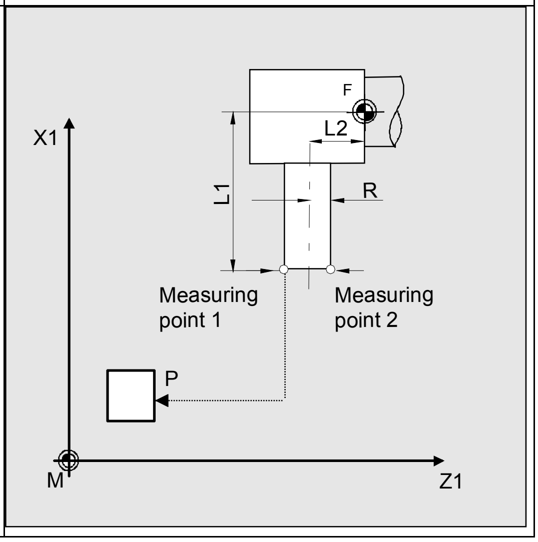


Measuring variant	Given geometry	Offset applied in	Milling tools, drills
Example: **Axial position,** R=0, Measuring without reversal, calculate length only **_MVAR=1** **_MA=1**	L1=... L2=... R=0	L2	

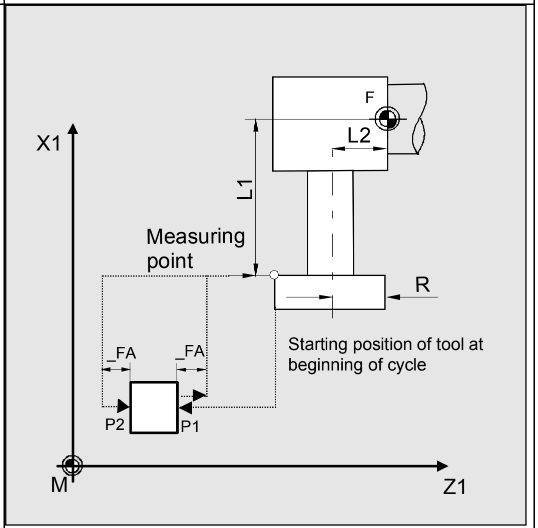
© Siemens AG, 2004. All rights reserved

Measuring variant	Given geometry	Offset applied 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=... L2=... R=...	L2	

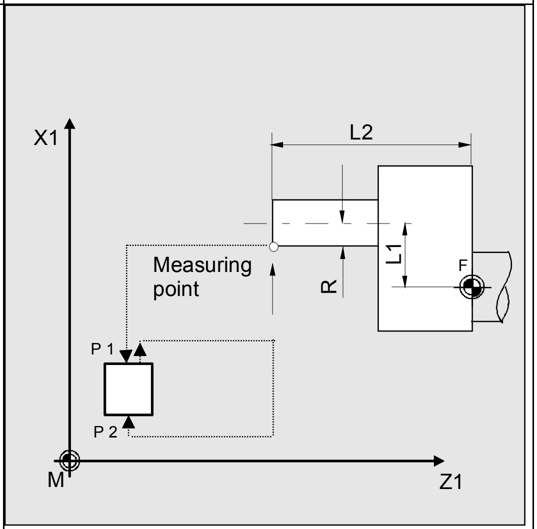
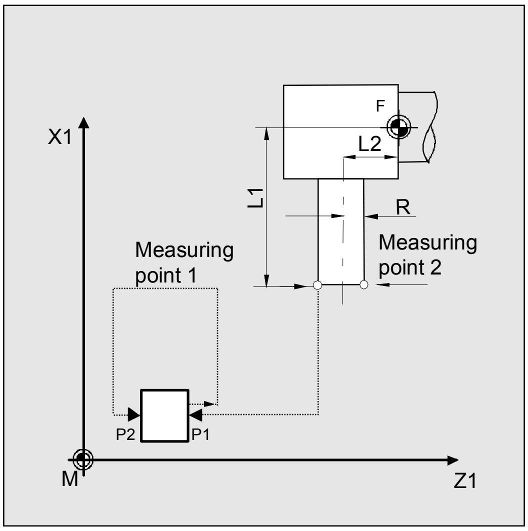
© Siemens AG, 2004. All rights reserved

Measuring variant	Given geometry	Offset applied in	Milling tools, drills
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	L1=... L2=... R=...	R R=ABS(P – 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=(P - R) Or other measuring direction: L2=(P + R)	

© Siemens AG, 2004. All rights reserved

Measuring variant	Given geometry	Offset applied in	Milling tools, drills
Example: **Radial position,** R ≠ 0, Measuring without reversal, calculate length and radius, 2 measuring points necessary **_MVAR=12001** **_MA=1** Notes: On starting, the measuring point must be outside the measurement cube coordinates in both coordinates. On the opposite side of the measuring cube (P2) 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=(P1 + P2)/2 R= ABS(P1-P2)/2	

© Siemens AG, 2004. All rights reserved

Measuring variant	Given geometry	Offset applied in	Milling tools, drills
Example: **Axial position,** R ≠ 0, Measuring without reversal, calculate length and radius, 2 measuring points necessary **_MVAR=2001** **_MA=2** P2 is measured with a rotated spindle (by 180 degrees), if measurement is performed with a stationary spindle.	L1=-.. L2=... R=...	L1 R L1=(P1 + P2)/2 R=ABS(P1-P2)/2	
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**	L1=... L2=... R=...	L2 R L2=(P1 + P2)/2 R=ABS(P1-P2)/2	

© Siemens AG, 2004. All rights reserved

Programming example

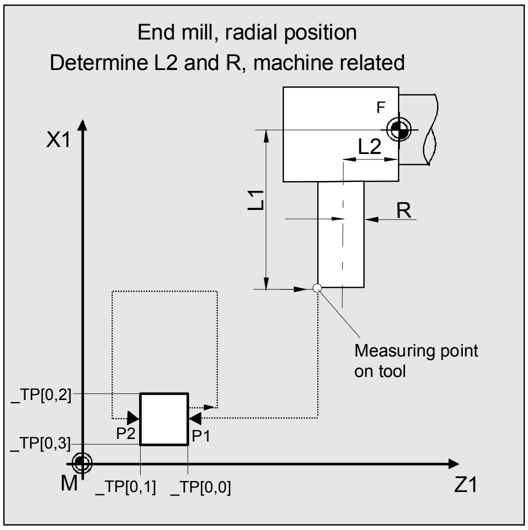
Measure milling tool in the radial position (machine-related)

For the end miller T3, D1 should be determined in the radial position when first measuring length L2 and radius R. Measurements will be without reversal. The cutting edge to be measured is in the milling spindle position 15 degrees.

The lengths and radius of tool T3 to be measured are roughly known and entered in offset field D1:

Tool type (DP1):	120
Cutting edge position (DP2)	
Length 1 - Geometry (DP3):	L1 = 60
Length 2 - Geometry (DP4):	L2 = 10
Radius - Geometry (DP6):	R = 14

Deviations from this value of less then 2.5 mm are expected.



The probe to be used is tool probe 1. This probe has already been completely calibrated (machine-related). The precise values are entered in array _TP[0,...] and are approximately:

_TP[0,0]=220, _TP[0,1]=200
_TP[0,2]=400, _TP[0,3]=380.

%_N_T3_MEAS_FR_MPF	
N1 G0 G18 G90 G94 DIAMOF	
N100 T3 D1	;Selection of the tool to be measured
N110 G0 SUPA Z285 X450	;Starting position for measurement ;Traversal without ZO
N120 _TZL=0.001 _TSA=3 _FA=3 _PRNUM=1 _VMS=0 _NMSP=1 **N121 _MA=1 _TDIF=2.5 _MVAR=12001 _STA1=15**	;Change of parameter definition for ;measurement, otherwise calibration
N130 _CHBIT[3]=0	;Offset in the geometry
N131 _CHBIT[20]=0	;Do not suppress _STA1
N140 CYCLE982	;Tool measurement L2, R
N180 G0 SUPA X450	;Retraction axis by axis
N190 SUPA Z285	
N200 M2	;End of program

© Siemens AG, 2004. All rights reserved

Explanation

The spindle is positioned at 15 degrees with SPOS.
Measuring point P1 is approached first.
The measuring process is initiated in the negative Z direction (_MA=1, starting position) 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=2 mm.
Otherwise, an alarm will be triggered.
Measurement is performed once (_NMSP=1).
After successful measurement, tool T3 is _FA=3 mm + tool radius in front of the probe.

The probe is then traveled around as shown in the figure.
On the opposite side of the probe (P2) measurement is performed with a rotated spindle (by 180 degrees).
The same cutting edge is then measured.
Measurement is performed with the spindle stopped and without reversal.
After successful measurement, tool T3 is _FA=3 mm + tool radius in front of the probe. The spindle remains in this position.

The radius and length L2 are determined precisely and tool parameters DP6 and DP4 of T3, D1 are entered. The values in result parameter array OVR[ ] are also entered.

In block N180, N190, the tool returns to the starting position and the program then ends.

© Siemens AG, 2004. All rights reserved

6.3.4 Automatic tool measurement

Function

This cycle and its various measuring variants are for measuring tools **automatically**:

_MVAR=2: Turning tools (machine-related)
_MVAR=12: Turning tools (workpiece-related)
_MVAR=xxx02: Milling tools, drills (machine-related)
_MVAR=xxx12: Milling tools, drills (workpiece-related)

Workpiece-related or machine-related measurement require a corresponding calibrated tool probe (see Subsection 6.3.2 or 6.2.2).
With milling tools/drills, the measurement is further specified with the 3rd to 5th decimal places of parameter _MVAR. Here, SD 42950: TOOL_LENGTH_TYPE must be = 2

The function is as for non-automatic measurement.
In automatic measurement, all offsets are determined.
These are then defined with the tool type:

- Turning tool: Both lengths (2 measurements), for cutting edge positions SL = 5, 6, 7, and 8, only one length.
- Drill: Length according to axial or radial position (1 measurement)
- Milling tool: Both lengths and radius (4 meas.), if the radius is R=0, both lengths only are calculated (2 measurements).

The calculated offsets are entered in the active D number of the active tool.
The offset strategy is defined via _CHBIT[3] as for measurement.

The measuring cycle generates the approach blocks to the probe and the traverse movements for measuring length 1. Length 2 and for the milling cutter, the radius itself.
The condition is a correctly selected starting position.

Precondition

As for non-automatic tool measurement

© Siemens AG, 2004. All rights reserved

Parameters

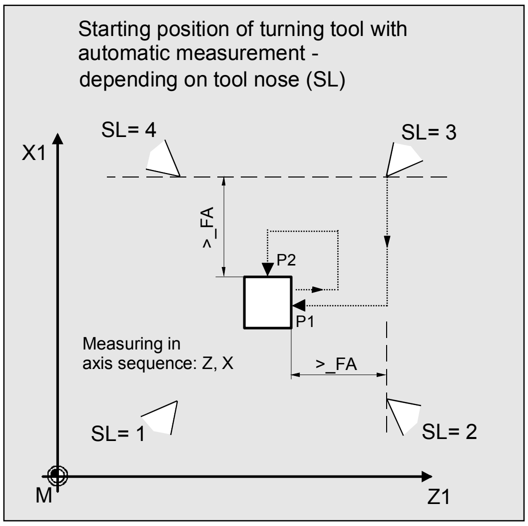
_MVAR	2 or xxx02 12 or xxx12	Measure tool automatically (machine-related) Measure tool automatically (workpiece-related) (> meas. cycle SW 6.3) More precise parameterization for milling tools is entered in the 3rd to 5th digits of _MVAR.
_MA	1, 2	Measuring axis
_STA1	REAL	For milling tools: Start angle
_CORA	REAL	For milling tools: Offset angle position after reversal (for measurement with reversal only _MVAR=xx1x1)

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

Sequence

Position before measuring cycle call

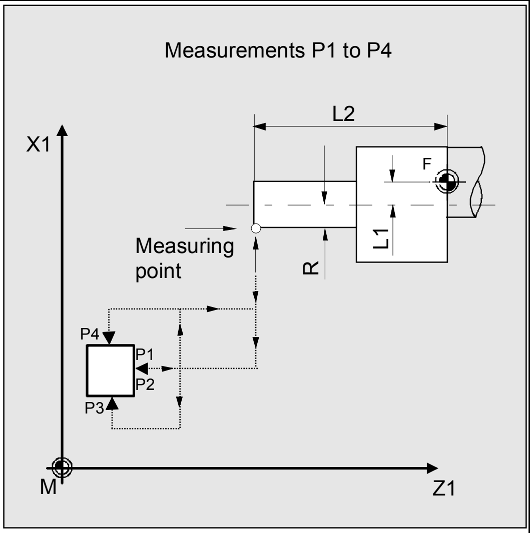
Before the cycle is called, the tool must be moved to the starting position, as shown in the diagram for turning tools. The measuring cycle then calculates the approach position automatically.
First the length (P1) in the abscissa (Z axis for G18) and then (P2) in the ordinate (X axis for G18) is measured. For turning tools, the measuring probe travels round the measuring cube at distance _FA.



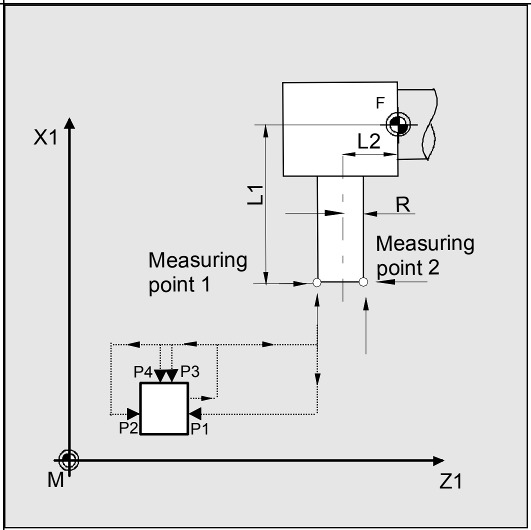
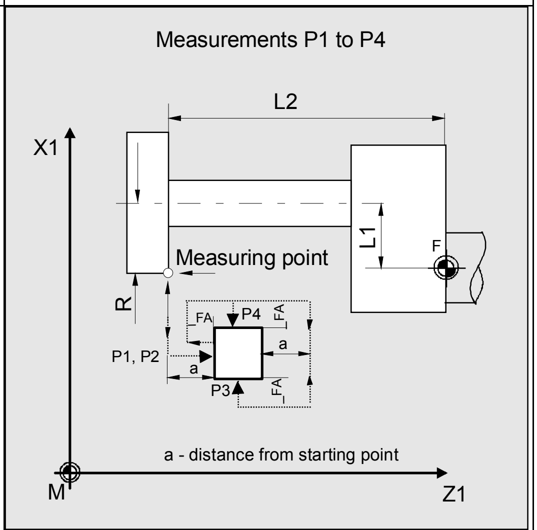
For milling tools, the measuring points on the tool are determined by entered lengths 1 and 2 (please note: SD 42950). If the radius value is not equal to zero, this is also a determining factor. The axial or radial position of the tool must be specified in _MVAR, and the starting position approached 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 or according to the starting point coordinate (see figs.).

© Siemens AG, 2004. All rights reserved

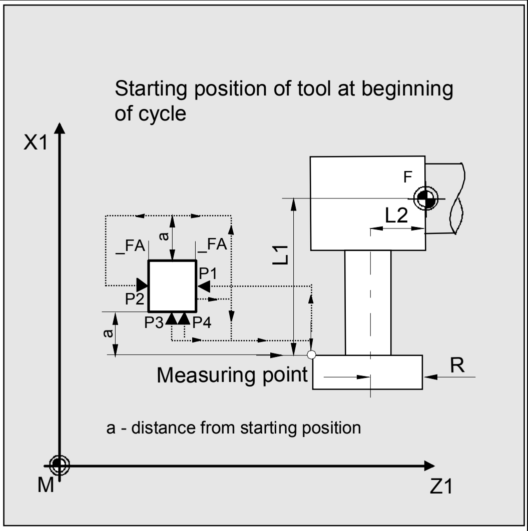
Position after end of measuring cycle
When the cycle is complete, the tool nose is again located at the starting point. A movement to this point is automatically generated in the cycle.

Measuring variant	Given geometry	Override applied in	Milling tools
Example: **Axial position,** R ≠ 0, Measuring without reversal, spindle stationary, 4 measurements necessary **_MVAR=2** **Sequence:** P1 is approached with the starting 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 P3 is approached and measured, after that the spindle is again rotated by 180 degrees and P4 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=... L2=... R=...	L1 L2 R L1=(P3x + P4x)/2 L2=(P1z + P2z)/2 R= ABS(P3x-P4x)/2	

© Siemens AG, 2004. All rights reserved

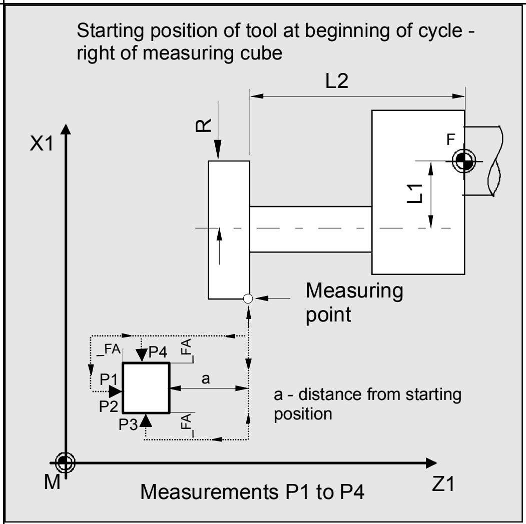
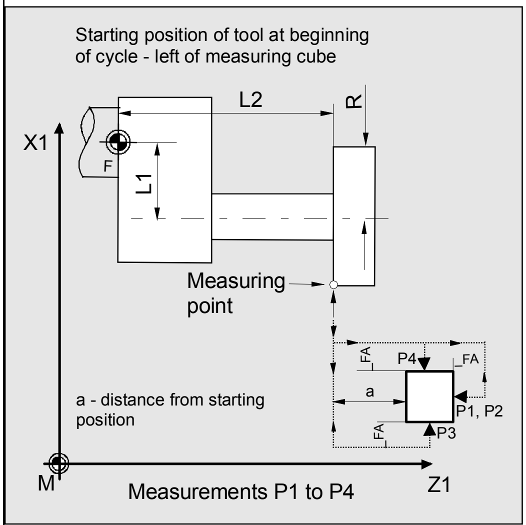
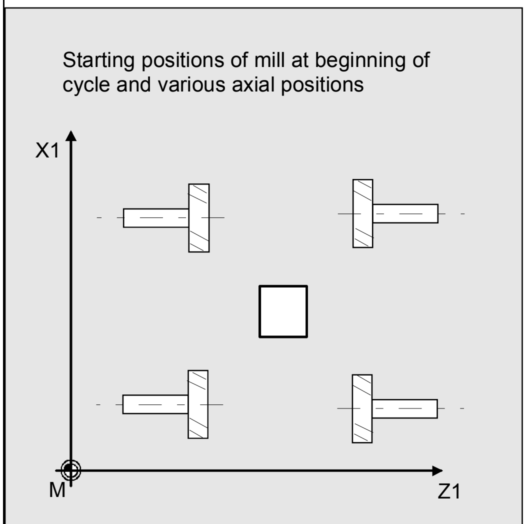
Measuring variant	Given geometry	Override applied in	Milling tools
Example: **Radial position,** R ≠ 0 measuring with reversal, 8 measurements necessary (P1 to P4 each with reversal) **_MVAR=10102**	L1=... L2=... R=...	L1 L2 R L1=(P3x + P4x)/2 L2=(P1z + P2z)/2 R=ABS(P1z-P2z)/2	
Example: **Axial position,** R ≠ 0, Measuring without reversal, 4 measurements necessary **_MVAR=3002** The probe travels around the measuring cube opposite the starting position side. Note: Length measurements for L2 (P1, P2) 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=(P3x + P4x)/2 L2=(P1z + P2z)/2 R=ABS(P3x-P4x)/2	

© Siemens AG, 2004. All rights reserved

Measuring variant	Given geometry	Override applied in	Milling tools
Example: **Radial position,** R ≠ 0 Measuring without reversal, 4 measurements necessary **_MVAR=13002** The probe travels around the measuring cube opposite the starting position side. Note: Length measurements for L1 (P3, P4) 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=(P3x + P4x)/2 L2=(P1z + P2z)/2 R= ABS(P1z-P2z)/2	

For measuring variants _MVAR=0x3xx2 and _MVAR=0x4xx2 (measure upper cutting edge automatically), the cutting mill must have suitable geometric dimension (end mill/cutter radius) for approaching the center of the probe with the cutting edge without collision.
No check is made for suitability of the milling cutter for this measuring variant. It is up to the user to ensure this.

© Siemens AG, 2004. All rights reserved

Measuring variant	Given geometry	Override applied in	Milling tools
Example: **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 (P1, P2) 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 starting position or dimension a!	L1=... L2=... R=...	L1 L2 R L1=(P3x + P4x)/2 L2=(P1z + P2z)/2 R= ABS(P3x-P4x)/2	  

© Siemens AG, 2004. All rights reserved

<table>
<tr><th>Measuring variant</th><th>Given geometry</th><th>Override applied in</th><th>Milling tools</th></tr>
<tr>
<td>Example:
Radial position,
R ≠ 0,
Measuring without reversal,
4 measurements necessary

_MVAR=14002

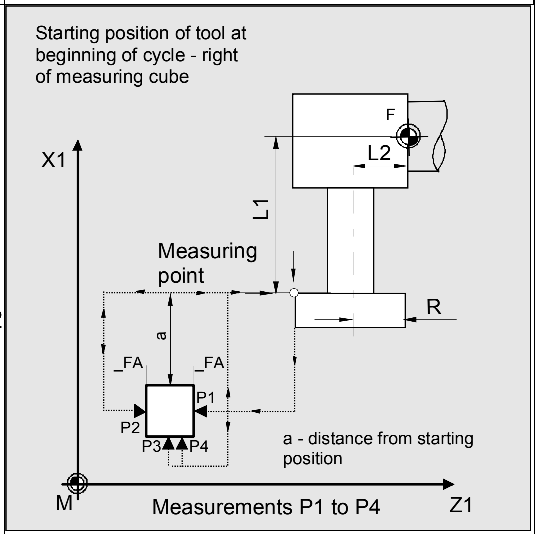
Direction of measurement for determining length L1 opposite to traversing direction, measuring procedure as for _MVAR=13002 but with different traversing motion

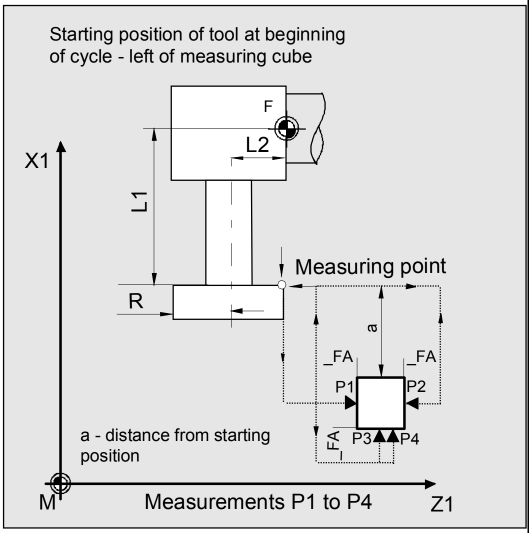
Notes:
Length measurements for L1 (P3, P4) are performed here at the same measuring point – without rotating the spindle by 180 degrees. The same cutting edge is always measured (starting angle _STA1).

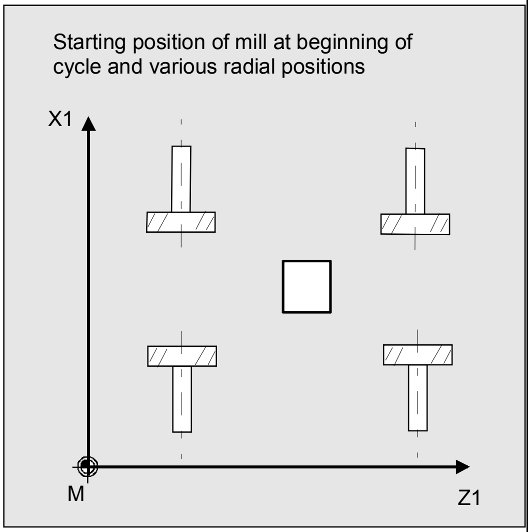
The width of the milling tool must be considered when selecting the starting position or dimension a!</td>
<td>L1=...
L2=...
R=...</td>
<td>L1
L2
R

L1=(P3x + P4x)/2

L2=(P1z + P2z)/2

R= ABS(P3z-P4z)/2</td>
<td>






</td>
</tr>
</table>

6.3.5 Incremental calibration

Function

A tool probe can be calibrated with measuring variant
_MVAR=100000 (machine related) or
_MVAR=100010 (workpiece-related)
incrementally with a calibration tool.

The switching positions of the probe are not known. The values entered in the array of the probe are not evaluated. The calibration tool must have been positioned in front of the probe manually (in JOG mode) before the cycle is called.
The cycle uses the calibration tool to ascertain the current distance dimensions between the zero and the probe trigger point and automatically loads them into the appropriate data area in data block GUD6 (field _TP [ ] or _TPW[ ]).
Values are corrected without empirical and mean values.

Precondition

The lateral surfaces of the tool probe must be aligned parallel to the relevant axes (machine or workpiece coordinate system in abscissa and ordinate).

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 cutting edge position SL 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.
Careful when positioning manually!
Damage to the probe must be prevented.

Parameters

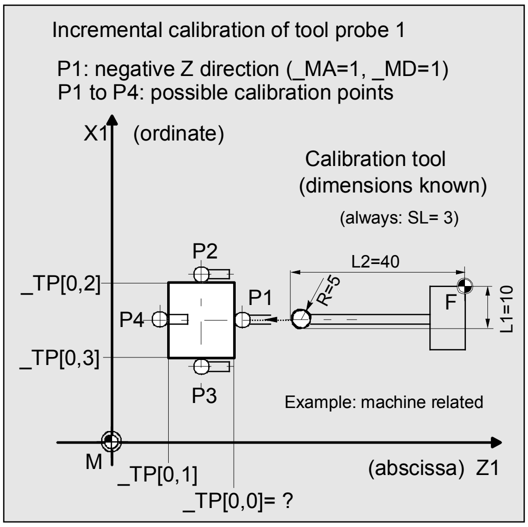
_MVAR	100000	Calibrate tool probes incrementally (machine-related)
	100010	Calibrate tool probe incrementally (workpiece-related)
_MA	1, 2	Measuring axis
_MD	0, 1	Measuring direction: 0 - positive, 1 - negative

The following additional parameters are also valid
_VMS, _FA, **_PRNUM**, and **_NMSP**.
Refer to Sections 2.2 and 2.3.

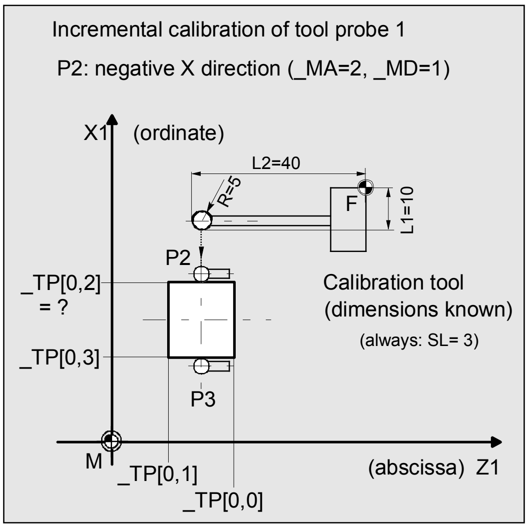
© Siemens AG, 2004. All rights reserved

Sequence

Position before measuring cycle call
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 **in front of** the measuring surface.
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 measuring direction (_MD) immediately on starting.

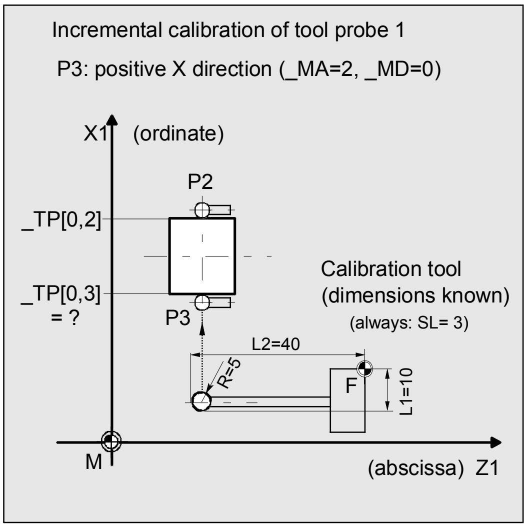


Position after end of measuring cycle
When the calibration procedure is completed the calibration tool is positioned on the starting position again.



Remark
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).
Calibration tool: See Subsection 6.2.2

However, it is not necessary to calibrate all 4 sides for **incremental** measurement. The side that is used for incremental measurement is sufficient.
That does not apply to automatic measurement. Here all 4 points must be calibrated or values entered for automatic central positioning of the tool to be measured.



© Siemens AG, 2004. All rights reserved

Programming example

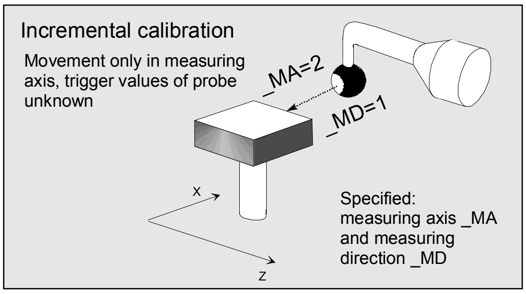
Calibrate tool probe incrementally

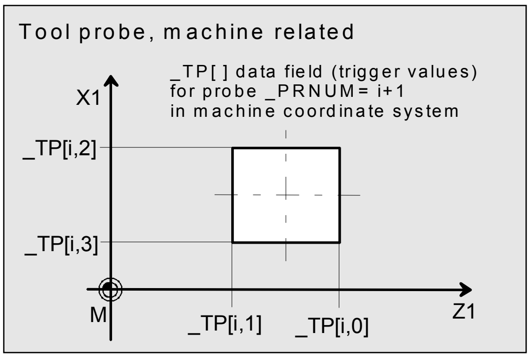
Tool probe 1 is in the machining area and is oriented parallel to the axis of the machine. Calibration is to be performed in the minus X direction and incrementally. The calibration tool is inserted in the turret as tool T7. Values of the calibration tool T7 D1:

Tool type (DP1):	500
Cutting edge position (DP2)	3
Length 1 - Geometry (DP3):	L1 = 10
Length 2 - Geometry (DP4):	L2 = 40
Radius - Geometry (DP6):	R = 5

Values of tool probe 1 in data block GUD6 before calibration:

_TP[0,0] = ?, _TP[0,1] = ?
_TP[0,2] = ?, _TP[0,3] = ?





%_N_INCR_CALIBRATE_MPF	
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 (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.

© Siemens AG, 2004. All rights reserved

6.3.6 Incremental measurement

Function

This cycle and its various measuring variants are for measuring tools **incrementally**:
_MVAR=100001: Turning tools (machine-related)
_MVAR=100011: Turning tools (workpiece-related)
_MVAR=1xxx01: Milling tools, drills (machine-related)
_MVAR=1xxx11: Milling tools, drills (workpiece-related).

Workpiece-related or machine-related measurement require an appropriately calibrated tool probe (refer to Subsection 6.3.5).
With milling tools/drills, the measurement is further specified in the 3rd to 5th digits of parameter _MVAR.

It is possible to measure single tool lengths, or alternatively for milling tools the cutter radius.
The calculated offsets are entered in the active D number. The offset is entered in the **geometry data** and the wear data are reset (irrespective of _CHBIT[3]).
Only the offset value that is in the measuring axis _MA and measuring direction _MD can be determined in a measurement.

If _CHBIT[20]=1, positioning of the milling spindle at the value of _STA1 can be suppressed (see Subsection 6.3.7).
This is possible for milling cutter measuring variants:
_MVAR= xxx001 (where x : 0 or 1, no other values).

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 offset (D number).
The tool type is entered in the offset data.
For a **milling cutter**, 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. The **drill** is also SD 42950: TOOL_LENGTH_TYPE = 0 possible (see Subsection 6.3.8).

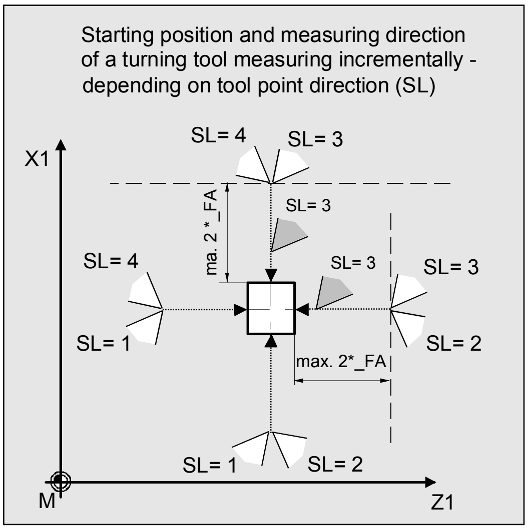
© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	1xxx01 1xxx11	Measure a tool incrementally – machine-related Measure a tool incrementally – workpiece-related (> meas. cycle SW 6.3) More precise parameterization for milling tools/drills is entered in the 3rd to 5th digits of _MVAR.
_MA	1, 2	Measuring axis
_MD	0, 1	Measuring direction: 0 - positive, 1 - negative
_STA1		Only for milling tools and if _CHBIT[20]=0: Starting angle of the milling spindle
_CORA		Only for milling tools and measurement with reversal: Offset angle position of the milling spindle after reversal

The following additional parameters are also valid
_VMS, _FA, _PRNUM and **_NMSP**.
Refer to Sections 2.2 and 2.3.

Sequence



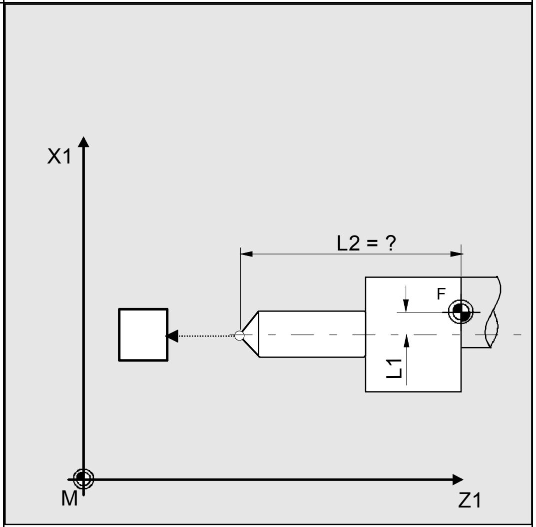
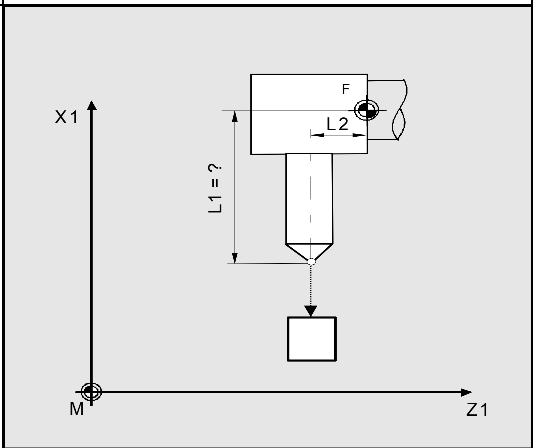
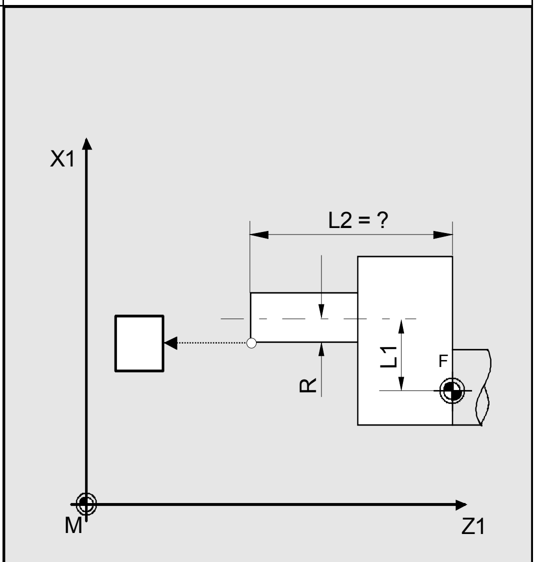
Position before measuring cycle call

Before the cycle is called, the tool must be moved to the starting position, as is shown in the diagram for **turning tools**, e.g.: with traversal in JOG:
The "tip" of the tool in the **measuring axis** _MA is within distance 2 · _FA **in front of** the measuring surface. 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!
If _CHBIT[20]=1, selected measuring variants are possible for a milling cutter without taking the starting angle _STA1 into account (see Subsection 6.3.7).

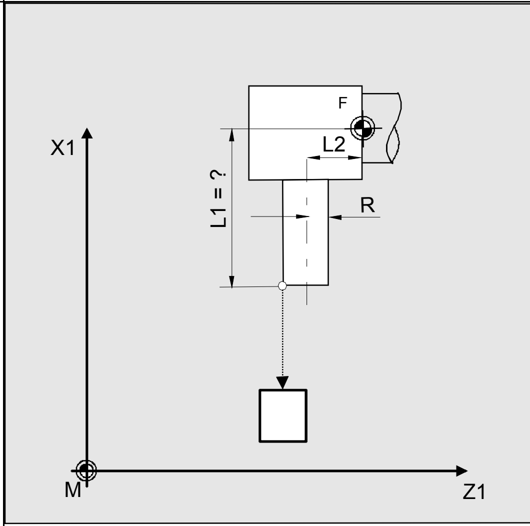
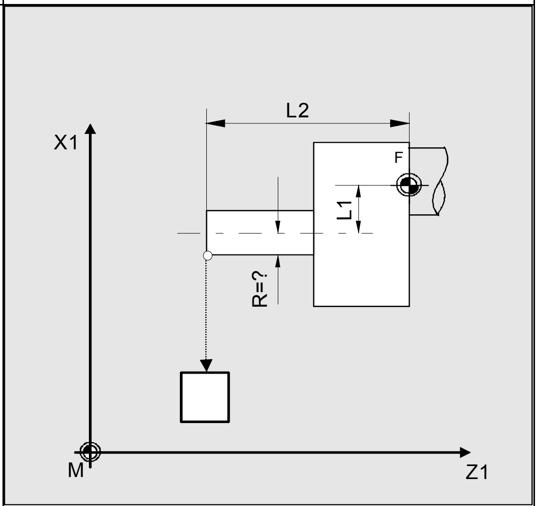
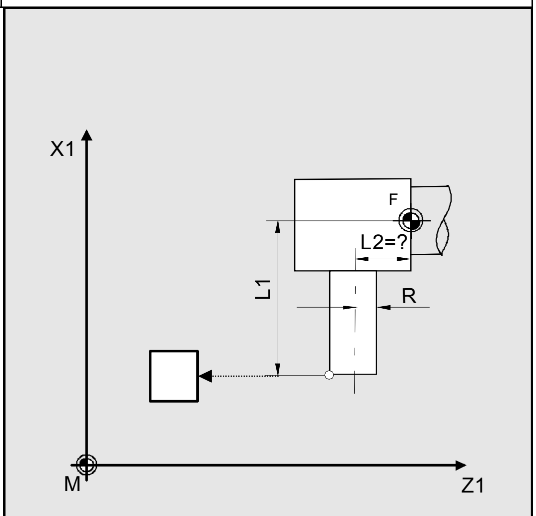
Position after end of measuring cycle

When the cycle is complete, the tool nose is again located at the starting position.

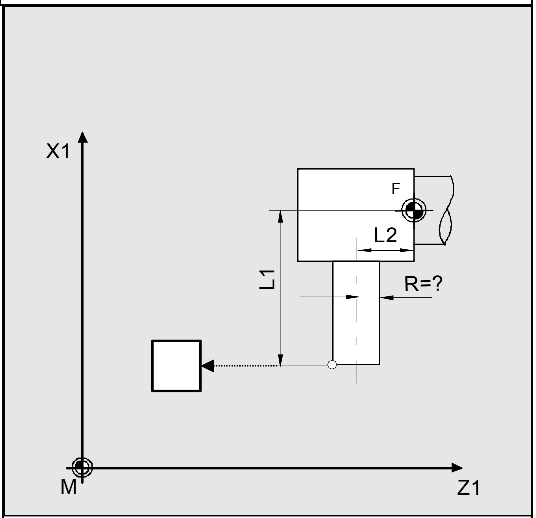
© Siemens AG, 2004. All rights reserved

Measuring variant	Specified geometry	Override applied in	Milling tools, drills
Example: **Axial position,** Drill, R=0, incremental measurement without reversal, calculation of the length in Z **_MVAR=100001** **_MA=1** Always position the drill tip in the center of the probe!	L1=... 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	
Example: **Axial position,** Miller, R ≠ 0, Measuring without reversal, calculation of the length in Z **_MVAR=100001** **_MA=1**	L1= -... L2=... R=...	L2	

© Siemens AG, 2004. All rights reserved

Measuring variant	Specified geometry	Override applied in	Milling tools, drills
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	
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	

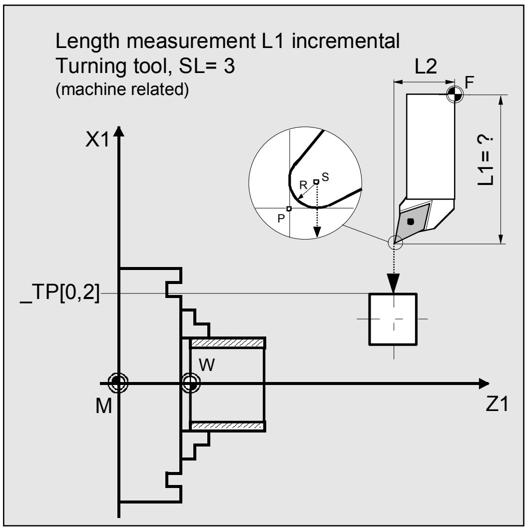
© Siemens AG, 2004. All rights reserved

Measuring variant	Specified geometry	Override applied in	Milling tools, drills
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	

Programming example

With the turning tool T3, D1 with SL=3, length L1 is to be determined **incrementally** and machine-related.

The probe to be used is tool probe 1. This probe is already calibrated in the minus X direction (machine-related). The precise values are entered in probe array _TP[0,2,...].



%_N_INCR_MEAS_MPF	
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 the 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.

© Siemens AG, 2004. All rights reserved

6.3.7 Milling tool: Suppression of start angle positioning _STA1

Function

To apply the angular position of the milling spindle (cutting edge of the miller contacting the probe) unchanged in the cycle and thus suppress the starting angle positioning with the value in _STA1, you can set

_CHBIT[20]=1

will be reached.

However, this only permits simple milling cutter measuring variants that do not have to use the starting angle in _STA1, e.g.: no 2nd measurement or no re-positioning after measurement with reversal. Otherwise milling cutter measuring variants are possible that are also permitted for incremental measurement.

If the machine does not feature an SPOS-capable milling spindle, it is also possible to measure milling cutters with these measuring variants and _CHBIT[20]=1.

Permissible measuring variants with milling cutter and _CHBIT[20]=1:
xxx0x1 (where x: 0 or 1, no other values)

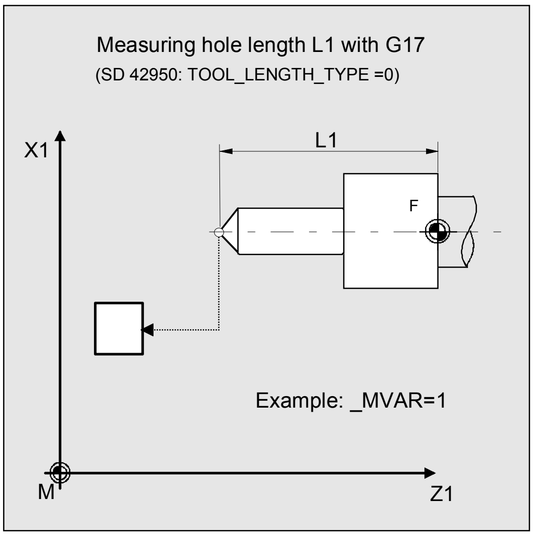
Other measuring variants with a miller will be rejected with an alarm message.

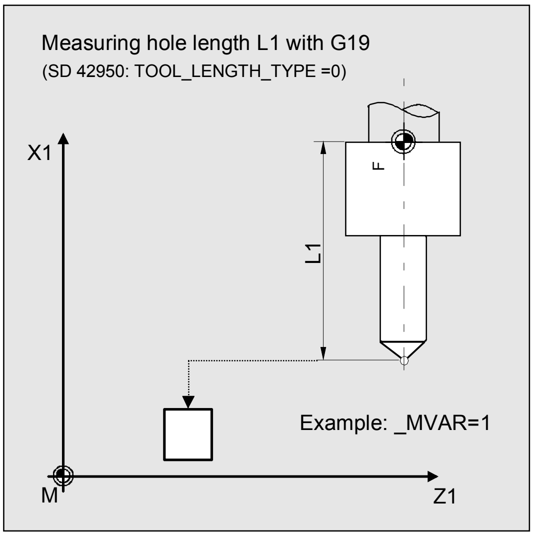
For measurement with rotating spindle and _CHBIT[20]=1, only these measuring variants are permitted. Measurement with reversal is not permitted.

© Siemens AG, 2004. All rights reserved

6.3.8 Measuring drills – special applications (from measuring cycles SW 6.3)

Function





If **drills** are used on lathes with a length correction (offset) as for milling machines (SD 42950: TOOL_LENGTH_TYPE=0), then a drill can also be measured (gauged) in this application.
Length L1 is always calculated in the applicate (tool offset axis) of the current plane G17 to G19.
This also characterizes the position of the tool. The usual position definition in _MVAR (5th digit) is no longer relevant and is ignored.
G17: L1 in Z axis (corresponds to axial position)
G18: L1 in Y axis (no turning machine application)
G19: L1 in X axis (corresponds to radial position)

Length L1 is determined if

- the active tool of type 2xyx (drill)
- SD 42950: TOOL_LENGTH_TYPE=0
- G17 and G19 are active and
- a measuring variant is set:

_MVAR=1:	Measuring (relative to the machine)
_MVAR=11:	Measuring (relative to the workpiece)
_MVAR=1000x1:	Incremental measurement, (referred to the machine or work-piece)

A measuring axis specified in _MA is ignored. The 3rd axis (applicate) is used within the cycle.
Otherwise the description of the measuring variant applies.

Requirement:
The tool probe has been calibrated with **G18** active as is usual for turning tools.

© Siemens AG, 2004. All rights reserved

6.4 CYCLE973 calibrate workpiece probe

6.4.1 Function overview

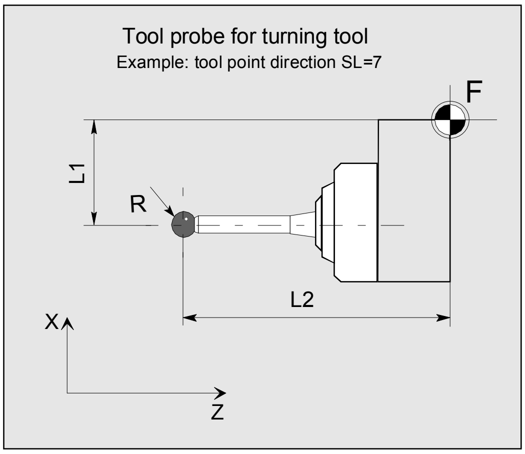
Programming

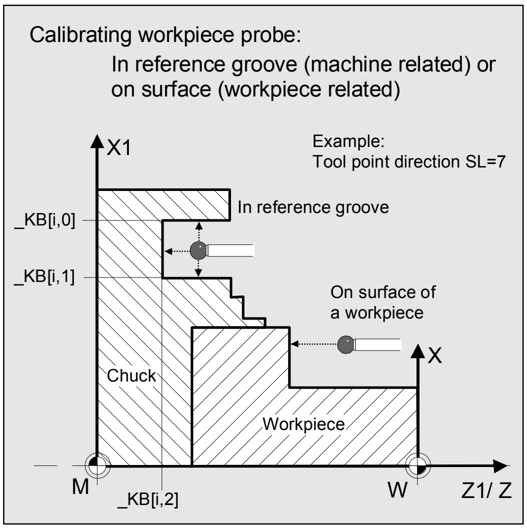
CYCLE973

Function

This cycle can calibrate a workpiece probe with various cutting edge positions in a

- **reference groove** or on a
- **surface**.





The surface for calibration on a "surface" is workpiece-related. It is only possible to calibrate in the selected axis and direction (perpendicular in front of the surface).

Calibration in a "reference groove" is relative to the machine. In this case, calibration is possible in one cycle call in both axis directions.
It is additionally possible select determining the skew of the probe in the idle position or determining the active probe ball diameter. The workpiece probe can only have cutting edge positions SL=7 or SL=8.

Calibration principle
The switching position of the workpiece probe in an axis is calculated into the measuring probe length. The trigger values calculated in this way (relative to ball center) is then entered in the corresponding array _WP[ ] of array GUD6.DEF for the associated probe **_PRNUM** (_WP[_PRNUM-1,...]).

For a complete description of the array _WP[ ] of a workpiece probe, refer to Subsection 9.2.3.

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE973 permits the following calibration variants that are specified with parameter _MVAR.

Digit					*Meaning*
5	**4**	**3**	**2**	**1**	
-	-	-	-	0	**Calibration on surface** (workpiece-related)
			1	3	**Calibrate in groove** (machine-related)
0			1	3	Without determining skew of probe
1			1	3	Determining skew of probe
	1		1	3	1 axis direction (specify meas. axis _MA and axis direction _MD)
	2		1	3	2 axis directions (specify measuring axis _MA)
		0	1	3	Without determining diameter of probe ball
		1	1	3	Determining diameter of probe ball

Note:
When _MVAR=x1x13 calibration is only performed in one direction. It is not possible to determine skew or calculate probe ball.

Result parameters

Measuring cycle CYCLE973 returns the following values in block GUD5 for **calibration**:

_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

© Siemens AG, 2004. All rights reserved

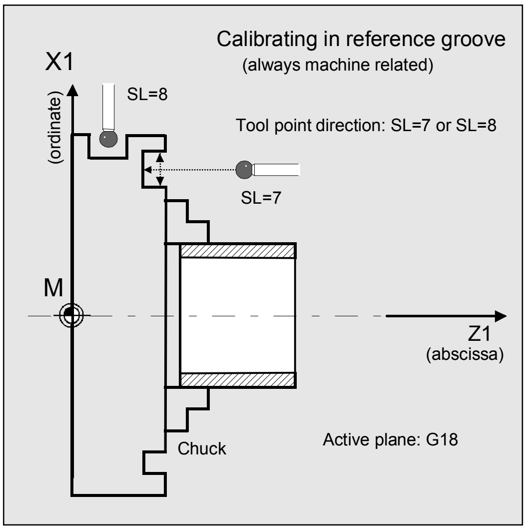
6.4.2 Calibrating in the reference groove

Function

With this measuring cycle and measuring variants

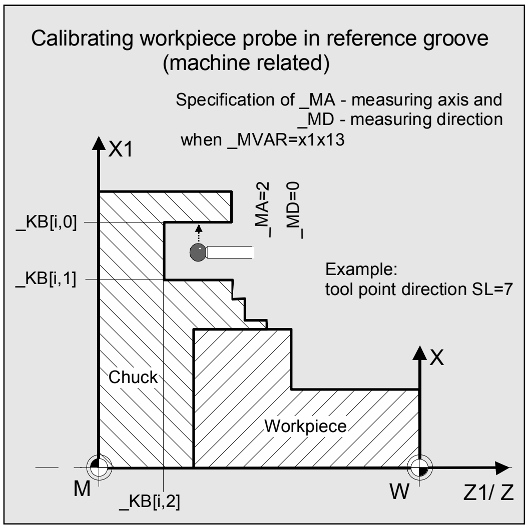
_MVAR=xxx13

it is possible to calibrate a workpiece probe with cutting edge position SL=7 or SL=8 in a **reference groove** machine-related in the axes of the plane (abscissa, ordinate).
Calibration is possible in one direction (_MVAR=x1x13) or in both directions of an axis (_MVAR=x2x13).



It is also possible to calibrate the skew of the probe and the active diameter of the probe ball in both directions.

The workpiece probe calibrated is selected with **_PRNUM.** The associated array _WP[ ] in data block GUD6.DEF is _WP[_PRNUM-1,...] (for a detailed description of the data array, refer to Subsection 9.2.3).

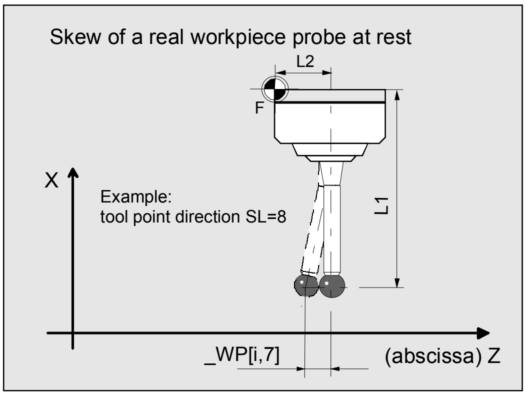


The reference groove is selected with **_CALNUM**. The associated array _KB[ ] in data block GUD6.DEF is _KB[_CALNUM-1,...].

Precondition

The dimensions of the reference groove must already be stored in array _KB[ ] of data block GUD6.DEF for the groove selected via **_CALNUM**.

The workpiece probe must be called as a tool with a tool offset.



© Siemens AG, 2004. All rights reserved

Parameters

Parameter						Value	Description
_MVAR							Calibration variant
				1	3		**Calibrate in groove** (machine-related)
	0			1	3		Without determining skew of probe
	1			1	3		Determining skew of probe
		1		1	3		1 axis direction (specify meas. axis _MA and axis direction _MD)
		2		1	3		2 axis directions (specify measuring axis _MA)
			0	1	3		Without determining diameter of probe ball
			1	1	3		Determining diameter of probe ball
_MA						1, 2	Measuring axis
_MD						0 positive axis direction 1 negative axis direction	Measuring direction (for _MVAR=x1x13 only)
_CALNUM						INTEGER	Number of reference groove (calibration groove)
_PRNUM						INTEGER	Probe number

The following additional parameters are also valid
_VMS, _TZL, _TSA, _FA, and **_NMSP**.
Refer to Sections 2.2 and 2.3.

Note:
When _MVAR=x1x13 calibration is only performed in one direction. It is not possible to determine skew or calculate probe ball.

Attention!
The first time calibration is performed the default setting in the array of the probe is still "0". For that reason _TSA> radius probe ball must be programmed to avoid alarm "Safe area violated".

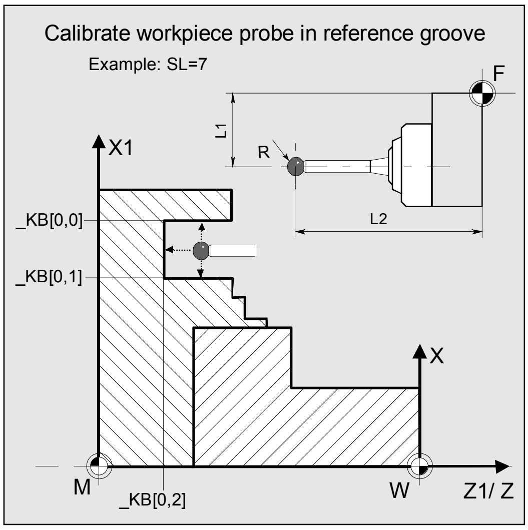
Sequence

Position before measuring cycle call
The starting point must be selected such that the selected workpiece probe can be positioned in the cycle into the reference groove selected via _CALNUM by the shortest path with paraxial collision-free movements in accordance with the active cutting edge position.

Position after end of measuring cycle
On completion of calibration, the probe is positioned facing the calibration surface at distance _FA.

© Siemens AG, 2004. All rights reserved

Programming example



Calibrate workpiece probe in reference groove

The workpiece probe 1 with cutting edge position SL=7 is to be calibrated in reference groove 1 in both axes and in both directions in X.
The probe is inserted as tool T8, D1.
Probe lengths L1 and L2 always refer to the probe ball center and must be entered in the tool offset memory before the measuring cycle is called, T8, D1:

Tool type (DP1):	580
Cutting edge position (DP2)	7
Length 1 - Geometry (DP3):	L1 = 40.123
Length 2 - Geometry (DP4):	L2 = 100.456
Radius - Geometry DP6):	3.000

The data for reference groove 1 have already been entered: _KB[0,0] = 60.123, _KB[0,1] = 50.054, _KB[0,2] = 15.021

%_N_CALIBRATE_IN_GROOVE_MPF	
N10 T8 D1	;Tool offset of the probe
N20 G0 SUPA G90 DIAMOF Z125 X95	;Position in front of cycle call (starting ;position), position without ZO
N30 _TZL=0 _TSA=1 _VMS=0 _NMSP=1 _FA=3 _PRNUM=1 N31 _MVAR=13 _MA=1 _MD=1 _CALNUM=1	;Set parameters for calibration, ;minus Z direction
N40 CYCLE973	;Cycle call
N50 _MVAR=02013 _MA=2	;In X axis, both directions
N60 CYCLE973	;Cycle call
N70 G0 SUPA Z125	;Retraction in Z
N80 SUPA X95	;Retraction in X
N100 M2	;End of program

Explanation

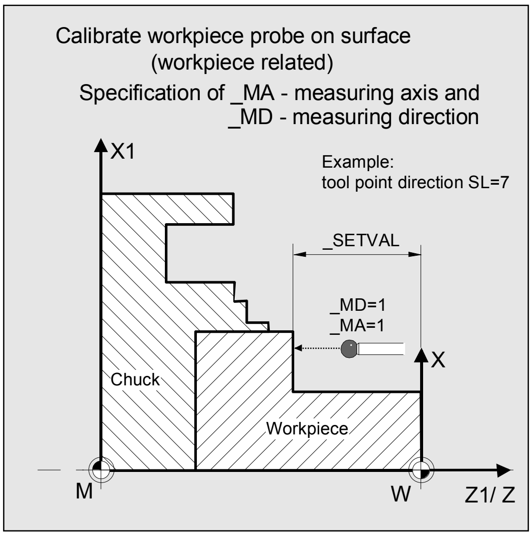
The cycle automatically approaches reference groove 1 from the starting position and performs calibration in both axes and in the X axis in a double cycle call.
The new trigger values are stored in the data of the workpiece probe 1
_WP[0,1], _WP[0,3], _WP[0,4].
At the end, result array _OVR[ ] contains the values of the 2nd cycle call.

© Siemens AG, 2004. All rights reserved

6.4.3 Calibration at a surface

Function

With this measuring cycle and measuring variant
_MVAR=0
it is possible to calibrate a workpiece probe with cutting edge positions SL=5 to 8 on a surface (workpiece-related) and therefore determine the probe trigger points.
The position of the surfaces is defined in workpiece coordinates in **_SETVAL**.
The workpiece probe calibrated is selected with **_PRNUM.** The associated array _WP[ ] in data block GUD6.DEF is _WP[_PRNUM-1,...].



Precondition
The surface must be parallel to an axis of the workpiece coordinate system and have low surface roughness.
The workpiece probe is called as a tool with tool offset and positioned opposite the calibration surface. 5xy should be entered as the tool type.

Parameters

_MVAR	0	Calibration on surface (workpiece-related)
_SETVAL	REAL	Setpoint referred to the workpiece zero, for facing axis in the diameter (DIAMON)
_MA	1, 2, 3[1)]	Measuring axis
_MD	0 positive axis direction 1 negative axis direction	Measuring direction
_PRNUM	INT	Probe number

1) It is also possible to calibrate in the 3rd axis (Y in G18), provided that this axis exists.

The following additional parameters are also valid **_VMS, _TZL, _TSA, _FA**, and **_NMSP**.
Refer to Sections 2.2 and 2.3.

Attention!
The first time calibration is performed the default setting in the array of the probe is still "0". For that reason _TSA> probe ball radius must be programmed to avoid alarm "Safe area violated".

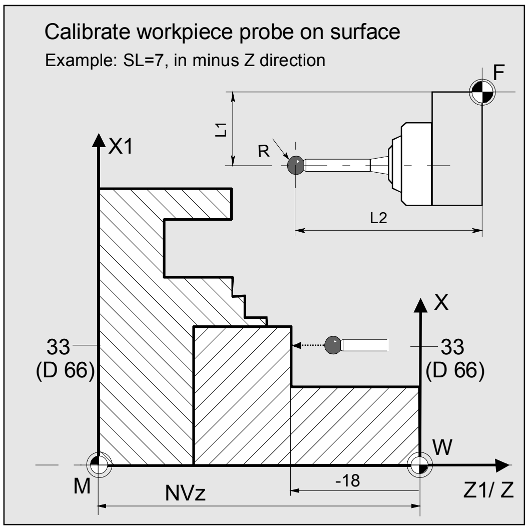
© Siemens AG, 2004. All rights reserved

Sequence

Position before measuring cycle call
The starting point must be a position facing the calibration surface.

Position after end of measuring cycle
On completion of calibration, the probe is positioned facing the calibration surface at distance _FA.

Programming example



Calibration of probe 1 on a surface
Workpiece probe 1 with cutting edge position SL=7 is to be calibrated on surface Z=-18 mm in direction minus Z.
The probe is inserted as tool T9, D1.
Probe lengths L1 and L2 always refer to the probe ball center and must be entered in the tool offset memory before the measuring cycle is called, T9, D1:

Tool type (DP1):	580
Cutting edge position (DP2)	7
Length 1 - Geometry (DP3):	L1 = 40.123
Length 2 - Geometry (DP4):	L2 = 100.456
Radius - Geometry DP6):	3.000

Zero offset, with settable ZO G54:
ZOz

%_N_CALIBRATE_IN_Z_MPF	
N10 G54 G90 G0 X66 Z90 T9 D1 DIAMON	;Activate ZO, select the tool offset ;of the probe ;Position before cycle call
N20 _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, _SETVAL is negative!
N30 CYCLE973	;Cycle call
N40 G0 Z90	;Retraction in Z
N50 X146	;Retraction in X
N100 M2	;End of program

Explanation

The surface with position Z=-18 is approached in the Z axis in the minus direction (_SETVAL=-18, _MA=1, _MD=1).

© Siemens AG, 2004. All rights reserved

Actual calibration starts _FA=3 mm in front of the surface. The workpiece probe is then calibrated and ends up facing the surface again at distance _FA from it.
The new trigger value in minus Z is entered in the data of probe 1 _WP[0,1] and in the result field _OVR[ ].

The original position is approached in block N40, N50.

© Siemens AG, 2004. All rights reserved

6.5 CYCLE974 workpiece: 1-point measurement

6.5.1 Function overview

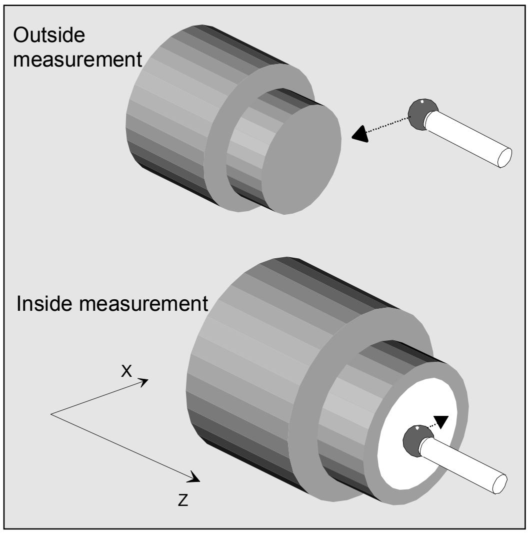
Programming

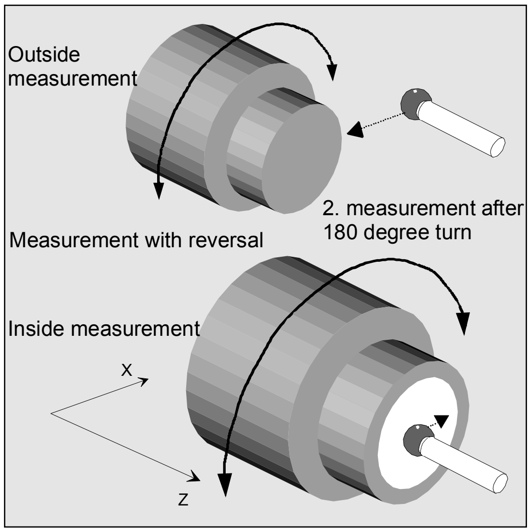
CYCLE974

Function

This measuring cycle can be used in various measurement variants to determine workpiece dimensions in a 1-point measurement.
It is also possible to determine a zero offset (ZO) or an automatic tool offset.

- **1-point measurement and ZO determination**
- **1-point measurement and tool offset**
- **1-point measurement with reversal and tool offset**





The measuring cycle determines the actual value of the workpiece with respect to the workpiece zero in the selected measuring axis _MA and calculates the difference from a defined setpoint (setpoint-actual value).
An empirical value stored in data block GUD5 can be included. For variants "with tool offset" it is also possible to calculate a mean value over several parts.
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.
If KNUM=0, there is no offset.

Precondition
The probe must be calibrated in the measuring direction and as a tool with tool offset. The tool type is 5xy.
The cutting edge position can be 5 to 8 and must be suitable for the measurement task.

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE974 permits the following measuring variants that are specified with parameter _MVAR.

Value	*Meaning*
0	1-point measurement and tool offset
100	1-point measurement and ZO determination
1000	1-point measurement with reversal and tool offset

Result parameters

Depending on the measuring variant, measuring cycle CYCLE974 returns the following result values in the GUD5 block:

_OVR [0]	REAL	Setpoint value 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)3)]	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	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 ZO correction only
3) From meas. cycles - SW 6.3: For "tool offset" the offset value always appears in _OVR[20] as radius dimension.

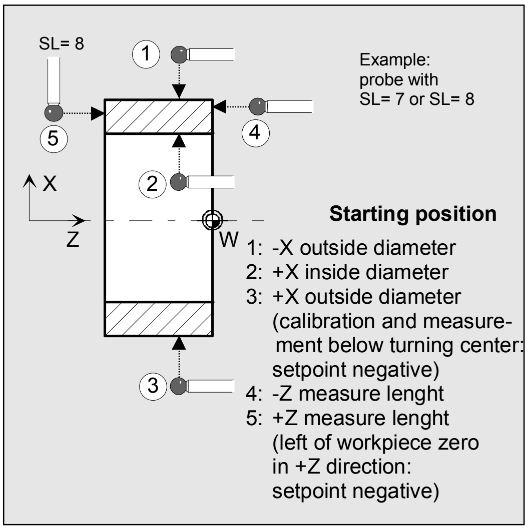
© Siemens AG, 2004. All rights reserved

Starting positions for various measuring tasks

The starting positions before cycle call depend on the measuring task, the value of the setpoint _SETVAL, the measuring axes, and the cutting edge position (SL) of the workpiece probe.

The probe must be positioned facing the point to be measured and is reached by traversing measuring axis _MA in the setpoint direction in the measuring cycle. The setpoint (position of the point) is defined by parameter _SETVAL.

Measurement is possible parallel to and in the direction of the axes permitted by the "cutting edge position" of the workpiece probe inserted.



© Siemens AG, 2004. All rights reserved

6.5.2 1-point measurement and ZO determination

Function

With this measuring cycle and measuring variant
_MVAR=100
the actual value of a workpiece is determined with reference to the workpiece zero in the selected measuring axis _MA.

An empirical value from data block GUD5 can be included with the correct sign.

The zero offset (ZO) is applied in such a way that the actual value adopts the required setpoint (_SETVAL) in the workpiece coordinate system when the corrected ZO (frame) is used.
Mirroring can be active in a frame of the frame sequence. Dimension factors must never be active.

The ZO to be corrected is specified in coded form with variable **_KNUM** >0.
The ZO can be specified and corrected by various methods, e.g. in various settable frames, in various basic frames, system frames, fine offset, or coarse offset, etc.

For detailed information on specifying _KNUM for the zero offset: Refer to Section 2.3.

Precondition
If necessary, the workpiece must be positioned in the correct angular spindle position with SPOS before the cycle is called.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	100	1-point measurement and ZO determination
_SETVAL	REAL[1)]	Setpoint, with reference to the workpiece zero
_MA	1, 2, 3[1)]	Measuring axis
_KNUM	0, >0	0: No automatic ZO correction >0: With automatic ZO correction (individual values: See Section 2.3, parameter _KNUM)

1) Measurement in the 3rd axis of the plane (with G18 in Y) is also possible, provided 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 (facing axis), if _CHBIT[19]=1 is set in block GUD6. The offset is then stored in the X component of the selected ZO memory.

The following additional parameters are also valid **_VMS, _TSA, _FA, _PRNUM, _EVNUM,** and **_NMSP**.
Refer to Sections 2.2 and 2.3.

Sequence

Position before measuring cycle call

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

Position after end of measuring cycle

On completion of measurement, the probe is positioned facing the measuring surface at distance _FA.

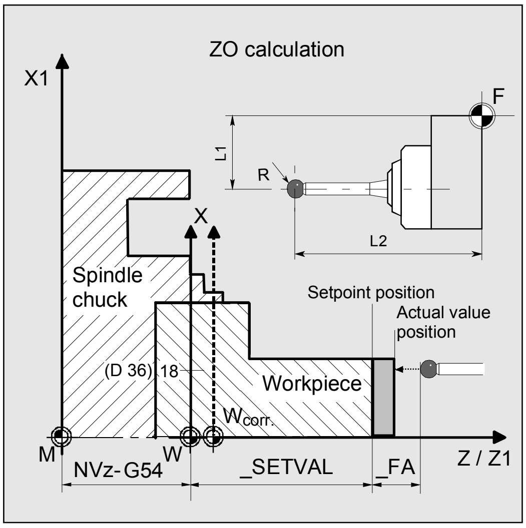
Attention!

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 spindle orientation must also be considered. Deviations can cause additional measuring errors.

© Siemens AG, 2004. All rights reserved

Programming example

ZO calculation at a workpiece



The intention is to determine the zero offset in the Z axis on a clamped workpiece with workpiece probe 1, inserted as tool T8, D1. The position determined should retain the value 60 mm in the new workpiece coordinate system for G54.
Measurement is also performed with G54.

The probe is already calibrated and the tool data are entered in T8, D1:

Tool type (DP1):	580
Cutting edge position (DP2)	7
Length 1 - Geometry (DP3):	L1 = 40.123
Length 2 - Geometry (DP4):	L2 = 100.456
Radius - Geometry DP6):	3.000

Zero offset, with settable ZO G54:
ZOz

%_N_ZO_DETERMINING_1_MPF	
N10 G54 G90 G18 DIAMON T8 D1	;Call ZO, tool = probe
N20 G0 X36 Z100	;Starting position before cycle call
N30 _MVAR=100 _SETVAL=60 _MA=1 _TSA=1 _KNUM=1 _EVNUM=0 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	;Parameters for cycle call
N40 CYCLE974	;Measurement in the Z direction
N50 G0 Z100	;Retraction in Z
N60 X114	;Retraction in X
N100 M2	;End of program

Note

If parameter _VMS has value 0, the default value of the measuring cycle is used for the variable measuring velocity:
if _FA=1: 150 mm/min.
if _FA>1: 300 mm/min.
(see Section 2.3)

© Siemens AG, 2004. All rights reserved

6.5.3 1-point measurement and tool offset

Function

With this measuring cycle and measuring variant
_MVAR=0
the actual value of a workpiece is determined with reference to the workpiece zero in the selected measuring axis.
An offset can also be determined for a tool depending on that. This tool is specified in _TNUM and _TNAME. The D number and type of offset are specified in coded form in variable _KNUM.

From measuring cycles SW 6.3, extended tool offset is available. With this function a tool from a particular stored tool environment _TENV, and additive, setup offsets can be corrected by specifying the DL number in _DLNUM.

Detailed information on the parameters:
Refer to Section 2.3.

Empirical values and mean values
An empirical value stored in data block GUD5 in array _EV[ ] can be included in calculation of the result after measurement is completed.
Optionally, averaging is performed over a number of parts (array _MV[ ]) and the tolerance bands are checked.
Both are activated in _EVNUM (see Section 2.3).

Precondition
If necessary, the workpiece must be positioned in the correct angular spindle position with SPOS before the cycle is called.

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	0	1-point measurement and tool offset
_SETVAL	REAL[2)]	Setpoint (according to drawing) (in the case of facing axis (X) and diameter programming, this is a diameter dimension)
_MA	1, 2, 3[1)]	Measuring axis
_KNUM	0, >0	0: Without automatic tool offset >0: With automatic tool offset (individual values: See Section 2.3, parameter _KNUM)
_TNUM	INTEGER, ≥0	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)
_DLNUM	INTEGER, ≥0	DL number for additive and setup offset (> meas. cycle SW 6.3)
_TENV	STRING[32]	Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)

1) Measurement can also be made in the 3rd axis of the plane (for G18 in Y) if this is available.

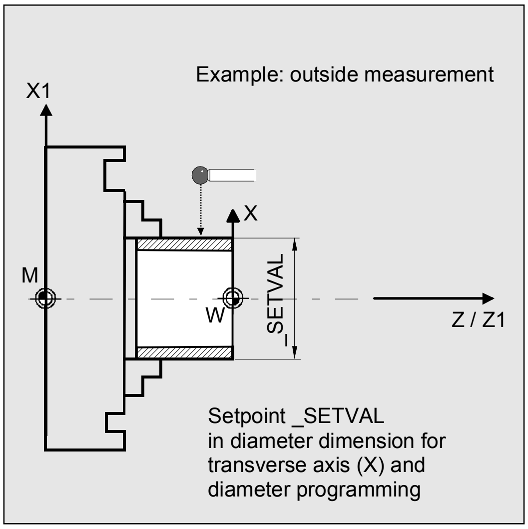
2) By setting _CHBIT[19] to 1 in the GUD6 block, the same parameterization can be used regarding the setpoint when measuring in the Y axis (3rd axis of the plane) for active G18 as when measuring in the X axis (transverse axis). In this case, the tool offset is also applied to L1 (effective length in X) if nothing different is specified by _KNUM.

The following additional parameters are also valid **_VMS, _TZL, _TMV, _TUL, TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP,** and **_K**.
Refer to Sections 2.2 and 2.3.

Sequence

Position before measuring cycle call
The probe must be positioned opposite the surface to be measured.

Position after end of measuring cycle
On completion of measurement, the probe is positioned facing the measuring surface at distance _FA.



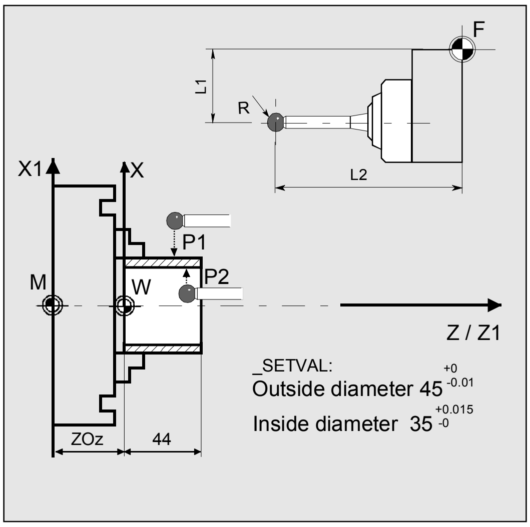
Attention!
Precise measurement is only possible with a probe calibrated under the measurement conditions, i.e. working plane and measuring velocity are the same for both measurement and calibration. If the probe is used in the spindle for a powered tool, the spindle orientation must also be considered. Deviations can cause additional measuring errors.

© Siemens AG, 2004. All rights reserved

Programming example

1-point measurements at outside and inside diameters with tool offsets

An outside diameter with tool T7, D1 and an inside diameter with tool T8, D1 has been machines on a workpiece. The set diameters have the dimensions shown in the figure.



If the absolute value of the difference determined is >0.002 mm, the length (in measuring axis _MA) of the tool is to be automatically offset in the wear.

The maximum permissible deviation is taken as max. 0.5 mm. Max. 0.04 mm is permissible.
To obtain a minimum measuring path of 0.5 mm, the measuring path is programmed as _FA=0.5+0.5=1 mm (max. total measuring path = 2 mm).

The offset must take the empirical value in memory _EV[12] into consideration for T 7, or _EV[13] for T 8. Mean value calculation _MV[12] or _MV[13] and inclusion in calculation are also to be used.

This tool offset will therefore affect the production of the next workpieces or possible remachining.

Clamping for workpiece:
Zero offset, with settable ZO G54:
ZOz

Workpiece probe 1, inserted as tool **T9, D1**, is to be used as probe.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	580
Cutting edge position (DP2)	7
Length 1 - Geometry (DP3):	L1 = 40.123
Length 2 - Geometry (DP4):	L2 = 100.456
Radius - Geometry DP6):	3.000

© Siemens AG, 2004. All rights reserved

%_N_ONE_POINT_MEAS_MPF	
N10 G54 G18 G90 T9 D1 DIAMON	;Call ZO, tool = probe
N20 G0 Z30 X90	;Preposition probe
N25 _CHBIT[4]=1	;With mean value calculation
N30 _TZL=0.002 _TMV=0.005 _TDIF=0.04 _TSA=0.5 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1 **N31 _MVAR=0 _SETVAL=45 _TUL=0 _TLL=-0.01 _TNUM=7 _KNUM=1 _EVNUM=13 _K=2 _MA=2**	;Parameters for cycle call
N40 CYCLE974	;Measurement on the outside diameter
N50 G0 Z60	;Place probe facing P2
N60 X0	
N70 Z40	
N80 _SETVAL=35 _TUL=0.015 _TLL=0 _TNUM=8 _EVNUM=14	
N90 CYCLE974	;Measurement on the inside diameter
N100 G0 Z110	;Retraction in Z
N110 X90	;Retraction in X
N200 M2	;End of program

Explanation

Measurement of outside diameter and offset T7

The difference calculated from the actual value and setpoint is compensated for by the empirical value in the empirical value memory _EV[12] and compared with the tolerance parameter:

- If it is more than 0.5 mm (_TSA), alarm "Safe area violated" is output and program execution cannot be continued.
- If it is more than 0.04 mm (_TDIF), no compensation is performed and alarm "Permissible dimensional difference exceeded" is output and the program continues.
- If values _TUL= -0.01, _TLL=0 are violated upward or downward, the length of T7 D1 is compensated 100% by this difference. Alarm "Oversize" or "Undersize" is displayed and the program is continued.
- If 0.005 mm (_TMV) is exceeded, the length in T7 D1 is compensated 100% by this difference.
  If it is less than 0.005 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[12] and by including weighting factor (_K=2).

© Siemens AG, 2004. All rights reserved

- If the mean value obtained is >0.002 (_TZL), the reduced compensation of the length 1 for T7 D1 is the mean value/2 and the mean value is deleted in _MV[12].
- If the mean value is <0.002 (_TZL), there is no offset but it is stored in the mean value memory _MV[12] if mean value storage (_CHBIT[4]=1) is active.

The results are entered in result array _OVR[ ]. The wear of the length 1 of T7, D1 is included if a change is necessary.

Measurement of inside diameter and offset T8
Procedure as described for "Measurement output diameter".
Offset of T8 with appropriate modified values _EV[13], _MV[13] (EVNUM=14), _TUL, _TLL, _SETVAL.

Note
The values of the workpiece tolerance parameters _TUL, _TLL were selected asymmetrically in the example. The result is then made symmetrical (see Subsection 2.3.11).

© Siemens AG, 2004. All rights reserved

6.5.4 1-point measurement with reversal and tool offset

Function

With this measuring cycle and measuring variant
_MVAR=1000
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.
The mean value is calculated from both measured values.

Otherwise, the same conditions and tool offset options apply as for measuring variant _MVAR=10 "1-point measurement and tool offset" (see Subsection 6.5.4).

Parameters

_MVAR	1000	1-point measurement with reversal and tool offset
_SETVAL	REAL 2)	Setpoint (according to drawing) (in the case of facing axis (X) and DIAMON , this is a diameter dimension)
_MA	1, 2, 3 1)	Measuring axis
_STA1	REAL, >=0	Starting angle (spindle position)
_KNUM	0, >0	0: Without automatic tool offset >0: With automatic tool offset (individual values: See Section 2.3, parameter _KNUM)
_TNUM	INTEGER, ≥0	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)
_DLNUM	INTEGER, ≥0	DL number for additive and setup offset (from meas. cycle SW 6.3)
_TENV	STRING[32]	Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)

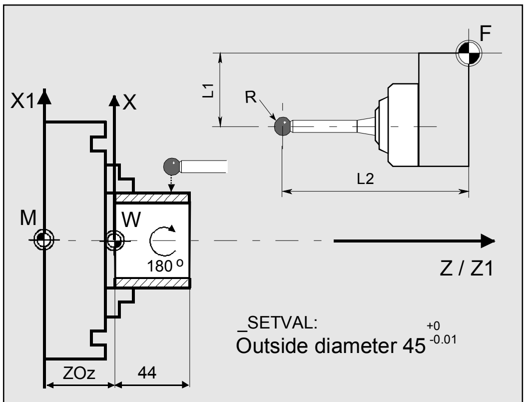
1) Measurement can also be made in the 3rd axis of the plane (for G18 in Y) if this is available.
2) By setting _CHBIT[19] to 1 in the GUD6 block, the same parameterization regarding the setpoint can be used when measuring in the Y axis (3rd axis of the plane) for active G18 as when measuring in the X axis (transverse axis).

© Siemens AG, 2004. All rights reserved

In this case, the tool offset is also applied to L1 (effective length in X) if nothing different is specified by _KNUM.

The following additional parameters are also valid
_VMS, _TZL, _TMV, _TUL, TLL, _TDIF, _TSA, _FA, _PRNUM, _EVNUM, _NMSP, and **_K**.
Refer to Sections 2.2 and 2.3.

Programming example



Outside measurement with spindle reversal

1-point measurement at outside diameter, measuring with reversal

An outside diameter with tool T7, D1 has been machined on a workpiece. The set diameter has the dimension shown in the figure.

This outside diameter is to be measured with reversal. The spindle is SPOS-capable.

If the absolute value of the difference determined is >0.002, the length (in measuring axis _MA) of the tool is to be automatically offset in the wear.

The maximum permissible deviation is taken as max. 1 mm. Max. 0.4 mm is permissible.
To obtain a minimum measuring path of 1 mm, the measuring path is programmed as _FA=1+1=2 mm (max. total measuring path = 4 mm).

The offset is not to consider an empirical value and no mean value is calculated or used.

Clamping for workpiece:
Zero offset, with settable ZO G54:
ZOz

Workpiece probe 1, inserted as tool **T9, D1**, is to be used as probe.
The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T9, D1 in the tool offset memory:

Tool type (DP1):	580
Cutting edge position (DP2)	7
Length 1 - Geometry (DP3):	L1 = 40.123
Length 2 - Geometry (DP4):	L2 = 100.456
Radius - Geometry DP6):	3.000

© Siemens AG, 2004. All rights reserved

%_N_REVERSALMEAS_MPF	
N10 G54 G90 G18 T9 D1 DIAMON	;Call ZO, tool = probe
N20 G0 Z30 X90	;Preposition probe
N30 _MVAR=1000 _SETVAL=45 _TUL=0 _TLL=-0.01 _MA=2 _STA1=0 _KNUM=1 _TNUM=7 _EVNUM=0 _TZL=0.002 _TDIF=0.4 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=2	;Parameters for cycle call
N40 CYCLE974	;Measuring cycle call
N50 G0 Z110	;Retraction in Z
N60 X90	;Retraction in X
N100 M2	;End of program

Note

The values of the workpiece tolerance parameters _TUL, _TLL were selected asymmetrically in the example. The result is then made symmetrical (see Subsection 2.3.11).

© Siemens AG, 2004. All rights reserved

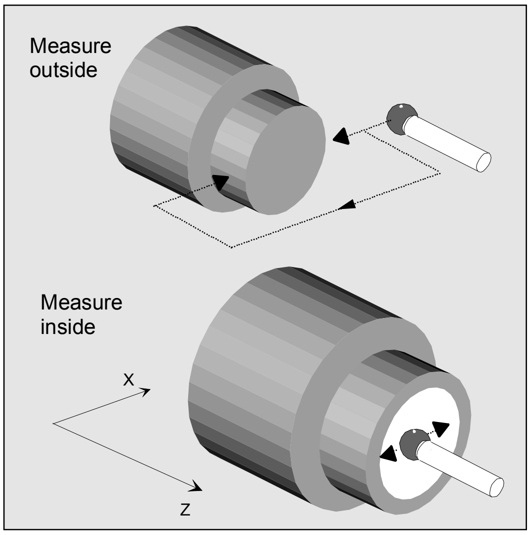
6.6 CYCLE994 workpiece: 2-point measurement

Programming

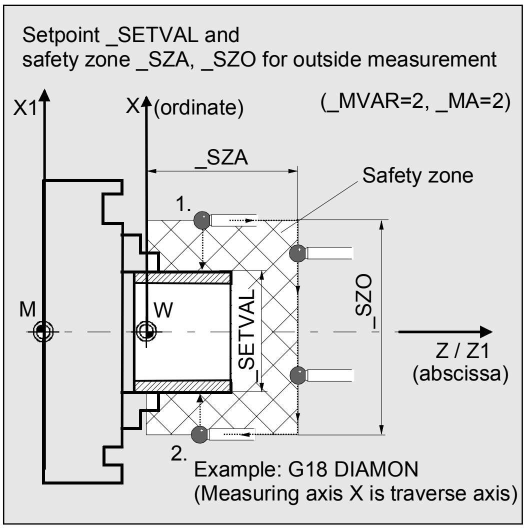
CYCLE994

Function

This measuring cycle can be used to determine workpiece dimensions in 2-point measurements with various measuring variants.
Automatic tool offset is also possible.
The measuring cycle determines the actual value of the workpiece with respect to the workpiece zero in the selected measuring axis _MA and calculates the difference from a defined setpoint (setpoint-actual value).
An empirical value stored in data block GUD5 can be included. It is also possible to calculate a mean value over several parts.
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.
If KNUM=0, there is no offset.



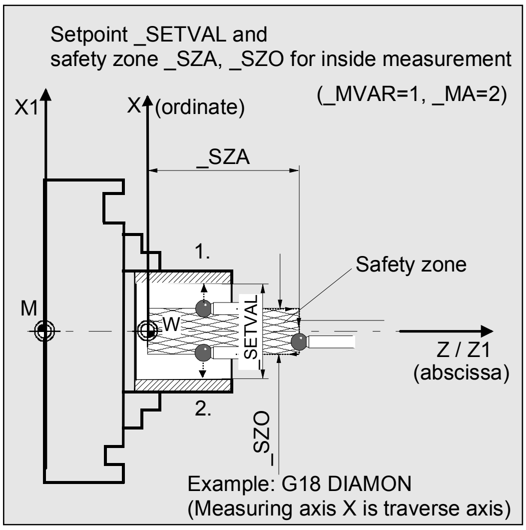
Two opposite measuring points in measuring axis _MA are approached symmetrically to the workpiece zero at the distance of the setpoint _SETVAL.
Sequence of operations: 1. measuring point positive, 2nd measuring point negative.



A safety zone is programmed in parameters _SZA and _SZO. This is considered in traversal with the measuring variant. The probe ball radius must also be considered by the user.

Precondition
The probe must be calibrated in the measuring direction (if _CHBIT[7] = 0) and called as a tool with tool offset.
The tool type is 5xy. The cutting edge position can be 5 to 8 and must be suitable for the measurement task.

© Siemens AG, 2004. All rights reserved



The measuring cycle can be used for measurement **without** previous **calibration**.
Instead of the trigger values in _WP[ ], the probe ball diameter entered in the probe array _PRNUM (_WP[_PRNUM-1,0]) is used in the calculation.
The function is controlled via bit:

_CHBIT[7]=1: Probe not calibrated (without use of trigger values). Use of ball probe diameter _WP[_PRNUM-1,**0**])

_CHBIT[7]=0: Probe calibrated, use of trigger values in _WP[_PRNUM-1,**...**])

Tool offset
An offset can be applied for the tool that machined the workpiece. This tool is specified in _TNUM and _TNAME. The D number and type of offset are specified in coded form in variable _KNUM.

From measuring cycles SW 6.3, extended tool offset is available. With this function a tool from a particular stored tool environment **_TENV**, and additive, setup offsets can be corrected by specifying the DL number in **_DLNUM**.

Detailed information on the parameters:
Refer to Section 2.3.

Empirical values and mean values
An empirical value stored in data block GUD5 in array _EV[ ] can be included in calculation of the result after measurement is completed.
Optionally, averaging is performed over a number of parts (array _MV[ ]) and the tolerance bands are checked.
Both are activated in _EVNUM (see Section 2.3).

© Siemens AG, 2004. All rights reserved

Measuring variants

Measuring cycle CYCLE994 permits the following measuring variants that are specified in parameter _MVAR.

Value	*Meaning*
1	2-point measurement with programmed safety zone (This measuring variant is only for inside measurement!)
2	2-point measurement with programmed safety zone (For inside measurement without safety zone in this measuring variant!)

Result parameters

The measuring cycle CYCLE994 supplies the following values as results in the GUD5 block:

_OVR [0]	REAL	Setpoint diameter or setpoint as radius dimension (note _MA)	
_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 [3]	REAL	Setpoint diameter/radius in applicate	→ with _MA=3 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 [7]	REAL	Actual value diameter/radius in applicate	→ with _MA=3 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 [19]	REAL	Difference diameter/radius in applicate	→ with _MA=3 only
_OVR [20][1)]	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	
_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	

When measuring in the traverse axis and for diameter programming (DIAMON), all of the dimensioned parameters are diameter dimensions, otherwise radius dimensions.

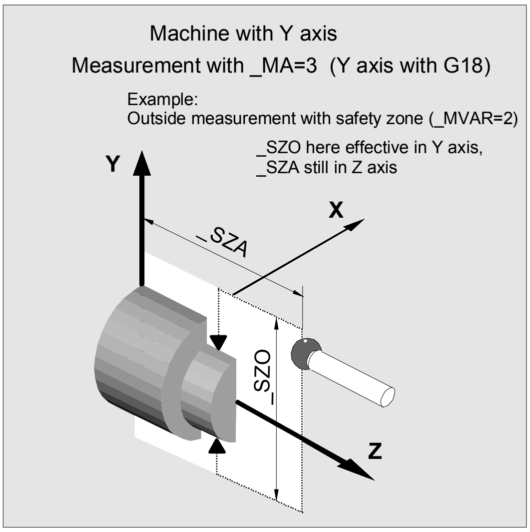
1) from meas. cycles - SW 6.3: For "tool offset" the offset value always appears in _OVR[20] as radius dimension, independent of DIAMON or DIAMOF

© Siemens AG, 2004. All rights reserved

Parameters

_MVAR	1 or 2	1: Inside measurement, 2-point measurement with safety zone 2: 2-point measurement, safety zone only for outside measurement
_SETVAL	REAL2)	Setpoint If measurement is made in the transverse axis and diameter programming (DIAMON) is active, then _SETVAL is a diameter dimension, otherwise a radius dimension around the workpiece zero.
_MA	1, 2, 31)	Measuring axis
_SZA	REAL	Protection zone at the workpiece in the abscissa2) If the abscissa is a transverse axis and diameter programming (DIAMON) is active, then _SZA is a diameter dimension, otherwise a radius dimension around the workpiece zero.
_SZO	REAL	Protection zone at the workpiece in the ordinate2) If the ordinate is a transverse axis and diameter programming (DIAMON) is active, then _SZO is a diameter dimension, otherwise a radius dimension around the workpiece zero.
_KNUM	0, > 0	0: Without automatic tool offset >0: with automatic tool offset (individual values: See Section 2.3, parameter _KNUM)
_TNUM	INTEGER, ≥0	Tool number for automatic tool offset
_TNAME	STRING[32]	Tool name for automatic tool offset (alternative to _TNUM with tool management active)
_DLNUM	INTEGER, ≥0	DL number for additive and setup offset (from meas. cycle SW 6.3)
_TENV	STRING[32]	Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)

1) Measurement in the 3rd axis of the plane is also possible, provided this axis exists (_MA=3: if G18 is in the Y axis).
2) For measurement in the 3rd axis (in G18 in Y _SZO applies in this axis. _SZA still applies in the 1st axis in the plane (Z axis in G18). Travel around is performed in the 1st axis of the plane (Z axis in G18).
Setting _CHBIT[19]=1 in block GUD6 enables the same setpoint and safety 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 (facing 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**.
Refer to Sections 2.2 and 2.3.

© Siemens AG, 2004. All rights reserved

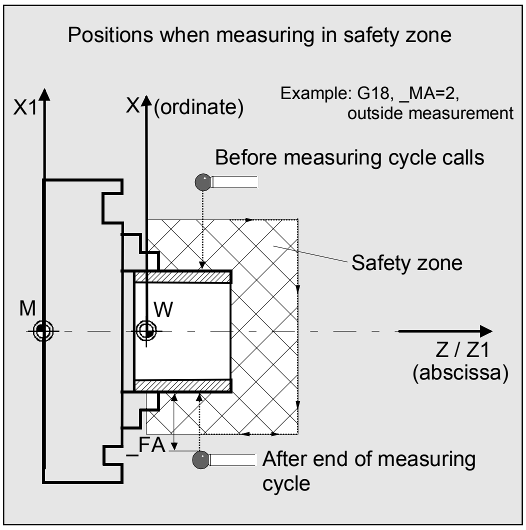
Sequence

Position before measuring cycle call

The probe must be positioned opposite the **positive** measuring point.

Position after end of measuring cycle

After the end of measurement, the probe is facing the **negative** measuring point at distance _FA.



Attention!

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 spindle orientation must also be considered. Deviations can cause additional measuring errors.

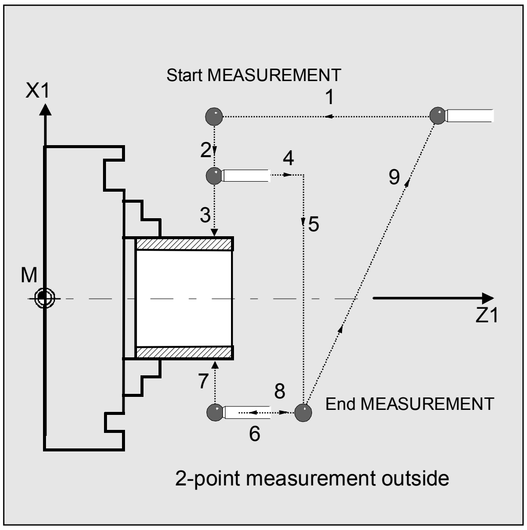
Procedure for outside measurement with _MVAR=2, _MA=2:

(safety zone _SZA, _SZO active)

1: Approach path outside diameter (user)

2 to 7: Traverse paths generated by the cycle for Measurement at the outer diameter taking into account the protection zone _SZA, _SZO (4 to 6)

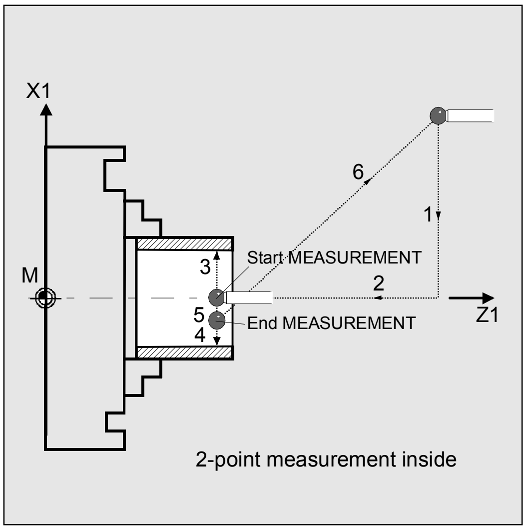
8 to 9: Retraction to the original point (user)



© Siemens AG, 2004. All rights reserved

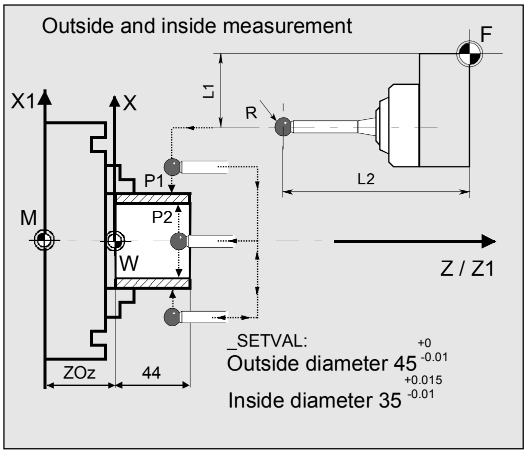
Procedure for inside measurement with _MVAR=2, _MA=2:
(no safety zone active)

1, 2: Approach paths inside diameter (user)
3 to 5: Traverse paths generated by the cycle for Measurement on the inside diameter
6: Retraction paths to the original point (user)



Programming example

Two-point measurement, outside and inside
An outside diameter with tool T8, D1 and an inside diameter with tool T9, D1 has been machines on a workpiece. The set diameters have the dimensions shown in the figure.



If the absolute value of the difference determined is >0.002 mm, the length (in measuring axis _MA) of the tool is to be automatically offset in the wear.

The maximum permissible deviation is taken as max. 0.5 mm. Max. 0.04 mm is permissible.
To obtain a minimum measuring path of 0.5 mm, the measuring path is programmed as _FA= 0.5+0.5=1 mm (max. total measuring path = 2 mm).

The offset must take the empirical value in memory _EV[2] into consideration for T 8, or _EV[3] for T 9. Mean value calculation _MV[2] or _MV[3] and inclusion in calculation are also to be used.

This tool offsets will therefore affect the production of the next workpieces or possible remachining.

Clamping for workpiece:
Zero offset, with settable ZO G54:
ZOz
The workpiece probe 1, used as tool **T1, D1**, should be used as probe.

© Siemens AG, 2004. All rights reserved

The probe is already calibrated. Arrays for workpiece probe 1: _WP[0, ...]
The following is entered under T1, D1 in the tool offset memory:

Tool type (DP1):	580
Cutting edge position (DP2)	7
Length 1 - Geometry (DP3):	L1 = 40.123
Length 2 - Geometry (DP4):	L2 = 100.456
Radius - Geometry DP6):	3.000

%_N_TWO_POINT_MEAS_MPF	
N10 T1 D1 DIAMON	;Call tool = probe (MT)
N20 G0 G54 Z30 X60	;ZO selection, Position probe facing P1
N25 _CHBIT[4]=1 _CHBIT[7]=0	;With mean value calc., calibrated MT
N30 _TLL=-0.01 _MA=2 _SZA=55 _SZO=55 _KNUM=1 _K=3 _TZL=0.002 _TMV=0.005 _TDIF=0.04 _TSA=0.5 _VMS=0 _NMSP=1 _FA=1 _MVAR=2 N31 _SETVAL=45 _TUL=0 _TNUM=8 _EVNUM=3	;Parameter assignment for 1st cycle call ;(outside measurement)
N40 CYCLE994	;2-point measurement outside ;with safety zone (P1)
N50 G0 Z55	;Position probe facing P2
N60 X20	
N70 Z30	
N80 _SETVAL=35 _TUL=0.015 _TNUM=9 _EVNUM=4	;Parameter change for 2nd cycle call ;(inside measurement)
N90 CYCLE994	;2-point measurement inside ;without safety zone (P2)
N100 G0 Z110	;Retraction in Z
N110 X60	;Retraction in X
N200 M2	;End of program

=?

Explanation

Measurement of outside diameter and offset T8

The difference calculated from the actual value and setpoint is compensated for by the empirical value in the empirical value memory _EV[2] and compared with the tolerance parameter:

- If it is more than 0.5 mm (_TSA), alarm "Safe area violated" is output and the program can no longer be executed.
- If it is more than 0.04 mm (_TDIF), no compensation is performed and alarm "Permissible dimensional difference exceeded" is output and the program continues.
- If values _TUL= -0.01, _TLL=0 are violated upward or downward, the length of T8 D1 is compensated 100% by this difference. Alarm "Oversize" or "Undersize" is displayed and the program is continued.

© Siemens AG, 2004. All rights reserved

- If 0.005 mm (_TMV) is exceeded, the length in T8 D1 is compensated 100% by this difference.
- If it is less than 0.005 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[2] and by including weighting factor (_K=3).
  - If the mean value obtained is >0.002 (_TZL), the reduced offset of length 1 for T8 D1 is the mean value/2 and the mean value is deleted in _MV[2].
  - If the mean value is <0.002 (_TZL), there is no offset but it is stored in the mean value memory _MV[2] if mean value storage (_CHBIT[4]=1) is active.

The results are entered in result field _OVR[ ]. The wear of the length L1 (KNUM=1, _MA=2) of T8, D1 is included if a change is necessary.

Measurement of inside diameter and offset T9
Procedure as described for "Measurement output diameter".

Offset of T8 with appropriate modified values _EV[3], _MV[3] (EVNUM=4), _TUL, _SETVAL.

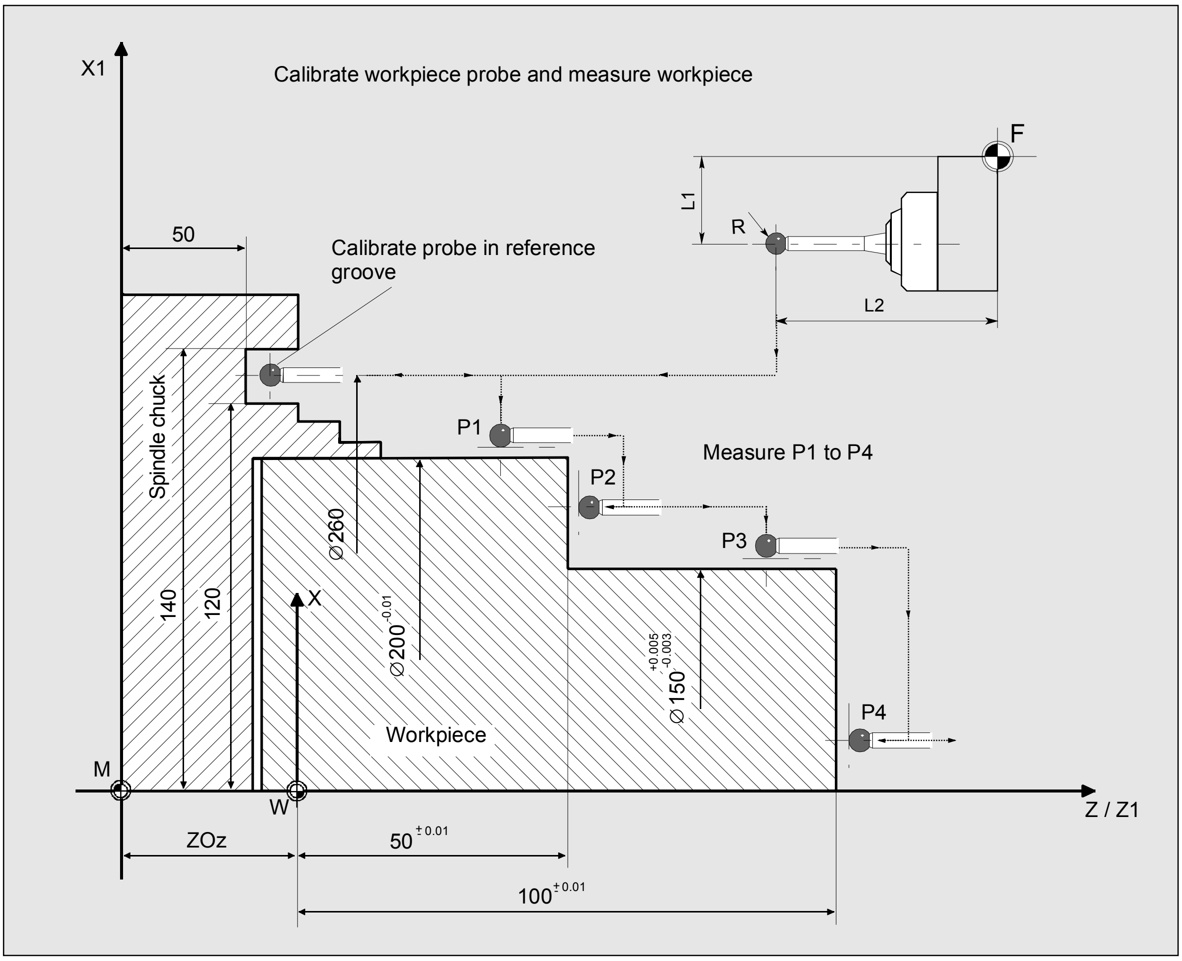
Note
The values of the workpiece tolerance parameters _TUL, _TLL were selected asymmetrically in the example. The result is then made symmetrical (see Subsection 2.3.11).

© Siemens AG, 2004. All rights reserved

6.7 Complex example for workpiece measurement

Explanation

The workpiece shown in the figure is to be measured with workpiece probe 1 with cutting edge position 7, inserted as tool T8, D1, in **CYCLE974**.
This tool master is previously calibrated with **CYCLE973** in reference groove 1 in both axes in the negative direction.
Measuring points P1 to P4 were machined with different tools T1, D1 to T4, D1. These tools are to be automatically corrected in the length (according to measuring axis _MA) without empirical and mean values.



Programming example

Calibration with workpiece probe CYCLE973, measurement of workpiece with CYCLE974

%_N_PART_1_MEAS_MPF	
N10 T8 D1 DIAMON	;Select tool = probe
N20 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
N30 _MVAR=13 _MA=1 _MD=1 _CALNUM=1 _TZL=0 _TSA=1 _PRNUM=1 _VMS=0 _NMSP=1 _FA=1	;Parameters for calibration in ref. groove
N40 CYCLE973	;Calibrate probe in the minus Z direction
N50 _MA=2	;Another measuring axis
N60 CYCLE973	;Calibrate probe in the minus X direction
N70 G54 G0 Z40	;Select zero offset, traverse to measuring ;point in the Z axis
N80 X220	;Position probe facing P 1
N100 _TUL=0 _TLL=-0.01 _TZL=0.002 _EVNUM=0 _TDIF=0.2 _TSA=0.3 _PRNUM=1 _MVAR=0 _SETVAL=200 _MA=2 _TNUM=1 _KNUM=1	;Define parameters for measurement
N110 CYCLE974	;Measure P1
N120 G0 Z70	;Position probe facing P2
N130 X175	
N140 _MA=1 _SETVAL=50 _TUL=0.01 _TNUM=2 _KNUM=1	;Define parameters for measurement in ;another axis
N150 CYCLE974	;Measure P2
N160 G0 Z180	;Position probe opposite P3
N170 _MA=2 _SETVAL=150 _TUL=0.005 _TLL=-0.003 _TNUM=3 _KNUM=1	;Change parameters for measurement
N180 CYCLE974	;Measure P3
N190 G0 Z150	;Position probe opposite P4
N200 X50	
N210 _MA=1 _SETVAL=100 _TUL=0.01 _TLL=-0.01 _TNUM=4 _KNUM=1	;Change parameters for measurement
N220 CYCLE974	;Measure P4
N230 G0 SUPA Z250	;Retraction in Z
N240 SUPA X280	;Retraction in X
N300 M2	;End of program

■

© Siemens AG, 2004. All rights reserved

Miscellaneous Functions

© Siemens AG, 2004. All rights reserved

7.1 Logging the measurement results

The measuring cycles support measuring cycle logging into a file on the control.
There are no special hardware requirements for logging measurement results. It is executed solely by the software.

7.1.1 Overview of the logging cycles

CYCLE100	Log ON
CYCLE101	Log OFF
CYCLE105	Generate log contents
CYCLE106	Log sequential controller
CYCLE113	Internal subroutine: Log
CYCLE118	Format real values

7.1.2 Log file

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. 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 limited in MD 11420: LEN_PROTOCOL_FILE. 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 following message is output:
"New log file has been created".

In this way, up to 9 subsequent logs can be stored in the control. After the 10th log operation is halted and the following message is output:
"Please specify new log name".

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.

© Siemens AG, 2004. All rights reserved

7.1.3 Handling the 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 can be exported from the part program memory to diskette (MMC102/103 and HMI Advanced only) or via RS-232-C.

Sequence

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 7.1.5 to 7.1.7.
Logging is executed by CYCLE105,
CYCLE106, CYCLE113, and CYCLE118. These cycles are called up in conjunction with 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 be called separately, if necessary.

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.

CYCLE101 Log OFF

Disables the logging function and resets the internal flag.

CYCLE105(PAR1) Generate log contents

- This cycle generates up to 4 lines of log contents (lines of values) according to the entries in the variables of the data block (GUD6). It allows you to generate only value lines or only the log header depending on the setting for PAR1.

© Siemens AG, 2004. All rights reserved

Parameters

PAR1	INTEGER	0	Output value block
		1	Output header

CYCLE106(PAR1)
Log sequential controller

- This cycle controls how logging is executed.

Parameters

PAR1	INTEGER	1	Output header
		2	Output value block

The CYCLE106 cycle is called by CYCLE100 automatically when the log is activated. CYCLE106 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(PAR1,PAR2)
Read time and date from system

Parameters

PAR1	INTEGER	1	Read date and return in PAR2
		2	Read time and return in PAR2
PAR2	STRING[10]		Specify the length with the STRING type

CYCLE118(PAR1,PAR2,PAR3,PAR4,PAR5)

The CYCLE118 cycle formats real numbers (data type REAL) for a set number of decimal places.
It can be called to format just one number of to format several numbers.

Formatting one number:

- The input and output values are specified directly in the first 3 parameters passed on cycle call, i.e. the number to be formatted (PAR1), the number of digits (PAR2), and the name of a string variable for the formatted string that is returned.
- For the return value, up to 12 characters are provided, i.e. the return variable is of type STRING[12].

© Siemens AG, 2004. All rights reserved

Formatting more than one number:

- Up to 10 numbers can be formatted; the actual number of numbers to be formatted is passed to the cycle in PAR4.
- The cycle takes the numbers from consecutive R parameters, .e.g. R11 to R20.
- It is possible to start with any R parameter. It is passed to the cycle on calling (PAR5).
- The number of decimal places is specified in the same way as for formatting one number (PAR2).
- The return values are in the predefined variables **_TXT[0]** up to **_TXT[9]** (variable field data block GUD6).

Parameters

PAR1	REAL		Number to be formatted
PAR2	INTEGER	≥0	Number of decimal places
PAR3	STRING[12]		Formatted return as STRING
PAR4	INTEGER	≥1	Number of numbers to be formatted
PAR5	INTEGER	≥0	Number of the first R parameter only relevant if PAR4>1

Programming example

Example 1: Formatting a single number

The value of **one** variable of type REAL is to be formatted for 2 decimal places and displayed:

```
DEF STRING[12] TEXTVAR
DEF REAL VAR1
VAR1=100/$PI                              ;$PI - circle constant PI = 3.1415927
CYCLE118(VAR1,2,TEXTVAR)
MSG("VAR1:"<<VAR1<<" TEXTVAR="<<TEXTVAR)
M0                                        ;Stop – view value: " TEXTVAR=31.83"
M30
```

Example 2: Formatting three numbers

The intention is to format the values of R parameters **R11 to R13** for the decimal places in _DIGIT and display them.

_DIGIT is a variable in data block GUD6 for setting the decimal places in the measuring cycles. The value in the default setting is _DIGIT=3.

$PI – PI = 3.1415927. The results in STRING format are supplied in _TXT[0] to _TXT[2]; the value for the last parameter (R13) also in TEXTVAR.

© Siemens AG, 2004. All rights reserved

`DEF STRING[12] TEXTVAR`	
`R11=0.1/$PI R12=1/$PI R13=10/$PI`	
`CYCLE118(0,_DIGIT,TEXTVAR,3,11)`	
`MSG(‟_TXT[0]=‟<<_TXT[0]<<‟ _TXT[1]=‟_TXT[1]<<‟ _TXT[2]=‟<<TXT[2])`	
`M0`	;Stop – view values: ;‟_TXT[0]=0.032 _TXT[1]=0.318 _TXT[2]=3.183‟
`M30`	

7.1.4 Variables when logging

Function

With these parameters you can

- select the content of the log,
- format the log,
- determine the content of the log header.

In the measuring cycle, data logging is controlled via the following data bit:

_CBIT[6]=	0	With measuring cycle name and measuring variant
	1	Without measuring cycle name and measuring variant
_CBIT[11]=	0	Standard log header
	1	User-defined log header

The following variables (data block GUD6) describe the contents of the measuring log:

Variable	*Type*	*Default value*	*Table of Contents*
_PROTNAME[2]	STRING[32]	Blank string	_PROTNAME[0] = Name of the main program from which the log is written
			_PROTNAME[1] = Name of the log file
_HEADLINE[10]	STRING[80]	Blank string	_HEADLINE[0] ... _HEADLINE[9] The user can enter customized texts in these STRINGs; they are included in the log
Variable	*Type*	*Default value*	*Table of Contents*
_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
		3	_PROTFORM[3] = Number of customized header lines
		1	_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]		_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
_TXT[100]	STRING[12]		Field for formatted STRINGs

© Siemens AG, 2004. All rights reserved

7.1.5 Selecting log content

Function

The measurement result log contains parts that are fixed and some that can be set. It always contains:

- Measuring cycle (cycle name)
- Measuring variant (value of `_MVAR`)

Output of the measuring cycle and measuring variant can be suppressed by setting _CBIT[6].

The following additional data can be included in a log:

- Time (specification `_TIME`)
- Axis of the associated measuring axes
  - Specification _AXIS:
    The axis name is entered automatically according to the measuring axis entered in _MA.
  - or specification _AXIS1... 3:
    _AXIS1: Axis name of the abscissa in the selected plane
    _AXIS2: Axis name of the ordinate in the selected plane
    _AXIS3: Axis name of the applicate in in the selected plane
- All result data provided by the measuring cycle in the `_OVR[ ]` array.
- R parameters
- Comment texts and
- STRINGs stored in _TXT[ ] (GUD6)

The logging values to be selected must correspond to the measuring cycle and the selected measuring variant. This makes for versatile adaptation of the contents of the log to meet your requirements.

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 7.1.9, Lines 8 - 10).
`_PROTVAL[2]...[5]` specify the line contents of the individual log lines.

© Siemens AG, 2004. All rights reserved

If you change the measuring cycle or the measuring variant, you may have to adapt `_PROTVAL[2]...[5]` (see example in Subsection 7.1.9).

The comma is used as the separator between variables.

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], Metr”

R27 is just an example of an R parameter. 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.

© Siemens AG, 2004. All rights reserved

7.1.6 Log format

Programming

The following values can be specified for the log format:

_PROTFORM[0]	INTEGER	Number of lines per page including log header
_PROTFORM[1]	INTEGER	Number of characters per line
_PROTFORM[2]	INTEGER	First page number
_PROTFORM[3]	INTEGER	Number of customized log header lines
_PROTFORM[4]	INTEGER	Number of value lines in the log
_PROTFORM[5]	INTEGER	Column width/variable column width
_PROTSYM[0]	CHAR	Separators between the values in the log
_PROTSYM[1]	CHAR	Special characters for identification when tolerance limits are exceeded
_DIGIT	INTEGER	Number of decimal places

Explanation

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.

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. 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.

The number of decimal places can be set via the variable _DIGIT in GUD6 data block (display precision).

© Siemens AG, 2004. All rights reserved

7.1.7 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.

The selection is made via measuring cycle data bit `_CBIT[11]`.
`_CBIT[11]=0`: Default log header
`_CBIT[11]=1`: User-defined log header

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

User-defined log header
The content of the string array `_HEADLINE[ ]` is entered in line 1 ff. The number of header lines can be defined by the user (according to the length of the `_HEADLINE` array).

Default log header
The default log header contains fixed and variable components.
All variable components are in bold formatting, that is:

Line 1	Number of pages,
Line 3	Program name,
Line 5, 6, 7	(`_HEADLINE[0-2]`) ff. and
Line 9	(`_PROTVAL[0]`)
Line 10	(`_PROTVAL[1]`)

Line 1 is fixed and predefined.

© Siemens AG, 2004. All rights reserved

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	**Processed by: Müller**	**Phone: 1234**				
Line 8	--					
Line 9	**Meas.**	**, 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: Müller              Phone: 1234"
_PROTVAL[0]="Meas.   , Axis   , Set    , Act. value, Difference  , Time"
_PROTVAL[1]="point   ,        , value
```

7.1.8 Example: Creating a measurement result log

Log view

%N_LOG_1_MPF

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	**Processed by: Müller**	**Phone: 1234**				
Line 8	--					
Line 9	**Meas.**	**, 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**	

© Siemens AG, 2004. All rights reserved

Programming

The log with the default log header shown above is created using the following program. The example shows the user how to handle the log.

%_N_MEASPROGRAM_1_MPF	
;$PATH=/_N_MPF_DIR	
;Measure shaft with measuring log	
DEF INT PARTNUM, JOBNUM, MP_COUNTER	
; ----------- Set parameters for log -----------------	
_CBIT[11]=0	;Log with default header
; ------------------------- Log header ------------------------	
PARTNUM=123456789 JOBNUM=6878 _PROTNAME[0]="MEASPROGRAM_1"	;Name of calling program
_PROTNAME[1]="LOG_1"	;Name of log file
_HEADLINE[0]="Part number: "<<PARTNUM	
_HEADLINE[1]="Job number:"<<JOBNUM	
_HEADLINE[2]="Supervisor: Müller Phone: 1234"	
; --------------------- Log format ----------------------	Formats: Default values from
_PROTSYM[0]="," _PROTSYM[1]="*"	;Define separators and special
_PROTFORM[0]=60	;60 lines per page
_PROTFORM[1]=80	;80 characters per line
_PROTFORM[2]=1	;Start of page 1
_PROTFORM[3]=3	;Three customized log header
_PROTFORM[4]=1	;One value line
_PROTFORM[5]=12	;12 characters per line
; --------------------- Log contents ----------------------	
; Header lines	
_PROTVAL[0]="Meas. , Axis , Set , Act. value, Difference , Time"	
_PROTVAL[1]="point , , value"	
; ---------------- Other value assignments -----------------	
MP_COUNTER=1 _TXT[0]=<<MP_COUNTER	;Assign counter for measurement log
; -------- Perform measurements with log -------------	
N100 G0 G17 G90 T3 D1 Z100 F1000	;Approach start position for
N110 X70 Y90	
.	
_MVAR=100 _SETVAL=80 _MA=3 _TSA=2 _FA=2	;Set measuring cycle
...	;Measuring variant: Measure
; Contents of the value lines	
_PROTVAL[2]="_TXT[0],_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
...	;Measuring variant: Measure shaft
_PROTFORM[4]=2	;two value lines
_PROTVAL[2]="_TXT[0],_AXIS1,_OVR[1],_OVR[5],_OVR[17],_TIME"	
_PROTVAL[3]=" ,_AXIS2,_OVR[2],_OVR[6],_OVR[18]"	

© Siemens AG, 2004. All rights reserved

MP_COUNTER=MP_COUNTER+1 _TXT[0]=<<MP_COUNTER	;Increase user-def. counter for measurements
N190 CYCLE977	;Measure shaft
N210 CYCLE101	;Deactivate log
N220 Z200	;Retraction in Z
N290 M2	

© Siemens AG, 2004. All rights reserved

7.2 Measuring cycle support in the program editor (up to measuring cycles SW 5.4)

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 softkeys. The softkey menu is divided up according to measuring tasks, e.g. "Calibration" and then "Calibration in hole" or "Tool probe". This results in a one-to-one assignment between softkeys and measuring cycles.

As from SW 5 of the MMC100/100.2, MMC102/103 and HMI Advanced/Embedded, measuring cycle support will be accessible via softkeys

[>] → [Support] → [New cycle] → [Measuring cycles]

from the extended menu of the editor.

In the edited program there are calls with parameter list,

e.g. CYCLE_PARA(...) for additional parameter assignment

CYCLE_976(...) for calibrating in a hole,

CYCLE_CAL_TOOLSETTER(...) for calibrating the tool probe.

© Siemens AG, 2004. All rights reserved

7.2.1 Files of the measuring cycle support

Function

Measuring cycle support requires the following files:

- COV.COM
  Configuring the softkeys for cycle selection
- SC.COM
  Configuring the input screens for the individual parameters
- Auxiliary cycle*.spf
  Additional cycles with parameter list that transfer the input parameters to the data blocks (GUD) of the measuring cycles 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 measuring 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 7.2.3) must be transferred to the NCU with "Load".

The Power ON is executed.

© Siemens AG, 2004. All rights reserved

7.2.3 Assigning 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
Auxiliary parameters	-	CYCLE_PARA(...)
Calibrating tool probes	CYCLE971, CYCLE972, CYCLE982	CYCLE_CAL_TOOLSETTER(...)
Calibrate 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 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, specifying angles	CYCLE961	CYCLE_961_W
Measure corner, specifying points	CYCLE961	CYCLE_961_P
Single-point measurement turning	CYCLE974	CYCLE_974(...)
Two-point measurement	CYCLE994	CYCLE_994(...)

© Siemens AG, 2004. All rights reserved

7.2.4 Description of the parameterizing 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 screen form for particular functions.

Setting additional parameters – CYCLE_PARA

With CYCLE_PARA, you can program all generally valid parameters of the measuring cycles. These parameters do not depend on the individual measuring variant. If an entered parameter has the value 0, the target parameter is not overwritten, i.e. its previous value is retained.

Parameters

_FA	REAL	>0	Measurement path in mm
_VMS	REAL	≥0	Variable measuring speed
_NMSP	INTEGER	>0	Number of measurements at the same location
_RF	REAL	>0	**CYCLE979 only:** Feedrate in circular-path programming
_PRNUM	INTEGER	>0	Probe number
_CORA	REAL		**Only if mono probe is used:** Offset angle position
_TZL	REAL	≥0	Tolerance range for zero offset
_TMV	REAL	≥0	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	>0	Safe area for measuring result
_EVNUM	INTEGER	≥0	Number of empirical value memory that is calculated
_K	INTEGER	≥1	Weighting factor for mean value calculation
_TDIF	REAL	>0	Tolerance range for dimensional difference check

© Siemens AG, 2004. All rights reserved

Calibrating a tool probe – CYCLE_CAL_TOOLSETTER

With CYCLE_CAL_TOOLSETTER
softkeys [Measurem. milling] → [Calibrat. TL probe] (CYCLE971)
or
softkeys [Measurem. turning] → [Calibrat. TL probe] (CYCLE972)
measuring cycles CYCLE971 and CYCLE972 can be parameterized to calibrate a tool measuring probe.

Parameters

	INTEGER		Selection: Cycle number 971… for CYCLE971 (milling machine), 972... for CYCLE972 (turning machine)
_MA	INTEGER	>0	Number of measuring axis and for CYCLE972 also the offset axis
_PRNUM	INTEGER	>0	Probe number
	INTEGER		**For CYCLE971 only** Selection: Measuring variant 0...absolute calibration/1...incremental calibration
_FA	REAL	>0	Measurement path

Calibration on surface – CYCLE_CAL_PROBE

With CYCLE_CAL_PROBE
softkeys [Measurem. turning] → [Probe calibrat.] → [Plane]
or
softkeys [Measurem. milling] → [Probe calibrat.] → [Plane]
measuring cycles CYCLE973 and CYCLE976 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	1...3	Number of the measuring axis
_MD	INTEGER	0, 1	Measuring direction
_PRNUM	INTEGER	>0	Probe number
_MVAR	INTEGER	≥0	Selection: Measuring variant (for CYCLE976 only) 0: Calibration on any surface 10000: Calibrating in the 3rd axis with calculation of probe length

© Siemens AG, 2004. All rights reserved

Calibration in groove – CYCLE_973

With CYCLE_973

softkeys [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	>0	Number of measuring axis 1, 2
_MD	INTEGER	≥0	Determines measuring direction 0…in positive direction / 1...in negative direction
_CALNUM	INTEGER	≥0	Selection of calibration groove with number
_PRNUM	INTEGER	>0	Probe number

Calibration in hole – CYCLE_976

With CYCLE_976

softkeys [Measurem. milling] → [Probe calibrat.]

CYCLE976 can be parameterized to calibrate a 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	>0	Number of measuring axis 1, 2
_MD	INTEGER	≥0	Determines measuring direction 0…in positive direction / 1...in negative direction
_STA1	REAL		Angle (for calibration at an angle only)
_PRNUM	INTEGER	>0	Probe number
			Selection: Hole type 0…hole center known / 1...unknown

© Siemens AG, 2004. All rights reserved

Measuring milling tools – CYCLE_971

With CYCLE_971

softkeys Measurem. milling → Tool measurem.

CYCLE971 can be parameterized for tool measurement.

Parameters

_MVAR	INTEGER	>0	Measuring variant
_MA	INTEGER	>0	Number of the measuring axis
_ID	REAL	>0	Offset
_PRNUM	INTEGER	>0	Probe number
_MFS[0]	REAL	>0	Feed 1st probing (only with _CBIT[12]=1)
_MFS[1]	REAL	>0	Speed 1st probing (only with _CBIT[12]=1)
_MFS[2]	REAL	≥0	Feed 1st probing (only with _CBIT[12]=1)
_MFS[3]	REAL	≥0	Speed 1st probing (only with _CBIT[12]=1)
_MFS[4]	REAL	≥0	Feed 1st probing (only with _CBIT[12]=1)
_MFS[5]	REAL	≥0	Speed 1st probing (only with _CBIT[12]=1)

Measuring turning tools – CYCLE_972

CYCLE_972 can be used to parameterize CYCLE976 to gauge tools.

Parameters

_MA	INTEGER	>0	Number of the measuring axis
_PRNUM	INTEGER	>0	Probe number

© Siemens AG, 2004. All rights reserved

Measure hole, shaft – CYCLE_977_979A

With CYCLE_977_979A softkeys [Measurem. milling] → [Workpiece measure] → [Hole Shaft] measuring 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 >0	Measuring variant
_SETVAL	REAL	Setpoint
_ID	REAL	Infeed path
_SZA	REAL	Safety zone (for paraxial measurement only)
_TNUM	INTEGER ≥0	Tool number for automatic offset
_TNAME	STRING	Tool name with active tool management
_KNUM	INTEGER ≥0	Offset number D number for measurement/ZO number for calculating zero offset
_CPA	REAL	Center point hole, shaft 1st axis
_CPO	REAL	Center point hole, shaft 2nd axis
_STA1	REAL	Start angle
_INCA	REAL	Indexing angle
_PRNUM	INTEGER >0	Measuring probe number **CYCLE979 only**: The number of measuring points is assigned from the thousands digit; 0…3 measuring points, 1…4 measuring points

Measure groove, web – CYCLE_977_979B

With CYCLE_977_979B softkeys [Measurem. milling] → [Workpiece measure] → [Slot or web] measuring 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 >0	Measuring variant
_SETVAL	REAL	Setpoint
_ID	REAL	Infeed path
_MA	INTEGER >0	Number of measuring axis 1, 2
_TNUM	INTEGER ≥0	Tool number for automatic offset

© Siemens AG, 2004. All rights reserved

_TNAME	STRING	Tool name with active tool management
_KNUM	INTEGER ≥0	Offset number D number for measurement/ZO number for calculating zero offset
_CPA	REAL	Center groove, web 1st axis
_CPO	REAL	Center groove, web 2nd axis
_STA1	REAL	Start angle
_SZA	REAL	Safe area (for paraxial measurement only)
_PRNUM	INTEGER >0	Probe number

Measure rectangle – CYCLE_977_979C

With CYCLE_977_979C softkeys [Measurem. milling] → [Workpiece measure] → [>>] → [Rectangle] measuring variants xxx5 and xxx6 of measuring cycle CYCLE977 can be parameterized.

Parameters

_MVAR	INTEGER >0	Measuring 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	INTEGER ≥0	Tool number for automatic offset
_TNAME	STRING	Tool name with active tool management
_KNUM	INTEGER ≥0	Offset number D number for measurement/ZO number for calculating zero offset

© Siemens AG, 2004. All rights reserved

Single-point measurement – CYCLE_978

With CYCLE_978
softkeys [Measurem. milling] → [Workpiece measure] → [Plane]
CYCLE978 can be parameterized.

Parameters

_MVAR	INTEGER ≥0	Measuring variant
_SETVAL	REAL	Setpoint
_MA	INTEGER >0	Measuring axis 1, 2, 3
_TNUM	INTEGER ≥0	Tool number for automatic offset
_TNAME	STRING	Tool name with active tool management
_KNUM	INTEGER ≥0	Offset number D number for measurement/ZO number for calculating zero offset

Angle measurement – CYCLE_998

With CYCLE_998
softkeys [Measurem. milling] → [Workpiece measure] → [Angle]
CYCLE998 can be parameterized.

Parameters

_MVAR	INTEGER >0	Measuring variant
_SETVAL	REAL	Setpoint
_ID	REAL	To be controlled
_RA	INTEGER ≥0	Number of rotary axis
_MA	INTEGER >0	Number of measuring axis and offset axis 102...302
_KNUM	INTEGER ≥0	ZO number
_STA1	REAL	Angle
_PRNUM	INTEGER >0	Probe number

© Siemens AG, 2004. All rights reserved

Corner measurement, specifying angles – CYCLE_961_W

With CYCLE_961_W

softkeys [Measurem. milling] → [Workpiece measure] → [Corner]

measuring variants 105 to 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	>0	Distance between starting point and measuring point 2, without sign
_SETV[1]	REAL	>0	Distance between starting point and measuring point 4, without sign
_ID	REAL	>0	Retraction path in 3rd axis (applicate), for outside corner only, without sign
_STA1	REAL		Enter approx. angle between 1st axis (abscissa) and 1st edge, in clockwise direction with negative sign
_INCA	REAL	<>0	Angle between 1st and 2nd edge of workpiece, in clockwise direction with negative sign
_KNUM	INTEGER	≥0	ZO number
_SETV[4]	REAL		**For 3 measuring points only:** Selection: Override 1…measured corner entered as zero point 2…measured corner is entered in 1st axis offset by the value in _SETV[3] and as a zero point 3…measured corner is offset in both axes and entered as zero point 4…measured corner is offset in 2nd axis by the value from _SETV[3] and entered as 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	>0	Probe number

© Siemens AG, 2004. All rights reserved

Corner measurement, specifying angles – CYCLE_961_P

With CYCLE_961_P

softkeys [Measurem. milling] → [Workpiece measure] → [Corner]

measuring variants 117 to 118 for CYCLE961 can be parameterized.

Parameters

	INTEGER		Selection: Outside or inside corner 0…inside corner / 1...outside corner
_ID	REAL	>0	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	≥0	ZO number
_PRNUM	INTEGER	>0	Probe number

Single-point measurement – CYCLE_974

With CYCLE_974

softkeys [Measurem. turning] → [Workpiece measure] → [1-point measurem.]

CYCLE974 can be parameterized.

Parameters

_MVAR	INTEGER	≥0	Measuring variant
_SETVAL	REAL		Setpoint
_MA	INTEGER	>0	Number of measuring axis 1, 2, 3
_TNUM	INTEGER	≥0	Tool number for automatic offset
_TNAME	STRING		Tool name with active tool management
_KNUM	INTEGER	>0	Offset number D number for measurement/ZO number for calculating zero offset
_PRNUM	INTEGER	>0	Probe number
_STA1	REAL		Start angle

© Siemens AG, 2004. All rights reserved

Two-point measurement – CYCLE_994

With CYCLE_994

softkeys Measurem. turning → Workpiece measure → Two-point measurem.

CYCLE994 can be parameterized.

Parameters

_MVAR	INTEGER 1, 2	Measuring variant
_SETVAL	REAL	Setpoint
_MA	INTEGER >0	Number of measuring axis 1, 2, 3
_TNUM	INTEGER ≥0	Tool number for automatic offset
_TNAME	STRING	Tool name with active tool management
_KNUM	INTEGER ≥0	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 >0	Probe number

© Siemens AG, 2004. All rights reserved

7.3 Measuring cycle support in the program editor (from measuring cycles SW 6.2)

From measuring cycles SW 6.2, the program editor provides extended measuring cycle support for inserting measuring cycle calls into the program.

Requirement:
HMI Advanced/Embedded as from SW 6.2 required.

Function

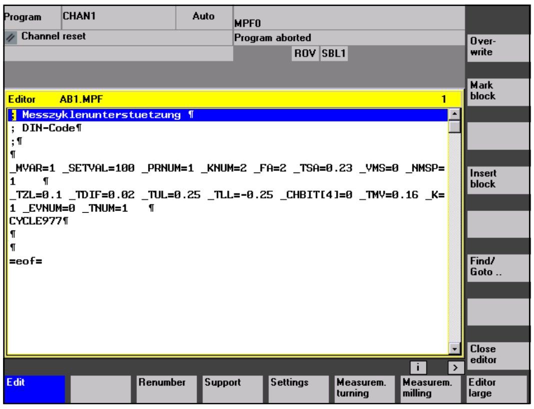
This measuring cycle support provides the following functionality:

- Measuring cycle selection via softkeys
- Input screen forms for parameter assignment with help displays
- Retranslatable code is generated from the individual screen forms.

7.3.1 Menus, explanation of the cycles

Explanation

The input screens for the measuring cycles are selected depending on the technology being used via horizontal softkeys.



Measurem. turning

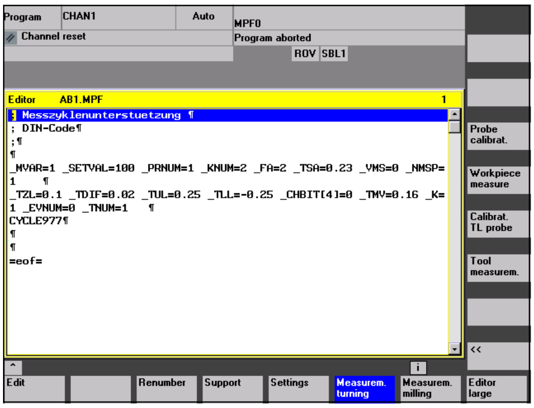
Input screen forms for measuring cycles for turning technology.

Measurem. milling

Input screens for measuring cycles for milling technology.

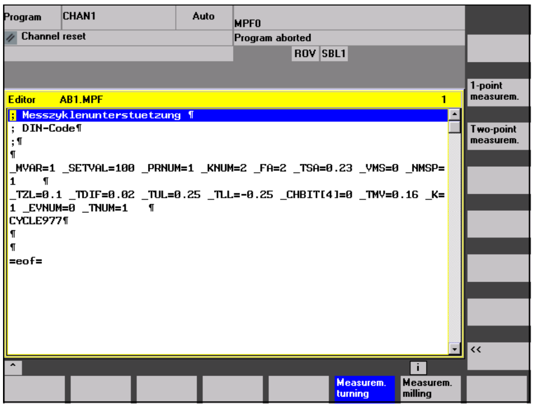
© Siemens AG, 2004. All rights reserved

Vertical softkey menu for turning technology



Probe calibrat.	Call screen form for CYCLE973 Calibrate workpiece probe for turning machines.
Workpiece measure	Call new vertical softkey 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
<<	Back

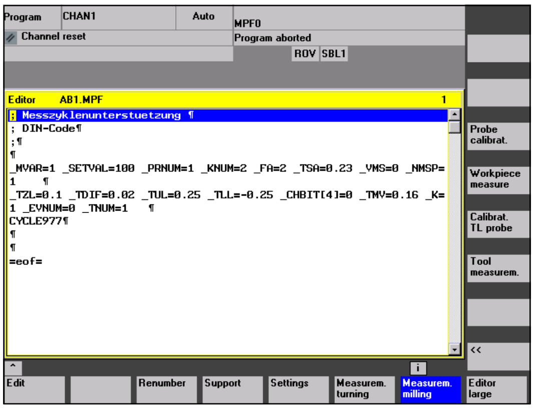
Vertical softkey menu for measure workpiece, turning



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.
<<	Back to selection menu turning.

© Siemens AG, 2004. All rights reserved

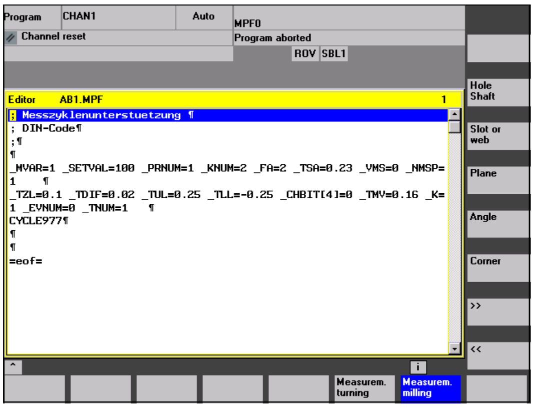
Vertical softkey menu for milling technology



Probe calibrat.	Call screen form for CYCLE976 Calibrate workpiece probe for milling machines.
Workpiece measure	Call new vertical softkey menu for selection "measure workpiece".
Calibrat. TL probe	Call screen form for CYCLE971 Calibrate tool probe for milling machines.
Tool measurem.	Call screen form for CYCLE971 Gauge milling tools on milling machines.
<<	Back

© Siemens AG, 2004. All rights reserved

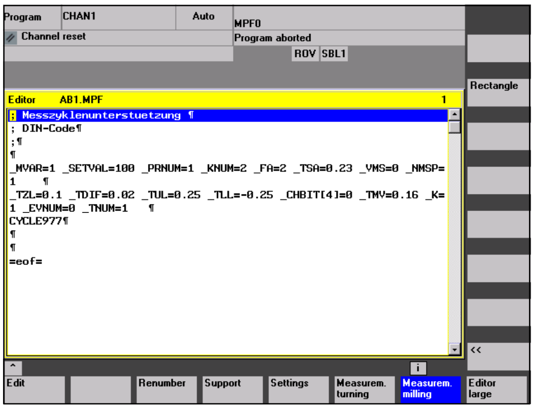
Vertical softkey menu for workpiece measurement milling



Hole Shaft	Call screen form for workpiece measurement for milling machines CYCLE977/CYCLE979 hole/shaft. Hole/shaft and paraxial/at an angle switchover are performed in the screen form.
Slot or web	Call screen form for workpiece measurement for milling machines CYCLE977/CYCLE979 groove/web. Slot/web and paraxial/at an angle switchover 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.
<<	Back to selection menu milling.

© Siemens AG, 2004. All rights reserved

Vertical advancement menu for workpiece measurement milling



Rectangle	Call screen form for workpiece measurement for milling machines CYCLE977 rectangle internal/external.
Ball	Call screen form for workpiece measurement for milling machines CYCLE997 measure ball and ZO determination (from measuring cycles SW 6.3).
<<	Back to selection list measure workpiece milling.

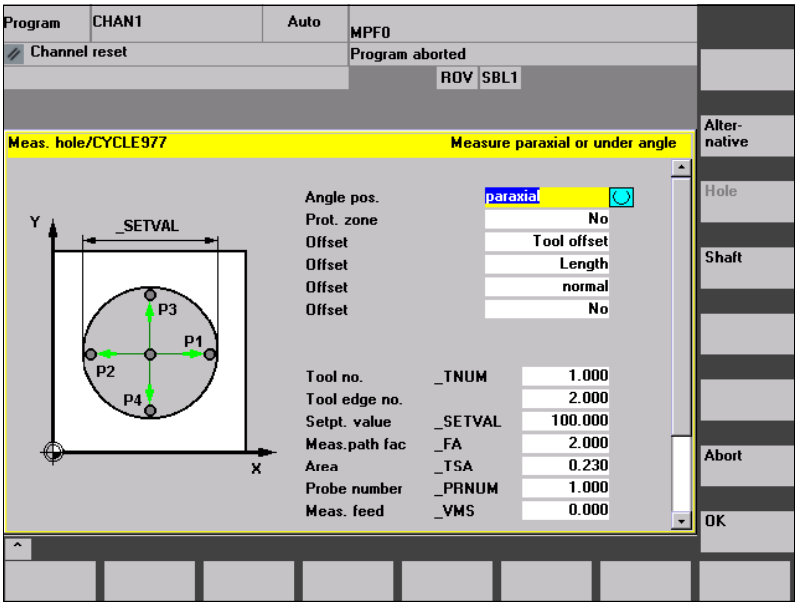
Programming example

Measuring a hole parallel to the axis with protection zone

(generated with measuring cycle support)

```
N100 G17 G0 G90 Z20 F2000 S500 M3                ;Main block
N110 T7 M6                                       ;Insert probe
N120 G17 G0 G90 X50 Y50                          ;Position probe in X/Y plane
                                                 ;on hole center point
N130 Z20 D1                                      ;Position Z axis in hole
; from the measuring cycle support using the
; NC code _MZ_MASK[0]=1 generated using the
; input screen
N130 _MVAR=1001 _SETVAL=100.000 _PRNUM=101       ;Parameter passing to
     _KNUM=2002 _FA=2 _TSA=0.23                  ;measuring cycle
     _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
     CYCLE977                                    ;Call measuring cycle
;* end of NC code generated by measuring cycle ;
;  support
...
N200 M30                                         ;End of program
```

Input screen for measuring a hole, parallel with the axis, with protection zone (CYCLE977).



© Siemens AG, 2004. All rights reserved

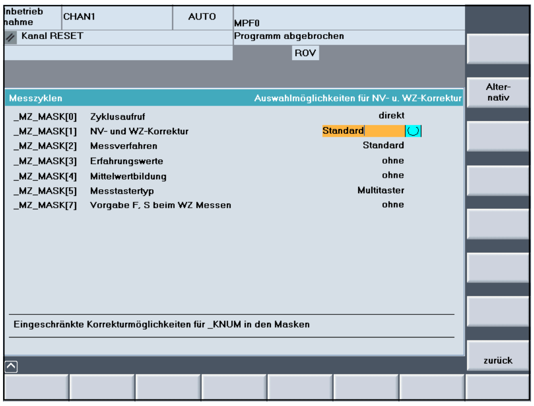
7.3.2 Pre-setting of the measuring cycle support

Explanation

A field _MZ_MASK is declared in the data block (GUD6) in which the screen forms can be adapted:

- technological measuring conditions
- to measuring variants

The settings in the data block for the _MZ_MASK field can be changed in a screen form in "Setup" operating area.



Variable	*Value*	*Default*	*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	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	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-specific) 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

© Siemens AG, 2004. All rights reserved

Variable	*Value*	*Default*	*Meaning*
_MZ_MASK[2]	0	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 input fields for parameters: • _VMS: Measuring velocity • _NMSP: Number of measurements at the same location
_MZ_MASK[3]	0	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 an input field for the following parameters: • _EVNUM: Number of empirical value memory
_MZ_MASK[4]	0	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 calculation • 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 with 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 calculation • _EVNUM: mean value memory number The following is also entered in the NC code: • _CHBIT[4]=1
_MZ_MASK[5]	0	0	Probe type for workpiece measurement is a multi probe
	1	–	Probe type for workpiece measurement is a mono probe The relevant screen forms show an input field for the offset angle _CORA.
_MZ_MASK[6]			Reserved

© Siemens AG, 2004. All rights reserved

Variable	*Value*	*Default*	*Meaning*
_MZ_MASK[7]	0	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. The following is also entered in the NC code: • _CBIT[12]=0
	1	–	Screen form for CYCLE971 – tool measurement/milling contains input fields for feedrate and spindle speed. The following is also entered in the NC code: • _CBIT[12]=1

References: /BEM/, HMI Embedded Operator's Guide
/IAM/, Installation and Start-Up Guide HMI/MMC
IM2 "Installation and Start-up of HMI Embedded"

Retranslation

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.

© Siemens AG, 2004. All rights reserved

7.4 Measuring result screens

Function

Measurement result displays will be shown automatically during measuring cycle runtime if _CHBIT[10]=1. If _CHBIT[10]=0 (default setting), the measurement result displays are now shown.

Depending on the setting in _CHBIT[11] and _CHBIT[18]

- the measurement result displays are automatically deselected at the end of measuring cycle (_CHBIT[11]=0, _CHBIT[18]=0)
  or
- the measurement result displays must be acknowledged with the NC start key (_CHBIT[11]=1, _CHBIT[18]=0). In this case, the measuring cycle outputs the message: **"Please acknowledge meas. result display with NC start"**
  or
- The measurement result displays are retained until the next measuring cycle call (_CHBIT[11]=0, _CHBIT[18]=1).

Explanation

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

- Calibrating tool probes
- Tool measurement
- Calibrating workpiece probes
- Workpiece measurement

The result displays contain the following data:

Calibrating tool probes

- Measuring cycle and measuring variant
- Trigger values of axis directions and differences
- Probe number
- Safe area

© Siemens AG, 2004. All rights reserved

Tool measurement

- Measuring cycle and measuring variant
- Actual values and differences for tool offsets
- Safe area and permissible dimensional difference
- T-, D-number

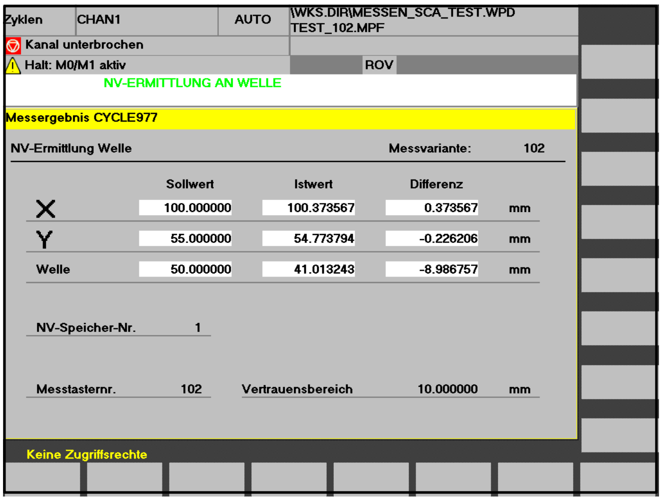
Calibrating workpiece probes

- Measuring cycle and measuring variant
- Trigger values of axis directions and differences
- Positional deviation during calibration in the plane
- Probe number
- Safe area

Workpiece measurement

- Measuring cycle and measuring variant
- Setpoints, actual values and their differences
- Tolerance upper and lower limits (for tool offset)
- Offset value
- Probe number
- Safe area and permissible dimensional difference
- T number, D number, and DL number or ZO memory number for automatic offset

Example of measurement result display



■

Notes

© Siemens AG, 2004. All rights reserved

Part 2 Description of Functions

Hardware, Software and Installation

© Siemens AG, 2004. All rights reserved

8.1 Hardware prerequisites

8.1.1 General hardware prerequisites

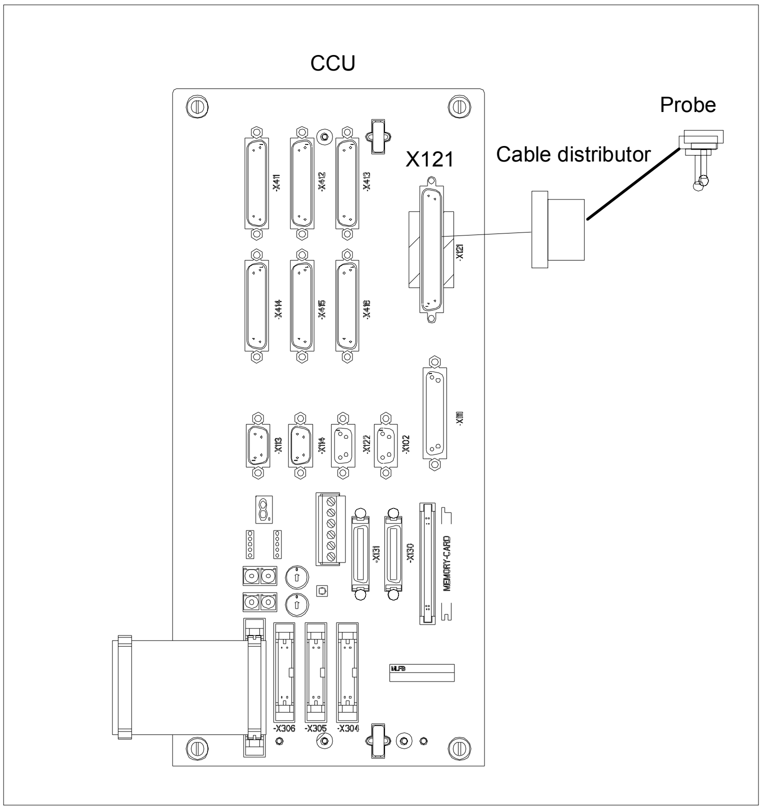
Axis assignment
For correct execution of the measuring cycles the machine axes must be assigned according to DIN 66217.

Probes that can be used
The measuring cycles require connection of switching probes; see the descriptions in Section 1.5.

8.1.2 Measuring probe connection

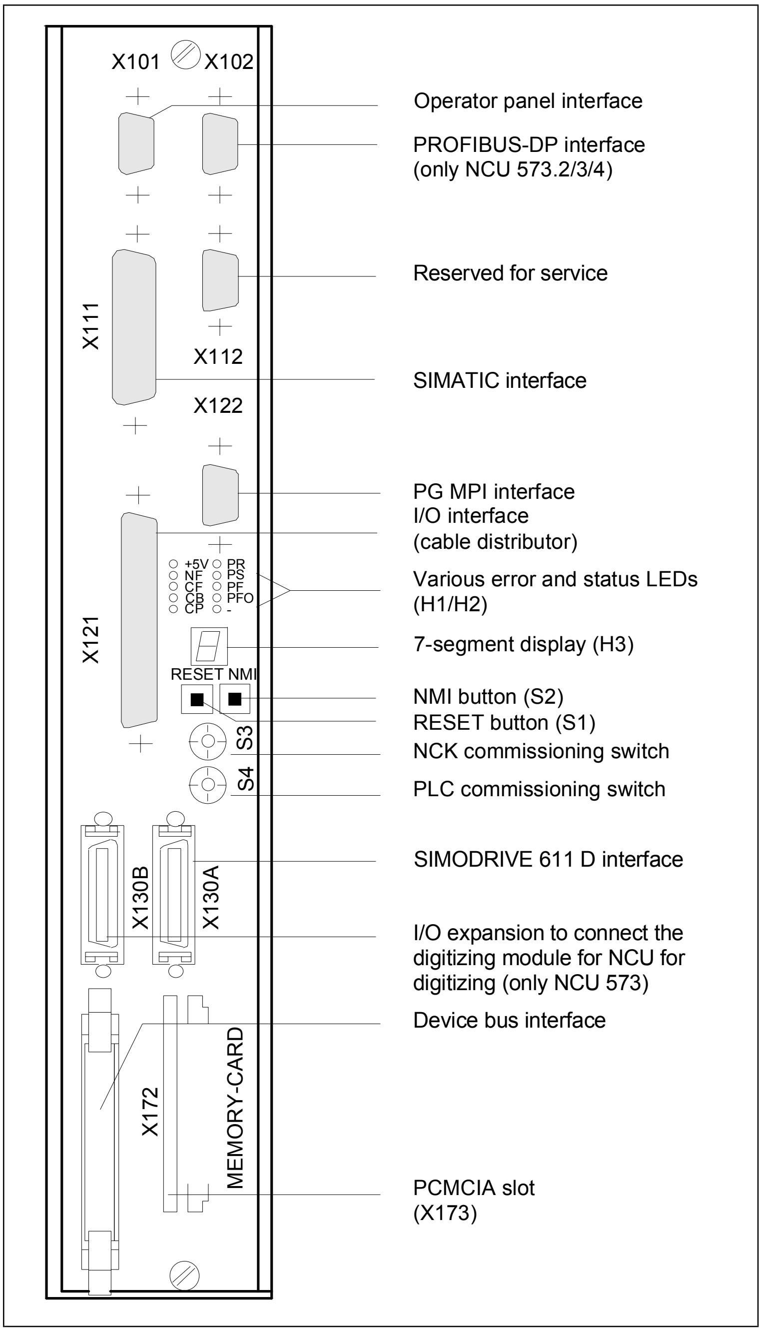
On the SINUMERIK 840D, 840Di and 810D, the probe is connected via the I/O interface X121 which is located on the front panel of the NCU/CCU module.

Example: Probe connection to X121 on CCU module



© Siemens AG, 2004. All rights reserved

Interfaces, control and display elements of NCU module



© Siemens AG, 2004. All rights reserved

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		*Description*
1	M24EXT	External power supply External ground
2	M24EXT	External ground
...	...	...
9	MEPUS 0	Connection of 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 of probe 2 Measuring pulse signal input
29	MEPUC 1	Measuring pulse common input
...	...	...

The interfaces (e.g. pin assignment) are described in detail in

References: /PHD/, Hardware Configuring Guide

8.1.3 Measuring in JOG

Explanation

Measuring in JOG is only possible with

- SINUMERIK 840D
- SINUMERIK 810D
- HMI Advanced (PCU 50) or MMC 103

© Siemens AG, 2004. All rights reserved

8.2 Software prerequisites

How the measuring cycles are supplied

The measuring cycle software is supplied on diskette.

8.2.1 Software versions NC and MMC/HMI

NC software version

The measuring cycles can only be used with NCK SW 6.3 and higher.

MMC SW

The measuring cycles require an MMC/HMI software version as from SW 4.4; see compatibility list below.

PLC program

No adaptation of the PLC user program is required to run the measuring cycles.

Compatibility of measuring cycle versions – NCU/CCU, HMI/MMC

The following table provides an overview of the software versions of the NCU/CCU and HMI/MMC components and their compatibility with the measuring cycles.
The table indicates whether the basic functionality of the measuring cycle version or the subfunction listed can be executed with the corresponding other system component.
Some cycles or functions may have additional software requirements. These are described in the description of the cycle or function.

© Siemens AG, 2004. All rights reserved

	Meas. cycle SW 5.3/5.4	Meas. cycle SW 6.2	Meas. in JOG SW 5.3/5.4	Meas. in JOG SW 6.2	Meas. cycle support SW 6.2
NCU					
SW 4.4	x	n	x	n	No requirements on NCU, pure HMI function
SW 5.3	x	n	x	n	
SW 6.2	x	n	x	n	
SW 6.3	x	x	x	x	
SW 6.4	x	x	x	x	
CCU					
SW 2.4	x	n	x	n	No requirements on NCU, pure HMI function
SW 3.3	x	n	x	n	
SW 4.2	x	n	x	n	
SW 6.3	x	x	x	x	
SW 6.4	x	x	x	x	
HMI Embedded					
SW 6.1	x	x	Measuring in JOG not possible with HMI Embedded		n
SW 6.2	x	x			x
SW 6.3	x	x			x
MMC 100.2					
SW 4.4	x	x	Measuring in JOG not possible with MMC100.2		Not possible with MMC100.2
SW 5.1	x	x			
SW 5.2	x	x			
SW 5.3	x	x			
HMI Advanced					
SW 6.1	x	x	x	x	n
SW 6.2	x	x	x	x	x
SW 6.3	x	x	x	x	x
MMC 103					
SW 4.4	x	x	n	n	Not possible with MMC103
SW 5.1	x	x	n	n	
SW 5.2	x	x	n	n	
SW 5.3	x	x	x	x	

x executable
n not executable

8.2.2 Measuring in JOG

Options

"Measurement in JOG" can only be used if the "Inter-modal actions" option (ASUPs and synchronized actions in all modes) is active.

© Siemens AG, 2004. All rights reserved

8.3 Function test

Function

Measure command
The control has the command MEAS for generating a measuring block.

The measuring input number is set in the command parameters.

Saving measurement results
The results of the measurement command are stored in the system data of the NCK and can be accessed from the program.

They 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: /PG/, Programming Guide

PLC service display
The functional test for the probe is conducted via an NC program.

The measuring signal can be controlled via the diagnostics menu "PLC status".

Status display for measurement signal

Probe 1 deflected	DB10	DB X107.0
Probe 2 deflected	DB10	DB X107.1

© Siemens AG, 2004. All rights reserved

Example of functional check

%_N_TEST_PROBE_MPF

;$PATH=/_N_MPF_DIR		
;Testing program probe connection		
N05	DEF INT MTSIGNAL	;Flag for trigger status
N10	DEF INT ME_NR=1	;Measurement input number
N30	G17 T1 D1	;Select tool offset for probe
N40	G0 G90 X0 F150	;Starting position and meas. velocity
N50	MEAS=ME_NR G1 X100	;Measurement at measuring input 1 ;in the X axis
N60	STOPRE	;Preprocessing stop
N70	MTSIGNAL=$AC_MEA[ME_NR]	;Read software switching signal ;at 1st measurement input
N80	IF MTSIGNAL == 0 GOTOF _ERROR1	;Evaluation of signal
N90	R1=$AA_MW[X]	;Read measured value in workpiece ;coordinates
N95	M0	;View measured value in R1
N100	M2	
N110	_ERROR1: MSG ("Probe not switching!")	;Display "Probe not switching"
N120	M0	
N130	M2	

■

© Siemens AG, 2004. All rights reserved

Data Description

© Siemens AG, 2004. All rights reserved

9.1 Machine data for executing measuring cycles

9.1.1 Machine data that configure the memory

Function

The measuring cycles use their own **GUD** and **LUD variables** (**G**lobal **U**ser **D**ata and **L**ocal **U**ser **D**ata).
The memory provided is configured via machine data in the control. The values must be increased.

The memory requirement for GUD variables arises on loading data blocks GUD5 and GUD6; and if the "**Measurement in JOG**" function is installed, also on loading GUD7 (from SW 6.3: GUD7_MC).
For special data for "measuring in JOG", refer to Section 9.3

LUD variables are created during program and cycle runtime. So the memory space is not used until execution. A sufficient reserve must be available because different amounts of memory are required at different times.

System frames are used in the cycles. The memory space is provided via special machine data settings in the control and set by default.

Notice
The following information in the machine data **only** covers use of SIEMENS measuring cycles.
When using **ShopMill** or **ShopTurn**, comply with the information for these products.

Note:
Future versions measuring cycle versions may require more memory. For current data, see file SIEMENSD.TXT or SIEMENSE.TXT on supplied diskette 1.

© Siemens AG, 2004. All rights reserved

Notice
Machine data whose name starts "_MM" are memory configuring data. Changing these data reallocates the memory. Data it contains will be lost.
If this memory is in the non-volatile memory range of the control (**SRAM**), the **user data** (loaded programs and definitions, R-parameters, tool offsets, zero offsets, etc.) it contains will be lost.
Therefore: When performing initial start-up, set these machine data **at the beginning** of installation.
If data is already contained in the user memory, all data of the user memory must be saved before changing the machine data. After a successful change followed by a new control start-up, this data can be downloaded again.

Memory configuring machine data, SRAM

18118 MD number	**MM_NUM_GUD_MODULES** Number of data blocks	
Default setting: 7 **When using measuring cycles: 7**	Minimum input limit: 1	Maximum input limit: 9
Changes effective after POWER ON	Protection level: 2/7	Unit: -
Data type: DWORD	Applies from SW 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 variable names in the control	
Default setting: 10 **When using measuring cycles: 30**	Minimum input limit: 0	Maximum input limit: plus
Changes effective after POWER ON	Protection level: 2/7	Unit: -
Data type: DWORD	Applies from SW version: SW 1	
Meaning:	Memory reservation for GUD variable names, NCK (SRAM) range **Note:** The number of GUDs may also be limited by MD 18150.	

18130 MD number	**MM_NUM_GUD_NAMES_CHAN** Number of GUD variable names per channel	
Default setting: 40 **When using measuring cycles: 100** **with measurement in JOG: 130**	Minimum input limit: 0	Maximum input limit: plus
Changes effective after POWER ON	Protection level: 2/7	Unit: -
Data type: DWORD	Applies from SW version: SW 1	
Meaning:	Memory reservation for GUD variable names in each channel, CHAN (SRAM) range **Note:** The number of GUDs may also be limited by MD 18150.	

© Siemens AG, 2004. All rights reserved

18150 MD number	MM_GUD_VALUES_MEM Memory for values of the GUD variables	
Default setting: 12 / 16[1)] **When using measuring cycles: 28 / 32[1)]** (see Table below)	Minimum input limit: 0	Maximum input limit: plus
Changes effective after POWER ON	Protection level: 2/7	Unit: KB
Data type: DWORD	Applies from SW version: SW 1	
Meaning:	Memory for user variables (SRAM): Provides indication of the number of GUD variables that can be created. The FRAME data type requires almost 1 Kbyte. It is the largest data type. Data type BOOLEAN requires only 1 byte; type REAL 8 bytes. An array of variables requires only one name (MD 18120, MD 18130) – but memory space for all elements. Here, the total memory requirement is entered for all GUD variables, for NCK area, and for all channels. **Note:** The number of GUDs may also be limited by MD 18120 or MD 18130 .	

1) NCU-specific

Memory requirement of GUD variables for meas. cycles

The machine data settings recommended above provide overall values that enable work with measuring cycles. Deviating settings may be required for particular applications. The memory requirement of each measuring cycle is therefore specified here (approximate values). It may be higher in future measuring cycle versions.
The resulting memory requirement must be added to other user memory requirements.

Range	Number of names	Memory in Kbytes MD 18150
NCK	MD 18120: **20**	**7**
CHAN (GUD5, GUD6 only)	MD 18130: **60**	Two channels: **8** (number of channels * 4)
CHAN (from GUD7, also for "measuring in JOG")	MD 18130: **30**	Two channels: **1** (number of channels * 0.5)
		Example: Measuring cycles with 2 channels and "measuring in JOG": **16**

28082 MD number	MM_SYSTEM_FRAME_MASK Configuration screen form for channel-specific system frames	
Default setting: 21hex (bit 0=1, bit 5=1)	Minimum input limit: 0	Maximum input limit: 7Fhex
Changes effective after POWER ON	Protection level: 2/7	Unit: -
Data type: INT	(valid from meas. cycles SW 6.2)	
Meaning:	Bit 0: 1 system frames for preset mode and scratching Bit 5: 1 system frames for cycles (must always be available for cycles)	

© Siemens AG, 2004. All rights reserved

Memory configuring machine data, DRAM

18170 MD number	**MM_NUM_MAX_FUNC_NAMES** Number of names of miscellaneous functions - cycles					
Default setting: 40 **When using measuring cycles: 80**		Minimum input limit: 0		Maximum input limit: plus		
Changes effective after POWER ON			Protection level: 2/7		Unit: -	
Data type: DWORD				Applies from SW version: SW 1		
Meaning:	Number of cycles with input parameters (DRAM)					

18180 MD number	**MM_NUM_MAX_FUNC_PARAM** Number of parameters in the miscellaneous functions - cycles					
Default setting: 300 **When using measuring cycles: 800**		Minimum input limit: 0		Maximum input limit: plus		
Changes effective after POWER ON			Protection level: 2/7		Unit: -	
Data type: DWORD				Applies from SW version: SW 1		
Meaning:	Number of parameters for cycles acc. to MD 18170 (DRAM)					

28020 MD number	**MM_NUM_LUD_NAMES_TOTAL** Number of LUD variable names (in total in all program levels)					
Default setting: 200 **When using measuring cycles: 200**		Minimum input limit: 0		Maximum input limit: plus		
Changes effective after POWER ON			Protection level: 2/7		Unit: -	
Data type: DWORD				Applies from SW version: SW 3.2		
Meaning:	Number of local user variables – for each channel (DRAM) **Note:** The number of LUDs may also be limited by MD 28040.					

28040 MD number	**MM_NUM_LUD_VALUES_MEM** Memory for values of the LUD variables (DRAM)					
Default setting: 12 / 25 1) **When using measuring cycles: 16 / 29 1)**		Minimum input limit: 0		Maximum input limit: plus		
Changes effective after POWER ON			Protection level: 2/7		Unit: KB	
Data type: DWORD				Applies from SW version: SW 3.2		
Meaning:	Provides indication of the number of LUD variables that can be created. The FRAME data type requires almost 1 Kbyte. It is the largest data type. The BOOLEAN data type, however, requires only 1 byte. An array of variables requires only one name (MD 18120, MD 18130) – but memory space for all elements. **Note:** The number of LUDs may also be limited by MD 28020.					

1) NCU-specific

© Siemens AG, 2004. All rights reserved

9.1.2 Additional machine data

Machine data for adapting the probe

13200 MD number	**MEAS_PROBE_LOW_ACTIVE[n]** Switching characteristics of the measuring probe (n=0: Measuring input 1, n=1: Measuring input 2				
Default setting: FALSE **When using measurement cycles: FALSE**		Minimum input limit: FALSE		Maximum input limit: TRUE	
Changes effective after POWER ON			Protection level: 2/7		Unit: -
Data type: BOOLEAN				Applies from SW version: 2.2	
Meaning:	Value 0 (FALSE): non-deflected state 0 V (low) deflected state 24V (high) Value 1 (TRUE) non-deflected state 24 V (high) deflected state 0 V (low) **Note:** When programming MEAS, the data always refers to "not deflected", "deflected" – but not to the voltage level. The voltage level is set with this MD.				

Machine data for adapting MMC commands in cycles

10132 MD number	**MMC_CMD_TIMEOUT** Monitoring time for MMC command in part program				
Default setting: 1.0 **When using measuring cycles: 3.0**		Minimum input limit: 1.0		Maximum input limit: 100.0	
Changes effective after POWER ON			Protection level: 2/7		Unit: s
Data type: DOUBLE				Applies from SW 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				
Default setting: 1 **When using measuring cycles: 5**		Minimum input limit: 1		Maximum input limit: 1000 000	
Changes effective after POWER ON			Protection level: 2/7		Unit: KB
Data type: DWORD				Applies from SW version: SW4.3	
Meaning:	Maximum size of protocol files that are created with WRITE command.				

© Siemens AG, 2004. All rights reserved

9.2 Cycle data

9.2.1 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 block for measuring cycle users
- GUD6.DEF Data module for machine manufacturers

Data block GUD5.DEF

The input and output parameters for measuring cycles are stored in the data block GUD5.DEF; their status flags and arrays for the empirical and mean values are also defined here.

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.

In the as-delivered state, for example, the following settings are active:

_EV[20]	REAL	Number of empirical values
_MV[20]	REAL	Number of mean values

© Siemens AG, 2004. All rights reserved

Data block GUD6.DEF

The general, global, and channel-specific measuring cycle data are configured in the GUD6.DEF data block.

This block 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.

In the as-delivered state, the following settings are active:

Global data

_CVAL[4]=(3,3,3,0)	INTEGER	3 arrays each for • tool probe, machine-related (_TP[3,10]) • workpiece probe (_WP[3,11]) • gauging block (_KB[3,7]) • tool probe, workpiece-related (_TPW[3,10])
_TP[3,10]	REAL	3 arrays for tool probes, machine-related
_WP[3,11]	REAL	3 arrays for workpiece probes
_KB[3,7]	REAL	3 arrays for gauging blocks
_TPW[3,10]	REAL	3 arrays for tool probes, workpiece-related (from measuring cycles SW 6.3)
_CM[8]=(100,1000,1, 0.005,20,4,10,0),	REAL	Only active if _CBIT[12]=0 Monitoring data for tool measurement with rotating spindle and cyclic calculation: • max. peripheral speed 100 m/min • max. speed 1000 rev/min • F_{min}=1mm/min • measuring accuracy 0.005 mm • F_{max} for probing 20 mm/min • direction of rotation M4 • probing twice with feed factor 10 on first probing
_MFS[6]	REAL	Only active if _CBIT[12]=1 Speed and feed set by user during tool measurement with rotating spindle
_CBIT[16]=(0,0,0,1,0, 0,0,0,1,0,0,0,0,0,0,0)	BOOL	Central bits [0]: 0: A measurement is not repeated after a limit has been violated dimensional difference and safe area [1]: 0: No M0 on measurement repeat [2]: 0: No M0 on "oversize", "undersize", "dim. difference" [3]: 1: Metric basic system [4]: 0: Internal data item

© Siemens AG, 2004. All rights reserved

		[5]: 0: Tool measurement and calibration with CYCLE982 performed in basic coordinate system (machine coordinate system with switched-off kinematic transformation) [6]: 0: Logging with information about the measuring cycle measuring variant [7]: 0: Internal data item [8]: 1: Offset for mono probe setting with _CORA, [9]: 0: Logging off [10]: 0: Internal data item [11]: 0: Use of standard log header [12]: 0: Cycle internal calculation of feedrate and feedrate for tool measurement with rotating spindle [13]: 0: Without deleting fields _TP[ ], _TPW[ ], _WP[ ], _KB[ ], _EV[ ] and _MV[ ] [14]: 0: Length of the workpiece measuring probe, referred to the center of the probe sphere (ball) [15]: 0: Internal data item
_SI[3]=("","6","")	STRING[8]	Central strings (system information) • Internal data item • Software version of the control • Internal data item
_PROTNAME[2]	STRING[32]	• Name of main program the log is from, • Name of log file
_HEADLINE[10]	STRING[80]	Strings for log header
_PROTFORM[6]=(60,80, 1,3,1,12),	INTEGER	Log formatting • 60 lines per page • 80 characters per line • First page number is 1 • Number of header lines is 3 • Number of value lines in the log is 1 • Number of characters per column is 12
_PROTSYM[2]=(";","#")	CHAR	• Separator in the log is ";" • Special character for identification of measuring cycles is "#"
_PROTVAL[13]	STRING[100]	Title line in log; Specification of the values to be logged • 0...1: two title lines • 2...5: up to 4 value lines • 6...12: internal fields
_PMI[4]	INTEGER	Field for internal flags for logging
_SP_B[20]	INTEGER	Variable column width
_TXT[100]	STRING[12]	Field for formatted strings
_DIGIT=3	INTEGER	Number of decimal places is 3

© Siemens AG, 2004. All rights reserved

_MZ_MASK[20] =(1,0,0,0,0,0,0,0)	INTEGER	Setting data for measuring cycle support • generation of a direct measuring cycle call • without extended ZO/tool offset • without input field for measuring speed and measuring feed • without empirical values • without mean value calculation • workpiece probe is a multidirectional probe, • internal data item • cycle-internal calculation of feedrate and spindle rotation for tool measurement with rotating spindle (CYCLE971)

Channel-specific data

_EVMVNUM[2]=(20,20)	INTEGER	Number of empirical values and mean values • 20 memories for empirical values • 20 memories for mean values
_SPEED[4] =(50,1000,1000,900),	REAL	Traversing velocities for intermediate positioning • 50% rapid traverse velocity • positioning feedrate in the plane 1000 mm/min • positioning feedrate in infeed axis 1000 mm/min • fast measurement feedrate 900 mm/min
_CHBIT[30] =(0,1,1,0,0,0,0,0,0,0,0 ,0,0,0,0,0,0,0,0,0,0,0, 0,0,1,0,0,0,0,0)	BOOL	[0]: 0: Measuring input 1 for connecting a workpiece probe [1]: 1: Measuring input 2 for connecting a tool probe [2]: 1: Collision monitoring active for traversing blocks generated by measuring cycles [3]: 0: Tool data are entered into the geometry memory when the tool is measured [4]: 0: No mean value storage [5]: 0: The empirical value is subtracted from the measured actual value [6]: 0: When measuring workpieces with automatic tool offset, an additive offset/correction is made in the wear memory [7]: 0: CYCLE994 uses the trigger values for offset [8]: 0: When measuring workpieces with automatic tool offset, an additive offset is made in the total offset [9]: 0: Internal data item [10]: 0: No measurement result display [11]: 0: Measurement result display is deselected at the end of cycle [12]: 0: Internal data item [13]: 0: No coupling of the spindle position with coordinate rotation in the plane [14]: 0: Spindle positioning acc. to default [15]: 0: Up to 5 measurement attempts [16]: 0: Retraction from the measuring location with the same velocity as for the intermediate positioning

© Siemens AG, 2004. All rights reserved

		[17]: 0: Measurement feed only defined by _VMS [18]: 0: Automatic de-selection of the measuring result screen with end of cycle [19]: 0: Normal handling of the Y axis for rotating meas. cycles, [20]: 0: Spindle positioning with measurement with CYCLE982 [21]: 0: Internal data item [22]: 0: Last measurement with reduced speed for tool measurement with rotating spindle (CYCLE971) [23]: 0: Internal data item [24]: 1: Metric basic system [25...29]: 0: Internal data item
_IP_CF=0	INTEGER	No tool probe manufacturer specified (type)
_MT_COMP=0	INTEGER	No additional offset of the measurement result display on tool measurement with rotating spindle (CYCLE971)
_MT_EC_R[6,5]=(0,...,0)	REAL	User-defined array for offsetting the measurement result on tool radius measurement and rotating spindle (CYCLE971)
_MT_EC_L[6,5]=(0,...,0	REAL	User-defined array for offsetting the measurement result on tool length measurement and rotating spindle (CYCLE971)
_JM_I[5]=(0,1,1,17,0)	INTEGER	INT value field for JOG measurement • no set array number for probes like in ShopMill • number of array for workpiece probe is 1 • number of array for tool probe is 1 • working plane for measurement in JOG is G17 • active ZO number on measurement in JOG is 0 (G500)
_JM_B[7]= (0,1,0,0,0,0,0)	BOOL	BOOL value field for JOG measurement • offset in geometry on tool measurement • 1 measurement attempt • retraction from meas. point at same velocity as intermediate positioning • no fast measurement feed
_MC_MTL[3]= (33.3,33.3,33.3)	INTEGER	Measuring probe offset (correction) for sphere measurement (only relevant in CYCLE997 for the measurement variant with "determining the sphere diameter") Ratio between the tracer length/pin-sphere (ball) radius ($TC_DP6) Pre-assignment (default) for 3 measuring probes: 100/3 If this variable is not available, then the trigger values are not corrected. [Array index]: _PRNUM-1 (from meas. cycles SW 6.3)
_MC_SIMSIM=1	INTEGER	0: Skip measuring cycles during simulation 1: Run measuring cycles during simulation (from meas. cycles SW 6.3)
_MC_SIMDIFF=0	REAL	Value for simulated measuring difference (from meas. cycles SW 6.3)

© Siemens AG, 2004. All rights reserved

9.2.2 Data adjustment to a specific machine

Function

There are two main steps for adapting the data to a specific machine:

1. **Adapting the data definitions** in the GUD blocks and loading them in the PLC.
2. **Subsequent adjustment** of certain values in part program.

1. Adapting the data definitions

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:

- and 2 data fields (data arrays) for using tool probes
- and 2 data fields (data arrays) for using workpiece probes
- without calibration groove pair,
- only 10 empirical and mean values

The following example only shows the lines to be changed!

Example:

%_N_GUD6_DEF
;$PATH=/_N_DEF_DIR
;(date) adaptation to a 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)]
;N13 DEF NCK REAL _KB[3,7][1)]
N40 DEF CHAN INT _EVMVNUM[2]=(**10**,**10**)[1)]
...

M17

1) Characters and digits in bold (highlighted) have been changed with respect to the status when first supplied

© Siemens AG, 2004. All rights reserved

2. Subsequence adaptation of 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:

- A tool probe with a disk in the XY plane is being used with an active disk diameter of 20 mm for tool length measurement.
- Measurement repetition when the permissible dimensional difference or the safe area are exceeded, but without M0.
- Length of workpiece probe relative to end of probe ball.
- Static measurement result display.
- No repeat of an unsuccessful attempted measurement.
- Retraction of the probe from the measuring point at 80% of rapid traverse velocity.

%_N_MZ_VALUE ASSIGNMENT_MPF	
;$PATH=/_N_MPF_DIR	
;(date) pre-assignment of meas. cycle data to machine_1	
N10 _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
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 meas. result.
N50 _CHBIT[15]=1	;Measurement abort after unsuccessful ;attempt
N60 _CHBIT[16]=1	;Retraction from measuring point at % ;rapid traverse velocity defined in ;_SPEED[0]
N65 _SPEED[0]=80	;Reduction of rapid traverse to 80 %
N99 M2	

© Siemens AG, 2004. All rights reserved

9.2.3 Central values

Data block GUD6.DEF

	_CVAL[ ] Number of elements, arrays		
	Minimum input limit: -	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: INT	Applies from SW version: 3.2 / 6.3		
Meaning:			Preset default
	_CVAL[0]	Number of arrays for tool probes _TP[ ] (machine-related)	3
	_CVAL[1]	Number of arrays for workpiece probes _WP[ ]	3
	_CVAL[2]	Number of arrays for gauging blocks _KB[ ]	3
	_CVAL[3]	Number of arrays for tool probes _TPW[ ] (relative to the workpiece, from meas. cycles SW 6.3)	3

© Siemens AG, 2004. All rights reserved

	_TP[ ] Array for tool probes (machine-related)
	Minimum input limit: - / Maximum input limit: -
Changes valid after value assignment	Protection level: - / Unit: -
Data type: REAL	Applies from SW version: 3.2
Meaning:	Preset default

Index "k" stands for the number of the current array (_PRNUM-1)

Assignment on milling

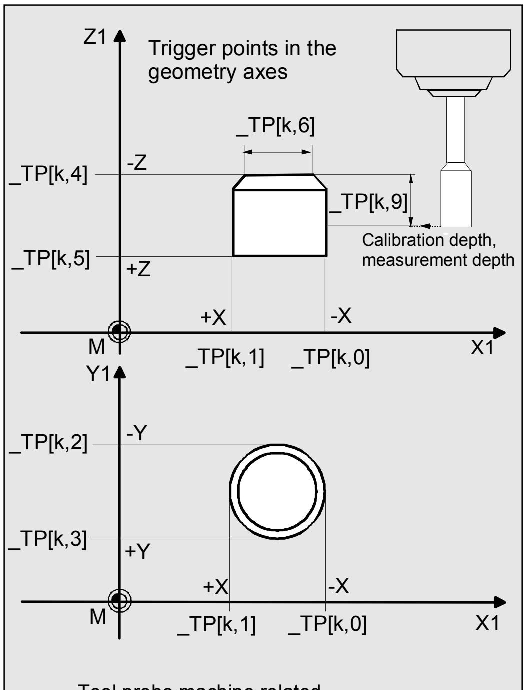
Element	Meaning	Preset default
_TP[k,0]	Trigger point in minus direction **X** (1st geometry axis)	0
_TP[k,1]	Trigger point in plus direction **X** (1st geometry axis)	0
_TP[k,2]	Trigger point in minus direction **Y** (2nd geometry axis)	0
_TP[k,3]	Trigger point in plus direction **Y** (2nd geometry axis)	0
_TP[k,4]	Trigger point in minus direction **Z** (3rd geometry axis)	0
_TP[k,5]	Trigger point in plus direction **Z** (3rd geometry axis)	0
_TP[k,6]	Edge length/disk diameter	0
_TP[k,7]	Axes and directions for "automatic calibration" (from meas. cycles SW 6.3)	133
_TP[k,8)	Probe type 0: cube 101: disk in XY 201: disk in ZX 301: disk in YZ	0
_TP[k,9]	Distance between upper edge of tool probe and lower edge of tool (= calibration depth, measurement depth for cutter radius)	2

Assignment for turning (probe type: primarily cube)

Element	Meaning	Preset default
_TP[k,0]	Trigger point in minus direction of abscissa	0
_TP[k,1]	Trigger point in plus direction of abscissa	0
_TP[k,2]	Trigger point in minus direction of ordinate	0
_TP[k,3]	Trigger point in plus direction of ordinate	0
_TP[k,4] to _TP[k,9]	irrelevant	0

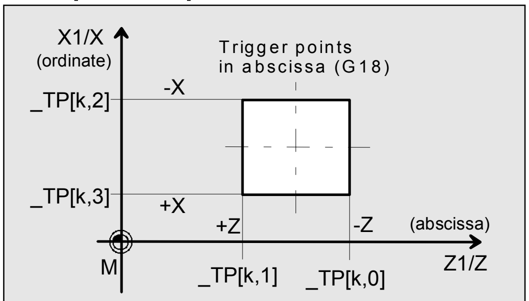
Tool probe on milling machine

Example: Probe type disk in XY (_TP[k,8]=101)



Tool probe machine related, disk-type in XY (_TP[k,8]=101)

Tool probe on turning machine
Example: G18 plane, values machine-related

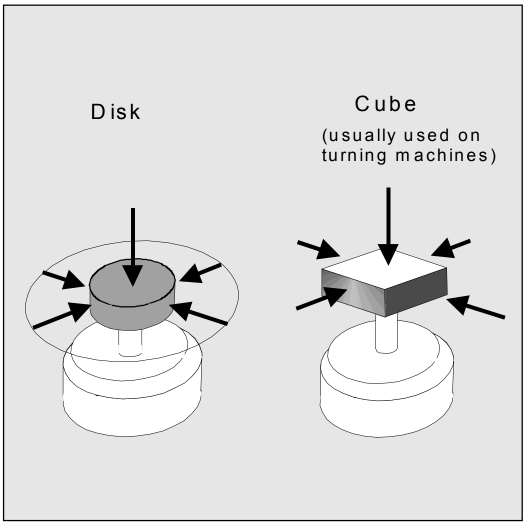


Tool probe machine related, cube type

	_TPW[ ] Array for tool probes (workpiece-related)
	Minimum input limit: - / Maximum input limit: -
Changes valid after value assignment	Protection level: - / Unit: -
Data type: REAL	Applies from SW version: 6.3
Meaning:	Preset default Index "k" stands for the number of the current array (_PRNUM-1) **Assignment on milling** _TPW[k,0] Trigger point in minus direction **X** (1st geometry axis) 0 _TPW[k,1] Trigger point in plus direction **X** (1st geometry axis) 0 _TPW[k,2] Trigger point in minus direction **Y** (2nd geometry axis) 0 _TPW[k,3] Trigger point in plus direction **Y** (2nd geometry axis) 0 _TPW[k,4] Trigger point in minus direction **Z** (3rd geometry axis) 0 _TPW[k,5] Trigger point in plus direction **Z** (3rd geometry axis) 0 _TPW[k,6] Edge length/disk diameter 0 _TPW[k,7] Axes and directions for "automatic calibration" 133 _TPW[k,8) Probe type 0: cube 0 101: disk in XY 201: disk in ZX 301: disk in YZ _TPW[k,9] Distance between upper edge of tool probe and lower edge of tool 2 (= calibration depth, measurement depth for cutter radius) **Assignment for turning** _TPW[k,0] Trigger point in minus direction of abscissa 0 _TPW[k,1] Trigger point in plus direction of abscissa 0 _TPW[k,2] Trigger point in minus direction of ordinate 0 _TPW[k,3] Trigger point in plus direction of ordinate 0 _TPW[k,4] irrelevant 0 to _TPW[k,9] irrelevant 0

For illustration, see analogous explanation of _TP[ ]

Tool probe types

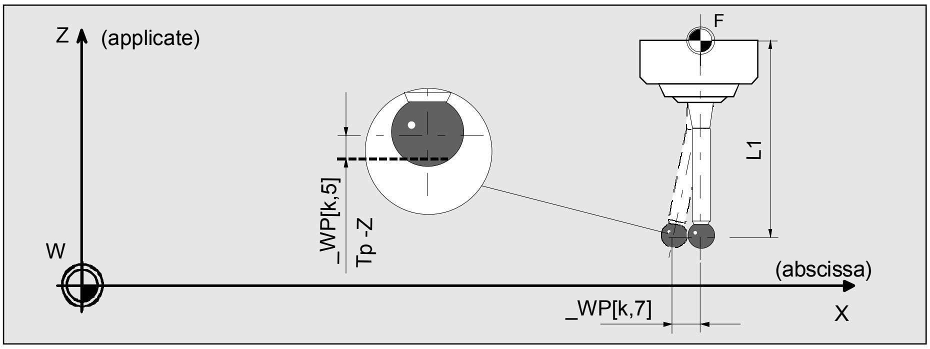


© Siemens AG, 2004. All rights reserved

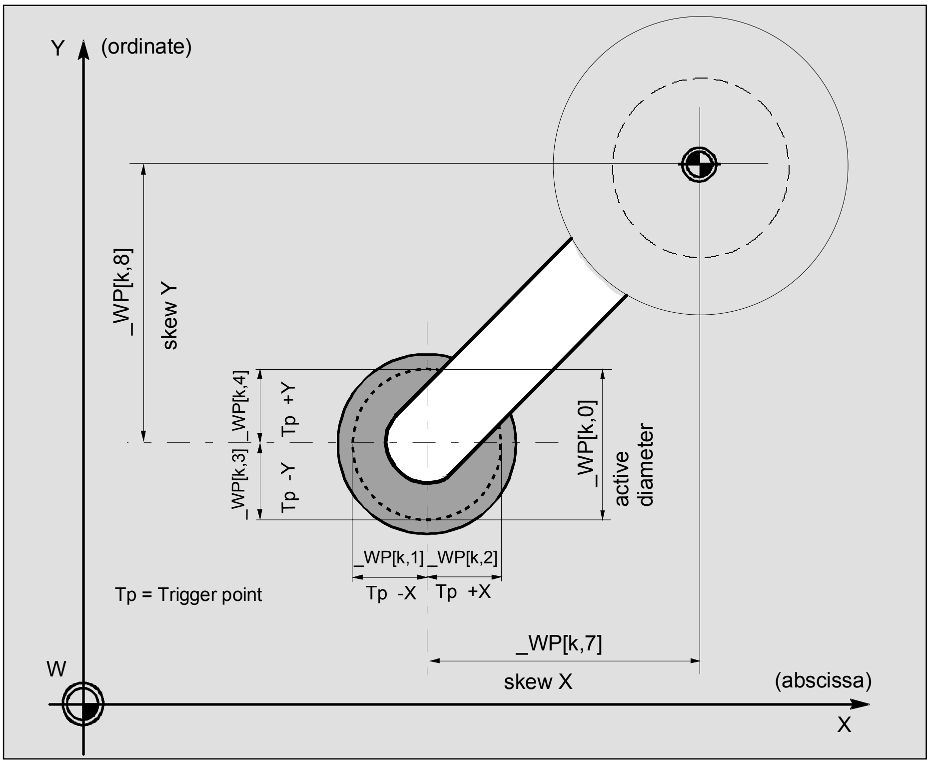
	_WP[ ] Workpiece probe				
		Minimum input limit: -		Maximum input limit: -	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: REAL				Applies from SW version: 3.2	
Meaning:	Index "k" stands for the number of the current array (_PRNUM-1)				Preset default
	_WP[k,0]	active ball diameter of the workpiece probe			0
	_WP[k,1]	Trigger point in minus direction of abscissa			0
	_WP[k,2]	Trigger point in plus direction of abscissa			0
	_WP[k,3]	Trigger point in minus direction of ordinate			0
	_WP[k,4]	Trigger point in plus direction of ordinate			0
	_WP[k,5]	Trigger point in minus direction of applicate			0
	_WP[k,6]	Trigger point in plus direction of applicate			0
	_WP[k,7]	Positional deviation abscissa (skew)			0
	_WP[k,8]	Positional deviation ordinate (skew)			0
	_WP[k,9]	Calibration status, coded			0
	_WP[k,10]	Calibration status, coded			0

Overview of workpiece probe data:
Example: G17, milling, **_CBIT[14]=0**

Skew of a real probe in rest position and trigger point Tp in –Z



Skew and trigger point Tp in X and Y (enlarged view)

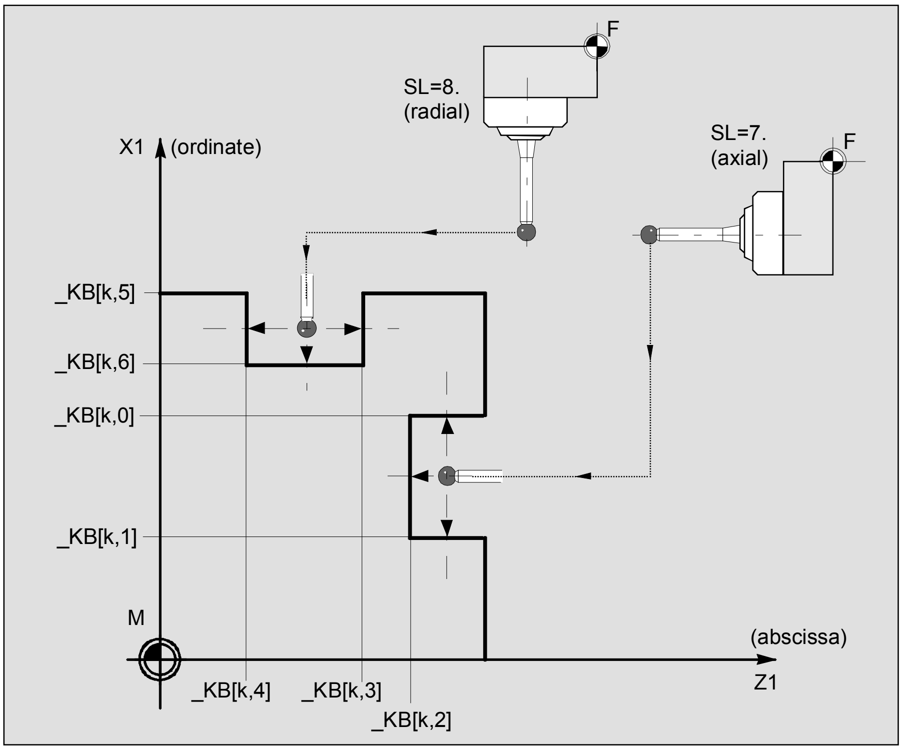


© Siemens AG, 2004. All rights reserved

	_KB[ ] Gauging block (pair of reference grooves)				
		Minimum input limit: -		Maximum input limit: -	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: REAL				Applies from SW version: 3.2	
Meaning:	Preset default Index "k" stands for the number of the current array (_CALNUM-1) Reference slot for calibration of a workpiece probe with cutting edge position SL=7 (tool type: 5xy) _KB[k,0] Groove edge in plus direction, ordinate 0 _KB[k,1] Groove edge in minus direction, ordinate 0 _KB[k,2] Groove base in abscissa 0 Reference groove for calibration of a workpiece probe with cutting edge position SL=8 (tool type: 5xy) _KB[k,3] Groove edge in plus direction, abscissa 0 _KB[k,4] Groove edge in minus direction, abscissa 0 _KB[k,5] Upper edge groove in ordinate 0 _KB[k,6] Groove base in ordinate 0 Notes: The values of the grooves are always machine-related and radius dimensions. Both grooves have the same depth.				

Overview of pair of reference grooves for calibration
(for turning only)

The representation refers to the working plane defined by G18.



© Siemens AG, 2004. All rights reserved

For tool measurement with CYCLE971 only

	_CM[ ] Monitoring functions for tool measurement with rotating spindle, only active if _CBIT[12]=0				
		Minimum input limit: -		Maximum input limit: -	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: REAL				Applies from SW version: 4.3	

Meaning:

		Preset default
_CM[0]	Max. permissible peripheral speed [m/min]/[feet/min]	100
_CM[1]	Max. permissible speed [rpm]	1000
_CM[2]	Minimum feedrate for 1st probing [mm/min]	1
_CM[3]	Required measuring accuracy [mm]	0.005
_CM[4]	Max. permissible feedrate for probing	20
_CM[5]	Direction of spindle rotation	4
_CM[6]	Feed factor 1	10
_CM[7]	Feed factor 2	0

For tool measurement with CYCLE971 only

	_MFS[ ] Feeds and speeds for tool measurement with rotating spindle, only active if _CBIT[12]=1				
		Minimum input limit: -		Maximum input limit: -	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: REAL				Applies from SW version: 4.3	

Meaning:

		Preset default
_MFS[0]	Speed 1st probing	0
_MFS[1]	Feed 1st probing	0
_MFS[2]	Speed 2nd probing	0
_MFS[3]	Feed 2nd probing	0
_MFS[4]	Speed 3rd probing	0
_MFS[5]	Feed 3rd probing	0

© Siemens AG, 2004. All rights reserved

9.2.4 Central bits

Data block GUD6.DEF

	_CBIT[ ] Central bits		
	Minimum input limit: 0	Maximum input limit: 1	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: BOOLEAN	Applies from SW version: 3.2		
Comment:			Preset default
	_CBIT[0]	Measurement repetition after violation of dimensional difference and Safe area 0: no measurement repeat 1: measurement repeat, up to 4	0
	_CBIT[1]	Alarm and M0 for measurement repeat with _CBIT[0]=1 0: no alarm, no M0 generated 1: M0 and an alarm are generated before each repeat	0
	_CBIT[2]	M0 on tolerance alarms "oversize", "undersize", "Permissible dimensional difference exceeded" 0: no generation of M0 for the above alarms 1: generation of M0 for the above alarms	0
	_CBIT[3]	Central marker for basic dimension system of the control 0: Basic system setting is inches 1: Basic system setting is metric	1
	_CBIT[4]	*currently not assigned*	0
	_CBIT[5] Note:	Tool measurement and calibration in the WCS in CYCLE982 0: machine-related measurement and calibration 1: workpiece-related measurement and calibration In both cases, the _TP[ ] field of the probe is used. From measuring cycles SW 6.3, function switchover via _MVAR is available.	0
	_CBIT[6]	Logging without output of the measuring cycle name and measuring variant 0: Measuring cycle name and measuring variant will be output. 1: These outputs will be suppressed.	0
	_CBIT[7]	*currently not assigned*	0
	_CBIT[8]	Offset of mono probe position 0: no offset 1: offset of spindle by angle _CORA,	1
	_CBIT[9]	Assigned internally	0
	_CBIT[10]	*currently not assigned*	0
	_CBIT[11]	Selection of log header for logging 0: Standard 1: user-defined	0
	_CBIT[12]	Feed and speed in CYCLE971 0: calculation by measuring cycle itself 1: set by user in array _MFS[ ]	0
	_CBIT[13]	Deletion of the measuring cycle arrays in the GUD6 0: no deletion 1: delete _TP[ ], _TPW[ ], _WP[ ], _KB[ ], _EV[ ], _MV[ ], _CBIT[13]	0
	_CBIT[14]	Length reference of the workpiece probe in milling measuring cycles 0: length relative to probe ball center 1: length relative to end	0
	_CBIT[15]	Transfer of workpiece probe data into the tool offset in CYCLE976 0: no transfer 1: result of probe ball calculation on **calibration** will be entered in the geometry memory of the workpiece probe (radius)	0

© Siemens AG, 2004. All rights reserved

Central bits – detailed description

Measurement repetition after violation of dimensional difference and safe area

_CBIT[0]=0: When the parameters for the dimensional difference (_TDIF) and safe area (_TSA) are exceeded, the measurement is not repeated. An appropriate alarm is displayed.

_CBIT[0]=1: When the parameters for the dimensional difference (_TDIF) and safe area (_TSA) are exceeded, a measurement is repeated. An alarm is also displayed in the repeat measurements if **_CBIT[1]** is set.

Alarm and M0 for measurement repeat with _CBIT[0]=1

_CBIT[1]=0: No alarm, no M0 in measurement repeats

_CBIT[1]=1: If the limits of the parameters for the dimensional difference and safe area are exceeded, M0 is generated and the repeat measurement must be started with NC-START. An alarm is displayed in the alarm line that does not have to be exceeded.

M0 for tolerance alarms "oversize", "undersize", or "permissible dimensional difference exceeded"

_CBIT[2]=0: When the alarms "oversize", "undersize", or "permissible dimensional difference exceeded" occur, no M0 is generated.

_CBIT[2]=1: M0 is generated when these alarms occur.

Central marker for basic dimension system of the control

_CBIT[3]=0: Basic system is based on inches

_CBIT[3]=1: The basic system is metric

When starting up the measuring cycles, this bit has to be set according to the basic settings of the PLC (MD 10240).
MD 10240: SCALING_SYSTEM_IS_METRIC must be set.

If modifying the basic settings of the control results in _CBIT[3] no longer matching MD 10240, arrays _TP[], _WP[], _KB[] and _EV[] will be converted the first time a measuring cycle is called after the modification and a message will be output.

The data for tool measurement with rotating spindle (_CM[], _MFS[]) are also converted as are the data in the _SPEED[ ] field.

© Siemens AG, 2004. All rights reserved

Tool measurement and calibration in the WCS in CYCLE982

_CBIT[5]=0: The tool is measured and the tool measuring probe is calibrated in the machine coordinate system.
The tool probe data are stored in the _TP[ ] field.

_CBIT[5]=1: The tool can be measured and the tool probe calibrated in the active WCS.
In this case, when calibrating and measuring, the same prerequisites must exist regarding the actual WCS. This means that tools with active transformation can also be measured, e.g. TRAANG.
Notice: When calibrating and measuring, the _TP[ ] field is also used here.

Note:
From measuring cycles SW 6.3, function switchover via _MVAR is available. A separate tool probe array is used there, the _TPW[ ] field, for calibration/measurement in the WCS.

Logging without output of the measuring cycle name and measuring variant

_CBIT[6]=0: When logging, the measuring cycle name and the measuring variant are written into the log.

_CBIT[6]=1: When logging, the measuring cycle name and the measuring variant are not output into the log.

Offset of the mono probe setting

_CBIT[8]=0: No compensation

_CBIT[8]=1: If the workpiece measuring probe is a mono probe, its position (spindle position) is corrected by the angular value in _CORA.

Selection of log header for logging

_CBIT[11]=0: The standard log header is used.

_CBIT[11]=1 A user-defined log header is used.

Feed and speed in CYCLE971

_CBIT[12]=0: When measuring milling tools with the spindle rotating, the **measuring cycle** calculates the feed and speed.

_CBIT[12]=1: The **user** enters the feed and speed in data field (data array) _MFS[ ].

© Siemens AG, 2004. All rights reserved

Deletion of the measuring cycle arrays in the GUD6

_CBIT[13]=0: No deletion

_CBIT[13]=1: For the following measuring cycle call, the data fields (data arrays) _TP[ ], _TPW[ ], _WP[ ], _KB[ ], _EV[ ], _MV[ ] and _CBIT[13] are set to zero.

Length reference of the workpiece probe in milling measuring cycles

_CBIT[14]=0: The length 1 of the measuring probe referred to the **center** of the probe sphere (ball) should be entered into the tool offset.

_CBIT[14]=1: The length 1 of the measuring probe referred to the **end** of the probe sphere (ball) should be entered into the tool offset.

Transfer of workpiece probe data into the tool offset in CYCLE976

_CBIT[15]=0: No transfer

_CBIT[15]=1: With measuring variant "Calibration with probe ball calculation", the determined "effective probe sphere diameter" (_WP[k,0]), converted as radius value is entered into the radius geometry memory of the active workpiece measuring probe as tool offset.

9.2.5 Central strings

Data block GUD6.DEF

	_SI[ ] Central strings (only up to measuring cycles SW 6.2)		
	Minimum input limit: -	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: STRING	Applies from SW version:		
Meaning:			Preset default
	_SI[0]	*currently not assigned*	
	_SI[1]	Software version	"6"
	_SI[2]	*currently not assigned*	

Software version

Here you have to enter the first digit of the version of the NCU software on the control, e.g. for SW 05.xx.xx, enter 5.

© Siemens AG, 2004. All rights reserved

9.2.6 Channel-oriented values

Data block GUD5.DEF

	_EV[ ] Empirical values		
	Minimum input limit: -	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: REAL	Applies from SW version: 3.2		
Meaning:	Index "k" stands for the number of the current array -1 _EV[k] Number of empirical values		Preset default 0

	_MV[ ] Mean values		
	Minimum input limit: -	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: REAL	Applies from SW version: 3.2		
Meaning:	Index "k" stands for the number of the current array _MV[k] Mean value		Preset default 0

Data block GUD6.DEF

	_EVMVNUM[ ] Number of empirical values and mean values		
	Minimum input limit: 0	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: INT	Applies from SW version: 3.2		
Meaning:	_EVMVNUM[0] Number of empirical values _EVMVNUM[1] Number of mean values		Preset default 20 20

	_SPEED[ ] Traversing velocities for intermediate positioning		
	Minimum input limit: 0	Maximum input limit: 100	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: REAL	Applies from SW version: 3.2		
Meaning:	_SPEED[0] Intermediate positioning in the measuring cycle with rapid traverse in % non-active collision monitoring (values between 1 and 100) _SPEED[1] Intermediate positioning in measuring cycle in the plane with active collision monitoring _SPEED[2] Intermediate positioning in the measuring cycle with positioning in the infeed axis with collision monitoring active _SPEED[3] Fast measuring feed Note: Adjust value to probe and machine used, if necessary! Too high values can damage the probe!		Preset default 50 1000 1000 900

© Siemens AG, 2004. All rights reserved

Traverse velocities for intermediate positioning _SPEED[0] to [2]

Intermediate positions before the actual measuring block are calculated in the measuring cycles. This positions can

- be approached **with collision monitoring** (_CHBIT[2]=1, default) or
- **without collision monitoring** (_CHBIT[2]=0).

The appropriate velocities are used for approach as specified in these settings.

With collision monitoring (_CHBIT[2]=1):
With **_SPEED[1]** the feedrate is applied to traversing in the plane and with **_SPEED[2]** to traversing in the infeed axis (applicate).
If the probe switches while approaching these intermediate positions, the movement is interrupted and the alarm "probe collision" is output.

Without collision monitoring (_CHBIT[2]=0):
The intermediate positions are approached with the **percentage maximum axis velocity** (rapid traverse) specified in **_SPEED[0]**. With _SPEED[0]=0 and _SPEED[0]=100 the maximum axis velocity applies.
Caution! The user must ensure that collisions are ruled out.

Measuring feed _VMS, fast measuring feed _SPEED[3]

Measurement is performed with the measuring feed of _VMS.
If _VMS=0 and _FA=1: 150 mm/min.
If _VMS=0 and _FA>1: 300 mm/min (see Section 2.3)
If **_CHBIT[17]=1** and **_FA>1** probing is performed twice. The **fast measuring feed _SPEED[3]** is used for the first probing. After the probe has switched, it is retracted by 2 mm. This is followed by actual measurement with the feedrate programmed in _VMS.

Measurement retraction velocity

Retraction from the measuring point is usually performed with the same speed (_SPEED[1], [2]) or percentage of the rapid traverse as approach to the intermediate position (see above).
However, while **collision monitoring** (_CHBIT[2]=1) is active, it is possible with **CHBIT[16]=1** to switch to the percentage of rapid traverse in **_SPEED[0]**.

© Siemens AG, 2004. All rights reserved

	_TP_CF Tool probe type (manufacturer)				
		Minimum input limit: 0		Maximum input limit: 2	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: INTEGER				Applies from SW version: 6.2	
Meaning:	Applies to tool measurement with rotating spindle (CYCLE971 only) 0: No specified validity 1: TT130 (Heidenhain) 2: TS27R (Renishaw)				Preset default 0

	_MT_COMP Measurement result offset for tool measurement with rotating spindle (CYCLE971 only)				
		Minimum input limit: 0		Maximum input limit: 2	
Changes valid after value assignment			Protection level: -		Unit: -
Data type: INTEGER				Applies from SW version: 6.2	
Meaning:	0: no offset 1: cycle-internal offset (only active if _TP_CF<>0) 2: offset via user-defined offset table				Preset default 0

	_MT_EC_R[6,5] Offset table for measurement result offset for tool radius measurement with rotating spindle (CYCLE971 only)				
		Minimum input limit: -		Maximum input limit: -	
Changes valid after value assignment			Protection level: -		Unit: mm
Data type: REAL				Applies from SW version: 6.2	
Meaning:	Measurement result offset for tool measurement with rotating spindle (CYCLE971 only) _MT_EC_R[0,1]..._MT_EC_R[0,4] 4 tool radii from small to large _MT_EC_R[1,0]..._MT_EC_R[5,0] 5 peripheral velocities from low to high are specified _MT_EC_R[i,k] where i=1...5, k=1...4 If _MT_COMP=2, then actual radius = measured radius - _MT_EC_R[i,k] where i=1...5 next lowest table value for peripheral velocity and K=1...4 next lowest table value for tool radius				Preset default 20 offset values

	_MT_EC_L[6,5] Offset table for measurement result offset for tool length measurement with rotating spindle (CYCLE971 only)				
		Minimum input limit: -		Maximum input limit: -	
Changes valid after value assignment			Protection level: -		Unit: mm
Data type: REALR				Applies from SW version: 6.2	
Meaning:	Measurement result offset for tool measurement with rotating spindle (CYCLE971 only) _MT_EC_L[0,1]..._MT_EC_L[0,4] 4 tool lengths from small to large _MT_EC_L[1,0]..._MT_EC_L[5,0] 5 peripheral velocities from low to high are specified _MT_EC_L[i,k] where i=1...5, k=1...4 If _MT_COMP=2, then actual length = measured length - _MT_EC_L[i,k] where i=1...5 next lowest table value for peripheral velocity and K=1...4 next lowest table value for tool radius				Preset default 20 offset values

© Siemens AG, 2004. All rights reserved

9.2.7 Channel-oriented bits

Data block GUD6.DEF

	_CHBIT Channel bits		
	Minimum input limit: -	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: BOOLEAN	Applies from SW version: 3.2		
Meaning:			Preset default
	_CHBIT[0]	Measurement input for workpiece measurement 0: measurement input 1 1: measurement input 2	0
	_CHBIT[1]	Measurement input for tool measurement: 0: measurement input 1 1: measurement input 2	1
	_CHBIT[2]	Collision monitoring for intermediate positioning 0: OFF 1: ON	1
	_CHBIT[3]	Tool offset mode with tool measurement 0: first-time measurement (determining geometry) 1: remeasuring (determining wear)	0
	_CHBIT[4]	Mean value for workpiece measurement with automatic tool offset (_EVNUM>0) 0: no mean value derivation over several parts 1: with mean value formation and calculation	0
	_CHBIT[5]	Inclusion of empirical value (_EVNUM>0) 0: subtraction of actual value 1: addition to actual value	0
from meas. cycles - SW 6.3	_CHBIT[6]	Tool offset mode with workpiece measurement with automatic tool offset 0: Offset in wear 1: offset in geometry, delete wear For additive and setup offset and _CHBIT[8]=0: 0: offset in additive offset 1: offset in set-up offset, delete additive offset	0
	_CHBIT[7]	Measured value offset in CYCLE994 0: use of trigger values of the probe _WP[k,1] ... 1: use of the active ball diameter of the probe _WP[k,0]	0
from meas. cycles - SW 6.3	_CHBIT[8]	Offset mode for workpiece measurement with automatic tool offset 0: additive, setup offset according to _CHBIT[6] 1: offset additive in set-up offset, independent of _CHBIT[6]	0
	_CHBIT[9]	*currently not assigned*	0
	_CHBIT[10]	Measuring result display 0: OFF 1: ON	0
	_CHBIT[11]	Acknowledgment measurement result screen with NC start 0: OFF (If _CHBIT[18]=0, the display is automatically deselected at end of cycle.) 1: ON (M0 is generated in the cycle.)	0
	_CHBIT[12]	*currently not assigned*	
	_CHBIT[13]	Coupling spindle position with coordinate rotation in active plane for workpiece measurement with multi probe 0: OFF 1: ON	0
	_CHBIT[14]	Adapt spindle positioning, if _CHBIT[13]=1 0: according to default 1: adapted angle values	0
	_CHBIT[15]	Number of measurements on failure to switch 0: up to 5 measurements 1: only 1 measurement	0

© Siemens AG, 2004. All rights reserved

	_CHBIT[16]	Retraction velocity from the measuring point 0: velocity as for intermediate positioning 1: with percentage of rapid traverse velocity (_SPEED[0]) (only active with collision monitoring ON: _CHBIT[2]=1)	0
	_CHBIT[17]	Feed during measurement 0: with feed in _VMS 1: on 1st measurement feed in _SPEED[3] on 2nd measurement feed in _VMS	0
	_CHBIT[18]	Static measurement result display 0: effect as set in _CHBIT[11]. 1: only active if _CHBIT[11]=0: Measuring result display remains until next measuring cycle is called	0
	Only active for CYCLE974 or CYCLE994! _CHBIT[19]	Special treatment of Y axis with G18 0: no special treatment 1: setpoint setting and parameterization (_SETVAL, _TUL, _TLL, SZO) for the Y axis (applicate) as for the parameterization of the ordinate (X axis) The tool offset is in the length that is active in the ordinate (X axis) (L1 usually) – as long as via _KNUM no other length settings are made. The ZO correction is applied in the specified ZO memory in the ordinate component (X axis).	0
	Only active for CYCLE982! _CHBIT[20]	Suppression of the starting angle positioning _STA1 0: suppression OFF 1: suppression ON	0
	Only active for CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE9997! _CHBIT[21]	ZO mode 0: offset additive in FINE 1: offset in COARSE, delete FINE	0
	Only active for CYCLE971! _CHBIT[22]	Speed reduction for tool measurement with rotating spindle and multiple measurement 0: last measurement with reduced speed at _CBIT[12]=0 1: no speed reduction	0
from meas. cycles - SW 6.3	Only active for CYCLE982! _CHBIT[23]	Recoding of tool point direction during tool measurement 0: no recoding 1: internal recording (tool point direction mirroring about X)	0
	_CHBIT[24]	Channel-oriented marker for basic dimension system (For deviations from MD 10240, cycle parameters are internally converted and this bit is again set.) 0: The basic system is inch 1: The basic system is metric	1

© Siemens AG, 2004. All rights reserved

Channel-oriented bits – detailed description

Measurement input for workpiece measurement

_CHBIT[0]=0: Meas. input 1 is activated for workpc. measurement.
_CHBIT[0]=1: Meas. input 2 is activated for workpc. measurement.

Measurement input for 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 for intermediate positioning

_CHBIT[2]=0: OFF
_CHBIT[2]=1: ON
Intermediate positions that are calculated and approached by the measuring cycles are interrupted as soon as the measuring probe supplies a switching signal. When aborted/interrupted (collision), an alarm message is generated.

Tool offset mode with tool measurement

_CHBIT[3]=0: First-time measurement
The determined tool data (length and radius) are written into the geometry data of the tool. The wear is deleted.
_CHBIT[3]=1: Remeasurement
The difference that is determined is entered into the appropriate wear data of the tool.
The geometry data remains unchanged.

Mean value for workpiece measurement with automatic tool correction (_EVNUM >0)

_CHBIT[4]=0: No mean value derivation over several parts
For the formula used to calculate the mean value (refer to Section 1.7), 0 is used as the old mean value. The calculated mean value is not saved.
_CHBIT[4]=1: When calculating the mean value, the value from the mean value memory _MV[ ] programmed using _EVNUM, is used for the calculation and then the new mean value is saved in this mean value memory.

Inclusion of empirical value (_EVNUM >0)

_CHBIT[5]=0: The empirical value _EV[ ] is subtracted from the measured actual value.
_CHBIT[5]=1: The empirical value _EV[ ] is added to the measured actual value.

© Siemens AG, 2004. All rights reserved

Tool offset mode for workpiece measurement with automatic tool offset

_CHBIT[6]=0: The determined offset value is added in the **wear** memory (length and radius) of the specified tool and is incorporated in the D number specified using _KNUM.

_CHBIT[6]=1: The length and the radius of the specified tool is corrected by the determined offset value and entered into the appropriate **geometry** memory. The corresponding wear memory is calculated and then set to zero.

From measuring cycles SW 6.3, an offset in the set-up/additive offset can also be programmed. The type of calculation is also defined by **_CHBIT[6]** and **_CHBIT[8]**:

_CHBIT[6]=0 The calculated offset value is included additively in
_CHBIT[8]=0 the appropriate additive offset value memory.

_CHBIT[6]=1 The determined offset value is incorporated, taking into account
_CHBIT[8]=0 the appropriate additive offset value, into the setting-up offset memory and the additive offset memory is deleted.

_CHBIT[8]=1 Independent of _CHBIT[6], the determined offset value is additively incorporated into the appropriate setting-up offset memory.

Measured value offset in CYCLE994

_CHBIT[7]=0: In order to determine the actual value, the **trigger values** of the measuring probe, saved in the _WP[_PRNUM-1,1...4) are used.

_CHBIT[7]=1: In order to determine the actual value, the effective **diameter** of the measuring probe, saved in the _WP[_PRNUM-1,0] is used.

Offset mode for workpiece measurement with automatic tool offset

_CHBIT[8]: Explanation --> see _CHBIT[6]

Measuring result display

_CHBIT[10]=0: OFF

_CHBIT[10]=1: ON – After measuring or calibrating, a measurement result screen is automatically displayed.

© Siemens AG, 2004. All rights reserved

Acknowledgment measurement result screen with NC start

_CHBIT[11]=0: The measurement result screen is automatically de-selected at the end of the cycle. Also _CHBIT[18] must be =0, otherwise the effect as described for **_CHBIT[18]**=1 is obtained.

_CHBIT[11]=1: After the measurement result screen is displayed, the cycle generates M0. The measurement cycle is continued and the screen is de-selected after the NC start.

Static measurement result display

_CHBIT[18]=0: Effect is defined by **_CHBIT[11]**.

_CHBIT[18]=1: The measurement result screen display is kept until the next measuring cycle is called. The NC program processing is not interrupted. **_CHBIT[10]** must be set, _CHBIT[11] must be 0!

Coupling spindle position with coordinate rotation in active plane for workpiece measurement with multi probe

_CHBIT[13]=0: OFF
There is no coupling between the spindle position and active coordinate rotation in the plane.

_CHBIT[13]=1: ON
When multi-probes are being used, the spindle is positioned depending on the active coordinate rotation in the plane (rotation around the applicate (feed axis)) so that the same positions of the probe sphere are probed when calibrating and measuring.
Note: Pay attention to **_CHBIT[14]**!

Notice: If additional rotations are active in the other planes/ axes, then this function is not effective!

Adapt spindle positioning, if _CHBIT[13]=1

_CHBIT[14]=0: Spindle positioning is performed acc. to default.
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 pos. is performed in 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 active plane is:

- 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,

© Siemens AG, 2004. All rights reserved

Number of measurements on failure to switch

_CHBIT[15]=0: A maximum of 5 measuring attempts are made before alarm "Probe not switching" is generated.

_CHBIT[15]=1: After **one** unsuccessful measurement attempt, the fault "measuring sensor does not switch" is generated.

Retraction velocity from the measuring point

_CHBIT[16]=0: The retraction from the measuring point is realized with the same velocity as for an intermediate positioning operation.

_CHBIT[16]=1: The retraction velocity is realized with the percentage rapid traverse velocity, defined in SPEED[0] and is only effective when collision monitoring is active (**_CHBIT[2]=1**).

Feed during measurement

_CHBIT[17]=0: Measurement is performed with the feed programmed in _VMS. Pay attention to consequences if _VMS=0!

_CHBIT[17]=1: Initially, the axis traverses with the measuring feed _SPEED[3], after switching, there is a retraction of 2 mm from the measuring position and the actual measurement starts with the feed from _VMS.
Measurement with the feed from _SPEED[3] is only realized for a measurement distance/ travel > 2 mm.

Static measurement result display

_CHBIT[18]: Explanation -> see _CHBIT[10], _CHBIT[11]

Special treatment of Y axis with G18 in CYCLE974 or CYCLE994

_CHBIT[19]=0: No special treatment for Y axis (applicate)

_CHBIT[19]=1: Setpoint setting and parameterization (_SETVAL, _TUL, _TLL, SZO) for the Y axis (applicate) are performed as for parameterization of the ordinate (X axis).
The tool offset is applied to the length that is active in the ordinate (X axis) (usually L1), as long as other lengths are not specified by _KNUM.
The ZO correction is applied in the specified ZO memory in the ordinate component (X axis).

© Siemens AG, 2004. All rights reserved

Suppression of the starting angle positioning _STA1 in CYCLE982

_CHBIT[20]=0: For certain measuring variants, the milling spindle is positioned with _STA1.
_CHBIT[20]=1: When measuring milling tools, for basic measuring variants, it is possible to suppress positioning of the milling spindle to the value of the starting angle _STA1. This is possible for the following measuring variants to measure milling tools: _MVAR=xxx001 (where x: 0 or 1, no other values)

ZO mode in CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE997

_CHBIT[21]=0: The offset is applied additively in FINE, if MD 18600: MM_FRAME_FINE_TRANS=1, otherwise in COARSE.
_CHBIT[21]=1: The offset is applied in COARSE. FINE is taken into account and is then subsequently deleted.

Speed reduction in tool measurement in CYCLE971

_CHBIT[22]=0: When measuring tools with rotating spindle and when the speed (**_CBIT[12]=0**) is calculated in the cycle, for multiple measurements, the last measurement is carried-out at a reduced speed.
_CHBIT[22]=1: For multiple measurements with the spindle rotating and calculation in the cycle, the speed remains constant.

Re-coding of the cutting edge position for tool measurement in CYCLE982 (from measuring cycles SW 6.3)

_CHBIT[23]=0: Standard setting, no recoding
_CHBIT[23]=1: Internal recording, tool point direction mirroring about X axis (tool turret -180 degrees, Z not mirrored)

Channel-oriented marker for basic dimension system

_CHBIT[24]=0: Basic system is based on inches
_CHBIT[24]=1: The basic system is metric

When starting up the measuring cycles, this bit has to be set according to the basic settings of the PLC (MD 10240). (MD 10240: SCALING_SYSTEM_IS_METRIC) must be set.
If modifying the basic settings of the control results in _CBIT[24] no longer matching MD 10240, the channel-specific dimensioned values are converted in each channel the first time a measuring cycle is called.

© Siemens AG, 2004. All rights reserved

9.3 Data for measuring in JOG

9.3.1 Machine data for ensuring ability to function

11602 MD number	**ASUP_START_MASK** Ignore stop conditions for ASUB	
Default setting: 0 **for measuring in JOG: 1, 3 (Bit0=1)**	Minimum input limit: 0	Maximum input limit: 3
Changes effective after POWER ON	Protection level: 2/7	Unit: -
Data type: DWORD	Applies from SW 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 effective	
Default setting: 0 **for measuring in JOG: 1 - 64H**	Minimum input limit: 0	Maximum input limit: 64H
Changes effective after POWER ON	Protection level: 2/7	Unit: HEX
Data type: INT	Applies from SW version: SW 4.1	
Meaning:	"ASUP_START_MASK" included from ASUB priority "64H" to ASUB priority 1.	

20050 MD number	**AXCONF_GEOAX_ASSIGN_TAB[i]** Assignment of geometry axes to channel axes [i = Geometry axis index]: 0 ... 2	
Default setting: 1, 2, 3 **for measuring in JOG: all 3 geometry axes must be present.**	Minimum input limit: 0	Maximum input limit: 15
Change applies after POWER_ON:	Protection level: 2/7	Unit:
Data type BYTE	Applies from SW version: SW 1	
Meaning:	The MD specifies channel axes onto which the axes of the Cartesian coordinate system (WCS) are mapped. Index i assumes values 0, 1, 2 and refers to the 1st to 3rd geometry axis.	

20110 MD number	**RESET_MODE_MASK** Define control default setting after power-up and RESET	
Default setting: 0 **for measuring in JOG: min. 4045H (Bit0=1, Bit2=1, Bit6=1, Bit14=1)**	Minimum input limit: 0	Maximum input limit: 07FFFH
Changes effective after RESET	Protection level: 2/7	Unit: HEX
Data type: DWORD	Applies from SW version: SW 2	
Meaning:	Bit 0: 1 tool length offset active Bit 2: 1 } The tool length correction last selected is active after POWER ON Bit 6: 1 } Bit 14: 1 The current setting of the basic frame is retained.	

20112 MD number	**START_MODE_MASK** Define control default setting after part program start	
Default setting: 400H **for measuring in JOG 400H (Bit6=0)**	Minimum input limit: 0	Maximum input limit: 07FFFH
Changes effective after RESET	Protection level: 2/7	Unit: HEX
Data type DWORD	Applies from SW version: SW 3.2	
Meaning:	Bit 6: 0 active tool length correction is retained	

© Siemens AG, 2004. All rights reserved

20310 MD number	**TOOL_MANAGEMENT_MASK** Channel-specific activation of tool management Note: MD only relevant if tool management is active (Bit0=1)				
Default setting: 0 if Bit0=1 **for measuring in JOG: min. 4001H (Bit0=1, Bit14=1)**		Minimum input limit: 0		Maximum input limit: 0FFFFhex	
Changes effective after POWER ON			Protection level: 2/7		Unit: HEX
Data type DWORD				Applies from SW version: SW 2	
Meaning:	Bit 0: 1 Tool management active Bit 14: 1 Automatic tool change on RESET and START according to MD 20110: RESET_MODE_MASK				

24006 MD number	**CHSFRAME_RESET_MASK** Reset behavior of the channel-specific system frame, actual value setting and scratching (basis reference) Note: MD only relevant if frames are configured (MD 28082 SYSTEM_FRAME_MASK)				
Default setting: 0 **for measuring in JOG: min. 1 (Bit0=1)**		Minimum input limit: 0		Maximum input limit: 07FFFH	
Changes effective after RESET			Protection level: 2/7		Unit: HEX
Data type: DWORD				Applies from SW version: SW 2	
Meaning:	Bit 0: 1 system frame for PRESET and scratching (basis reference) is active after reset.				

24007 MD number	**CHSFRAME_RESET_CLEAR_MASK** Clear system frames on reset Note: MD only relevant if frames are configured (MD 28082 SYSTEM_FRAME_MASK)				
Default setting: 0 **for measuring in JOG: Bit 0=0**		Minimum input limit: 0		Maximum input limit: 07FFFH	
Changes effective after RESET			Protection level: 2/7		Unit: HEX
Data type: DWORD				Applies from SW version: SW 2	
Meaning:	Bit 0:0 system frame for PRESET and scratching (basis reference) is not deleted at reset.				

© Siemens AG, 2004. All rights reserved

9.3.2 Modifying the GUD7 data block

Function

Measurement in JOG requires data definitions in data block GUD7.DEF.
Different procedures are followed depending on the configuration of the overall system.

1. If the "ShopMill" programming interface is installed on the control, then all of the required GUD data are already active. No reloading or additional modification of definition blocks is required.
2. From SW 6.02.17 of the measuring cycle package, the definition files GUD7.DEF and SMAC.DEF – are split-up. Now a differentiation is made between basis definition files and application-specific definition files. All applications, measuring cycles, technology cycles, ShopMill etc. only incorporate their own application-specific definitions. For this purpose, new cycle files GUD7_xxx.DEF and SMAC_xxx.DEF have been introduced, and are located in the HMI data management, in the definition directory DEF.DIR. This means, when measuring in JOG, from measuring cycles SW 6.02.17, the data block GUD7_MC.DEF supplied must be used. A secondary condition/limitation is that file GUD7.DEF must be used – also for measuring cycles from SW 6.02.17.
   For installation, refer to Section 10.1.

Commissioning "measuring in JOG" for the first time, up to measuring cycles SW 6.2.16

3. In this case, definition block GUD7.DEF must be modified. Select definition file GUD7.DEF in menu "Services" in directory "Definitions" with the arrow keys and unload it by pressing the softkey "Unload". Before modifying data in GUD7, the "old" data should be backed-up (archived). 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 affects the definition lines as from the comment "Definitions for measurement in JOG and ShipMill" (starting with data item E_MESS_IS_METRIC).

© Siemens AG, 2004. All rights reserved

```
DEF CHAN INT      E_MESS_IS_METRIC=1
DEF CHAN INT      E_MESS_IS_METRIC_SPEZ_VAR=1
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_F_FAK1=10
DEF CHAN REAL     E_MESS_F_FAK2
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_MT_D=10
DEF CHAN REAL     E_MESS_MT_FM=100
DEF CHAN INT      E_MESS_MT_CF
DEF CHAN INT      E_MESS_MT_COMP
DEF CHAN REAL     E_MESS[3]
DEF CHAN REAL     E_MEAS
DEF CHAN BOOL     E_MESS_SETT[10]
```

Possibility of minimizing the memory requirements

4. The number of available data fields (data arrays) regarding the measuring probes that can be connected, can be adapted by the machinery manufacturer to the specific relationships and situations. When supplied, 3 data fields (data arrays) are available for tool measuring probes, workpiece measuring probes and calibration gauging blocks. If, for example, in the specific application case, only one data field (data array) is required for the workpiece and the tool measuring probe, and there is no gauging block available, then the appropriate parameters can be adapted in data block GUD6 and in GUD7.DEF and GUD7_MC.DEF. The details can be taken from the following example.

Example:

There is only one tool and workpiece measuring probe on a specific milling machine. The tool is to be only gauged in the interpolation plane G17.

In order to minimize the NC memory requirement in the SINUMERIK, the following changes are required:

- GUD7.DEF or GUD7_MC.DEF

```
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)
DEF CHAN INT      E_MESS_CALL_D[1]=SET(0) from measuring cycles SW 6.3
DEF CHAN INT      E_MESS_CALL_L[1]=SET(0) from measuring cycles SW 6.3
```

© Siemens AG, 2004. All rights reserved

- GUD6.DEF
N10 DEF NCK INT _CVAL[4]=(**1**,**1**,0,0) ; 1x tool and 1x workpiece measuring probe data set
N11 DEF NCK REAL _TP[**1**,10]=(0,0,0,0,0,0,0,133,0,2)
N12 DEF NCK REAL _WP[**1**,11]
;N13 DEF NCK REAL _KB[3,7]
;N111 DEF NCK REAL _TPW[3,10]

The definition blocks should be opened in the HMI area Services\Definitions using the operator panel "INPUT key" and after editing, re-closed using the vertical softkey "Close editor"; i.e. to save and to activate.
When supplied, the following basic settings are active:

E_MESS_IS_METRIC=1 E_MESS_IS_METRIC_SPEZ_VAR=1	INT	All dimensioned data are metric
E_MESS_MS_IN=0	BOOL	Work piece probe connected to measuring input 1
E_MESS_MT_IN=1	BOOL	Tool probe connected to measuring input 2
E_MESS_D=5	REAL	Internal data for measuring in JOG not relevant
E_MESS_D_M=50	REAL	Measuring path for manual measuring [mm] (in front of and behind the measuring point)
E_MESS_D_L=2	REAL	Measuring path for length measurement [mm] for tool measurement (in front of and behind the measuring point)
E_MESS_D_R=1	REAL	Measuring path for radius measurement [mm] for tool measurement (in front of and behind the measuring point)
E_MESS_FM=300	REAL	Measuring feedrate [mm/min] for workpiece measurement and calibration
E_MESS_F=2000	REAL	Plane feedrate for collision monitoring [mm/min]
E_MESS_FZ=2000	REAL	Infeed feedrate for collision monitoring [mm/min]
E_MESS_CAL_D=[3] (from measuring cycles SW 6.3)	REAL	Diameter, calibration ring [mm] 0
E_MESS_CAL_L=[3] (from measuring cycles SW 6.3)	REAL	Calibration height in the feed axis (referred to the WCS) [mm] 0
E_MESS_MAX_V=100	REAL	Max. peripheral speed for measuring with rotating spindle [m/min]
E_MESS_MAX_S=1000	REAL	Max. spindle speed for measuring with rotating spindle [rpm]
E_MESS_MAX_F=20	REAL	Max. feedrate for measuring with rotating spindle [mm/min]
E_MESS_MIN_F=1	REAL	Min. feedrate for measuring with rotating spindle on 1st probing [mm/min]
E_MESS_F_FAK1=10	REAL	On tool measurement with rotating spindle, traversal with 10 times measuring feed is performed in the 1st probing (limitation by E_MESS_MAX_F)

© Siemens AG, 2004. All rights reserved

E_MESS_F_FAK2=0	REAL	For tool measurement with rotating spindle, 2nd probing is performed with measuring feed. There is no 3rd probing.
E_MESS_MIN_D=0.01	REAL	Measuring accuracy for measuring with rotating spindle [mm/min]
E_MESS_MT_TYP[3]=SET(0,0,0)	INT	3 arrays for tool measuring probe; tool measuring probe type; cube
E_MESS_MT_AX[3]=SET(133,133,133)	INT	Permissible axis directions for tool probe in X and Y in plus and minus direction, in Z in minus direction only
E_MESS_MT_DL[3]	REAL	Active diameter of tool measuring probe for length measurement 0
E_MESS_MT_DR[3]	REAL	Active diameter of tool measuring probe for radius measurement 0
E_MESS_MT_DZ[3]=SET(2,2,2)	REAL	Distance between upper edge of tool probe and lower edge of tool probe [mm] when measuring the tool radius 2
E_MESS_MT_DIR[3]=SET(-1,-1,-1)	INT	Approach direction in plane of tool measuring probe for tool measurement (minus direction in 1st plane axis) -1
E_MESS_MT_D=10	REAL	Calibrate measurement path for tool probe and tool measurement with motionless spindle (before and after expected switching position)
E_MESS_MT_FM=100	REAL	Calibrate measuring feed for tool probe and tool measurement with motionless spindle
E_MESS_MT_CF=0	INT	No tool probe make (manufacturer) specified
E_MESS_MT_COMP=0	INT	No offset of the measurement result on tool measurement with rotating spindle
E_MESS[3]	REAL	Internal values
E_MEAS	REAL	Internal values
E_MESS_RETT	FRAME	Internal values
E_MESS_SETT[10]	BOOL	Field for settings
E_MESS_AM	BOOL	Internal values

Attention!

Arrays E_MESS_MT_DL[] and E_MESS_DR[] (active diameter, width of tool measuring probe for length/radius measurement) must be assigned.

	E_MESS_SETT[ ] Field for settings		
	Minimum input limit: 0	Maximum input limit: 1	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: BOOL	Applies from SW version: 6.2		
Meaning:	E_MESS_SETT[0] 0: Spindle coupling with coordinate rotation 1: Spindle position on cycle call is initial position E_MESS_SETT[1...9] internal date	Preset default 0	

9.3.3 Settings in data block GUD6

Function

The channel specific arrays _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[10]=SET(0,1,1,17,100,0,0,0,0,0)

	_JM_I[ ] INT value field for JOG measurement		
	Minimum input limit: -	Maximum input limit: -	
Changes valid after value assignment	Protection level: -	Unit: -	
Data type: INT	Applies from SW version: 5.3		
Meaning:			Preset default
	_JM_I[0]	Setting for the workpiece probe number 0: set by _JM_I[1] 1: set as with ShopMill G17 plane Workpiece probe number 1 G18 plane Workpiece probe number 2 G19 plane Workpiece probe number 3	0
	_JM_I[1]	Probe number and probe type for workpiece measurement Only active if _JM_I[0]=0	1
	_JM_I[2]	Probe number for tool measurement	1
	_JM_I[3]	plane 17: measurement in G17 plane 18: measurement in G18 plane 19: measurement in G19 plane Each other value: Measurement made in the plane defined by the machine data	17
	_JM_I[4]	Definition of the active ZO on measurement 0: measurement made with G500 1...99: Measurement is made with the defined, settable zero offset G54...G57 or G505...G599 where 1: G54...4: G57 5...99: G505...G599 100: measurement made with ZO defined in machine data	0
As from measuring cycle version SW 6.3	_JM_I[5]	0: Standard offset (only measurement, basis reference, settable ZO 1: Extended offset (additional global and channel-specific basis frame)	0
	_JM_I[6]	Internal copy of _JM_I[4]	
	_JM_I[7]	Reserved	
	_JM_I[8]	Reserved	
	_JM_I[9]	Reserved	

© Siemens AG, 2004. All rights reserved

	_JM_B[ ] BOOL value field for JOG measurement
	Minimum input limit: - / Maximum input limit: -
Changes valid after value assignment	Protection level: - / Unit: -
Data type: BOOLEAN	Applies from SW version: 5.3
Meaning:	Preset default _JM_B[0] Tool offset mode for tool measurement — 0 0: offset in geometry on tool measurement 1: Offset in wear _JM_B[1] Number of measurement attempts — 1 0: 5 measurement attempts 1: 1 measurement attempt _JM_B[2] Retraction from the measuring location — 0 0: retraction as for intermediate positioning 1: retraction with rapid traverse _JM_B[3] Fast measuring feed — 0 0: measuring with measuring feed 1:1. measuring with feed in _SPEED[3] 2. measuring with measuring feed _JM_B[5] Not assigned — 0 _JM_B[6] Internal data item — 0
Up to measuring cycle version SW 6.2	_JM_B[4] 0: Selection "basic frame" in the screen means offset in last channel-specific basic frame — 0 1: Selection of "basis" in screen form means offset in system frame "set zero" ($P_SETFRAME)

9.3.4 Loading files for measuring in JOG

Function

The files stored in the JOG_MEAS\CYCLES\SPFFILES directory in the software supplied

CYC_JM.SPF	Auxiliary cycle for measuring
CYC_JMA.SPF	Aux. cycle for activation (as from SW 6.3)
CYC_JMC.SPF	Aux. cycle for calculation
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_MS_POC.SPF	For measurement of a rectangular pocket (from measuring cycles SW 6.3)
E_MS_SPI.SPF	For measurement of a rectangular spigot (from measuring cycles SW 6.3)
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
E_SP_NPV.SPF	Auxiliary cycle for activating the ZO that is active during measurement

© Siemens AG, 2004. All rights reserved

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.

When performing a software upgrade, make sure the variable values in the existing DEF files are kept. Back them up first, if necessary!

The other files

JOG_MEAS.COM (up to meas. cycles SW 6.2) COM_FILE.COM (from meas. cycles SW 6.3)	Configuring file for measuring in JOG user interface
MA_JOG.COM (up to meas. cycles SW 6.2) COM_FILE.COM (from meas. cycles SW 6.3)	Configuring file for the softkeys for measuring in JOG in the JOG basic display
BMP_FILE.ARC (up to measuring cycles SW 6.2) BMP_FILE.EXE	Help displays for measuring in JOG
MC_CST0.ARC (from measuring cycles SW 6.3) or MC_CST1.ARC	All screens/displays

must also be transferred to the control.

BMP_FILE.EXE contains:

- BMPJ640.EXE — Aux. displays, resolution 640 · 480
- BMPJ800.EXE — Aux. displays, resolution 800 · 600
- BMPJ1024.EXE — Aux. displays, resolution 1024 · 768

Note.
Up-to-date information is provided in the file SIEMENSD.TXT or SIEMENSE.TXT of the software being supplied.

■

© Siemens AG, 2004. All rights reserved

Start-Up

© Siemens AG, 2004. All rights reserved

10.1 Commissioning measuring cycles for the first time

Preconditions

- The hardware and software requirements of the measuring cycle version have been fulfilled (see Chapter "Hardware, software and installation").
- The probe is functional (Functional check probe connection → see Chapter "Hardware, software and installation").
- Be sure to archive the existing state before beginning installation; export series start-up archive now, if necessary.

NOTICE!
Please heed the information on measuring cycle start-up in connection with ShopMill / ShopTurn in file SIEMENSD.TXT / SIEMENSE.TXT!

Step 1 – set up memory configuring machine data

- Set memory configuring machine data for measuring cycles (→ for data see Section 9.1 "Machine data for executing measuring cycles" and the **up-to-date information** in SIEMENSD.TXT / SIEMENSE.TXT included with the delivered software).
- The following machine data must be observed:
  - Machine data for GUDs
    MD18118: MM_NUM_GUD_MODULES
    MD18120: MM_NUM_GUD_NAMES_NCK
    MD18130: MM_NUM_GUD_NAMES_CHAN
    MD18150: MM_GUD_VALUES_MEM
  - Machine data for file systems
    MD18280: MM_NUM_FILES_PER_DIR
    MD18320: MM_NUM_FILES_IN_FILESYSTEM
  - Further machine data
    MD28082: MM_SYSTEM_FRAME_MASK
- Export and reimport NC series start-up.

© Siemens AG, 2004. All rights reserved

Step 2 – set up other machine data

- Set other machine data for measuring cycles (→ for data see Section 9.1 "Machine data for executing measuring cycles" and the **up-to-date information** in SIEMENSD.TXT / SIEMENSE.TXT included with the delivered software).
- The following machine data must be observed:
  - Machine data for cycle interface
    MD18170: MM_NUM_MAX_FUNC_NAMES
    MD18180: MM_NUM_MAX_FUNC_PARAM
    MD28020: MM_NUM_LUD_NAMES_TOTOL
    MD28040: MM_NUM_LUD_VALUES_MEM
    MD10132: MMC_CMD_TIMEOUT
    MD11420: LEN_PROTOCOL_FILE
    MD13200: MEAS_PROBE_LOW_ACTIV
- Perform an NC reset

Step 3 – load definition files

- Load GUD5.DEF and GUD6.DEF via "Services", "Data in" from diskette or via RS232
- Select blocks singly and press softkey "Activate"
- Go to step 4 without measuring in JOG

Only for measuring in JOG up to measuring cycles SW 6.02.16:

- Adjust GUD7.DEF (→ refer to Subsection 9.3.2 "Modifying the GUD7 data block")
- If a GUD7.DEF block was already active in the NCU, save the current values of existing GUD variables from the NCU (with "Services", "Data out", "NC-active data" NCK/channel, select block) and then reload.

Only when measuring in JOG from measuring cycles SW 6.3 (also from patch measuring cycles SW 6.02.17) with HMI Advance from SW 6.3:

- If GUD7.DEF is already active in the NCU, unload it (but first save current values of existing GUD variables)
- Import file GUD7_MC.DEF from diskette and activate it
- Re-activate GUD7.DEF (from Patch measuring cycles SW 6.02.17) or if previously not available, download and activate from "Archive\cycles archive".
- GUD7.DEF and GUD7_MC.DEF must always have the same version.

© Siemens AG, 2004. All rights reserved

Step 4 – load cycle programs

- Import cycle archive files (or single cycles) from diskette or via RS232.
- Load new cycles into the NCU (in the case of HMI Advanced: if now archive file was used or cycles were added).
- NC reset to update the cycle interface

Step 5 – set measuring cycle data

- Check default values of the GUD variables and set other values, if necessary.
- That is done by selecting and changing the variables in "Parameters", "User data" … or using a program (see Section 9.3).
- Data description (see → Section 9.2 "Cycle data", Section 9.3 "Data for measuring in JOG" and **up-to-date information** in SIEMENSD.TXT / SIEMENSE.TXT included with the software delivered).

Step 6 – load measuring cycle texts

When using measuring cycles with HMI up to SW 6.2, the measuring cycle text files have to be loaded afterward. If you require languages other than the standard languages, please order the appropriate HMI language versions.

Step 6 for HMI Advanced

- Load the archive file
  HMI_ADV\MC_TEXT.COM
  from diskette or via RS232.
- It contains the measuring cycle text files in the five standard languages.

Step 6 for HMI Embedded

- The text files for HMI Embedded are supplied singly (→ see diskette structure SIEMENSD.TXT / SIEMENSE.TXT).
- Out of the language-specific files ALMC.TXT and ALZ.XT, select the files of the required language path.
- The text files have to be linked in the HMI Software (→ see Upgrade Instructions for HMI Embedded).

© Siemens AG, 2004. All rights reserved

Step 7 – load files for measurement result display

Step 7 for HMI Advanced

- Load the files for HMI Advanced
  - HMI_ADV\MCRESULT.COM,
  - HMI_ADV\BMP_RESU.ARC
  via RS232 or from floppy disk (from measuring cycles SW 6.3: Pictures/images are contained in file MC_CSTx.ARC).
- After loading the files for the measurement result displays, check and adjust the entry for the text format in system file HMI **MMC2\MBDDE.INI**
  Change the entry to display the measurement result displays from
  - MEASURE_CYCLES_RESULT=DOS
  to
  - MEASURE_CYCLES_RESULT=
    (delete "DOS"!).

Step 7 for HMI Embedded

- Link the files for HMI Embedded
  - HMI_EMB\MCRESULT\MZBILD01.COM
  - HMI_EMB\MCRESULT\MZBILD02.COM
  - HMI_EMB\MCRESULT\MZBILD03.COM
  - HMI_EMB\MCRESULT\MZBILD04.COM
  into the HMI software via the application.
- Integrate the measurement result displays into the display file for programming a flash card:
  - Requirement:
    - The system and application diskette has already been installed on your PC.
    - The MZBILDnn.COM files have been packed as MZBILDnn.CO_.

      Commands for packing:
      arj a MZBILD01.CO_ MZBILD01.COM
      arj a MZBILD02.CO_ MZBILD02.COM
      arj a MZBILD03.CO_ MZBILD03.COM
      arj a MZBILD04.CO_ MZBILD04.COM
  - Sequence:
    - Change to directory "INSTUTIL" in your application path and start "APP_INST.EXE". The selection menu for software installation is displayed.
    - Select menu item < 2 > "Modify configuration".

© Siemens AG, 2004. All rights reserved

A further selection menu appears. Here, select item < 6 > "Add *.* Files ...". As the file name enter your graphics files path and file name in the input screen form.
e.g.: A: \HMI_EMB\MCRESULT\MZBILD01.CO_
- Press the Return key to confirm your input.
- The "Esc" key will take you back to the basic menu and you can now create your software as "PCU_20.ABB". That programs the flash card.
Now proceed according to the start-up instructions for HMI Embedded.

Step 8 – load and activate measuring cycle support

Step 8 for HMI Advanced

- From the software supplied located in subdirectory hmi_emb, the following files are required from directory HMI_ADV:
  - HMI_ADV\BMP_xxx.EXE (from measuring cycles SW 6.3: Pictures/images are contained in file MC_CSTx.ARC) xxx stands for the screen resolution
  - HMI_ADV\MCSUPP.COM
    HMI_ADV\AEDITOR.COM
    HMI_ADV\STARTUP.COM
- Handling the EXE file:
  - This file is not loaded into the control, but initially has to be loaded onto a PC. Starting this file automatically unpacks it, creating the unpacked archives:
    MZ_BMP01.ARC ... MZ_BMPxx.ARC
  - These must be copied onto floppy disk and downloaded into the HMI or via V.24 (binary format). These archives contain the bitmaps for measuring cycles support.
  - Load MCSUPP.COM, AEDITOR.COM, and STARTUP.COM from diskette or via RS232, then restart HMI.

Note:
Up-to-date information is provided in the file SIEMENSD.TXT or SIEMENSE.TXT of the software being supplied.

© Siemens AG, 2004. All rights reserved

Step 8 for HMI Embedded

- To pack the COM files, a tool is provided in MAKE_COM.BAT.
- Sequence:
  Set-up an empty directory and copy the following files there:
  - HMI_EMB\MCSUPP*.COM
  - HMI_EMB\TOOLS\ARJ.EXE
  - HMI_EMB\TOOLS\MAKE_COM.BAT
- Start MAKE_COM.BAT
- The procedure to follow with the *.CO_ files is the same as that for linking the measuring result displays.
- Also add the archive file of bitmaps supplied CST.ARJ or CST_10.ARJ (depending on screen resolution) via menu item < 6 > "Add *.* Files ...".
- Adapt file common.com and then load the following file into the NCU via RS232
  - HMI_EMB\MCSUPP\COMMON.COM
  - Adaptation: Remove semicolon in front of SC617, SC326, and SC327

Step 9 – configure measuring cycle support

The measuring cycle screen forms can be set via GUD field _MZ_MASK. It is of type integer and is located in the GUD6 under NCK global data. In the "Start-up" operating area under softkey "Meas. cycles", "Manage data", a start-up screen form appears allowing you to make the settings.

Step 10 – load and activate measurement in JOG (with HMI Advanced only)

- Set machine data for measurement in JOG
  (→ for data see Section 9.3 "Data for measurement in JOG")
- The following machine data must be observed:
  MD11602: ASUP_START_MASK
  MD11604: ASUP_START_PRIO_LEVEL
  MD20110: RESET_MODE_MASK
  MD20112: START_MODE_MASK
- Load the files for measurement in JOG (→ see Subsection 9.3.4 "Loading files for measurement in JOG")
- Activate softkeys for measurement in JOG in file MA_JOG.COM (by deleting comments)
- Restart the HMI

© Siemens AG, 2004. All rights reserved

10.2 Upgrading measuring cycles

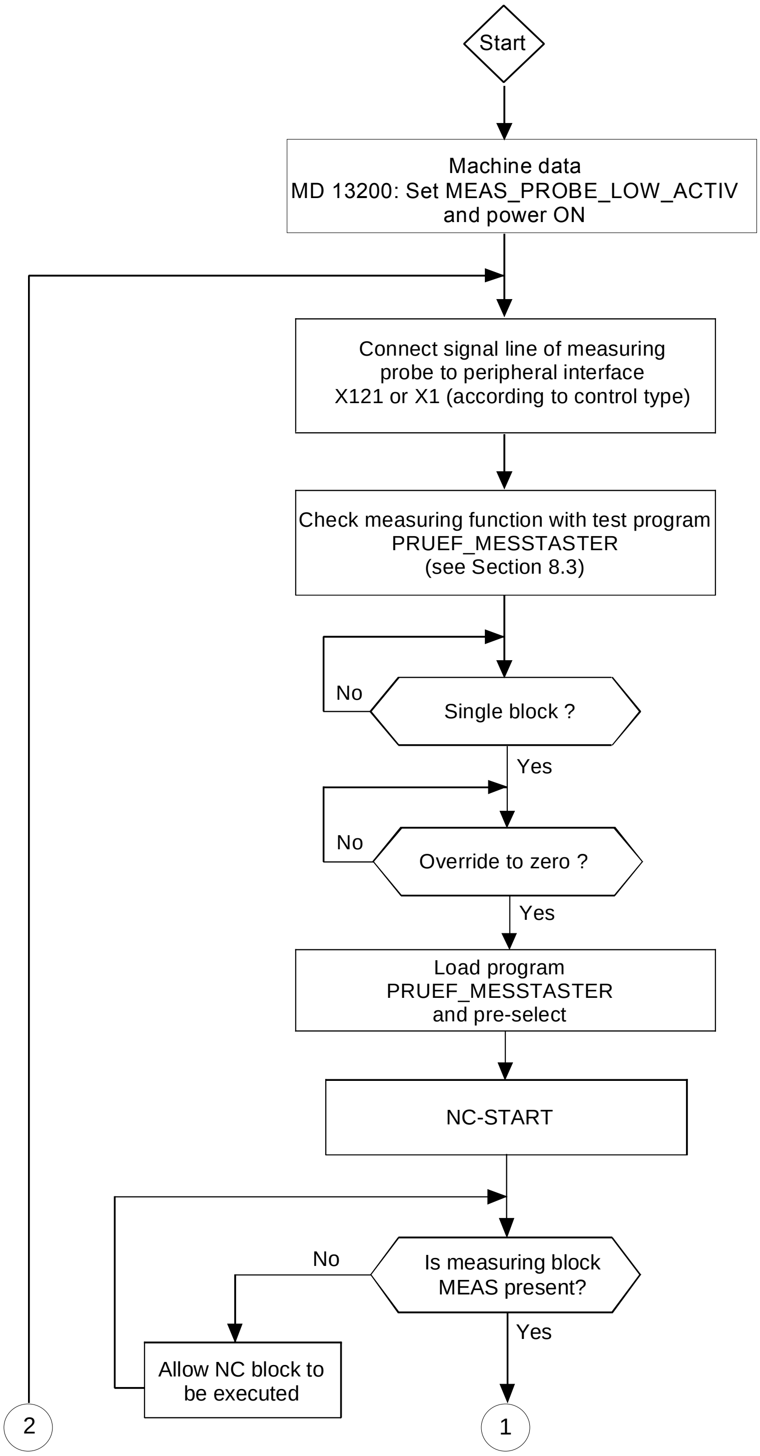
Explanation

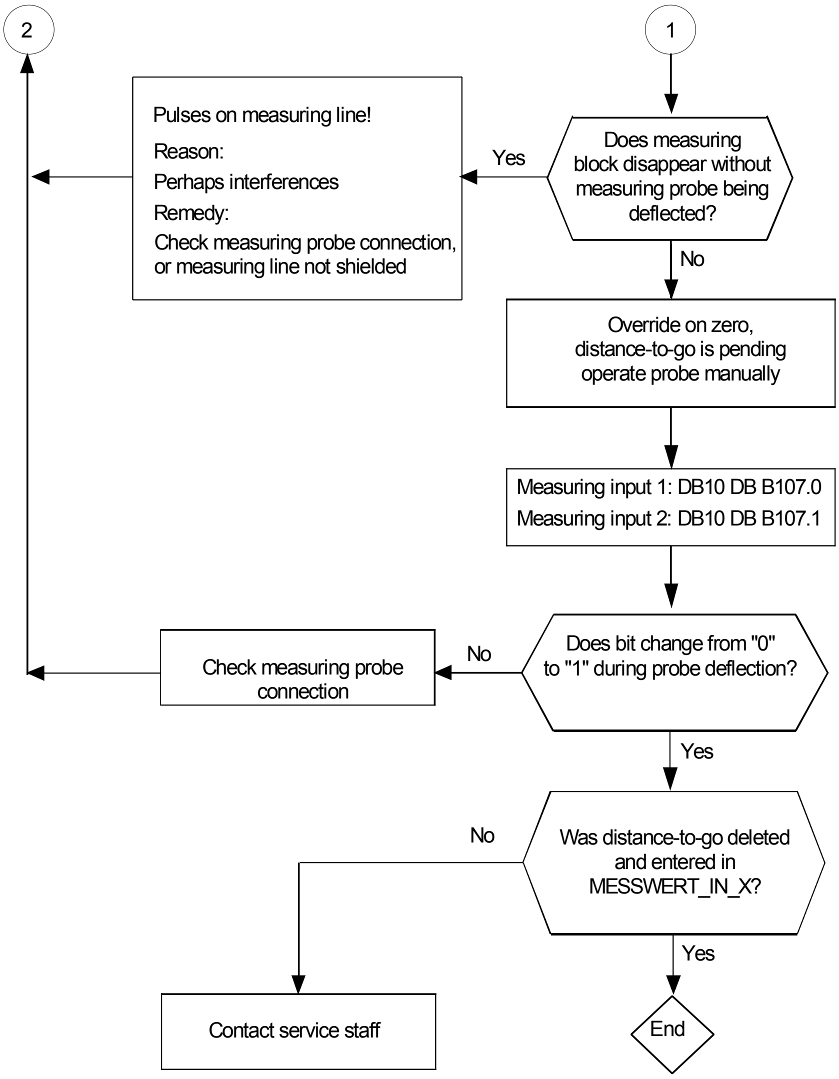
Upgrading measuring cycles is basically performed in the same order as initial installation.
Special notes:

- Only use files with the same measuring cycle software version. Do not mix files with different versions!
- Always heed the up-to-date information in SIEMENSD.TXT / SIEMENSE.TXT.
- Whether you need to increase the memory configuring machine data (step 1) depends on the state of the machine and the previous measuring cycle version.
- When performing a software upgrade, make sure the variable values in the existing DEF files are preserved. If necessary, save them first (step 3).
- Adaptation of the measuring cycle data usually only necessary when upgrading an original version 5.x (or 4.x); when upgrading within SW 6 it is enough to reconstruct the set values by previously archiving of the NC-active data (except where other settings are explicitly prescribed in SIEMENSD.TXT / SIEMENSE.TXT).

© Siemens AG, 2004. All rights reserved

10.3 Sequence for probe installation





© Siemens AG, 2004. All rights reserved

10.4 Example of calculating the repeat accuracy

Function

Test program

This program allows the measuring scatter (repeat accuracy) of the entire measuring system (machine-probe-signal transmission to NC) to be calculated.

In the example, ten measurements are taken in the X axis and the measured value recorded in the workpiece coordinates.
It is therefore possible to determine the random dimensional deviations, i.e. those that are not subject to any trend.

Example:

%_N_CHECK_ACCURATE_MPF		
;$PATH=/_N_MPF_DIR		
N05	DEF INT SIGNAL, KK	;Variable definition
N10	DEF REAL MEASVALUE_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 X50	;Measurement at 1st measurement ;input with switching signal from "not ;deflected" to "deflected" in the X axis. ;Probe switching expected between X0 and ;X50.
N30	STOPRE	;Stop decoding for subsequent ;evaluation of result (pre-processing stop)
N35	SIGNAL= $AC_MEA[1]	;Read software switching signal at ;1st measurement input
N37	IF SIGNAL == 0 GOTO _ERROR1	;Check switching signal
N40	MEASVALUE_IN_X[kk]=$AA_MW[X]	;Read measured value in workpiece ;coordinates
N50	KK=KK+1	
N60	IF KK<10 GOTOB ANF	;Repeat 10 times
N65	M0	
N66	STOPRE	;Preprocessing stop
N70	M2	
N80	_ERROR1: MSG ("Probe is not switching")	
N90	M0	
N95	M2	

© Siemens AG, 2004. All rights reserved

Explanation

After the parameter display (user-defined variables) have been selected, the measurement results can be read in field MEASVALUE_IN_X[10] provided that the program is still being processed.

■

© Siemens AG, 2004. All rights reserved

Alarms

General information

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 range is again subdivided according to alarm responses and acknowledgment criteria.
The text displayed with the alarm number provides an explanation of the cause of the error.

Alarm number	Delete criterion	Alarm responses
61000 ... 61999	NC_RESET	Block preprocessing in the NC is aborted
62000 ... 62999	Backspace key	Program execution is not interrupted; display only.

© Siemens AG, 2004. All rights reserved

Overview of 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	Explanation, remedy
61016	"System frame for cycles missing"	All	MD 28082: MM_SYSTEM_FRAME_MASK, set bit 5=1
61230	"Tool probe diameter too small"	E_MT_CAL E_MT_RAD E_MT_LEN	Correct in the data block GUD7 variable E_MESS_MT_DR[n] or E_MESS_MT_DL[n] for probe n+1 (measuring in JOG)
61301	"Probe not switching"	All	• Check measuring 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"	CYCLE971 CYCLE972 CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE982 CYCLE994	• Check setpoint • Increase parameter _TDIF
61307	"Incorrect measuring 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 CYCLE982	Tool type of workpiece probe in TO memory is not allowed
		CYCLE971	No permissible tool probe type entered in _TP[x,8], or in the case of tool probe type "disk" check the permissible working plane G17...G19.
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 array _TP[] or _WP[] for additional tool or workpiece probes and adapt _CVAL[0]/_CVAL[1] accordingly.

© Siemens AG, 2004. All rights reserved

Alarm number	Alarm text	Source	Explanation, remedy
61314	"Check selected tool type"	CYCLE971 CYCLE972 CYCLE982	Tool probe not permitted for tool measurement/tool probe calibration.
61315	"Check tool edge position"	CYCLE972 CYCLE973 CYCLE974 CYCLE982 CYCLE994	Check cutting 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 fault 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"	CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE994 CYCLE998	The ZO with the number specified in _KNUM does not exist.
61322	"Check 4th digit in _KNUM"	CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE994 CYCLE998 CYCLE114	The specified digit of _KNUM contains invalid values. Also check _MVAR!
61323	"Check 5th digit in _KNUM"		
61324	"Check 6th digit in _KNUM"		
61325	"Check measuring axis/offset axis"	All, except 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

© Siemens AG, 2004. All rights reserved

Alarm number	Alarm text	Source	Explanation, 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/_SZO 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 fault 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"	All	Up to meas. cycles SW 6.2 _SI[1] in GUD6 has either no value or a value < 3; from measuring cycles SW 6.3: Upgrade NCK-SW release

© Siemens AG, 2004. All rights reserved

Alarm number	Alarm text	Source	Explanation, 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
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	4. parameter for CYCLE118 is greater than 10.

© Siemens AG, 2004. All rights reserved

Alarm number	Alarm text	Source	Explanation, remedy
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 array _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	Assign values to _PROTVAL[0...5].
61371	"The log produced by the column width and number columns exceed 200 characters per line"	CYCLE105	Reduce the column width or number of columns.
61372	"Selected measuring variant requires an SPOS-capable spindle"	All	Change measuring variant or check machine equipment.
61373	"Mono probe requires an SPOS-capable spindle"	All	Check machine equipment

© Siemens AG, 2004. All rights reserved

Alarm number	Alarm text	Source	Explanation, remedy
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.
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 measuring variants shaft/web. Infeed in the infeed axis caused the sensor response.
61403[1)]	"Internal cycle error in frame calculation"	All	Call SIEMENS hotline
61404[1)]	"Internal cycle error on tool offset"	All	Check dependent tool data
61405[1)]	"Tool environment does not exist in _TENV"	All	Correct name or create this environment
61406[1)]	“Check DL_NUMMER in _DLNUM“	All	
61407[1)]	"Check the 7th digit and higher of _KNUM"	All	
61408[1)]	"Sum offset not available"	All	MD 18080, set bit 8=1
61409[1)]	"Setup offsets not available"	All	MD 18112, set bit 4=1
61410[1)]	"Option or offset size not available"	All	The magnitude to be corrected requires an option or increased MD values.
61411[1)]	"Frame calculation not possible, check values"	CYCLE997 CYCLE119	Check setpoints, actual values
61412[1)]	"Channel basic frame not available"	CYCLE997 CYCLE119	Set MD 28081>0, $P_CHBFRMASK>0
61413[1)]	“Check setpoint ball diameter, _SETVAL <=0"	CYCLE997	Check setpoint for ball diameter
61414[1)]	"Distortion of the triangle over limit"	CYCLE997 CYCLE119	Check setpoints, actual values
61415[1)]	"Check probe/machining plane"	CYCLE971	Use probe permitted for machining plane (_TP[x,8], _TPW[x,8]) or change machining plane
61416[1)]	“Adjust array size _TP[ ]/_CVAL[0]!" or “Adjust array size _WP[ ]/_CVAL[1]!" or “Adjust array size _KP[ ]/_CVAL[2]!" or “Adjust array size _TWP[ ]/_CVAL[3]!"	All	Adjust _CVAL entry for number of probe or gauging block data arrays

1) from measuring cycles SW 6.3

© Siemens AG, 2004. All rights reserved

Alarm number	Alarm text	Source	Explanation, remedy
62303	"Safe area violated"	All	• Check setpoint • Increase parameter _TSA
62304	"Allowance"	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"	CYCLE971 CYCLE972 CYCLE974 CYCLE977 CYCLE978 CYCLE979 CYCLE982 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.
62310	"The max. number of characters per line is limited to 200"	CYCLE105	The max. number of characters per line has been limited to 200.
62311	"The max. number of characters per line _PROTFORM[1] is adjusted"	CYCLE105	The max. number of characters per line _PROTFORM[1] is adjusted.

■

© Siemens AG, 2004. All rights reserved

Adaptation of the Measuring Cycles to Previous Software Versions

© Siemens AG, 2004. All rights reserved

12.1 Adaptation of the measuring cycle version to NC SW versions

Function

Parameter _SI[1] in data block GUD6 is used to adapt measuring cycle versions up to SW 5 to previous NC software versions.
In the delivery status of the measuring cycles, the current software status of the control is entered in parameter _SI[1], i. e. 5 for SW 5. This parameter must be changed to match the measuring cycles <SW 6 to older software releases.
Example:
When using measuring cycles status 5.x.x on a control with SW 4, → _SI[1] = 4
Requirement:
To use the measuring cycles, the software status of the control must be ≥ 3.

© Siemens AG, 2004. All rights reserved

12.2 CYCLE103: Cycle for parameter definition (only up to measuring cycles SW 4.5)

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.

Never ignore information provided by the machine manufacturer!

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 Section 7.2 for a detailed description.

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.

© Siemens AG, 2004. All rights reserved

12.3 Measuring cycle sub-routines

Function

These measuring cycle subroutines are called directly by the cycles. With the exception of CYCLE100, CYCLE101, and CYCLE116, these subroutines cannot be executed by a direct call.

Programming

Cycle	*Function*	*≥ SW 4*	*≥ SW 4.5*	*≥ SW 6.2*
CYCLE100	Activate logging	X		
CYCLE101	Deactivate logging	X		
CYCLE102	Measuring result display			
CYCLE103	Parameter setting in interactive mode			
CYCLE104	Internal subroutine: Measuring cycle interface			
CYCLE105	Internal subroutine: Log	X		
CYCLE106	Internal subroutine: Log	X		
CYCLE107	Output of measuring cycle messages	only up to measuring cycles – SW 6.2		
CYCLE108	Output of measuring cycle alarms	only up to measuring cycles – SW 6.2		
CYCLE109	Internal subroutine: Data transfer		X	
CYCLE110	Internal subroutine: Plausibility checks			
CYCLE111	Internal subroutine: Measuring functions			
CYCLE112	Internal subroutine: Measuring functions			
CYCLE113	Internal subroutine: Log	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: Log	X		

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.

© Siemens AG, 2004. All rights reserved

Programming

CYCLE103

Programming example

Workpiece probe calibration

CALIBRATE_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 length compensation, position probe in hole
N20 CYCLE103	The operator can assign the parameters for calibration cycle CYCLE976 in interactive mode
N25 CYCLE976	Measuring cycle call for calibration in the X-Y plane
N50 M30	End of program

© Siemens AG, 2004. All rights reserved

12.4 Start-up of the measuring cycle user interface for MMC 102 (only up to measuring cycles SW 4.4)

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 some adjustments 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.

Measuring cycle user interface test
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.

■

© Siemens AG, 2004. All rights reserved

Appendix

© Siemens AG, 2004. All rights reserved

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 measuring variant, other parameters, or on the machine configuration

CYCLE 961			Workpiece measurement					
Parameters GUD5	Type	Comparable parameter 840C	Automatic setup of 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	Measurement path in mm Only included if calculated larger than internal value					
_ID	REAL	R19		Retraction in infeed axis, incremental for overtravel of corner if _ID=0 travels around the corner		Retraction in infeed axis, incremental for overtravel of corner if _ID=0 travels around 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						

© Siemens AG, 2004. All rights reserved

CYCLE 961			Workpiece measurement					
Parameters GUD5	**Type**	**Com-parable parameter 840C**	**Automatic setup of 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**			
_KNUM	INTEGER >=0	R10	Without/with automatic 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 **1011...1026** automatic ZO correction in 1st to 16th basic frame **2000** automatic ZO in system frame					
_MA	INTEGER	R30						
_MD	INTEGER	R31						
_MVAR	INTEGER >0	R23	Measuring 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 abscissa 3: Offset in abscissa and ordinate 4: Offset in ordinate				Coordinates of point P3 in the active workpiece coordinate system (abscissa)	

© Siemens AG, 2004. All rights reserved

CYCLE 961			Workpiece measurement					
Parameters GUD5	Type	Comparable parameter 840C	Automatic setup of 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
_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 (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))					

© Siemens AG, 2004. All rights reserved

CYCLE971			**Tool measurement of milling tools on milling machines**			
Parameters GUD5	**Type**	**Comparable Parameters 840C**	**Possible axes** **Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3)** **for G17: X=1 Y=2 Z=3** **for G18: Z=1 X=2 Y=3** **for G19: Y=1 Z=2 X=3**			
			Calibrating tool probes		**Measuring the tool**	
			Relative to machine	**Relative to workpiece**	**Relative to machine**	**Relative to workpiece**
_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	Measurement path in mm 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) Also possible for calibration in plane: 102: a) calculation of center in 1 (abscissa) b) calibration in 2 (ordinate) 201: a) calculation of center in 2 (ordinate) b) calibration in 1 (abscissa) **Not for incremental calibration!**		1: meas. of radius in direction 1 (abscissa) 2: meas. of radius in direction 2 (ordinate) 3: meas. of length at center point of the tool probe 103: meas. of the length, offset about radius 1 (abscissa) 203: meas. of the length, offset about radius 2 (ordinate)	
_MD	INTEGER	R31				
_MVAR	INTEGER >=0	R23	Measuring variant Calibration in measurement axis acc. to previous positioning on center of measurement cube		Measure of length of radius with motionless spindle	
			0	10	1	11
			Incremental calibration, traverse motion in measuring axis only		Measurement with rotating spindle; direction of rotation before cycle call, if spindle is already rotating. For motionless spindle direction in **_CM[5]**	
			10000	10010	2	12
			from meas. cycles - SW 6.3: Automatic calibration			
			100000	100010		
_NMSP	INTEGER >0	R27	Number of measurements at the same location			

© Siemens AG, 2004. All rights reserved

CYCLE971			**Tool measurement of milling tools on milling machines**			
Parameters GUD5	**Type**	**Com-parable Para-meters 840C**	**Possible axes** **Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3)** **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**	
			Relative to machine	**Relative to workpiece**	**Relative to machine**	**Relative to workpiece**
_PRNUM	INTEGER >0	R22	Tool probe number (number of the data field assigned to the workpiece probe **GUD6:**			
			_TP[_PRNUM-1,i])	**_TPW[_PRNUM-1,i] (as from meas. cycle SW 6.3))**	**_TP[_PRNUM-1,i])**	**_TPW[_PRNUM-1,i] (as from meas. cycle SW 6.3))**
_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			Dimension 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 (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))			
_CM[] GUD6 data item	REAL	REAL			Cycle-internal calculation of **S, F** from monitoring data in **_CM[]** **Only active if _CBIT[12]=0**	
_MFS[] GUD6- data item	REAL	REAL			Specification of **S**, **F** by user in **_MFS[ ]** **Only active if _CBIT[12]=1**	

© Siemens AG, 2004. All rights reserved

CYCLE 972			Tool measurement of turning tools with cutting edge position 1 – 8 on turning machines (machine-related)	
Parameters GUD5	**Type**	**Comparable Parameters 840C**	**Possible axes** **Abscissa (_MA=1) / Ordinate (_MA=2)** for G17: X=1 Y=2 **for G18: Z=1 X=2** for G19: Y=1 Z=2	
			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	Measurement path in mm	
_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	Measuring 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		Dimension 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 (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))	

© Siemens AG, 2004. All rights reserved

CYCLE 973			Workpiece measurement	
Parameters GUD5	**Type**	**Com-parable Para-meters** 840C	**Possible axes** **Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3)** for G17: X=1 Y=2 Z=3 **for G18: Z=1 X=2** Y=3 for G19: Y=1 Z=2 X=3	
			Calibrating workpiece probes	
			Relative to machine	**Relative to workpiece**
			Reference 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	Measurement path in mm	
_ID	REAL	R19		
_INCA	REAL	R26		
_K	INTEGER	R29		
_KNUM	INTEGER	R10		
_MA	INTEGER >0	R30	Measuring axis 1, 2	Measuring axis 1..3
_MD	INTEGER	R31	Measuring direction (0 = positive / 1 = negative)	
_MVAR	INTEGER >=0	R23	Measuring variant xxx13 **5 4 3 2 1** |		|_|___ **1 3** Reference groove |	
	|_______**0** Without incl. probe ball in calc. |	**1** Including probe ball in calc. |	(for calibration in groove only) | |________**1** One axis direction (specify | meas. axis and axis direction) | **2** Two axis directions | (specify measuring axis) |_________ **0** Without position calculation **1** With position calculation (for Calibration in groove only)	0 Surface
_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		

© Siemens AG, 2004. All rights reserved

CYCLE 973			Workpiece measurement	
Parameters GUD5	Type	Com-parable Para-meters 840C	Possible axes Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3) for G17: X=1 Y=2 Z=3 **for G18: Z=1 X=2** Y=3 for G19: Y=1 Z=2 X=3	
			Calibrating workpiece probes	
			Relative to machine	Relative to workpiece
			Reference 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 (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))	

© Siemens AG, 2004. All rights reserved

CYCLE 974 CYCLE 994			Workpiece measurement			
Para-meters GUD5	Type	Com-parable Para-meters 840C	Possible measuring axes Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3) for G17: X=1 Y=2 Z=3 **for G18: Z=1 X=2** Y=3 for G19: Y=1 Z=2 X=3			
			Measuring with automatic ZO correction	**Measuring with automatic tool offset**		
			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	Measurement path in mm			
_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 **1011...1026** automatic ZO correction in 1st to 16th channel basic frame **1051...1066** automatic ZO correction in 1st to 16th global basic frame **2000** automatic ZO correction in system frame **9999** automatic ZO correction in active frame G54...G57, G505...G599 or with active G500 in the last active channel-specific basic frame	Normal D number structure **7 6 5 4 3 2 1** digits 3 2 1: D number digit 4: **0/1** length offset in the measuring axis or setup or additive offset (from meas. cycles SW 6.3); **2** radius offset or setup or additive offset (from meas. cycles SW 6.3) digit 5: **0** offset normal; **1** offset inverted digit 6: **0** offset according to 4. digit; **1** offset of L1; **2** offset of L2; **3** offset of L3; **4** Radius compensation digit 7: **0** offset in length / radius (> meas. cycle SW 6.3); **1** offset in setup or additive offset; **2** offset in length or radius acc. to _TENV; **3** offset in setup or additive offset acc. to _TENV		Flat D number structure **9 8 7 6 5 4 3 2 1** digits 5 4 3 2 1: D number digit 6: **0/1** length offset in the measuring axis or setup or additive offset (from meas. cycles SW 6.3); **2** radius offset or setup or additive offset (from meas. cycles SW 6.3) digit 7: **0** offset normal; **1** offset inverted digit 8: **0** offset according to 6. digit; **1** offset of L1; **2** offset of L2; **3** offset of L3; **4** Radius compensation digit 9: **0** offset in length / radius (> meas. cycle SW 6.3); **1** offset in setup or additive offset; **2** offset in length or radius acc. to _TENV; **3** offset in setup/ or additive offset acc. to _TENV

© Siemens AG, 2004. All rights reserved

CYCLE 974 CYCLE 994			Workpiece measurement			
Para-meters GUD5	Type	Com-parable Para-meters 840C	Possible measuring axes **Abscissa (_MA=1) / Ordinate (_MA=2) /** Applicate (_MA=3) for G17: X=1 Y=2 Z=3 **for G18: Z=1 X=2** Y=3 for G19: Y=1 Z=2 X=3			
			Measuring with automatic ZO correction	**Measuring with automatic tool offset**		
			CYCLE974	CYCLE974		CYCLE994
			1 point	1 point	1 point with reversal	2 point on diameter
_MA	INTEGER >0	R30	Measuring axis 1...3			
_MD	INTEGER	R31				
_MVAR	INTEGER >0	R23	Measuring 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 (according to drawing)		
_SETV[8]	REAL					
_STA1	REAL 0...360 degrees	R26			Start angle	
_SZA	REAL	R19				Safety zone abscissa
_SZO	REAL	R18				Safety zone ordinate
_TDIF	REAL >0	R37		Dimension difference check		
_TNAME	STRING[ ]			Tool name (alternative for "_TNUM" if tool management active)		
_TENV	STRING[ ]			Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)		
_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 (according to drawing)		
_TLL	REAL	R41		Lower tolerance limit (according to drawing)		
_TSA	REAL >0	R36	Safe area			
_TZL	REAL >=0	R33		Zero offset area		
_VMS	REAL >=0	R25	Variable measuring velocity (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))			

© Siemens AG, 2004. All rights reserved

<table>
<tr><td colspan="3">CYCLE 976</td><td colspan="3">Workpiece measurement</td></tr>
<tr><td rowspan="3">Para-
meters
GUD5</td><td rowspan="3">Type</td><td rowspan="3">Com-
parable
Para-
meters
840C</td><td colspan="3">Possible measuring axes
Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3)
for G17: X=1 Y=2 Z=3
for G18: Z=1 X=2 Y=3
for G19: Y=1 Z=2 X=3</td></tr>
<tr><td colspan="3">Calibrating workpiece probes</td></tr>
<tr><td>Drill-hole
with known center</td><td>Drill-hole with
unknown center</td><td>Surface</td></tr>
<tr><td>_CALNUM</td><td>INTEGER</td><td>R12</td><td></td><td></td><td></td></tr>
<tr><td>_CORA</td><td>REAL
0...359.5</td><td>R13</td><td colspan="3">Offset angular position
(only active if mono probe)</td></tr>
<tr><td>_CPA</td><td>REAL</td><td>R20</td><td></td><td></td><td></td></tr>
<tr><td>_CPO</td><td>REAL</td><td>R21</td><td></td><td></td><td></td></tr>
<tr><td>_EVNUM</td><td>INTEGER</td><td>R11</td><td></td><td></td><td></td></tr>
<tr><td>_FA</td><td>REAL
>0</td><td>R28</td><td colspan="3">Measurement path in mm</td></tr>
<tr><td>_ID</td><td>REAL</td><td>R19</td><td></td><td></td><td></td></tr>
<tr><td>_INCA</td><td>0...360
degrees</td><td>R26</td><td></td><td></td><td></td></tr>
<tr><td>_K</td><td>INTEGER</td><td>R29</td><td></td><td></td><td></td></tr>
<tr><td>_KNUM</td><td>INTEGER</td><td>R10</td><td></td><td></td><td></td></tr>
<tr><td>_MA</td><td>INTEGER
>0</td><td>R30</td><td colspan="3">Measuring axis</td></tr>
<tr><td>_MD</td><td>INTEGER
>0</td><td>R31</td><td colspan="3">Measuring direction (0 = positive / 1 = negative)</td></tr>
<tr><td rowspan="3">_MVAR</td><td rowspan="3">INTEGER
>0</td><td rowspan="3">R23</td><td colspan="3">Measuring variant</td></tr>
<tr><td>xxxx01</td><td>xxxx08</td><td>x0000</td></tr>
<tr><td colspan="2">Calibration in plane
6 5 4 3 2 1
| | | | | |__ 1 calibration in hole with known center
| | | | | 8 calibration in hole with unknown center
| | | | |
				___ 0 			
			_____ 0 without including probe ball in calculation 			1 including probe ball in calculation (for meas. in plane) 		
		_______ 0 4 axis directions 		1 1 axis direction (specify measuring axis and axis direction) 		2 2 axis directions (specify measuring axis) 	
	________ 0 without position calculation 		1 with position determination
__________ 0 Paraxial calibration (in the plane) 1 Calibration at any angle (in the plane)</td><td>Calibration on surface _MVAR=0 calibration on surface _MVAR=10000 calibration on surface with calculation of the probe length only permissible with _MA=3!</td></tr>
<tr><td>_NMSP</td><td>INTEGER
>0</td><td>R27</td><td colspan="3">Number of measurements at the same location</td></tr>
</table>

© Siemens AG, 2004. All rights reserved

CYCLE 976			Workpiece measurement		
Para-meters GUD5	Type	Com-parable Para-meters 840C	Possible measuring axes abscissa/ ordinate Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3) for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3		
			Calibrating workpiece probes		
			Drill-hole with known center	Drill-hole with unknown center	Surface
_PRNUM	INTEGER >0	R22	Probe type / workpiece probe number 3 2 1 		_	_______ 2-digit number
___________ 1 Mono probe 0 Multi probe (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	Start 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 (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))		

© Siemens AG, 2004. All rights reserved

<table>
<tr><td colspan="3">CYCLE 977</td><td colspan="8">Workpiece measurement</td></tr>
<tr><td rowspan="3">Para-meters GUD5</td><td rowspan="3">Type</td><td rowspan="3">Com-parable Para-meters 840C</td><td colspan="8">Possible measuring axes
Abscissa (_MA=1) / Ordinate (_MA=2)
for G17: X=1 Y=2
for G18: Z=1 X=2
for G19: Y=1 Z=2</td></tr>
<tr><td colspan="4">Measuring with automatic tool offset</td><td colspan="4">Measuring with automatic ZO correction</td></tr>
<tr><td>Hole</td><td>Shaft</td><td>Groove</td><td>Web</td><td>Hole</td><td>Shaft</td><td>Groove</td><td>Web</td></tr>
<tr><td>_CALNUM</td><td>INTEGER</td><td>R12</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>_CORA</td><td>REAL 0...359.5</td><td>R13</td><td colspan="8">Offset angular position
(only active if mono probe)</td></tr>
<tr><td>_CPA</td><td>REAL</td><td>R20</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>_CPO</td><td>REAL</td><td>R21</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">_EVNUM</td><td rowspan="2">INTEGER >=0</td><td rowspan="2">R11</td><td colspan="4">Empirical value memory number
number of data field GUD5:_EV[_EVNUM-1]</td><td colspan="4"></td></tr>
<tr><td colspan="4">Mean value memory number
number of the data field GUD5:_MV[_EVNUM-1]
Only active if GUD6:_CHBIT[4]=1</td><td></td><td></td><td></td><td></td></tr>
<tr><td>_FA</td><td>REAL >0</td><td>R28</td><td colspan="8">Measurement path in mm</td></tr>
<tr><td>_ID</td><td>REAL</td><td>R19</td><td colspan="8">Infeed applicate</td></tr>
<tr><td>_INCA</td><td>REAL 0...360 degrees</td><td>R26</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>_K</td><td>INTEGER >=0</td><td>R29</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="3">_KNUM</td><td rowspan="3">INTEGER >=0</td><td rowspan="3">R10</td><td colspan="4">Without / with automatic tool offset (D number)
0 without tool offset</td><td colspan="4" rowspan="2">Without/with automatic offset of the ZO memory
0 without offset</td></tr>
<tr><td colspan="2">Normal D number structure</td><td colspan="2">Flat D number structure</td></tr>
<tr><td colspan="2">7 6 5 4 3 2 1
| | | | |_|_|_ D number[1]
| | | |______0/1
| | | length offset in the measuring axis or setup or additive offset (as from meas. cycle SW 6.3)
| | | 2 radius offset or setup or additive offset (as from meas. cycle SW 6.3)
| | |_______0 offset normal
| | 1 offset inverted
| |________ 0 offset according to 4th digit
| 1 offset of L1
| 2 offset of L2
| 3 offset of L3
| 4 Radius compensation
|_________ 0 offset in length or radius
(> meas. cycle SW 6.3) 1 offset in setup or Sum offset
2 offset in length or radius acc. to _TENV
3 offset in setup or additive offset acc. to _TENV
1) if MD 18105 ≤9 single-digit, >9<1000, three-digit D number</td><td colspan="2">9 8 7 6 5 4 3 2 1
| | | | |_|_|_|_ |_ D number[2]
| | | |____0/1
| | | length offset in the measuring axis or setup or additive offset (from meas. cycles SW 6.3)
| | | 2 radius offset or setup or additive offset (from meas. cycles SW 6.3)
| | |_____ 0 offset normal
| | 1 offset inverted
| |______ 0 offset according to 6. digit
| 1 offset of L1
| 2 offset of L2
| 3 offset of L3
| 4 Radius compensation
|_______ 0 offset in length / radius
(> meas. cycle SW 6.3) 1 offset in setup or additive offset
2 offset in length or radius acc. to _TENV
3 offset in setup or additive offset acc. to _TENV
2) if MD 18105 >999 also valid for normal D number structure</td><td colspan="4">1...99
automatic offset in ZO G54...G57 G505...G599
1000
automatic offset in basic frame G500
1011...1026
automatic ZO correction in 1st to 16th channel basic frame
1051...1066
automatic ZO correction in 1st to 16th global basic frame
2000
automatic ZO in system frame
9999
automatic ZO correction in active frame G54...G57, G505...G599 or with active G500 in the last active channel-specific basic frame</td></tr>
</table>

© Siemens AG, 2004. All rights reserved

CYCLE 977			Workpiece measurement							
Parameters GUD5	Type	Comparable Parameters 840C	Possible measuring axes Abscissa (_MA=1) / Ordinate (_MA=2) for G17: X=1 Y=2 for G18: Z=1 X=2 for G19: Y=1 Z=2							
			Measuring with automatic tool offset				Measuring with automatic ZO correction			
			Hole	**Shaft**	**Groove**	**Web**	**Hole**	**Shaft**	**Groove**	**Web**
_MA	INTEGER >0	R30			Measuring axis 1...2				Measuring axis 1...2	
_MD	INTEGER	R31								
_MVAR	INTEGER >0	R23	Measuring variant 1xxx measurement traveling around or taking account of a safety zone							
			1	2	3	4	101	102	103	104
_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 Mono probe 0 Multi probe (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 (according 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	Dimension difference check							
_TMV	REAL >0	R34	Offset range with mean value calculation							
_TNAME	STRING[ ]		Tool name (alternative to "_TNUM" with tool management active)							
_TENV	STRING[ ]		Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)							
_TNUM	INTEGER >=0	R9	Tool number for automatic tool compensation							
_TUL	REAL	R40	Upper tolerance limit (according to drawing)							
_TLL	REAL	R41	Lower tolerance limit (according to drawing)							
_TSA	REAL >0	R36	Safe area							
_TZL	REAL >=0	R33	Zero offset area							
_VMS	REAL >=0	R25	Variable measuring velocity (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))							

© Siemens AG, 2004. All rights reserved

<table>
<tr><td colspan="3">CYCLE 978</td><td colspan="3">Workpiece measurement</td></tr>
<tr><td rowspan="2">Parameters GUD5</td><td rowspan="2">Type</td><td rowspan="2">Comparable Parameters 840C</td><td colspan="3">Possible measuring axes
Abscissa (_MA=1) / Ordinate (_MA=2)
for G17: X=1 Y=2
for G18: Z=1 X=2
for G19: Y=1 Z=2</td></tr>
<tr><td colspan="2">Measuring with automatic tool offset</td><td>Measuring with automatic ZO correction</td></tr>
<tr><td>_CALNUM</td><td>INTEGER</td><td>R12</td><td colspan="2"></td><td></td></tr>
<tr><td>_CORA</td><td>REAL
0...359.5</td><td>R13</td><td colspan="3">Offset angular position
only active if mono probe</td></tr>
<tr><td>_CPA</td><td>REAL</td><td>R20</td><td colspan="2"></td><td></td></tr>
<tr><td>_CPO</td><td>REAL</td><td>R21</td><td colspan="2"></td><td></td></tr>
<tr><td rowspan="2">_EVNUM</td><td rowspan="2">INTEGER
>=0</td><td rowspan="2">R11</td><td colspan="3">Empirical value memory number
Number of data field GUD5:_EV[_EVNUM-1]</td></tr>
<tr><td colspan="2">Mean value memory number
number of the data field GUD5:_MV[_EVNUM-1]
Only active if GUD6:_CHBIT[4]=1</td><td></td></tr>
<tr><td>_FA</td><td>REAL
>0</td><td>R28</td><td colspan="3">Measurement path in mm</td></tr>
<tr><td>_ID
_INCA</td><td>REAL
INTEGER</td><td>R19
R26</td><td colspan="2"></td><td></td></tr>
<tr><td>_K</td><td>INTEGER
>0</td><td>R29</td><td colspan="2">Weighting factor k for mean value calculation</td><td></td></tr>
<tr><td rowspan="3">_KNUM</td><td rowspan="3">INTEGER
>=0</td><td rowspan="3">R10</td><td colspan="2">Without / with automatic tool offset (D number)
0 without tool offset</td><td rowspan="2">Without/with automatic offset of the ZO memory
0 without offset</td></tr>
<tr><td>Normal D number structure</td><td>Flat D number structure</td></tr>
<tr><td><u>7 6 5 4 3 2 1</u>
| | | | |_|_|_ D number[1]
| | | |______0/1
| | | length offset in the measuring axis or setup or additive offset (as from meas. cycle SW 6.3)
| | | 2 radius offset or setup or additive offset (as from meas. cycle SW 6.3)
| | |______ 0 offset normal
| | 1 offset inverted
| |_______ 0 offset according to 4. digit
| 1 offset of L1
| 2 offset of L2
| 3 offset of L3
| 4 Radius compensation
|_________ 0 offset in length or radius
(> meas. cycle SW 6.3) 1 offset in setup or Sum offset
2 offset in length or radius acc. to _TENV
3 offset in setup or additive offset acc. to _TENV
1) if MD 18105 ≤9 single-digit, >9<1000, three-digit D number</td><td><u>9 8 7 6 5 4 3 2 12</u>
| | | | |_|_|_|_ |_ D number[2]
| | | |______0/1
| | | length offset in the measuring axis or setup or additive offset (as from meas. cycle SW 6.3)
| | | 2 radius offset or setup or additive offset (as from meas. cycle SW 6.3)
| | |______ 0 offset normal
| | 1 offset inverted
| |_______ 0 offset according to 6. digit
| 1 offset of L1
| 2 offset of L2
| 3 offset of L3
| 4 Radius compensation
|_________ 0 offset in length or radius
(> meas. cycle SW 6.3) 1 offset in setup or Sum offset
2 offset in length or radius acc. to _TENV
3 offset in setup or additive offset acc. to _TENV
2) if MD 18105 >999 also valid for normal D number structure</td><td>1...99
automatic offset in ZO
G54...G57
G505...G599
1000
automatic offset in basic frame G500
1011...1026
automatic ZO correction in 1st to 16th basic frame
1051...1066
automatic ZO correction in 1st to 16th global basic frame
2000
automatic ZO correction in system frame
9999
automatic ZO correction in active frame
G54...G57, G505...G599
or with active G500
in the last active channel-specific basic frame</td></tr>
</table>

© Siemens AG, 2004. All rights reserved

CYCLE 978			Workpiece measurement	
Parameters GUD5	**Type**	**Comparable Parameters 840C**	**Possible measuring axes** **Abscissa (_MA=1) / Ordinate (_MA=2)** **for G17: X=1 Y=2** **for G18: Z=1 X=2** **for G19: Y=1 Z=2**	
			Measuring with automatic tool offset	**Measuring with automatic ZO correction**
			CYCLE978	CYCLE978
			1 point	1 point
_MA	INTEGER >0	R30	Measuring axis 1..3	
_MD	INTEGER	R31		
_MVAR	INTEGER >=0	R23	Measuring variant 0	100
			1000 1)	1100 1)
_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 Mono probe 0 Multi probe (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM(2-digit)-1]**)	
_SETVAL	REAL	R32	Setpoint (acc. to the drawing)	Setpoint
_RA	INTEGER	R31		
_RF	REAL	R31		
_STA1	REAL 0...360 degrees	R24		
_SZA	REAL	R19		
_SZO	REAL	R18		
_TDIF	REAL >0	R37	Dimension difference check	
_TMV	REAL >0	R34	Offset range with mean value calculation only active if **GUD6:_CHBIT[4]=1**	
_TNAME	STRING[ ]		Tool name (alternative for "_TNUM" if tool management active)	
_TENV	STRING[ ]		Name of tool environment for automatic tool compensation (from measuring cycles SW 6.3)	
_TNUM	INTEGER >=0	R9	Tool number for automatic tool offset	
_TUL	REAL	R40	Upper tolerance limit (according to drawing)	
_TLL	REAL	R41	Lower tolerance limit (according to drawing)	
_TSA	REAL >0	R36	Safe area	
_TZL	REAL >=0	R33	Zero offset area	
_VMS	REAL >=0	R25	Variable measuring velocity (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))	

1) Difference measurement (not with mono probe)

© Siemens AG, 2004. All rights reserved

<table>
<tr><td colspan="3">CYCLE 979</td><td colspan="8">Workpiece measurement</td></tr>
<tr><td rowspan="3">Para-meters GUD5</td><td rowspan="3">Type</td><td rowspan="3">Comparable Para-meters 840C</td><td colspan="8">Possible measurements
G17: X -Y plane
G18: Z -X plane
G19: Y -Z plane</td></tr>
<tr><td colspan="4">Measuring with automatic tool offset</td><td colspan="4">Measuring with automatic ZO correction</td></tr>
<tr><td>Hole</td><td>Shaft</td><td>Groove</td><td>Web</td><td>Hole</td><td>Shaft</td><td>Groove</td><td>Web</td></tr>
<tr><td>_CALNUM</td><td>INTEGER</td><td>R12</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>_CORA</td><td>REAL
0...359.5</td><td>R13</td><td colspan="8">Offset angular position
only active if mono probe</td></tr>
<tr><td>_CPA</td><td>REAL</td><td>R20</td><td colspan="8">Center abscissa (with reference to the workpiece zero)</td></tr>
<tr><td>_CPO</td><td>REAL</td><td>R21</td><td colspan="8">Center ordinate (with reference to the workpiece zero)</td></tr>
<tr><td>_EVNUM</td><td>INTEGER
>=0</td><td>R11</td><td colspan="4">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</td><td></td><td></td><td></td><td></td></tr>
<tr><td>_FA</td><td>REAL
>0</td><td>R28</td><td colspan="8">Measurement path in mm</td></tr>
<tr><td>_ID</td><td>REAL</td><td>R19</td><td></td><td></td><td></td><td>Infeed applicate</td><td></td><td></td><td></td><td>Infeed appli-cate</td></tr>
<tr><td>_INCA</td><td>REAL</td><td>R26</td><td colspan="2">Indexing angle</td><td></td><td></td><td colspan="2">Indexing angle</td><td></td><td></td></tr>
<tr><td>_K</td><td>INTEGER
>0</td><td>R29</td><td colspan="4">Weighting factor k for mean value calculation</td><td></td><td></td><td></td><td></td></tr>
<tr><td>_KNUM</td><td>INTEGER
>=0</td><td>R10</td><td colspan="4">without / with automatic tool offset (D number)
0 without tool offset

Normal D number structure:
7 6 5 4 3 2 1
| | | | |_|_|_ D number[1]
| | | |______0/1
| | | length offset in the measuring axis or setup or additive offset (as from meas. cycle SW 6.3)
| | | 2 radius offset or setup or additive offset (as from meas. cycle SW 6.3)
| | |______ 0 offset normal
| | 1 offset inverted
| |_______ 0 offset according to 4. digit
| 1 offset of L1
| 2 offset of L2
| 3 offset of L3
| 4 Radius compensation
|_________ 0 offset in length or radius
(> meas. cycle SW 6.3) 1 offset in setup or Sum offset
2 offset in length or radius acc. to _TENV
3 offset in setup or additive offset acc. to _TENV
1) if MD 18105 ≤9 single-digit, >9<1000, three-digit D number

Flat D number structure:
9 8 7 6 5 4 3 2 1 2
| | | | |_|_|_|_ |_ D number[2]
| | | |______0/1
| | | length offset in the measuring axis or setup or additive offset (as from meas. cycle SW 6.3)
| | | 2 radius offset or setup or additive offset (as from meas. cycle SW 6.3)
| | |______ 0 offset normal
| | 1 offset inverted
| |_______ 0 offset according to 6. digit
| 1 offset of L1
| 2 offset of L2
| 3 offset of L3
| 4 Radius compensation
|_________ 0 offset in length or radius
(> meas. cycle SW 6.3) 1 offset in setup or Sum offset
2 offset in length or radius acc. to _TENV
3 offset in setup or additive offset acc. to _TENV
2) if MD 18105 >999 also valid for normal D number structure</td><td colspan="4">without/with automatic 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
1011...1026
automatic ZO correction in 1st to 16th basic frame
1051...1066
automatic ZO correction in 1st to 16th global basic frame
2000
automatic ZO correction in system frame
9999
automatic ZO correction in active frame G54...G57, G505...G599 or with active G500 in the last active channel-specific basic frame</td></tr>
</table>

© Siemens AG, 2004. All rights reserved

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							
			Measuring with automatic tool offset				Measuring with automatic ZO correction			
			Hole	Shaft	Groove	Web	Hole	Shaft	Groove	Web
_MA	INTEGER	R30								
_MVAR	INTEGER >0	R23	Measuring 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 1: 2-digit number 3: 1 Mono probe; 0 Multi probe 4: 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 interpolation				Velocity for circular interpolation			
_SETVAL	REAL	R32 R42	Setpoint (according to drawing)				Setpoint			
_SETV[8]	REAL									
_STA1	REAL	R24	Start angle							
_SZA	REAL	R19								
_SZO	REAL	R18								
_TDIF	REAL >0	R37	Dimension difference check							
_TMV	REAL >0	R34	Offset range with mean value calculation only active if **GUD6:_CHBIT[4]=1**							
_TNAME	STRING[ ]		Tool name (alternative to "_TNUM" with tool management active)							
_TENV	STRING[ ]		Name of tool environment for automatic tool compensation (from measuring cycles SW 6.)							
_TNUM	INTEGER >=0	R9	Tool number for automatic tool offset							
_TUL	REAL	R40	Upper tolerance limit (according to drawing)							
_TLL	REAL	R41	Lower tolerance limit (according to drawing)							
_TSA	REAL >0	R36	Safe area							
_TZL	REAL >=0	R33	Zero offset area							
_VMS	REAL >=0	R25	Variable measuring velocity (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))							

© Siemens AG, 2004. All rights reserved

CYCLE982			Tool measurement of turning, drilling, and milling tools for turning machines					
Parameters GUD5	**Type**	**Comparable Parameters 840C**	**Possible measuring axes** **Abscissa (_MA=1) / Ordinate (_MA=2)** for G17: X=1 Y=2 **for G18: Z=1 X=2** for G19: Y=1 Z=2					
			Calibrate tool probe		**Measure tool**		**Measuring the tool automatically**	
			Relative to machine	**Relative to workpiece**	**Relative to machine**	**Relative to workpiece**	**Relative to machine**	**Relative to workpiece**
_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	Measurement path in mm					
_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	Measuring variant					
			0	10	xxxx01	xxxx11	xxxx02	xxxx12

```
6 5 4 3 2 1
| | | | | |___   0  Calibration
| | | | |        1  Measurement of turning (SL 1-8), milling, and drilling tools
| | | | |           measurement in _MA
| | | | |        2  Automatic measurement in abscissa and ordinate
| | | | |____    0  Relative to machine
| | | |          1  Relative to the workpiece (from meas. cycles SW 6.3)
| | | |        ------------------------------------------------------------
| | | |          For milling tools only (also automatically), setting data SD42950=2!
| | | |_____    0  Measuring without reversal
| | |            1  Measuring with reversal
| | |_______     0  Measurement: only correct length, if the 1st digit=1
| |              1  Measurement: only correct radius, if 1st digit=1
| |              2  Measurement: correct length and radius, if 1st digit=1
| |              3  Automatic measurement: correct length and radius, if 1st digit=2
| |                 travel round measuring cube opposite the starting position side.
| |              4  Automatic measurement: correct length and radius, if 1st digit=2
| |                 measuring direction opposite to approach direction
| |_________     0  Axial position of milling tool/drill
|                   (radius in ordinate, for G18: X axis)
|                1  Radial position of milling tool/drill
|                   (radius in abscissa, for G18: Z axis)
|___________     0  Measurement and calibration
                 1  Incremental measurement or calibration (as from meas. cycle SW 6.2)
```


© Siemens AG, 2004. All rights reserved

CYCLE982			Tool measurement of turning, drilling, and milling tools for turning machines					
Para-meters GUD5	Type	Com-parable Para-meters 840C	Possible measuring axes Abscissa (_MA=1) / Ordinate (_MA=2) for G17: X=1 Y=2 **for G18: Z=1 X=2** for G19: Y=1 Z=2					
			Calibrate tool probe		Measure tool		Measuring the tool automatically	
			Relative to machine	Relative to workpiece	Relative to machine	Relative to workpiece	Relative to machine	Relative to workpiece
_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 workpiece probe **GUD6:**					
			_TP[_PRNUM-1,i]	**_TPW[_PRNUM-1,i] (as from meas. cycle SW 6.3)**	**_TP[_PRNUM-1,i]**	**TPW[_PRNUM-1,i] (as from meas. cycle SW 6.3)**	**_TP[_PRNUM-1,i]**	**TPW[_PRNUM-1,i] (as from meas. cycle SW 6.3)**
_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			Dimension 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 (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))					

© Siemens AG, 2004. All rights reserved

CYCLE997			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** **Measuring with automatic ZO correction**	
			1 sphere	**3 spheres**
_FA	REAL >0	R28	Measurement path in mm	
_INCA	REAL	R26	Incremental angle (for _MVAR=xx1109 only, measuring at an angle)	
_KNUM	INTEGER >=0	R10	without/with automatic 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 **1011...1026** automatic ZO correction in 1. up to 16th channel, basic frame **1051...1066** automatic ZO correction in 1. up to 16th global basic frame **2000** automatic ZO correction in system frame **9999** automatic ZO correction in active frame G54...G57, G505...G599 or with G500 active in last active channel-specific basic frame	**1..99** automatic offset in ZO G54...G57 G505...G599 **1000** automatic offset in basic frame G500 **1011...1026** automatic ZO correction in 1. up to 16th channel, basic frame **2000** automatic ZO correction in system frame **9999** automatic ZO correction in active frame G54...G57, G505...G599 or with G500 active in last active channel-specific basic frame
_MVAR	INTEGER >0	R23	Measuring variant	
			x0xxxx	x1xxxx
			7 6 5 4 3 2 1 Digits 3 2 1: **109** measure sphere and ZO determination without measurement repeat; **119** measure sphere and ZO determination with measurement repeat Digit 4: **0** measurement paraxial (to axes of the WCS); **1** measurement at an angle (intermediate positioning on circular path) Digit 5: **0** measure 1 sphere; **1** measure 3 spheres Digits 7 6: **01** 3 meas. points on circle determination, for measuring at an angle only; **11** 4 meas. points for circle determination, for measuring at an angle only Digit 7: **0** without diameter determination (sphere diameter known); **1** with diameter determination	
_NMSP	INTEGER >0	R27	Number of measurements at the same location	
_PRNUM	**INTEGER >0**	R22	Workpiece probe number (for multi probe only) **Values: 1 up to 99** (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM–1]**	
_RF	REAL	R31	Velocity for intermediate paths on circular path (G2 or G3) (for _MVAR=xx1109 only, measuring at an angle)	
_STA1	REAL	R26	Starting angle (for _MVAR=xx1109 only, measuring at an angle)	
_TNVL	REAL			Limit for triangle distortion (for _MVAR=x1x109 only, measure 3 spheres and ZO correction)
_SETV[8]	REAL		Setpoints, center point of the spheres (balls)	
_TSA	REAL >0	R36	Safe area	
_VMS	REAL >=0	R25	Variable measuring velocity (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))	

© Siemens AG, 2004. All rights reserved

CYCLE 998			Workpiece measurement	
Parameters GUD5	Type	Com-parable Para-meters 840C	Possible measuring axes Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3) for G17: X=1 Y=2 Z=3 for G18: Z=1 X=2 Y=3 for G19: Y=1 Z=2 X=3	
			Measuring with automatic ZO correction	
			1 angle	2 angle
_CALNUM	INTEGER	R12		
_CORA	REAL 0...359.5	R13	Offset angular position (only active if mono probe)	
_CPA	REAL	R20		
_CPO	REAL	R21		
_EVNUM	INTEGER >=0	R11		
_FA	REAL >0	R28	Measurement path in mm	
_ID	REAL >0	R19	Distance between measuring points P1 and P2 in offset axis	Distance between measuring points P1 and P2 in abscissa
_INCA	IREAL	R26		Setpoint or angle in ordinate
_K	INTEGER >0	R29		
_KNUM	INTEGER >=0	R10	without/with automatic 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 **1011...1026** automatic ZO correction in 1st to 16th basic frame **2000** automatic ZO in system frame **9999** automatic ZO correction in active frame G54...G57, G505...G599 or with G500 active in last active channel-specific basic frame	
_MA	INTEGER >0	R30	Offset axis/measuring axis 102...301 | |__ Measuring axis |_______ Offset axis	
_MD	INTEGER	R31	for _MVAR=1xx10x only	for _MVAR=1xx10x only
_MVAR	INTEGER >=0	R23	Measuring variant 1xx10x paraxial positioning between measuring points 105	106
			1105[1)]	
_NMSP	INTEGER	R27	Number of measurements at the same location	

1) Difference measurement (not with mono probe)

© Siemens AG, 2004. All rights reserved

CYCLE 998			Workpiece measurement	
Parameters GUD5	**Type**	**Com-parable Para-meters 840C**	**Possible measuring axes** **Abscissa (_MA=1) / Ordinate (_MA=2) / Applicate (_MA=3)** **for G17: X=1 Y=2 Z=3** **for G18: Z=1 X=2 Y=3** **for G19: Y=1 Z=2 X=3**	
			Measuring with automatic ZO correction	
			1 angle	2 angle
_PRNUM	INTEGER >0	R22	Probe type / workpiece probe number 3 2 1 | |_ |_______ 2-digit number | |__________ 1 Mono probe 0 Multi probe (number of the data field assigned to the workpiece probe **GUD6:_WP[_PRNUM(2-digit)-1]**)	
_SETVAL	REAL	R32	Setpoint in measuring point 1 in meas. axis[1)]	Setpoint in measuring point P1 in applicate[1)]
_SETV[0][2)]	REAL >0			Distance between measuring points P1 and P3 in ordinate
_RA	INTEGER	R31	_RA=0: coordinate system is rotated _RA>0: Number of the rotary axis in which the offset is applied	
_RF	REAL	R31		
_STA1	REAL 0...360 degrees	R24	Setpoint angle	Setpoint for angle about the abscissa
_SZA	REAL	R19		
_SZO	REAL	R18		
_TDIF	REAL >0	R37		
_TMV	REAL >0	R34		
_TNAME	STRING[ ]			
_TNUM	INTEGER >=0	R9		
_TUL	REAL	R40		
_TLL	REAL	R41		
_TSA	REAL >0	R36	Safe area angle	
_TZL	REAL >=0	R33		
_VMS	REAL >=0	R25	Variable measuring velocity (for _VMS=0: 150 mm/min (if _FA=1); 300 mm/min (if _FA>1))	

1) not for _MVAR 1xx10x

© Siemens AG, 2004. All rights reserved

Result parameters calibration

							CYCLE971	CYCLE972 CYCLE982	CYCLE973	CYCLE976
GUD5	**Data type**	**Meaning**								
_OVR [0]	REAL									
_OVR [1]	REAL									
_OVR [2]	REAL									
_OVR [3]	REAL									
_OVR [4]	REAL	Actual point	Probe ball diameter							
_OVR [5]	REAL	Difference	Probe ball diameter							
_OVR [6]	REAL	Center of hole				Abscissa				
_OVR [7]	REAL	Center of hole				Ordinate				
_OVR [8]	REAL	Trigger point	Minus	direction	Actual point	Abscissa				
_OVR [9]	REAL	Trigger point	Minus	direction	Difference	Abscissa				
_OVR [10]	REAL	Trigger point	Plus	direction	Actual point	Abscissa				
_OVR [11]	REAL	Trigger point	Plus	direction	Difference	Abscissa				
_OVR [12]	REAL	Trigger point	Minus	direction	Actual point	Ordinate				
_OVR [13]	REAL	Trigger point	Minus	direction	Difference	Ordinate				
_OVR [14]	REAL	Trigger point	Plus	direction	Actual point	Ordinate				
_OVR [15]	REAL	Trigger point	Plus	direction	Difference	Ordinate				
_OVR [16]	REAL	Trigger point	Minus	direction	Actual point	Applicate				
_OVR [17]	REAL	Trigger point	Minus	direction	Difference	Applicate				
_OVR [18]	REAL	Trigger point	Plus	direction	Actual point	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	Angle at which the trigger points were determined								
_OVR [25]	REAL									
_OVR [26]	REAL									
_OVR [27]	REAL	Zero offset area								
_OVR [28]	REAL	Safe area								
_OVR [29]	REAL									
_OVI [0]	INT									
_OVI [1]	INT									
_OVI [2]	INT	Measuring cycle number								
_OVI [3]	INT	Measuring variant								
_OVI [4]	INT									
_OVI [5]	INT	Probe number								
_OVI [6]	INT									
_OVI [7]	INT									
_OVI [8]	INT									
_OVI [9]	INT	Alarm number								

© Siemens AG, 2004. All rights reserved

Result parameters measurement (turning machines)					
GUD5	**Data type**	**Meaning**	**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	Applicate	Applicate	
_OVR [4]	REAL	Actual point	Measuring axis	Diameter/radius	
_OVR [5]	REAL	Actual point	Abscissa	Abscissa	
_OVR [6]	REAL	Actual point	Ordinate	Ordinate	
_OVR [7]	REAL	Actual point	Applicate	Applicate	
_OVR [8]	REAL	Tolerance Upper limit [1]	Measuring axis	Diameter/radius	
		Actual point			Length L1
_OVR [9]	REAL	Difference			Length L1
_OVR [10]	REAL	Actual point			Length L2
_OVR [11]	REAL	Difference			Length L2
_OVR [12]	REAL	Tolerance Lower limit [1]	Measuring axis	Diameter/radius	
		Actual point			Radius only CYCLE982
_OVR [13]	REAL	Difference			Radius only CYCLE982
_OVR [14]	REAL				
_OVR [15]	REAL				
_OVR [16]	REAL	Difference	Measuring axis	Diameter/radius	
_OVR [17]	REAL	Difference	Abscissa	Abscissa	
_OVR [18]	REAL	Difference	Ordinate	Ordinate	
_OVR [19]	REAL	Difference	Applicate	Applicate	
_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 range [1]			
_OVR [28]	REAL	Safe area			
_OVR [29]	REAL	Permissible dimension difference [1]			
_OVR [30]	REAL	Empirical value			
_OVR [31]	REAL	Mean value [1]			
_OVI [0]	INT	D number / ZO number			
_OVI [1]	INT				
_OVI [2]	INT	Measuring cycle number			
_OVI [3]	INT	Measuring variant			
_OVI [4]	INT	Weighting factor [1]			
_OVI [5]	INT	Probe number			
_OVI [6]	INT	Mean value memory number [1]			
_OVI [7]	INT	Empirical value memory number			
_OVI [8]	INT	Tool number			
_OVI [9]	INT	Alarm number			
_OVI [11]	INT	Status offset request [2]			
_OVI [12]	INT	Internal error number			

1) for automatic tool offset only;
2) for automatic ZO correction only

© Siemens AG, 2004. All rights reserved

Result parameters measurement (milling and machining centers)						
GUD5	**Data type**	**Meaning**	**CYCLE961**	**CYCLE997**	**CYCLE998**	
					1 angle	2 angle
_OVR [0]	REAL	Setpoint		Sphere diameter 1st sphere	Angle	Angle about abscissa
_OVR [1]	REAL	Setpoint		Center point coordinate abscissa 1st sphere		Angle about ordinate
_OVR [2]	REAL	Setpoint		Center point coordinate ordinate 1st sphere		
_OVR [3]	REAL	Setpoint		Center point coordinate applicate 1st sphere		
_OVR [4]	REAL	Actual point	Angle with abscissa axis (WCS)	Sphere diameter 1st sphere	Angle	Angle about abscissa
_OVR [5]	REAL	Actual point	Corner point in abscissa (WCS)	Center point coordinate abscissa 1st sphere		Angle about ordinate
_OVR [6]	REAL	Actual point	Corner point in ordinate (WCS)	Center point coordinate ordinate 1st sphere		
_OVR [7]	REAL	Actual point		Center point coordinate applicate 1st sphere		
_OVR [8]	REAL	Difference		Sphere diameter 1st sphere		
_OVR [9]	REAL	Difference		Center point coordinate abscissa 1st sphere		
_OVR [10]	REAL	Difference		Center point coordinate ordinate 1st sphere		
_OVR [11]	REAL	Difference		Center point coordinate applicate 1st sphere		
_OVR [12]	REAL	Actual point		Sphere diameter 2nd sphere[1)]		
_OVR [13]	REAL	Actual point		Center point coordinate abscissa 2nd sphere[1)]		
_OVR [14]	REAL	Actual point		Center point coordinate ordinate 2nd sphere[1)]		
_OVR [15]	REAL	Actual point		Center point coordinate applicate 2nd sphere[1)]		
_OVR [16]	REAL	Difference		Sphere diameter 2nd sphere[1)]	Angle	Angle about abscissa
_OVR [17]	REAL	Difference		Center point coordinate abscissa 2nd sphere[1)]		Angle about ordinate
_OVR [18]	REAL	Difference		Center point coordinate ordinate 2nd sphere[1)]		
_OVR [19]	REAL	Difference		Center point coordinate applicate 2nd sphere[1)]		
_OVR [20]	REAL	Actual point	Angle with abscissa axis (MCS)	Sphere diameter 3rd sphere[1)]		
		Offset value			Angle	
_OVR [21]	REAL	Actual point	Corner point in abscissa (MCS)	Center point coordinate abscissa 3rd sphere[1)]		
		Offset value				Angle about abscissa
_OVR [22]	REAL	Actual point	Corner point in ordinate (MCS)	Center point coordinate ordinate 3rd sphere[1)]		
		Offset value				Angle about ordinate
_OVR [23]	REAL	Actual point		Center point coordinate applicate 3rd sphere[1)]		
		Offset value				Angle about applicate

1) for measuring variants _MVAR=x1x109 only, measure 3 spheres

© Siemens AG, 2004. All rights reserved

GUD5	Data type	Meaning	CYCLE961	CYCLE997	CYCLE998	
					1 angle	2 angle
_OVR [24]	REAL	Difference		Sphere diameter 3rd sphere[1]		
_OVR [25]	REAL	Difference		Center point coordinate abscissa 3rd sphere[1]		
_OVR [26]	REAL	Difference		Center point coordinate ordinate 3rd sphere[1]		
_OVR [27]	REAL	Difference		Center point coordinate applicate 3rd sphere[1]		
_OVR [28]	REAL	Safe area				
_OVR [30]	REAL	Empirical value				
_OVR [31]	REAL	Mean value				
_OVI [0]	INT	ZO number				
_OVI [1]	INT					
_OVI [2]	INT	Measuring cycle number				
_OVI [3]	INT	Measuring variant				
_OVI [4]	INT	Weighting factor				
_OVI [5]	INT	Probe number				
_OVI [6]	INT	Mean value memory number				
_OVI [7]	INT	Empirical value, memory number				
_OVI [8]	INT	Tool number				
_OVI [9]	INT	Alarm number				
_OVI[10						
_OVI [11]	INT	Status offset request				
_OVI12]	INT	Internal error number		Internal measurement evaluation		

1) for measuring variants _MVAR=x1x109 only, measure 3 spheres

© Siemens AG, 2004. All rights reserved

Result parameters measurement (milling and machining centers)						
GUD5	**Data type**	**Meaning**	**CYCLE977**		**CYCLE978**	**CYCLE979**
			_MVAR=xxx1 to _MVAR=xxx4	_MVAR=xxx5 _MVAR=xxx6		
_OVR [0]	REAL	Setpoint	Hole Shaft Groove Web		Measuring axis	Hole Shaft Groove Web
		Setpoint rectangle length		Abscissa		
_OVR [1]	REAL	Setpoint center point/center	Abscissa		Abscissa	Abscissa
		Setpoint rectangle length		Ordinate		
_OVR [2]	REAL	Setpoint center point/center	Ordinate		Ordinate	Ordinate
		Setpoint for rectangle center point		Abscissa		
_OVR [3]	REAL	Setpoint			Applicate	
		Setpoint for rectangle center point		Ordinate		
_OVR [4]	REAL	Actual value for diameter/width	Hole Shaft Groove Web		Measuring axis	Hole Shaft Groove Web
		Actual value rectangle length		Abscissa		
_OVR [5]	REAL	Actual value center point/center	Abscissa			Abscissa
		Actual value rectangle length		Ordinate		
_OVR [6]	REAL	Actual point	Ordinate			Ordinate
		Actual value rectangle center point		Abscissa		
_OVR [7]	REAL	Actual point				
		Actual value rectangle center point		Ordinate		
_OVR [8][1)]	REAL	Upper tolerance limit diameter/width	Hole Shaft Groove Web		Measuring axis	Hole Shaft Groove Web
		Upper tolerance limit rectangle length		Abscissa		
_OVR [9][1)]	REAL	Upper tolerance limit rectangle length		Ordinate		
_OVR [10]	REAL					
_OVR [11]	REAL					
_OVR [12][1)]	REAL	Lower tolerance limit diameter/width	Hole Shaft Groove Web		Measuring axis	Hole Shaft Groove Web
		Lower tolerance limit rectangle length		Abscissa		
_OVR [13][1)]	REAL	Lower tolerance limit rectangle length		Ordinate		
_OVR [14]	REAL					
_OVR [15]	REAL					
_OVR [16]	REAL	Difference diameter/width	Hole Shaft Groove Web		Measuring axis	Hole Shaft Groove Web
		Difference rectangle length		Abscissa		

1) for workpiece measurement with tool offset only

© Siemens AG, 2004. All rights reserved

Result parameters measurement (milling and machining centers) 1)						
GUD5	**Data type**	**Meaning**	**CYCLE977**		**CYCLE978**	**CYCLE979**
			_MVAR=xxx1 to _MVAR=xxx4	_MVAR=xxx5 _MVAR=xxx6		
_OVR [17]	REAL	Difference center point/center	Abscissa			Abscissa
		Difference rectangle length		Ordinate		
_OVR [18]	REAL	Difference center point/center	Ordinate			Ordinate
		Difference of rectangle center point		Abscissa		
_OVR [19]	REAL	Difference of rectangle center point		Ordinate		
_OVR [20]	REAL	Offset value	1)	1)	1)	1)
_OVR [21]	REAL					
_OVR [22]	REAL					
_OVR [23]	REAL					
_OVR [24]	REAL					
_OVR [25]	REAL					
_OVR [26]	REAL					
_OVR [27]	REAL	Zero offset area	1)	1)	1)	1)
_OVR [28]	REAL	Safe area				1)
_OVR [29]	REAL	Permissible dimension difference	1)	1)	1)	1)
_OVR [30]	REAL	Empirical value	1)	1)		1)
_OVR [31]	REAL	Mean value	1)	1)	1)	1)
_OVI [0]	INT	D number / ZO number				
_OVI [1]	INT					
_OVI [2]	INT	Measuring cycle number				
_OVI [3]	INT	Measuring variant				
_OVI [4]	INT	Weighting factor	1)	1)	1)	1)
_OVI [5]	INT	Probe number				
_OVI [6]	INT	Mean value memory number	1)	1)	1)	1)
_OVI [7]	INT	Empirical value memory number	1)	1)		1)
_OVI [8]	INT	Tool number				
_OVI [9]	INT	Alarm number				
_OVI [11]	INT	Status offset request (for ZO correction only)				
_OVI12]	INT					
_OVI13]	INT	DL number (from meas. cycles SW 6.3)	1)	1)	1)	1)

1) for workpiece measurement with tool offset only

© Siemens AG, 2004. All rights reserved

NC machine data					
MD number	**Name of identifier**	**Description**	**Max. input value**	**Default value**	**Value for meas. cycles**
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 characteristics of 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 variable names in the control	plus	10	30
18130	MM_NUM_GUD_NAMES_CHAN	Number of GUD variable names per channel	plus	40	130
18150	MM_GUD_VALUES_MEM	Memory space for the values of the GUD variables	plus0	12/16 1)	28/32 1)
18170	MM_NUM_MAX_FUNC_NAMES	Number of miscellaneous functions (cycles, DRAM)	plus	40	70
18180	MM_NUM_MAX_FUNC_PARAM	Number of miscellaneous functions (cycles, DRAM)	plus	300	600
28020	MM_NUM_LUD_NAMES_TOTAL	Number of LUD variables in total (in all program levels)	plus	200	200
28040	MM_NUM_LUD_VALUES_MEM	Memory for values of the LUD variables	plus	12/25 1)	14/27 1)
28082	MM_SYSTEM_FRAME_MASK (as from measuring cycle SW 6)	Channel-specific system frames	7FH	21H	21H (Bit0, 5=1)
NC machine data for measuring in JOG					
11602	ASUP_START_MASK	Ignore stop conditions for ASUB	3	0	1, 3 Bit0=1
11604	ASUP_START_PRIO_LEVEL	Priority for ASUP_START_MASK effective	64H	0	From 1 to 64H
20110	RESET_MODE_MASK	Define control default setting after power-up and RESET	07FFFH	0	min. 4045H (Bit0, 2, 6, 14=1)
20112	START_MODE_MASK	Define control default setting after part program start	07FFFH	400H	400H (Bit6=0)
1) for hardware NCU 3					

© Siemens AG, 2004. All rights reserved

Cycle data The measuring cycle data are stored in blocks GUD5 and GUD6.			
Central values			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_CVAL[ ]	**Number of elements**	
GUD6	_CVAL[0]	Number of data fields for tool probe, machine-related	3
GUD6	_CVAL[1]	Number of data fields for workpiece probe	3
GUD6	_CVAL[2]	Number of data fields for gauging block	3
GUD6	_CVAL[3]	Number of data fields for tool probe, tool-related (from measuring cycles SW 6.3)	
	_TP[ ]	**Tool probe** (machine-related)	
	Assignment for milling		
GUD6	_TP[k,0]	Trigger point in minus direction **X** (1st geometry axis)	0
GUD6	_TP[k,1]	Trigger point in plus direction **X** (1st geometry axis)	0
GUD6	_TP[k,2]	Trigger point in minus direction **Y** (2nd geometry axis)	0
GUD6	_TP[k,3]	Trigger point in plus direction **Y** (2nd geometry axis)	0
GUD6	_TP[k,4]	Trigger point in minus direction **Z** (3rd geometry axis)	0
GUD6	_TP[k,5]	Trigger point in plus direction **Z** (3rd geometry axis)	0
GUD6	_TP[k,6]	Edge length/disk diameter	0
GUD6	_TP[k,7]	Assigned internally	133
GUD6	_TP[k,8]	Probe type 0: cube 101: disk in XY 201: disk in ZX 301: disk in YZ	0
GUD6	_TP[k,9]	Distance between upper edge of tool probe and lower edge of tool	2
	Assignment for turning		
GUD6	_TP[k,0]	Trigger point in minus direction, abscissa	0
GUD6	_TP[k,1]	Trigger point in plus direction, abscissa	0
GUD6	_TP[k,2]	Trigger point in minus direction, ordinate	0
GUD6	_TP[k,3]	Trigger point in plus direction, ordinate	0
GUD6	_TP[k,4]	Irrelevant	0
	to		
GUD6	_TP[k,9]	Irrelevant	0
	_TPW[ ]	**Tool probe** (workpiece-related).	
	Assignment for milling		
GUD6	_TPW[k,0]	Trigger point in minus direction **X** (1st geometry axis)	0
GUD6	_TPW[k,1]	Trigger point in plus direction **X** (1st geometry axis)	0
GUD6	_TPW[k,2]	Trigger point in minus direction **Y** (2nd geometry axis)	0
GUD6	_TPW[k,3]	Trigger point in plus direction **Y** (2nd geometry axis)	0
GUD6	_TPW[k,4]	Trigger point in minus direction **Z** (3rd geometry axis)	0
GUD6	_TPW[k,5]	Trigger point in plus direction **Z** (3rd geometry axis)	0
GUD6	_TPW[k,6]	Edge length/disk diameter	0
GUD6	_TPW[k,7]	Assigned internally	133
GUD6	_TPW[k,8]	Probe type 0: cube 101: disk in XY 201: disk in ZX 301: disk in YZ	0
GUD6	_TPW[k,9]	Distance between upper edge of tool probe and lower edge of tool	2

© Siemens AG, 2004. All rights reserved

	Assignment for turning		
GUD6	_TPW[k,0]	Trigger point in minus direction, abscissa	0
GUD6	_TPW[k,1]	Trigger point in plus direction, abscissa	0
GUD6	_TPW[k,2]	Trigger point in minus direction, ordinate	0
GUD6	_TPW[k,3]	Trigger point in plus direction, ordinate	0
GUD6	_TPW[k,4]	irrelevant	0
	to		
GUD6	_TPW[k,9]	irrelevant	0
	_WP[ ]	**Workpiece probe**	
GUD6	_WP[k,0]	Ball diameter	6
GUD6	_WP[k,1]	Trigger point in minus direction of abscissa	3
GUD6	_WP[k,2]	Trigger point in plus direction of abscissa	-3
GUD6	_WP[k,3]	Trigger point in minus direction of ordinate	3
GUD6	_WP[k,4]	Trigger point in plus direction of ordinate	-3
GUD6	_WP[k,5]	Trigger point in minus direction of applicate	3
GUD6	_WP[k,6]	Trigger point in plus direction of applicate	-3
GUD6	_WP[k,7]	Position deviation abscissa	0
GUD6	_WP[k,8]	Position deviation ordinate	0
GUD6	_WP[k,9]	Calibration status, coded	0
GUD6	_WP[k,10]	Calibration status, coded	0

Central values			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_KB[ ]	**Gauging block**	
GUD6	_KB[k,0]	Groove edge in plus direction, ordinate	0
GUD6	_KB[k,1]	Groove edge in minus direction, ordinate	0
GUD6	_KB[k,2]	Groove base in abscissa	0
GUD6	_KB[k,3]	Groove edge in plus direction, abscissa	0
GUD6	_KB[k,4]	Groove edge in minus direction, abscissa	0
GUD6	_KB[k,5]	Upper edge groove in ordinate	0
GUD6	_KB[k,6]	Groove base in ordinate	0
	_CM[ ]	**Monitoring functions _CBIT[12] = 0**	
GUD6	_CM[k,0]	Max. permissible peripheral speed [m/min]/[feet/min]	60
GUD6	_CM[k,1]	Max. permissible speed [rpm]	2000
GUD6	_CM[k,2]	Minimum feedrate for probing [mm/min]	1
GUD6	_CM[k,3]	Required measuring accuracy [mm]	0.005
GUD6	_CM[k,4]	Max. permissible feedrate for probing	20
GUD6	_CM[k,5]	Direction of spindle rotation	4
GUD6	_CM[k,6]	Feed factor 1	10
GUD6	_CM[k,7]	Feed factor 2	0
	_MFS[ ]	**Speed and feedrate _CBIT[12] = 1**	
GUD6	_MFS[k,0]	Speed 1st probing	0
GUD6	_MFS[k,1]	Feed 1st probing	0
GUD6	_MFS[k,2]	Speed 2nd probing	0
GUD6	_MFS[k,3]	Feed 2nd probing	0
GUD6	_MFS[k,4]	Speed 3rd probing	0
GUD6	_MFS[k,5]	Feed 3rd probing	0

© Siemens AG, 2004. All rights reserved

Central value for logging			
GUD6	**_PROTFORM**	Int field for formatting for log	
GUD6	_PROTFORM[0]	Number of line per page	60
GUD6	_PROTFORM[1]	Number of characters per line	80
GUD6	_PROTFORM[2]	First page number	1
GUD6	_PROTFORM[3]	Number of header lines	5
GUD6	_PROTFORM[4]	Number of value lines in the log	1
GUD6	_PROTFORM[5]	Number of characters per column	12
GUD6	**_PROTSYM**	Separator in the log	
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: Log header output	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

Central bits			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_CBIT[ ]	**Central bits**	
GUD6	_CBIT[0]	Measurement repetition after violation of dimensional difference and safe area 0: no measurement repeat 1: measurement repeat, up to 4	0
GUD6	_CBIT[1]	Alarm and M0 for measurement repeat with _CBIT[0]=1 0: no alarm, no M0 generated 1: M0 and an alarm are generated before each repeat	0
GUD6	_CBIT[2]	M0 for tolerance alarms "oversize", "undersize", or "permissible dimensional difference exceeded" 0: no generation of M0 for the above alarms 1: generation of M0 for the above alarms	0
GUD6	_CBIT[3]	Central marker for basic dimension system of the control 0: Basic system setting is inches 1: Basic system setting is metric Note. If _CBIT[3] and MD 10240 are not equal, the value in fields TP[ ], _TPWW[ ], _KB[ ], _MV[ ], _CM, and _MFS are converted.	1
GUD6	_CBIT[4]	currently not assigned	0
GUD6	_CBIT[5]	Tool measurement and calibration in the WCS in CYCLE982 (from measuring cycles SW 5.4) 0: machine-related measurement and calibration 1: workpiece-related measurement and calibration Note: In both cases, the _TP[ ] field of the probe is used. From measuring cycles SW 6.3, function switchover via _MVAR is available.	0
GUD6	_CBIT[6]	Logging without output of the measuring cycle name and measuring variant (from measuring cycles SW 6.2) 0: Measuring cycle name and measuring variant will be output. 1: These outputs will be suppressed.	0
GUD6	_CBIT[7]	currently not assigned	0
GUD6	_CBIT[8]	Offset of the mono probe setting 0: No compensation 1: offset of spindle by angle _CORA,	0
GUD6	_CBIT[9]	Assigned internally	0
GUD6	_CBIT[10]	currently not assigned	0
GUD6	_CBIT[11]	Selection of log header for logging 0: Standard 1: user-defined	0

© Siemens AG, 2004. All rights reserved

GUD6	_CBIT[12]	Feed and speed in CYCLE971 0: calculation by measuring cycle itself 1: set by user in array _MFS[ ]	0
GUD6	_CBIT[13]	Deletion of the measuring cycle data fields in the GUD6 0: No deletion 1: delete _TP[ ], _TPW[ ], _WP[ ], _KB[ ], _EV[ ], _MV[ ], _CBIT[13]	0
GUD6	_CBIT[14]	Length reference of the workpiece probe in milling measuring cycles (from measuring cycles SW 4.5) 0: length relative to probe ball center 1: length relative to end	0
GUD6	_CBIT[15]	Transfer of workpiece probe data into the tool offset in CYCLE976 (from measuring cycles SW 4.5) 0: no transfer 1: result of probe ball calculation on **calibration** will be entered in the geometry memory of the workpiece probe (radius)	0

Central strings			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_SI	**Central strings**	
GUD6	_SI[0]	currently not assigned	0
GUD6	_SI[1]	Software version	6
Central strings for logging			
	_PROTNAME (32 chars)		
GUD6	_PROTNAME [0]	Name of the main program to log from (for log header)	
GUD6	_PROTNAME [1]	Name of the log file to be created	
GUD6	**_HEADLINE (80 chars)**	Strings for log header (80 chars)	
GUD6	_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)	

Channel-oriented values			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_EVMVNUM	**Number of empirical values and mean values**	
GUD6	_EVMVNUM[0]	Number of empirical values	20
GUD6	_EVMVNUM[1]	Number of mean values	20
	_SPEED	**Traversing velocities for intermediate positioning**	
GUD6	_SPEED[0]	Max. rapid traverse in % (only active with collision monitoring switched off, max 100 %)	100
GUD6	_SPEED[1]	Positioning velocity in the plane with collision monitoring active	1000
GUD6	_SPEED[2]	Positioning velocity applicate	1000
GUD6	_SPEED[3]	Fast measuring feed	900
	_EV	**Empirical values**	
GUD5	_EV[x]	Empirical value	0
	_MV	**Mean values**	
GUD5	_MV[x]	Mean value	0

© Siemens AG, 2004. All rights reserved

Channel-specific values (for measuring in JOG)			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_JM_I		
GUD6	**_JM_I** [0]	Setting for the workpiece probe number 0: set by *JM*I[1] 1: set by tool parameters (ShopMill)	0
GUD6	**_JM_I** [1]	Probe number for workpiece measurement (_PRNUM) only if *JM*I[0]=0	1
GUD6	**_JM_I** [2]	Probe number for tool measurement (_PRNUM)	1
GUD6	**_JM_I** [3]	Working plane 17: G17 18: G18 19: G19 Every other value for working plane is defined in the machine data.	17
GUD6	**_JM_I** [4]	Definition of the active ZO on measurement 0: G500 1: G54 ... 4: G57 5...: G505... 100: active ZO is defined in machine data	0
GUD6	**_JM_I** [5] (as from meas. cycle SW 6.3)	Offset 0: standard offset (measuring only, basic reference, settable ZO) 1: extended offset (additional global and channel-specific basic frame)	0
GUD6	**_JM_I** [6] (as from meas. cycle SW 6.3)	Internal copy of _JM_I[4]	0
GUD6	**_JM_I** [7] to **_JM_I** [9] (as from meas. cycle SW 6.3)	Reserved	-

© Siemens AG, 2004. All rights reserved

Channel-specific values (for measuring in JOG, from measuring cycles SW6.3, GUD7_MC)			
Block	**Name of identifier**	**Description**	**As-delivered value**
GUD7	E_MESS_IS_METRIC E_MESS_IS_METRIC_SPEZ_VAR=1	All dimensioned data are metric	1
GUD7	E_MESS_MS_IN	Measurement input 1 for workpiece measurement	0
GUD7	E_MESS_MT_IN	Measurement input 2 for tool measurement	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 meas. 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
GUD7	E_MESS_FM	Measuring feed [mm/rev]	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_CAL_D (from measuring cycles SW 6.3)	Diameter, calibration ring	0
GUD7	E_MESS_CAL_L (from measur. cycles SW 6.3)	Calibration dimension in the feed axis (referred to WCS)	0
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. feed for measuring with rotating spindle for the 1st probing [mm/min]	1
GUD7	E_MESS_MIN_F_FAK1	On tool measurement with rotating spindle, traversal with 10 times measuring feed is performed in the 1st probing (limitation by E_MESS_MAX_F) [mm/min]	10
GUD7	E_MESS_MIN_F_FAK2	For tool measurement with rotating spindle, 2nd probing is performed with measuring feed. There is no 3rd probing [mm/min]	0
GUD7	E_MESS_MIN_D	Measuring accuracy for measuring with rotating spindle [mm/min]	0.01
GUD7	E_MESS_MT_TYP[3]	Type of 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 probe for length measurement	0
GUD7	E_MESS_MT_DR[3] 1)	Diameter of tool probe for radius measurement	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_MT_D	Calibrate measurement path for tool probe and tool measurement with motionless spindle (before and after expected switching position)	10
GUD7	E_MESS_MT_FM	Calibrate measuring feed for tool probe and tool measurement with motionless spindle	100
GUD7	E_MESS_MT_CF	No tool probe make (manufacturer) specified	0
GUD7	E_MESS_MT_COMP	No offset of the measurement result on tool measurement with rotating spindle	0
GUD7	E_MESS[3]	Internal data item	
GUD7	E_MEAS	Internal data item	
GUD7	E_MESS_RETT	Internal data item	
GUD7	E_MESS_SETT[10]	Field for settings	
GUD7	E_MESS_AM	Internal data item	

1) During installation value input is mandatory here!

© Siemens AG, 2004. All rights reserved

Channel-oriented bits			
Block	**Name of identifier**	**Description**	**As-delivered value**
	_CHBIT	**Channel bits**	
GUD6	_CHBIT[0]	Measurement input for workpiece measurement 0: measuring input 1 1: measurement input 2	0
GUD6	_CHBIT[1]	Measurement input for tool measurement 0: measuring input 1 1: measurement input 2	1
GUD6	_CHBIT[2]	Collision monitoring for intermediate positioning 0: OFF 1: ON	1
GUD6	_CHBIT[3]	Tool offset mode for tool measurement 0: first-time measurement (determining geometry) 1: remeasuring (determining wear)	0
GUD6	_CHBIT[4]	Mean value for workpiece measurement with automatic tool correction (_EVNUM>0) 0: no mean value derivation over several parts 1: with mean value formation and calculation	0
GUD6	_CHBIT[5]	Inclusion of empirical value (_EVNUM>0) 0: subtraction of actual value 1: addition to actual value	0
GUD6	_CHBIT[6]	Tool offset mode for workpiece measurement with automatic tool offset 0: Offset in wear 1: offset in geometry, delete wear From meas. cycles SW 6.3: For additive and setup offset and _CHBIT[8]=0: 0: offset in additive offset 1: offset in set-up offset, delete additive offset	0
GUD6	_CHBIT[7]	Measured value offset in CYCLE994 0: use of trigger values of the probe _WP[k,1] ... 1: use of the active ball diameter of the probe _WP[k,0]	0
GUD6	_CHBIT[8]	From meas. cycles SW 6.3: Offset mode for workpiece measurement with automatic tool offset 0: additive, setup offset according to _CHBIT[6] 1: offset additive in set-up offset, irrespective of _CHBIT[6]	0
GUD6	_CHBIT[9]	currently not assigned	0
GUD6	_CHBIT[10]	Measuring result display 0: OFF 1: ON	0
GUD6	_CHBIT[11]	Acknowledgment measurement result screen with NC start 0: OFF (If _CHBIT[18]=0, the display is automatically deselected at end of cycle.) 1: ON (M0 is generated in the cycle.)	0
GUD6	_CHBIT[12]	currently not assigned	0
GUD6	_CHBIT[13]	Coupling spindle position with coordinate rotation in active plane for workpiece measurement with multi probe 0: OFF 1: ON	0
GUD6	_CHBIT[14]	Adapt spindle positioning, if _CHBIT[13]=1 0: according to default 1: adapted angle values	0
GUD6	_CHBIT[15]	Number of measurements on failure to switch 0: up to 5 measurements 1: only 1 measurement	0
GUD6	_CHBIT[16]	Retraction velocity from the measuring point 0: velocity as for intermediate positioning 1: with percentage of rapid traverse velocity (_SPEED[0])(only active with collision monitoring ON) _CHBIT[2]=1)	0
GUD6	_CHBIT[17]	Feed during measurement 0: with feed in _VMS 1: on 1st measurement feed in _SPEED[3] on 2nd measurement with feed in _VMS	0

© Siemens AG, 2004. All rights reserved

GUD6	_CHBIT[18]	Static measurement result display 0: effect as set in _CHBIT[11]. 1: only active if _CHBIT[11]=0: Measuring result display remains until next measuring cycle is called	0
GUD6	_CHBIT[19]	(CYCLE974 and CYCLE994 only): Special treatment of Y axis with G18 0: no special treatment 1: setpoint setting and parameterization (_SETVAL, _TUL, _TLL, SZO) for the Y axis (applicate) as for the parameterization of the ordinate (X axis) The tool offset is applied in the length that is active in the ordinate (X axis) (usually L1), as long as not other length has been set in _KNUM. The ZO correction is applied in the specified ZO memory in the ordinate component (X axis)	0
GUD6	_CHBIT[20]	(CYCLE982 only): Suppression of the starting angle positioning _STA1 0: suppression OFF 1: suppression ON	0
GUD6	_CHBIT[21]	(CYCLE974, CYCLE977, CYCLE978, CYCLE979, CYCLE9997 only) Mode of ZO correction 0: offset additive in FINE 1: offset in COARSE, delete FINE	0
GUD6	_CHBIT[22]	(CYCLE971 only): with rotating spindle and multiple measurement with rotating spindle and multiple measurement 0: last measurement with reduced speed if _CBIT[12]=0 1: no speed reduction	0
GUD6	_CHBIT[23]	From measuring cycles SW 6.3 (CYCLE982 only) Recoding of tool point direction during tool measurement 0: no recoding 1: internal recording (tool point direction mirroring about X)	0
GUD6	_CHBIT[24]	Channel-oriented marker for basic dimension system (On deviation from MD 10240, cycle parameters are converted internally and this bit is reset.) 0: The basic system is inch 1: The basic system is metric	1

Channel-specific bits (for measurement in JOG)			
Block	**Name of identifier**	**Description**	**Value for meas. cycles**
	_JM_B		
GUD6	**_JM_B**[0]	Tool offset mode for tool measurement 0: offset in geometry, wear is deleted	0
GUD6	**_JM_B**[1]	Number of measurement attempts	0
GUD6	**_JM_B**[2]	Retraction from measuring point in rapid traverse	0
GUD6	**_JM_B**[3]	Fast measuring feed	0
GUD6	**_JM_B**[4]	Selection offset frame	0
GUD6	**_JM_B**[5]	Currently not assigned	0
GUD6	**_JM_B**[6]	Internal data item	0

© Siemens AG, 2004. All rights reserved

B Abbreviations

ASUB	Asynchronous subroutine
CNC	Computerized Numerical Control
CPU	Central Processing Unit
DIN	Deutsche Industrie-Norm (German Industry Standard)
DOS	Disk Operating System
DRF	Differential Resolver Function (handwheel)
FM-NC	Function Module Numerical Control
GUD	Global User Data
I/O	Input/Output
IBN	Startup
JOG	Jogging: Setup mode
LUD	Local User Data
MCS	Machine coordinate system
MD	Machine data
MMC	Man-Machine Communication: User interface on numerical control systems for operator control, 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: NCK hardware unit
OI	User interface

© Siemens AG, 2004. All rights reserved

PCIN	Name of SW for data exchange with the control
PG	Programming device
PLC	Programmable Logic Control: Programmable logic control
RS-232-C	Serial interface (definition of the exchange lines between DTE and DCE)
SR	Subprogram
SW	Software
TO	Tool Offset
TOA	Tool Offset Active identifier (file type) for tool offsets
WCS	Workpiece coordinate system
ZO	Zero offset

© Siemens AG, 2004. All rights reserved

C Terms

Important terms are listed in alphabetical order. The symbol "->" precedes terms that are explained in a separate entry in this list.

A

Actual/set difference
Difference between measured and expected value.

Asynchronous subroutine
A parts program, which can be started asynchronously to (independently of) the current program status by an interrupt signal (e.g., "rapid NC input" signal).

B

Blank measurement
The blank measurement ascertains the position, deviation, and zero offset of the workpiece in the result of a -> workpiece measurement.

C

Calibrating 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).

Calibration
During calibration, the trigger points of the probe are ascertained and stored in the measuring cycle data in block GUD6.

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.

D

Data blocks for measuring cycles
Data blocks GUD5.DEF, GUD6.DEF, GUD7DEF and GUD7.MC.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.

© Siemens AG, 2004. All rights reserved

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.

E

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

F-K

L

Logging measurement results Measurement results can be optionally be logged in a file located in the part program memory. The log can be output from the control either via RS-232-C or on a diskette.

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.

Measurement at an 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 defined in the WCS.

© Siemens AG, 2004. All rights reserved

Measurement path
Measurement path _FA defines the distance between the starting position and the expected switching position (setpoint) of the probe. Always specify _FA in **mm**.

Measurement result display
Measurement result displays can be shown automatically during measuring cycle runtime.
Activation of this function depends on the settings in the measuring cycle data.

Measuring accuracy
The measurement accuracy that can be obtained is dependent on the following factors:

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

The repeat accuracy of the controls for "on-the-fly measurement" is ±1 μm.

Measuring in JOG
It contains 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 function is operated with softkeys and input displays.

Measuring the 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.

Measuring variant
The measuring 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 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.

Mono probe
A mono(directional) probe is a probe that can only deflect in one direction. It can only be used for workpiece measurement on milling machines and machining centers with slight limitations.

Multi probe
A multi(directional) probe is one that can deflect in three dimensions.

© Siemens AG, 2004. All rights reserved

Multiple measurement at the same location Parameter _NMSP can be used to determine the number of measurements at the same location.
The actual/set difference is determined arithmetically.

N

O

Offset angle position If a -> mono probe is used, the position of the probe can also be corrected for machine-specific reasons using the parameter _CORA.

Offset axis With some measuring variants, for example, in CYCLE998, positioning in another axis that must be defined, also called offset axis can be performed between measurements in the measuring axis. This is must be defined in parameter _MA with offset axis/measuring axis.

On-the-fly measurement This method processes the probe signal directly in the NC.

P

Paraxial measurement A measuring 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 (skew) 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 active diameter of the probe ball. 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.

© Siemens AG, 2004. All rights reserved

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

Tolerance bottom 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.

Tolerance top 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.

Tool environment

As from NCK SW 6.3, you can now save the operating environment of a particular tool you are using. This is to allow you to correct the tool used to measure a workpiece taking into account the operating conditions (environment: G commands, setting data, ...).
You then no longer have to specify the T, D, DL number in the offset explicitly. These are included in the stored tool environment. The name of a tool environment can have up to 32 characters.

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 block GUD6 for the axis direction.

U

© Siemens AG, 2004. All rights reserved

V

Variable measuring speed

The measuring speed can be freely selected by means of _VMS.
The maximum measuring velocity must be selected to ensure safe deceleration within the probe deflecting path. -> Measuring velocity

W

Weighting factor for mean value calculation

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 measuring cycles makes it possible to perform nearly all measurements required on a milling or turning machine.

X

Y

Z

Zero offset area

This tolerance range (lower limit _TZL) corresponds to the amount of maximum accidental dimensional deviations. If the absolute value of the actual/set difference is less than the zero offset range, the offset is not applied.

ZO determination

In the result of a measurement, the actual-setpoint value difference is stored in the data set of any settable zero offset.

© Siemens AG, 2004. All rights reserved

D References

General Documentation

/BU/ SINUMERIK & SIMODRIVE, Automation Systems for Machine Tools
Catalog NC 60
Order No.: E86060-K4460-A101-A9-7600

/IKPI/ Industrial Communications and Field Devices
Catalog IK PI
Order No.: E86060-K6710-A101-B2-7600

/ST7/ SIMATIC
Products for Totally Integrated Automation and Micro Automation
Catalog ST 70
Order No.: E86060-K4670-A111-A8-7600

/Z/ MOTION-CONNECT
Connections & System Components for SIMATIC, SINUMERIK, MASTERDRIVES, and SIMOTION
Catalog NC Z
Order No.: E86060-K4490-A001-B1-7600

Electronic Documentation

/CD1/ The SINUMERIK System (10.04 Edition)
DOC ON CD
(includes all SINUMERIK 840D/840Di/810D/802- and SIMODRIVE publications)
Order No.: 6FC5298-7CA00-0BG0

© Siemens AG, 2004. All rights reserved

User Documentation

/AUK/ SINUMERIK 840D/810D (09.99 Edition)
Short Guide **AutoTurn**
Order No.: 6FC5298-4AA30-0BP2

/AUP/ SINUMERIK 840D/810D (02.02 Edition)
Operator's Guide **AutoTurn Graphic Programming System**
Programming/Setup
Order No.: 6FC5298-4AA40-0BP3

/BA/ SINUMERIK 840D/810D (10.00 Edition)
Operator's Guide **MMC**
Order No.: 6FC5298-6AA00-0BP0

/BAD/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Operator's Guide **HMI Advanced**
Order No.: 6FC5298-6AF00-0BP3

/BAH/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Operator's Guide **HT 6**
Order No.: 6FC5298-0AD60-0BP3

/BAK/ SINUMERIK 840D/840Di/810D (02.01 Edition)
Short Guide Operation
Order No.: 6FC5298-6AA10-0BP0

/BAM/ SINUMERIK 810D/840D (08.02 Edition)
Operation/Programming **ManualTurn**
Order No.: 6FC5298-6AD00-0BP0

/BAS/ SINUMERIK 840D/840Di/810D (10.04 Edition)
Operation/Programming **ShopMill**
Order No.: 6FC5298-6AD10-0BP3

/BAT/ SINUMERIK 840D/810D (06.03 Edition)
Operation/Programming **ShopTurn**
Order No.: 6FC5298-6AD50-0BP2

/BEM/ SINUMERIK 840D/810D (03.04 Edition)
Operator's Guide **HMI Embedded**
Order No.: 6FC5298-6AC00-0BP3

/BNM/ SINUMERIK 840D/840Di/810D (10.04 Edition)
User's Guide **Measuring Cycles**
Order No.: 6FC5298-7AA70-0BP1

/BTDI/ SINUMERIK 840D/840Di/810D (06.04 Edition)
Motion Control Information System (MCIS)
User's Guide **Tool Data Information**
Order No.: 6FC5297-6AE01-0BP1

/CAD/ SINUMERIK 840D/840Di/810D (03.02 Edition)
Operator's Guide **CAD Reader**
Order No.: (included in online help)

/DA/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Diagnostics Guide
Order No.: 6FC5298-6AA20-0BP4

© Siemens AG, 2004. All rights reserved

/KAM/ SINUMERIK 840D/810D (04.01 Edition)
Short Guide **ManualTurn**
Order No.: 6FC5298-5AD40-0BP0

/KAS/ SINUMERIK 840D/810D (04.01 Edition)
Short Guide **ShopMill**
Order No.: 6FC5298-5AD30-0BP0

/KAT/ SINUMERIK 840D/810D (07.01 Edition)
Short Guide **ShopTurn**
Order No.: 6FC5298-6AF20-0BP0

/PG/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Programming Guide **Fundamentals**
Order No.: 6FC5298-7AB00-0BP0

/PGA/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Programming Guide **Advanced**
Order No.: 6FC5298-7AB10-0BP0

/PGA1/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Lists Manual **System Variables**
Order No.: 6FC5298-7AE10-0BP0

/PGK/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Short Guide **Programming**
Order No.: 6FC5298-7AB30-0BP0

/PGM/ SINUMERIK 840D/840Di/810D (11.02 Edition)
Programming Guide **ISO Milling**
Order No.: 6FC5298-6AC20-0BP2

/PGT/ SINUMERIK 840D/840Di/810D (11.02 Edition)
Programming Guide **ISO Turning**
Order No.: 6FC5298-6AC10-0BP2

/PGZ/ SINUMERIK 840D/840Di/810D (10.04 Edition)
Programming Guide **Cycles**
Order No.: 6FC5 298-7AB40-0BP1

/PI/ PCIN 4.4
Software for Data Transfer to/from **MMC Modules**
Order number: 6FX2060-4AA00-4XB0 (English, German, French)
Order from: WK Fürth

/SYI/ SINUMERIK 840Di (02.01 Edition)
System Overview
Order No.: 6FC5298-6AE40-0BP0

© Siemens AG, 2004. All rights reserved

Manufacturer/Service Documentation

a) Lists

/LIS/ SINUMERIK 840D/840Di/810D (03.04 Edition)
SIMODRIVE 611D
Lists
Order No.: 6FC5297-7AB70-0BP0

b) Hardware

/ASAL/ SIMODRIVE (10.03 Edition)
Planning Guide General Information for **Asynchronous Motors**
Order No.: 6SN1197-0AC62-0BP0

/APH2/ SIMODRIVE (10.03 Edition)
Planning Guide **Asynchronous Motors 1PH2**
Order No.: 6SN1197-0AC63-0BP0

/APH4/ SIMODRIVE (10.03 Edition)
Planning Guide **Asynchronous Motors 1PH4**
Order No.: 6SN1197-0AC64-0BP0

/APH7S/ SIMODRIVE (04.04 Edition)
Planning Guide **Asynchronous Motors 1PH7**
Order No.: 6SN1197-0AC65-0BP0

/APL6/ MASTERDRIVES VC/MC (03.04 Edition)
Planning Guide **Asynchronous Motors 1PL6**
Order No.: 6SN1197-0AC67-0BP0

/BH/ SINUMERIK 840D/840Di//810D (04.04 Edition)
Operator Components Manual (HW)
Order No.: 6FC5297-6AA50-0BP0

/BHA/ SIMODRIVE Sensor (03.04 Edition)
User Guide (HW) **Absolute Position Sensor with Profibus DP**
Order No.: 6SN1197-0AB10-0YP3

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE (03.04 Edition)
Planning Guide **EMC Installation Guide**
Order No.: 6FC5297-0AD30-0BP2

The current Declaration of Conformity is available under the following Internet address:
http://www4.ad.siemens.de

Please enter the ID no. 15257461 in the "Search" field (top right) and click on "go".

/GHA/ SINUMERIK/SIMOTION (02.03 Edition)
ADI4 - Analog Drive Interface for 4 Axes
Manual
Order No.: 6FC5297-0BA01-0BP1

/PFK6/ SIMODRIVE 611, SIMOVERT MASTERDRIVES (05.03 Edition)
Planning Guide **1FK6 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD05-0BP0

/PFK7/ SIMODRIVE 611, SIMOVERT MASTERDRIVES (01.03 Edition)
Planning Guide **1FK7 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD06-0BP0

© Siemens AG, 2004. All rights reserved

/PFS6/ SIMOVERT MASTERDRIVES (07.03 Edition)
Planning Guide **1FS6 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD08-0BP0

/PFT5/ SIMODRIVE (02.04 Edition)
Planning Guide **1FT5 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD01-0BP0

/PFT6/ SIMODRIVE 611, SIMOVERT MASTERDRIVES (01.04 Edition)
Planning Guide
1FT6 Three-Phase AC Servomotors
Order No.: 6SN1197-0AD02-0BP0

/PFU/ SINAMICS, SIMOVERT MASTERDRIVES, MICROMASTER (09.03 Edition)
SIEMOSYN Motors 1FU8
Order No.: 6SN1197-0AC80-0BP0

/PHC/ SINUMERIK 810D (11.02 Edition)
Configuring Manual CCU **(HW)**
Order No.: 6FC5297-6AD10-0BP1

/PHD/ SINUMERIK 840D (11.04 Edition)
Configuring Manual **NCU (HW)**
Order No.: 6FC5297-6AC10-0BP3

/PJAL/ SIMODRIVE 611, SIMOVERT MASTERDRIVES (01.03 Edition)
Planning Guide **Three-Phase Servomotors**
General Part for 1FT / 1FK Motors
Order No.: 6SN1197-0AD07-0BP1

/PJFE/ SIMODRIVE (02.03 Edition)
Planning Guide **1FE1 Built-In Synchronous Motors**
Three-Phase AC Motors for Main Spindle Drives
Order No.: 6SN1197-0AC00-0BP4

/PJF1/ SIMODRIVE (12.02 Edition)
Installation Guide **1FE1 051.-1FE1 147**. **Built-In Synchronous Motors**
AC Motors for Main Spindle Drives
Order No.: 610.43000.02

/PJLM/ SIMODRIVE (06.02 Edition)
Planning Guide **1FN1, 1FN3 Linear Motors**
ALL General Information about Linear Motor
1FN1 1FN1 Three-Phase AC Linear Motor
1FN3 1FN3 Three-Phase AC Linear Motor
CON Connections
Order No.: 6SN1197-0AB70-0BP3

/PJM/ SIMODRIVE 611, SIMOVERT MASTERDRIVES (11.00 Edition)
Planning Guide **Motors**
Three-Phase Motors for Feed and Main Spindle Drives
Order No.: 6SN1197-0AA20-0BP4

/PJM2/ SIMODRIVE (07.03 Edition)
Planning Guide **Servomotors**
Three-Phase Servo Motors for Feed and Main Spindle Drives
Order No.: 6SN1197-0AC20-0BP0

© Siemens AG, 2004. All rights reserved

/PJTM/ SIMODRIVE (11.03 Edition)
Planning Guide **1FW6 Built-In Torque Motors 1FW6**
Order No.: 6SN1197-0AD00-0BP2

/PJU/ SIMODRIVE 611 (10.04 Edition)
Planning Guide **Converters**
Order No.: 6SN1197-0AA00-0BP7

/PMH/ SIMODRIVE Sensor (07.02 Edition)
Configuring/Installation Guide
Hollow-Shaft Measuring System SIMAG H
Order No.: 6SN1197-0AB30-0BP1

/PMHS/ SIMODRIVE (12.00 Edition)
Installation Guide **Measuring System for Main Spindle Drives**
SIZAG2 Toothed-Wheel Encoder
Order No.: 6SN1197-0AB00-0YP3

/PMS/ SIMODRIVE (03.04 Edition)
Planning Guide
ECO Motor Spindle for Main Spindle Drives
Order No.: 6SN1197-0AD04-0BP1

/PPH/ SIMODRIVE (12.01 Edition)
Planning Guide **1PH2 / 1PH4 / 1PH7 Motors**
AC Induction Motors for Main Spindle Drives
Order No.: 6SN1197-0AC60-0BP0

/PPM/ SIMODRIVE (11.01 Edition)
Planning Guide
Hollow-Shaft Motors for **1PM4 and 1PM6** Main Spindle Drives
Order No.: 6SN1197-0AD03-0BP0

c) Software

/FB1/ SINUMERIK 840D/840Di/810D/FM-NC (03.04 Edition)
Description of Function **Basic Machine (Part 1)**
(the various sections are listed below)
Order No.: 6FC5297-7AC20-0BP0
A2 Various Interface Signals
A3 Axis Monitoring, Protection Zones
B1 Continous-Path Mode, Exact Stop and Look Ahead
B2 Acceleration
D1 Diagnostic Tools
D2 Interactive Programming
F1 Traverse to Fixed Stop
G2 Velocities, Setpoint/Actual Value Systems, Closed-Loop Control
H2 Output of Auxiliary Functions to PLC
K1 Mode Group, Channel, Program Operation Mode
K2 Axes, Coordinate Systems, Frames,
Actual-Value System for Workpiece, External Zero Offset
K4 Communication
N2 EMERGENCY STOP
P1 Traverse Axes
P3 Basic PLC Program
R1 Reference Point Approach
S1 Spindles
V1 Feeds
W1 Tool Offset

© Siemens AG, 2004. All rights reserved

/FB2/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Description of Functions **Extended Functions (Part 2)**
including FM-NC: Turning, Stepper Motor
(the various sections are listed below)
Order No.: 6FC5297-7AC30-0BP0
A4 Digital and Analog NCK I/Os
B3 Several Operator Panels and NCUs
B4 Operation via PG/PC
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 Measurement
N3 Software Cams, Position Switching Signals
N4 Punching and Nibbling
P2 Positioning Axes
P5 Oscillation
R2 Rotary Axes
S3 Synchronous Spindle
S5 Synchronized Actions (SW 3 and lower, higher /FBSY/)
S6 Stepper Motor Control
S7 Memory Configuration
T1 Indexing Axes
W3 Tool Change
W4 Grinding

/FB3/ SINUMERIK 840D/840Di/810D (08.04 Edition)
Description of Functions **Special Functions (Part 3)**
(the various sections are listed below)
Order No.: 6FC5297-7AC80-0BP1
F2 3-Axis to 5-Axis Transformation
G1 Gantry Axes
G3 Clock Times
K6 Contour Tunnel Monitoring
M3 Coupled Motion and Leading Value Coupling
S8 Constant Workpiece Speed for Centerless Grinding
T3 Tangential Control
TE0 Installation and Activation of Compile Cycles
TE1 Clearance Control
TE2 Analog Axes
TE3 Master-Slave for Drives
TE4 Transformation Package Handling
TE5 Setpoint Exchange
TE6 MCS Coupling
TE7 Retrace Support
TE8 Pulse-Independent Path-Synchronized Switching Signal Output
V2 Preprocessing
W5 3D Tool Radius Compensation

© Siemens AG, 2004. All rights reserved

/FBA/ SIMODRIVE 611D/SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **Drive Functions**
(the various sections are listed below)
Order No.: 6SN1197-0AA80-1BP1
DB1 Operating Messages/Alarm Reactions
DD1 Diagnostic Functions
DD2 Speed Control Loop
DE1 Extended Drive Functions
DF1 Enable Commands
DG1 Encoder Parameterization
DL1 Linear Motor MD
DM1 Calculating Motor/Power Section Parameters and Controller Data
DS1 Current Control Loop
DÜ1 Monitors/Limitations

/FBAN/ SINUMERIK 840D/SIMODRIVE 611 DIGITAL (02.00 Edition)
Description of Functions **ANA MODULE**
Order No.: 6SN1197-0AB80-0BP0

/FBD/ SINUMERIK 840D (07.99 Edition)
Description of Functions **Digitizing**
Order No.: 6FC5297-4AC50-0BP0
DI1 Start-Up
DI2 Scan with Tactile Sensor (scancad scan)
DI3 Scan with Laser (scancad laser)
DI4 Milling Program Generation (scancad mill)

/FBDM/ SINUMERIK 840D/840Di/810D (09.03 Edition)
Description of Functions DNC NC Program Management
Order No.: 6FC5297-1AE81-0BP0

/FBDN/ SINUMERIK 840D/840Di/810D (03.03 Edition)
Motion Control Information System (MCIS)
Description of Functions **DNC NC Program Management**
Order No.: 6FC5297-1AE80-0BP0
DN1 DNC Plant / DNC Cell
DN2 DNC IFC SINUMERIK, NC Data Transfer via Network

/FBFA/ SINUMERIK 840D/840Di/810D (11.02 Edition)
Description of Functions **ISO Dialects for SINUMERIK**
Order No.: 6FC5297-6AE10-0BP3

/FBFE/ SINUMERIK 840D/810D (04.03 Edition)
Description of Functions **Remote Diagnosis**
Order No.: 6FC5297-0AF00-0BP2
FE1 Remote Diagnosis ReachOut
FE3 Remote Diagnosis pcAnywhere

/FBH/ SINUMERIK 840D/840Di/810D
HMI Configuring Package (02.04 Edition)
Order No.: (supplied with the software)
Part 1 User's Guide
Part 2 Description of Functions

/FBH1/ SINUMERIK 840D/840Di/810D
HMI Configuring Package (03.03 Edition)
ProTool/Pro Option SINUMERIK
Order No.: (supplied with the software)

© Siemens AG, 2004. All rights reserved

/FBHL/ SINUMERIK 840D/SIMODRIVE 611 digital (10.03 Edition)
Description of Functions **HLA Module**
Order No.: 6SN1197-0AB60-0BP3

/FBIC/ SINUMERIK 840D/840Di/810D (06.03 Edition)
Motion Control Information System (MCIS)
Description of Functions **TDI Ident Connection**
Order No.: 6FC5297-1AE60-0BP0

/FBMA/ SINUMERIK 840D/810D (08.02 Edition)
Description of Functions **ManualTurn**
Order No.: 6FC5297-6AD50-0BP0

/FBO/ SINUMERIK 840D/810D (09.01 Edition)
Description of Functions Configuring **OP 030 Operator Interface**
(the various sections are listed below)
Order No.: 6FC5297-6AC40-0BP0
BA Operator's Guide
EU Development Environment (Configuring Package)
PSE Introduction to Configuring of Operator Interface
IK Screen Kit: Software Update and Configuration

/FBP/ SINUMERIK 840D (03.96 Edition)
Description of Functions **C-PLC Programming**
Order No.: 6FC5297-3AB60-0BP0

/FBR/ SINUMERIK 840D/810D (01.04 Edition)
IT Solutions
Description of Functions **RPC SINUMERIK Computer Link**
Order No.: 6FC5297-6AD61-0BP0
NFL Host Computer Interface
NPL PLC/NCK Interface

/FBSI/ SINUMERIK 840D/SIMODRIVE (11.03 Edition)
Description of Functions SINUMERIK **Safety Integrated**
Order No.: 6FC5297-6AB80-0BP2

/FBSP SINUMERIK 840D/840Di/810D (11.03 Edition)
Description of Functions **ShopMill**
Order No.: 6FC5297-6AD80-0BP2

/FBST/ SIMATIC (01.01 Edition)
Description of Functions **FM STEPDRIVE/SIMOSTEP**
Order No.: 6SN1197-0AA70-0YP4

/FBSY/ SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **Synchronized Actions**
Order No.: 6FC5297-7AD40-0BP2

/FBT/ SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **ShopTurn**
Order No.: 6FC5297-6AD70-0BP2

/FBTC/ SINUMERIK 840D/810D (01.02 Edition)
IT Solutions
Description of Functions **Tool Data Communication SinTDC**
Order No.: 6FC5297-5AF30-0BP0

© Siemens AG, 2004. All rights reserved

/FBTD/ SINUMERIK 840D/810D (02.01 Edition)
IT Solutions
Description of Functions **Tool Information System (SinTDI)**
with Online Help
Order No.: 6FC5297-6AE00-0BP0

/FBTP/ SINUMERIK 840D/840Di/810D (01.03 Edition)
Motion Control Information System (MCIS)
Description of Functions **TPM Total Productive Maintenance**
Order No.: Document is supplied with the software

/FBU/ SIMODRIVE 611 universal/universal E (10.04 Edition)
Description of Functions **Closed-Loop Control Component for Speed Control and Positioning**
Order No.: 6SN1197-0AB20-1BP1

/FBU2/ SIMODRIVE 611 **universal** (04.02 Edition)
Installation Guide
(enclosed with SIMODRIVE 611 universal)

/FBW/ SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **Tool Management**
Order No.: 6FC5297-6AC60-0BP1

/HBA/ SINUMERIK 840D/840Di/810D (03.02 Edition)
Manual **@Event**
Order No.: 6AU1900-0CL20-0BA0

/HBI/ SINUMERIK 840Di (03.04 Edition)
Manual **SINUMERIK 840Di**
Order No.: 6FC5297-7AE60-0BP0

/INC/ SINUMERIK 840D/840Di/810D (06.03 Edition)
System Description **Commissioning Tool SINUMERIK SinuCOM NC**
Order No.: (an integral part of the online help for the start-up tool)

/PJE/ SINUMERIK 840D/810D (08.01 Edition)
Description of Functions **Configuring Package HMI Embedded**
Software Update, Configuration, Installation
Order No.: 6FC5297-6EA10-0BP0

(the document PS Configuring Syntax is supplied with the software and available as a pdf file)

/POS1/ SIMODRIVE **POSMO A** (08.04 Edition)
User Guide's **Distributed Positioning Motor on PROFIBUS DP**
Order No.: 6SN2197-0AA00-0BP7

/POS2/ SIMODRIVE **POSMO A** (05.03 Edition)
Installation Guide (enclosed with POSMO A)

/POS3/ SIMODRIVE POSMO SI/CD/CA (10.04 Edition)
User Guide's **Distributed Servo Drive Systems**
Order No.: 6SN2197-0AA20-0BP7

/POS4/ SIMODRIVE **POSMO SI** (08.03 Edition)
Installation Guide (enclosed with POSMO SI)

© Siemens AG, 2004. All rights reserved

/POS5/ SIMODRIVE **POSMO CD/CA** (08.03 Edition)
Installation Guide (enclosed with POSMO CD/CA)

/S7H/ SIMATIC S7-300 (2002 Edition)
Installation Manual **Technological Functions**
Order No.: 6ES7398-8AA03-8BA0
- Reference Manual: CPU Data (HW Description)
- Reference Manual: Module Data

/S7HT/ SIMATIC S7-300 (03.97 Edition)
Manual **STEP 7, Fundamentals, V. 3.1**
Order No.: 6ES7810-4CA02-8BA0

/S7HR/ SIMATIC S7-300 (03.97 Edition)
Manual **STEP 7, Reference Manuals, V. 3.1**
Order No.: 6ES7810-4CA02-8BR0

/S7S/ SIMATIC S7-300 (04.02 Edition)
FM 353 Positioning Module **for Stepper Drive**
Order together with configuring package

/S7L/ SIMATIC S7-300 (04.02 Edition)
FM 354 Positioning Module **for Servo Drive**
Order together with configuring package

/S7M/ SIMATIC S7-300 (01.03 Edition)
FM 357.2 Multimodule for Servo and Stepper Drives
Order together with configuring package

/SP/ SIMODRIVE 611-A/611-D
SimoPro 3.1
Program for Configuring Machine-Tool Drives
Order number: 6SC6111-6PC00-0BA❑
Order from: WK Fürth

© Siemens AG, 2004. All rights reserved

d) Installation and Start-up

/BS/ SIMODRIVE 611 analog (10.00 Edition)
Description **Start-Up Software for Main Spindle and Asynchronous Motor Modules Version 3.20**
Order No.: 6SN1197-0AA30-0BP1

/IAA/ SIMODRIVE 611A (10.00 Edition)
Installation and Start-Up Guide
Order No.: 6SN1197-0AA60-0BP6

/IAC/ SINUMERIK 810D (11.02 Edition)
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up software)
Order No.: 6FC5297-6AD20-0BP1

/IAD/ SINUMERIK 840D/SIMODRIVE 611D (03.04 Edition)
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up software)
Order No.: 6FC5297-7AB10-0BP0

/IAM/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Installation and Start-Up Guide **HMI/MMC**
Order No.: 6FC5297-6AE20-0BP3

- AE1 Updates/Supplements
- BE1 Expanding the Operator Interface
- HE1 Online Help
- IM2 Start-Up HMI Embedded
- IM4 Start-Up HMI Advanced
- TX1 Creating Foreign Language Texts

© Siemens AG, 2004. All rights reserved

E Commands, 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
_DLNUM		DL number for set-up and total offsets
_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[20]		Field: Output values INT
_OVR[32]		Field: Output values REAL
_PRNUM	**Pr**obe type and probe **num**ber	
_PROTFORM[6]		Log formatting
_PROTNAME[2]		Name of log file
_PROTSYM[2]		Separator in the log

© Siemens AG, 2004. All rights reserved

_PROTVAL[13]		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[9]		Setpoint values for measuring rectangle
_SI[3]	**S**ystem **i**nformation	System information
_SPEED[4]		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	
_TENV		Name of tool environment
_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	
_TNVL		Limit value for distortion of triangle
_TP[3,10]		Field: Tool probe data with 6 elements each
_TPW[3,10]		3 data fields for tool probe, workpiece-related
_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 **me**a**s**uring velocity	
_WP[3,11]		Field: Workpiece probe data with 9 elements each

© Siemens AG, 2004. All rights reserved

Notes

© Siemens AG, 2004. All rights reserved

F Index

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

© Siemens AG, 2004. All rights reserved

<table>
<tr><td>To
SIEMENS AG

A&D MC BMS
Postfach 3180
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/Dept.
Address:<hr>Zip code: City:<hr>Phone: /<hr>Fax: /</td><td>User’s Guide

Order No.: 6FC5298-7AA70-0BP1
10.04 Edition<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 (10.2004)

General Documentation

- *SINUMERIK* 840D/810D — Brochure
- *SINUMERIK* 840D/840Di/810D/ — Catalog Ordering Info. NC 60 ***)**
- *Safety Integrated* Application Manual — **Safety Integrated** Application Manual

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 — **Programming Guide**
  – Short Guide
  – Fundamentals ***)**
  – Advanced ***)**
  – Cycles
  – Measuring Cycles
  – ISO Turning/Milling
  – Lists **System Variables**
- *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/840Di/810D — Description of Functions
  – ManualTurn
  – ShopMill
  – ShopTurn

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* 611D 840D/810D — Description of Functions **Synchronized Actions**
- *SINUMERIK* 840D/840Di/810D — Description of Functions **Drive Functions *)**
- *SINUMERIK* 840D/840Di/810D — **Description of Functions**
  – Basic Machine ***)**
  – Extended Functions
  – Special Functions
- *SINUMERIK* 840D/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 — **MCIS**
  – Computer Link
  – Tool Data Inform. Syst.
  – 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
- *MOTION CONTROL SYSTEMS* — **EMC** Guidelines

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**

Electronic Documentation

- *SINUMERIK SIMODRIVE* 840D/840Di/810D 611, Motors — **DOC ON CD *)** The SINUMERIK System

*) These documents are a minimum requirement

Siemens AG
Automation and Drives
Motion Control Systems
Postfach 3180
91050 ERLANGEN
GERMANY

www.siemens.com/motioncontrol

© Siemens AG, 2004
Subject to change without prior notice
Order No.: 6FC5298-7AA70-0BP1