SINUMERIK 840/840C
SINUMERIK 850
SINUMERIK 880/880 GA2
Measuring Cycles Version 20 and higher

User's Guide 08.96 Edition

User Documentation

SINUMERIK 840/840C
SINUMERIK 850
SINUMERIK 880/880 GA2
Measuring Cycles Version 20 and higher

User's Guide

User Documentation

Valid for:

Control	*Software Version*
SINUMERIK 840	from SW 01
SINUMERIK 840C	from SW 01
SINUMERIK 850	from SW 04
SINUMERIK 880	from SW 04
SINUMERIK 880 GA2	from SW 01

08.96 Edition

SINUMERIK® documentation

Printing history

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

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

Status code in "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 a page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks
07.90	6ZB5 410-0EX02-0BA0	**A**
10.91	6ZB5 410-0EX02-0BA1	**C**
01.93	6FC5 197-0AB70-0BP0	**C**
08.96	6FC5 197-0AB70-0BP1	**C**

Siemens quality for software and training
to DIN ISO 9001, Reg. No. 2160-01

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.

This publication was produced on the Siemens 5800 Office System.

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

We have checked that the contents of this publication agree with the hardware and software described herein. The information given in this publication is reviewed at regular intervals and any corrections that might be necessary are made in the subsequent printings. Suggestions for improvement are welcome at all times.

Subject to change without prior notice.

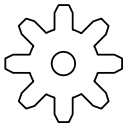
© Siemens AG 1992, 1993, 1994, 1995, 1996 All Rights Reserved

Order No. 6FC5 197-0AB70-0BP1
Printed in the Federal Republic of Germany

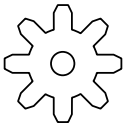
Preliminary Remarks

Technical Comments

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.



The symbol shown on the left appears in this documentation whenever the machine tool manufacturer has the possibility of influencing/modifying the functional behaviour described by changing a cycle machine data (MDC).



Since the setting up of the cycle machine data range and of the cycle setting data range is variable, the relevant data in the examples are explained on the basis of the standard settings.

This User's Guide is valid for:
SINUMERIK 850/880 as from Software Version 4,
SINUMERIK 880 GA2 as from Software Version 1,
SINUMERIK 840 as from Software Version 1,
SINUMERIK 840C as from Software Version 1,
Measuring Cycles as from Version 20!

Introduction	1
Defining Parameters	2
Measuring Cycle Auxiliary Programs	3
Measuring Cycles for Turning Machines	4
Measuring Cycles for Milling Machines and Machining Centres	5
Measuring Cycles for Turning and Milling Work	6
Measuring in JOG Mode (SINUMERIK 880/880 GA2)	7
Automatic Workpiece Measurement in JOG Mode (SINUMERIK 840 SW2)	8
Alarms	9
Lists	10
Abbreviations	11

Contents

		Page
1	**Introduction**	1–1
1.1	General	1–1
1.2	Measurement variants	1–2
1.2.1	Measurement variants for turning machines	1–2
1.2.2	Measurement variants for milling machines and machining centres	1–4
1.3	Probe	1–7
1.3.1	Multidirectional probe (3D)	1–8
1.3.2	Bidirectional probe	1–8
1.3.3	Monodirectional probe	1–8
1.4	Calibrating tool, workpiece probe in the TOA memory	1–9
1.4.1	Assignment of calibrating tool in the TOA memory for turning machines	1–9
1.4.2	Workpiece probe in the TOA memory for turning machines	1–10
1.4.3	Workpiece probe in the TOA memory for milling machines and machining centres	1–13
1.5	Measuring principle	1–14
1.5.1	Probe signal evaluation	1–14
1.5.2	In-process measurement	1–15
1.5.3	Measuring accuracy	1–17
1.5.4	Outline flowchart for workpiece measurement	1–18
1.6	Measuring strategy and correction value determination	1–20
1.6.1	Types of dimensional deviation	1–20
1.6.2	Averaging	1–21
1.7	Tolerance parameters (R33, R34, R36, R37, R40/R41)	1–23
1.7.1	Tolerance parameters for workpiece measurement	1–24
1.7.2	Tolerance parameters for tool measurement	1–25
1.8	Plane definition	1–26
1.9	Logging	1–27
1.9.1	Logging via logging module	1–27
1.9.2	Logging via CP315 with the SINUMERIK 840/880	1–28
1.9.2.1	General description	1–28
1.9.2.2	Configuring the form	1–29
1.9.2.3	Data transmission between NC and PLC	1–33
1.9.2.4	Description of transfer parameters	1–35
1.9.2.5	Error messages of logging function	1–39
1.9.2.6	Example demonstrating the logging of measurement results for the SINUMERIK 840/880	1–39
1.10	Notes to ensure smooth running of the measuring cycles	1–44
1.11	Reference points on machine and workpiece	1–45
2	**Defining Parameters**	2–1
2.1	R parameters used	2–1
2.2	R10 Offset number	2–1
2.2.1	R10 Tool offset memory number (workpiece measurement)	2–1
2.2.1.1	D number known: Relative D number	2–1
2.2.1.2	D number unknown	2–2
2.2.2	R10 ZO memory number (ZO determination)	2–2
2.3	R11 Empirical value/average value	2–2

2.4 R12 Number of calibrating elements 2–4
2.5 R13 Compensating angle position for mono-directional probe, angular offset for driven tools 2–4
2.6 R22 Probe type/probe number 2–5
2.7 R23 Measurement variant 2–5
2.7.1 L972/L982 Tool measurement (T) 2–5
2.7.2 L973 Calibrating the workpiece probe (T) 2–7
2.7.3 L974 Workpiece measurement (T) 2–7
2.7.4 L976 Calibrating the workpiece probe (M) 2–8
2.7.5 L977/L979 Workpiece measurement (M) 2–8
2.7.6 L978 Workpiece measurement (M) 2–8
2.7.7 L981 Workpiece measurement (T/M) 2–8
2.8 R25 Variable measuring speed 2–9
2.9 R27 Multiple measurement at same location 2–9
2.10 R28 Multiplication factor for measurement path 2a 2–9
2.11 R29 Weighting factor k for averaging 2–9
2.12 R30 Number of measuring axis 2–10
2.13 R33...R37 Tolerance parameters 2–11

3 Measuring Cycle Auxiliary Programs 3–1

3.1 Internal measuring cycle auxiliary programs 3–2
3.1.1 L931 Auxiliary cycle for tool or workpiece measuring cycles 3–2
3.1.2 L932 Check MDC and transfer parameters 3–2
3.1.3 L933 Auxiliary cycle for tool or workpiece measuring cycles 3–2
3.1.4 L934 Auxiliary cycle for tool or workpiece measuring cycles 3–2
3.1.5 L935 Measurement result display selection 3–2
3.1.6 L936 Measurement abort 3–3
3.1.7 L937 Auxiliary cycle for workpiece measuring cycles 3–3
3.1.8 L938 Auxiliary cycle for tool or workpiece measuring cycles 3–3
3.1.9 L939 Auxiliary cycle for tool measuring cycles 3–3
3.2 Internal/external measuring cycle auxiliary programs 3–3
3.2.1 L960 Transfer of ZO data blocks (external) 3–3
3.2.2 L961 Additive input of empirical values (external) 3–4
3.2.3 L962 Erase program EV/AV (external) 3–4
3.2.4 L963 Auxiliary cycle for workpiece measuring cycle (external) 3–5
3.2.5 L964 Auxiliary cycle for workpiece measuring cycles (internal) 3–5
3.2.6 L965 Determination of measuring plane (external) 3–5
3.2.7 L966 Auxiliary program for operator guidance macro (OGM) 3–6
3.2.8 L967 Presetting of transfer parameters (internal) 3–6
3.2.9 L969 Auxiliary cycle for coordinate rotation (internal) 3–6
3.2.10 L970 Prepositioning cycle (external) 3–6
3.2.11 L971 Auxiliary cycle for tool measuring cycles (internal) 3–9
3.2.12 L980 Auxiliary cycle for tool measuring cycle L981 (internal) 3–9
3.2.13 L988 Auxiliary cycle for workpiece measuring cycles (internal/external) . 3–9
3.2.14 L989 Auxiliary cycle for workpiece measuring cycle L979 3–9

4 Measuring Cycles for Turning Machines 4–1

4.1 L972/L982 Tool measurement 4–2
4.1.1 L972/L982 Calibrating the tool probe 4–10
4.1.2 L972/L982 Measure tool 4–13
4.1.3 L972/L982 Automatic tool measurement 4–17

4.2	L973 Calibrating the workpiece probe	4–20
4.2.1	L973 Calibrating on any surface	4–21
4.3	L974 Workpiece measurement	4–23
4.3.1	L974 1-point measurement ZO determination	4–24
4.3.2	L974 1-point measurement	4–26
4.3.3	L974 1-point measurement with reversal	4–30
4.3.4	L974 2-point measurement on diameter	4–34
4.3.5	L974 Multi-point measurement on circumference	4–38
4.3.6	L974 Multi-point measurement on cylinder	4–41
4.4	Examples of application for workpiece measurement (L973, L974)	4–44
4.5	Parameter recommendations for L973, L974	4–45
5	**Measuring Cycles for Milling Machines and Machining Centres**	5–1
5.1	L976 Calibrating the workpiece probe	5–2
5.1.1	L976 Calibrating the workpiece probe in reference hole (plane)	5–3
5.1.2	L976 Calibrating workpiece probe in reference hole (applicate)	5–6
5.1.3	L976 Calibrating workpiece probe in any hole (plane)	5–9
5.1.4	L976 Calibrating workpiece probe on any surface (applicate)	5–12
5.2	L977 Workpiece measurement hole/shaft/slots/ZO determination (paraxial)	5–14
5.2.1	L977 Measure hole (paraxial)	5–15
5.2.2	L977 Measure shaft (paraxial)	5–19
5.2.3	L977 Measure slot (paraxial)	5–23
5.2.4	L977 Measure web (paraxial)	5–27
5.2.5	L977 ZO determination in hole (paraxial)	5–31
5.2.6	L977 ZO determination on shaft	5–34
5.2.7	L977 ZO determination in slot (paraxial)	5–37
5.2.8	L977 ZO determination on a web (paraxial)	5–40
5.3	L979 Workpiece measurement hole/shaft/slot/web/ZO determination (at random angles)	5–43
5.3.1	L979 Measure hole (at random angles)	5–46
5.3.2	L979 Measure shaft (at random angles)	5–49
5.3.3	L979 Measure slot (at random angles)	5–52
5.3.4	L979 Measure web (at random angles)	5–55
5.3.5	L979 ZO determination in hole (at random angles)	5–58
5.3.6	L979 ZO determination at shaft (at random angles)	5–60
5.3.7	L979 ZO determination in slot (at random angles)	5–62
5.3.8	L979 ZO determination at web (at random angles)	5–64
5.4	L978 Workpiece measurement surface/angle /ZO determination (with and without differential measurement)	5–66
5.4.1	L978 ZO determination on surface (1-point measuring cycle)	5–67
5.4.2	L978 1-point measurement	5–70
5.4.3	L978 Paraxial multipoint measurement	5–73
5.4.4	L978 Angular measurement (ZO determination)	5–76

6 Measuring Cycles for Turning and Milling Work 6–1

6.1 L981 Searching for a hole/slot 6–1

7 Measuring in JOG Mode (SINUMERIK 880/880 GA2) 7–1

7.1 Tool measurement 7–1
7.1.1 Operating and function sequence 7–2
7.2 Workpiece measurement 7–9
7.2.1 Operating and function sequence 7–9

8 Automatic Workpiece Measurement in JOG Mode (SINUMERIK 840 SW2) 8–1

8.1 General 8–1
8.2 Setting the parameters for workpiece measurement and calibration 8–4
8.3 Calibrating probe in any hole 8–5
8.4 Workpiece measurement 8–7
8.4.1 Measuring a hole 8–9
8.4.2 Measuring a shaft 8–11
8.4.3 Measuring a slot 8–14
8.4.4 Measuring a web 8–17
8.4.5 Measuring a surface 8–20
8.4.6 Measuring an angle 8–23
8.4.7 Measuring a reference point on the workpiece 8–26

9 Alarms 9–1

9.1 Measuring cycle alarms 9–1
9.2 Measuring cycle alarms 9–4
9.3 Alarm description 9–7

10 Lists 10–1

10.1 Overview of transfer parameters 10–1
10.1.1 Compatibility list, Version 10 to Version 20 10–20
10.2 Result displays 10–21
10.3 Result parameters 10–23
10.4 Setting data for measuring cycles 10–32
10.4.1 Cycle setting data memory (SDC) 10–32
10.4.1.1 Channel-oriented values 10–32
10.4.1.2 Channel-oriented bits 10–32
10.4.2 SDC Channel-oriented values 10–33
10.4.3 Channel-oriented bits 10–33

11 Abbreviations 11–1

1 Introduction

1.1 General

The ever increasing rationalization and automation of manufacture makes additional demands on numerically controlled machine tools. Measuring on the machine is one of the new tasks to be coped with.

In order to be able:

- to use touch trigger probes,
- to use the path measuring systems of the machine for measuring actual values and
- to do without additional external electronic controls for measured value processing,

adapted functions are required in the control (CNC).

Before dealing in more detail with the measuring cycles created for measurement and measured value processing, some practical applications for measuring on the machine are given as an introduction to the subject.

The general purpose to be achieved by measurement on CNC turning machines and CNC milling machines and machining centres is to detect at an early stage impermissible dimensional deviations of the workpiece to enable suitable compensation measures to be initiated automatically.

Dimensional deviations are due to different causes:

- Tool wear and tool clamping
- Heat effects on the CNC machine, i.e. thermal expansion of the ball screw, machine bed or spindle head.

In addition to checking the workpiece for dimensional accuracy, measuring on a CNC machine offers further applications:

- Detection of tool breakage
- Measuring the tool geometry
- Determination of workpiece clamping tolerances
- Compensation for factors influencing the machining process (e.g. force and temperature)
- Identifying a pallet
- Determination of the centre point of a hole.

For this wide range of measuring tasks, specially tailored programmable measuring cycles are required putting the machine tool manufacturer and user in a position to achieve optimum solutions to the measuring tasks to be performed on his machine tool.

© Siemens AG 1990 All Rights Reserved

When measuring on the CNC machine, a distinction is made between two types of application, i.e. **tool measurement** and **workpiece measurement:**

Tool measurement

For tool measurement, the changed tool (seated in the turret or in the tool spindle) is moved onto the probe. The probe is either stationary or is swivelled or slid into the machine work area by a mechanical device. The automatically determined tool geometry is subsequently entered into a tool compensation memory provided for this purpose.

Workpiece measurement

For workpiece measurement, a probe is moved on the clamped workpiece in the same way as a tool. The probe is fitted in the turret or in the tool slide, depending on the design of the machine. The high flexibility of the measuring cycles enables nearly all measuring tasks capable of being performed on a turning or milling machine to be accomplished sucessfully.

On the following pages, different measurement variants are illustrated selected according to the criteria of flexibility, accuracy and duration of measurement.

1.2 Measurement variants

1.2.1 Measurement variants for turning machines

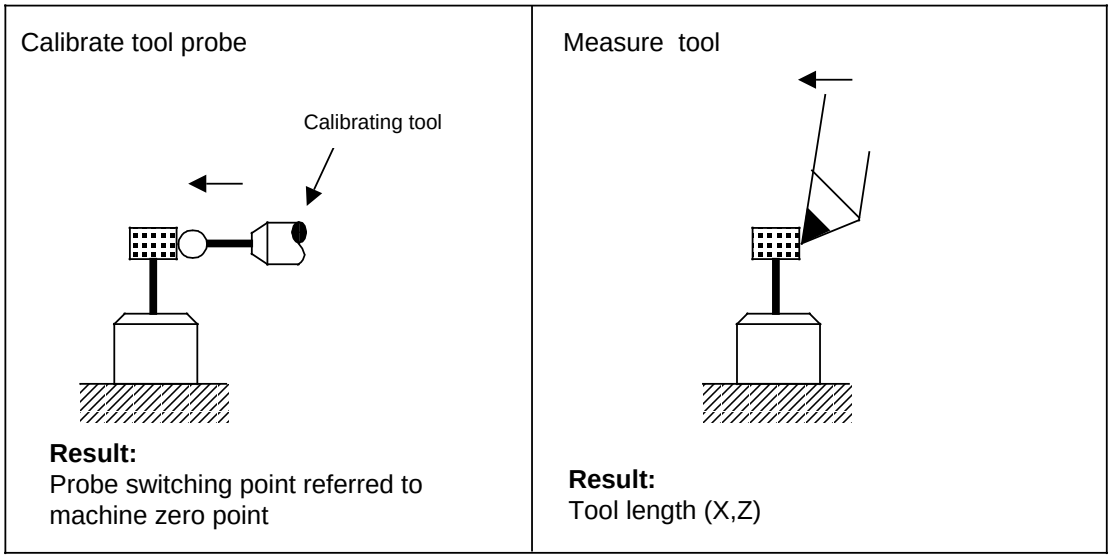
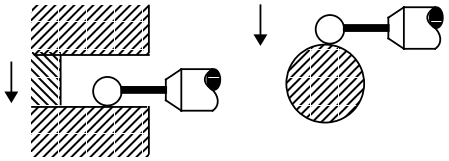


Fig. 1.1 Tool measurement

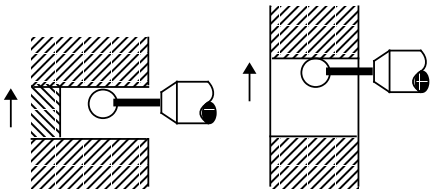
© Siemens AG 1990 All Rights Reserved

1-point measurement outside



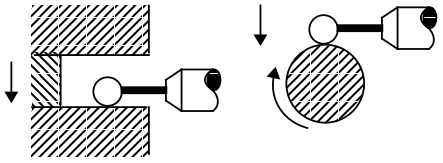
Result:
Actual dimension (diameter, length)
deviation,
tool offset,
zero offset (G58)

1-point measurement inside



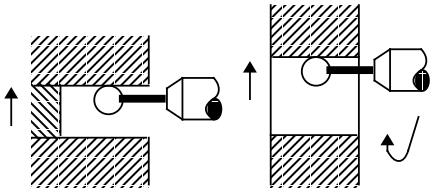
Result:
Actual dimension (diameter, length)
deviation,
tool offset,
zero offset (G58)

1-point measurement outside
with 180° spindle reversal



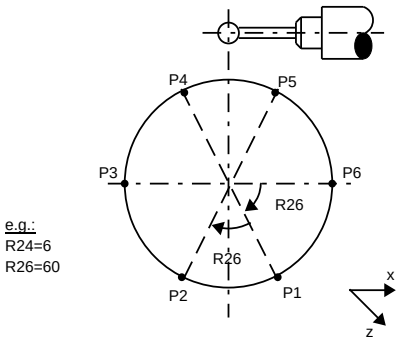
Result:
Actual dimension (diameter, length)
deviation,
tool offset

1-point measurement inside
with 180° spindle reversal



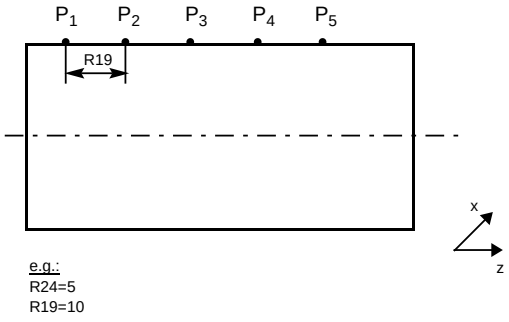
Result:
Actual dimension (diameter, length)
deviation,
tool offset

Multi-point measurement on
circumference



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

Multi-point measurement on
cylinder



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

Fig. 1.2 Workpiece measurement with calibration

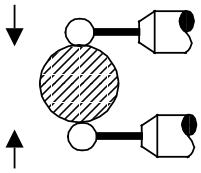
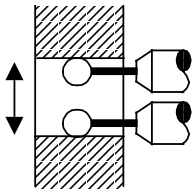
2-point measurement on diameter outside	2-point measurement on diameter inside
	
Result: Actual dimension (diameter) deviation, tool offset	**Result:** Actual dimension (diameter) deviation, tool offset

Fig. 1.3 Workpiece measurement without calibration

1.2.2 Measurement variants for milling machines and machining centres

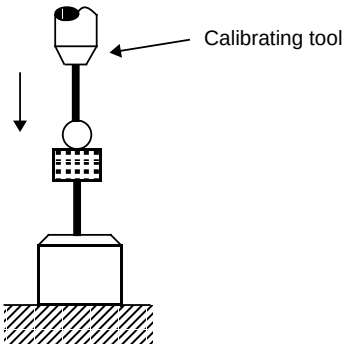
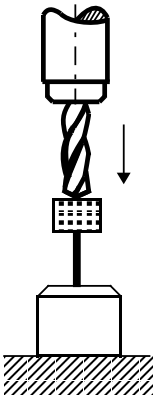
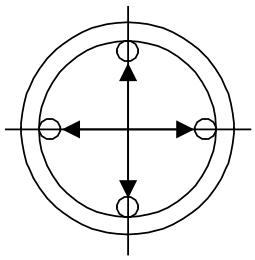
Calibrate tool probe	Measure tool
	
Result: Probe switching point referred to machine zero point	**Result:** Tool length, tool diameter

Fig. 1.4 Tool measurement

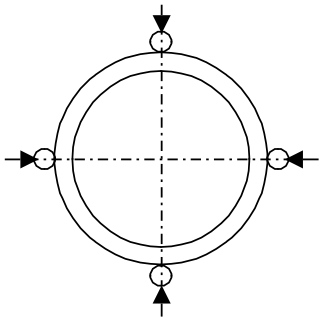
© Siemens AG 1990 All Rights Reserved

Bore:



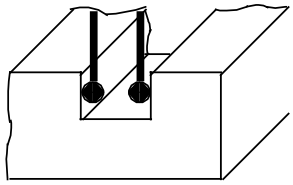
Result:
Actual dimension (diameter),
deviation, centre point,
tool offset,
zero offset (G58)

Shaft:



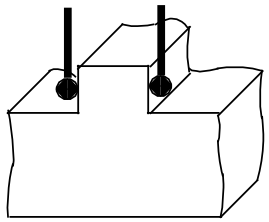
Result:
Actual dimension (diameter),
deviation, centre point,
tool offset,
zero offset (G58)

Slot:



Result:
Actual dimension (slot width),
deviation, slot centre,
tool offset,
zero offset (G58)

Web:

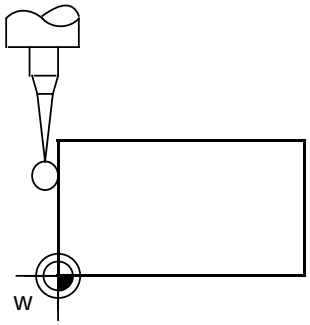


Result:
Actual dimension (web width),
deviation, web centre,
tool offset,
zero offset (G58)

Fig. 1.5 Workpiece measurement

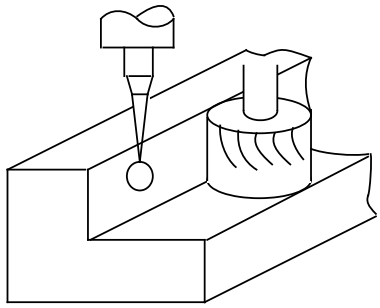
© Siemens AG 1990 All Rights Reserved

Blank gauging



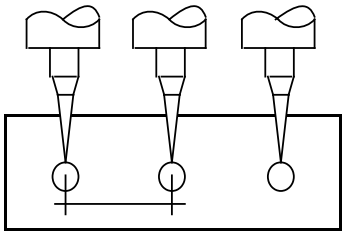
Result:
Position, deviation
zero offset (G58)

1-point measurement



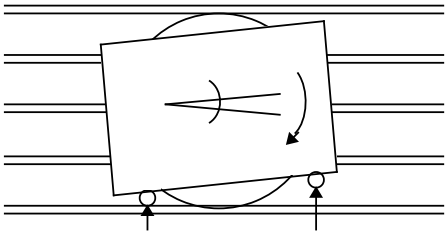
Result:
Actual dimension, deviation,
tool offset

Multi point measurement
paraxial



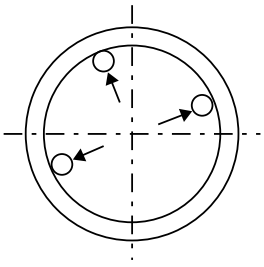
Result:
Actual dimension, deviation,
tool offset

Angular measurement



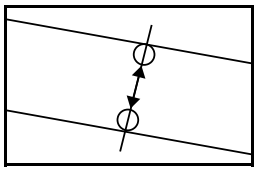
Result:
Actual dimension (angle), deviation,
zero offset (G58)

Measurement at random angles
with 3-point measurement:
hole, shaft, circular segment



Result:
Actual dimension (diameter),
deviation, centre point,
tool offset,
zero offset (G58)

Measurement at random angles
with 2-point measurement:
slot, web



Result:
Actual dimension (slot width, web width),
deviation, slot centre, web centre
tool offset,
zero offset (G58)

Fig. 1.6 Workpiece measurement

© Siemens AG 1990 All Rights Reserved

1.3 Probe

For determining tool and workpiece dimensions a touch trigger probe is required which supplies a constant signal (no pulse) when deflected.

The probe must switch with almost no bounce. This is generally achieved by adjusting the probe mechanically. In addition, ”Software debouncing” is carried out in the NC.

Various types of probe of different make are available on the market. Probes are therefore classified in three groups according to the number of directions in which the probe can be deflected (see figure below).

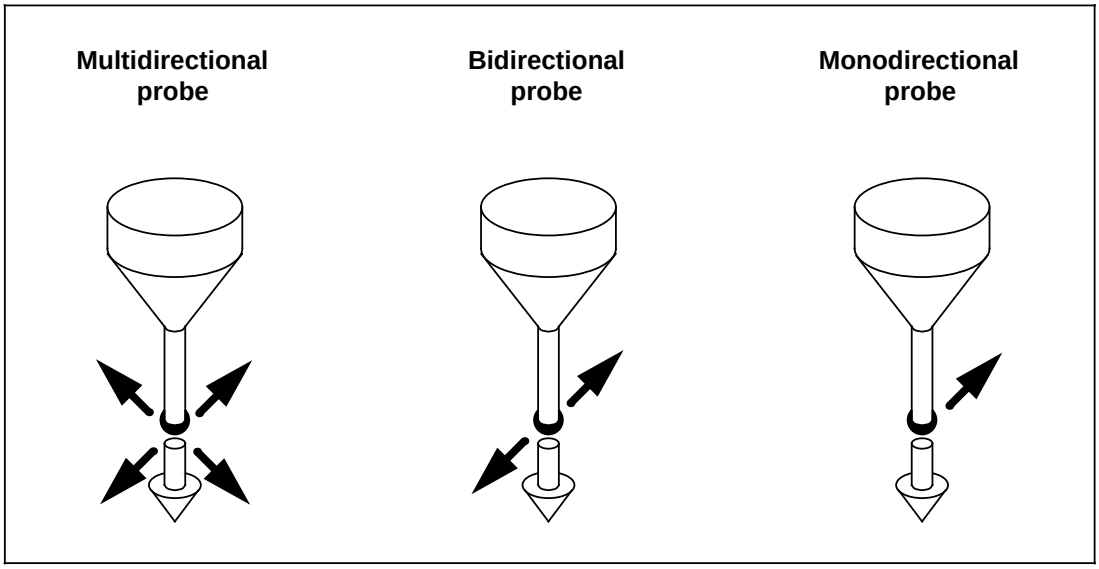


Fig. 1.7 Types of probe

	Turning machine		**Milling machines and machining centres**	
	Tool measurement	Workpiece measurement	Tool measurement	Workpiece measurement
Multidirectional probe	yes	yes	yes	yes
Bidirectional probe		yes		yes
Monodirectional probe				yes

While on turning machines a bidirectional probe can be used, milling machines and machining centres also permit the use of a monodirectional probe for workpiece measurement. In the cycles for milling machines and machining centres, the probe type is to be specified by an R parameter.

© Siemens AG 1990 All Rights Reserved

1.3.1 Multidirectional probe (3D)

With this type of probe, tool and workpiece measuring cycles can be used without restriction.

1.3.2 Bidirectional probe

This type of probe can be used on turning machines for workpiece measurement. When performing workpiece measurement on milling machines and machining centres, this probe type is treated as a monodirectional probe.

1.3.3 Monodirectional probe

This type cannot be used on turning machines. It can be used for workpiece measurement on milling machines and machining centres with some few restrictions (more information is given in the respective cycles).

It must be possible to position the spindle with the NC function ”M19” and to transmit the probe switching signal through 380° to the receiving station (on the machine column).

The probe must be aligned mechanically in the spindle in such a way that measurements can be performed in the following directions with the spindle positioned to 0°.

	Mesurement with 0 degr. spindle position
X-Y plane (G17)	Positive X direction
Z-X plane (G18)	Positive Z direction
Y-Z plane (G19)	Positive Y direction

Measurement takes longer with a monodirectional probe, because the spindle must be positioned in the cycle several times with M19.

© Siemens AG 1990 All Rights Reserved

1.4 Calibrating tool, workpiece probe in the TOA memory

1.4.1 Assignment of calibrating tool in the TOA memory for turning machines

On turning machines, the calibrating tool is treated same as a tool type 3 and, therefore, must be input in the TOA memory as such.

Dn		
P0		**Tool number**
P1	3	**Tool type**
P2	L1	**L1 Geometry**
P3	L2	**L2 Geometry**
P4	r	**Diameter/radius**
P5	0	**L1 Wear**
P6	0	**L2 Wear**
P7	0	**Diameter/radius**
P8	0	**Base (add. TO)**
P9	0	**Base (add. TO)**

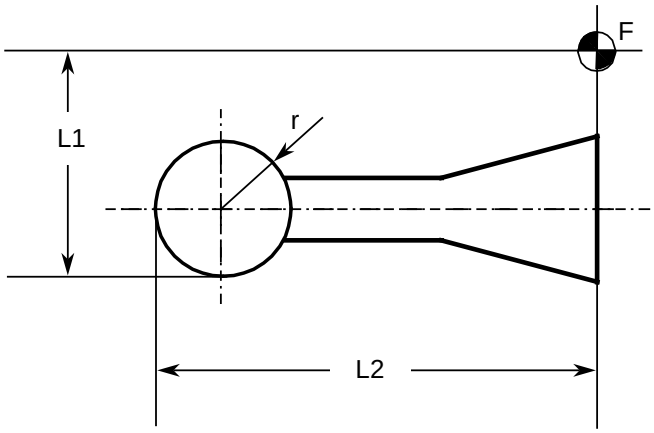


Fig. 1.8 Calibrating tool in the TOA memory for turning machines

© Siemens AG 1990 All Rights Reserved

1.4.2 Workpiece probe in the TOA memory for turning machines

On turning machines, measuring cycles are classified in the following types according to their respective locations:

Dn	Type 5	
P0		Tool number
P1	5	Tool type
P2	L1	L1 Geometry
P3	L2	L2 Geometry
P4	r	Diameter/radius
P5	0	L1 Wear
P6	0	L2 Wear
P7	0	Diameter/radius
P8	0	Base (add. TO)
P9	0	Base (add. TO)

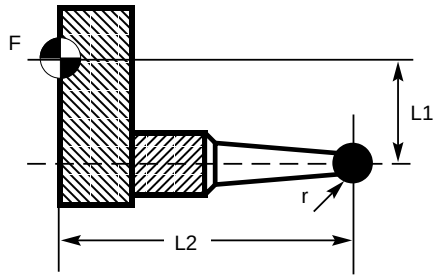


Fig. 1.9 Workpiece probe type 5 in the TOA memory for turning machines

Dn	Type 6 (8)	
P0		Tool number
P1	6	Tool type
P2	L1	L1 Geometry
P3	L2	L2 Geometry
P4	r	Diameter/radius
P5	0	L1 Wear
P6	0	L2 Wear
P7	0	Diameter/radius
P8	0	Base (add. TO)
P9	0	Base (add. TO)

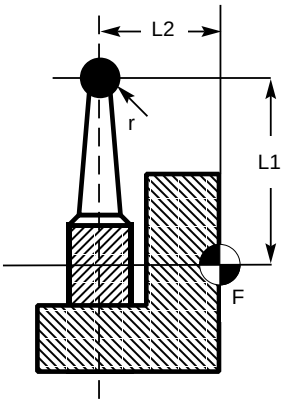


Fig. 1.10 Workpiece probe type 6 (8) in the TOA memory for turning machines
(type in parentheses = in front of turning centre)

© Siemens AG 1990 All Rights Reserved

Dn	Type 7	
P0		Tool Number
P1	7	Tool type
P2	L1	L1 Geometry
P3	L2	L2 Geometry
P4	r	Diameter/radius
P5	0	L1 Wear
P6	0	L2 Wear
P7	0	Diameter/radius
P8	0	Base (add. TO)
P9	0	Base (add. TO)

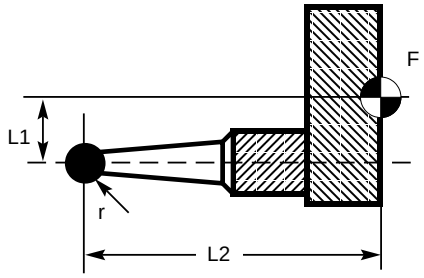


Fig. 1.11 Workpiece probe type 7 in the TOA memory for turning machines

Dn	Type 8 (6)	
P0		Tool number
P1	8	Tool type
P2	L1	L1 Geometry
P3	L2	L2 Geometry
P4	r	Diameter/radius
P5	0	L1 Wear
P6	0	L2 Wear
P7	0	Diameter/radius
P8	0	Base (add. TO)
P9	0	Base (add. TO)

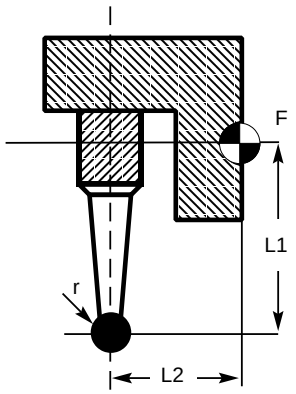
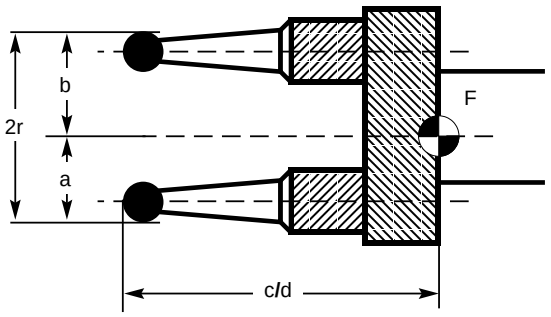


Fig. 1.12 Workpiece probe type 8 (6) in the TOA memory for turning machines (type in parentheses = in front of turning centre)

© Siemens AG 1990 All Rights Reserved

If double probes are used (Version 30 and higher) type 3 must be entered. The geometry data must be entered in two successive TOA memories, the data in D_n for measuring in the negative direction and the data in D_{n+1} for measuring in the positive direction.

	Dn	Dn+1	**Type 3 axial**
P0	P0		Tool number
P1	3	3	Tool type
P2	a	b	L1 Geometry
P3	c	d	L2 Geometry
P4	r	r	Diameter/radius
P5	0	0	L1 Wear
P6	0	0	L2 Wear
P7	0	0	Diameter/radius
P8	0	0	Base (add. TO)
P9	0	0	Base (add. TO)



	Dn	Dn+1	**Type 3 radial**
P0			Tool number
P1	3	3	Tool type
P2	e	f	L1 Geometry
P3	g	h	L2 Geometry
P4	r	r	Diameter/radius
P5	0	0	L1 Wear
P6	0	0	L2 Wear
P7	0	0	Diameter/radius
P8	0	0	Base (add. TO)
P9	0	0	Base (add. TO)

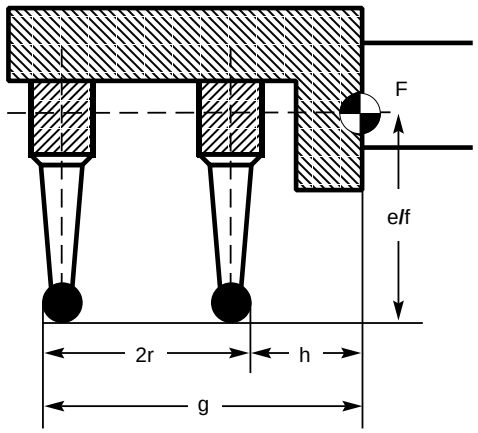


Fig. 1.13 Assignment of workpiece probes in the TOA memory for turning machines

© Siemens AG 1990 All Rights Reserved

1.4.3 Workpiece probe in the TO memory for milling machines and machining centres

On milling machines and machining centres, the probe is treated as a tool type 30 and, therefore, must be input in the TOA memory as such.

As from Version 3.5 it is now possible to define the length L1 to the end of the probe in cycle machine data 7004, bit 5. This setting takes effect in cycles L976/L978 calibration and measurement of the applicate (drilling axis).

Dn		
P0		Tool number
P1	30	Tool type
P2	L1	L1 Geometry
P3	L2	L2 Geometry
P4	r	Diameter/radius
P5	0	L1 Wear
P6	0	L2 Wear
P7	0	Diameter/radius
P8	0	Base (add. TO)
P9	0	Base (add. TO)

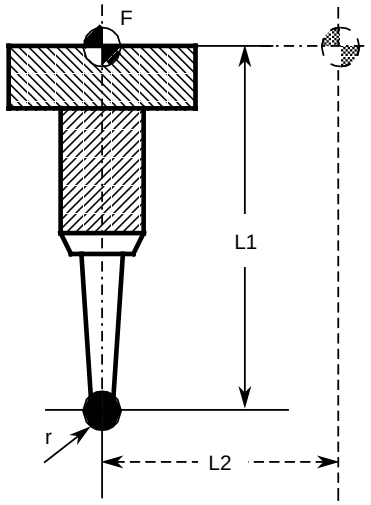


Fig 1.14 Workpiece probe in the TOA memory for milling machines and machining centres

© Siemens AG 1990 All Rights Reserved

1.5 Measuring principle

1.5.1 Probe signal evaluation

Two inputs for connecting touch trigger probes are provided on the SINUMERIK interface module.

If a measuring point is to be approached, a travel 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 the position setpoint value. The actual axis value at the time of the switching position is measured and the drive is stopped as soon as the probe makes contact. The remaining ”distance to go” is deleted.

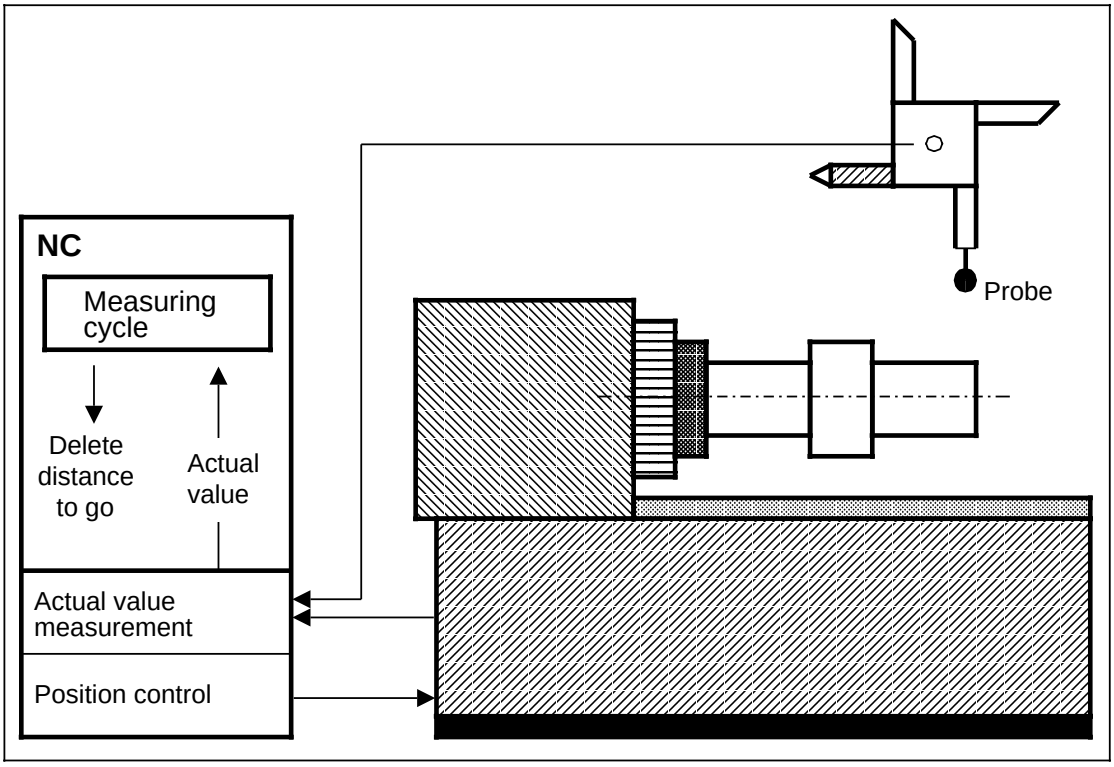


Fig. 1.15 Probe signal evaluation

© Siemens AG 1990 All Rights Reserved

1.5.2 In-process measurement

IThe principle of ”in-process measurement” has been implemented in the SINUMERIK control. The advantage of this method of measurement is that the probe signal is processed directly in the NC and not transferred to the NC via the PLC as previously done.

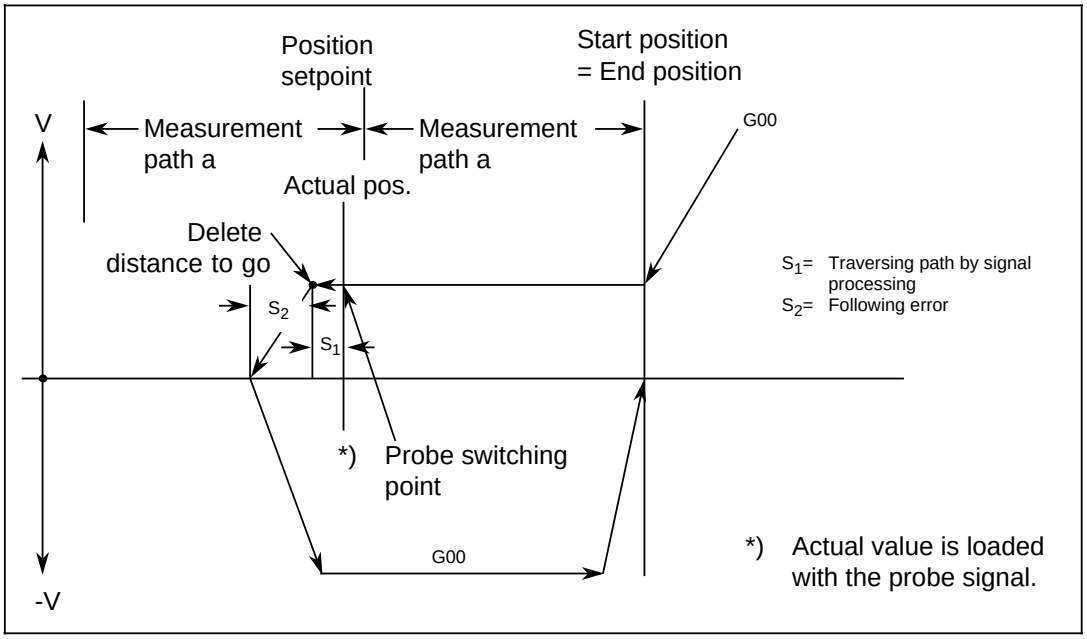


Fig. 1.16 In-process measurement

Start position/position setpoint

With the method of measurement used, a position is assigned to the cycle as set value at which the signal of the touched trigger probe is expected. Considering the fact, however, that it is very improbable for the probe to respond at precisely this point, the start position is displaced in front of the position setpoint by a settable distance a. This start position is approached by the control in rapid mode (G00). This approach is followed by a feed movement at the rate of F = 150 mm/min towards the position setpoint. The switching signal is now expected on a distance having the maximum length of 2a from the start position.

Loading the actual value/deleting distance to go

The moment the switching signal is transmitted by the probe, the current position is loaded into a register (R parameter) as actual value by in-process measurement. Following this, i.e. after a distance S_1 covered by the machine while the signal is being processed, the function ”Delete distance to go” is executed and the old start position is preset as the new set value. This end position is then approached in rapid mode.

The delay for actual value measurement is of the microsecond order. In other words, it has no influence on the attainable measuring accuracy when using comercially available probes (even in the case of increased approach speed).

© Siemens AG 1990 All Rights Reserved

Measurement path a/measuring speed

Normally, the path increment is 1 mm. It can, however, be increased or reduced when calling the measuring cycles with parameter R 28. The approach speed increases automatically from 155 mm/min to 300 mm/min when defining the value for a>1. This value has been chosen to make sure that the deceleration distance is less than the permissible probe deflection.

For measuring cycles the measuring speed can be preset via parameter R 25 as desired. However, safe deceleration within the deflection path of the probe should be ensured.

Hence, the maximum approach speed (measuring speed) depends only on

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

Example:

Deceleration path calculation

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

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

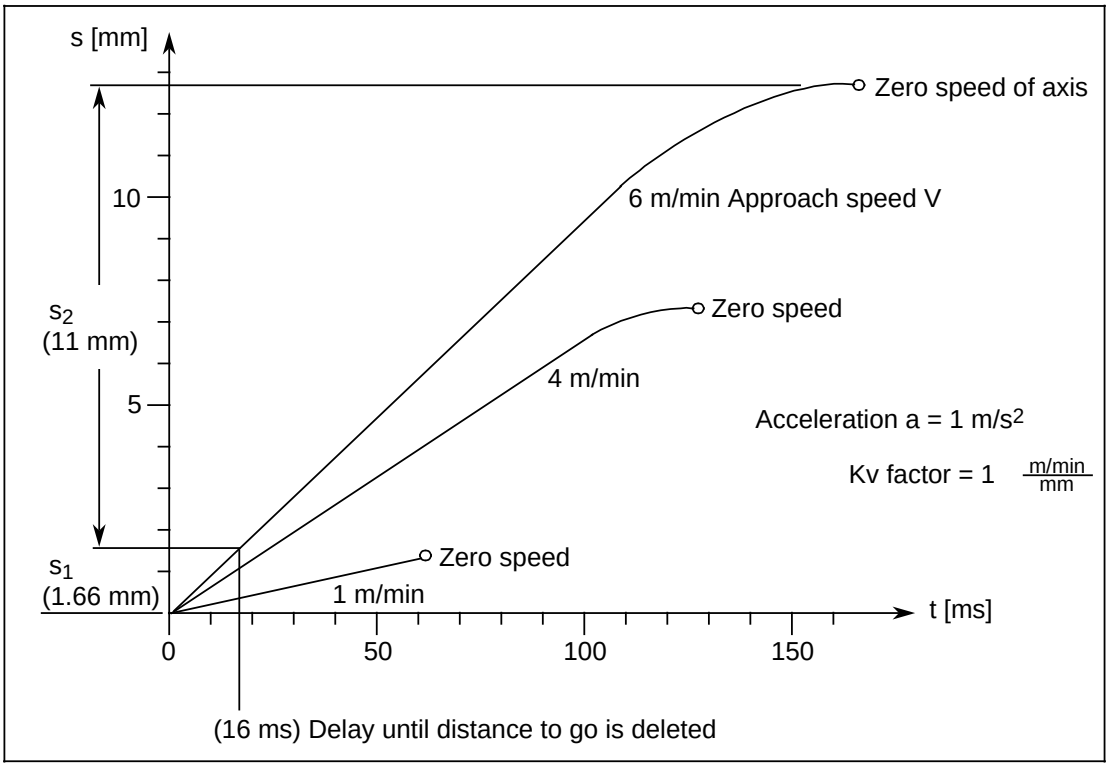


Fig. 1.17 Path/time diagram

The deflection of the probe until the axis reaches zero speed is approx. 12.6 mm in the case of an approach speed of 6 m/min and a delay of 1 m/s^2.

© Siemens AG 1990 All Rights Reserved

1.5.3 Measuring accuracy

The repeat accuracy of SINUMERIK controls for in-process measurement is ±1 µm.

The attainable measuring accuracy therefore depends on the following factors:

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

To give an example, the following series of measurements have been carried out on a turning machine:

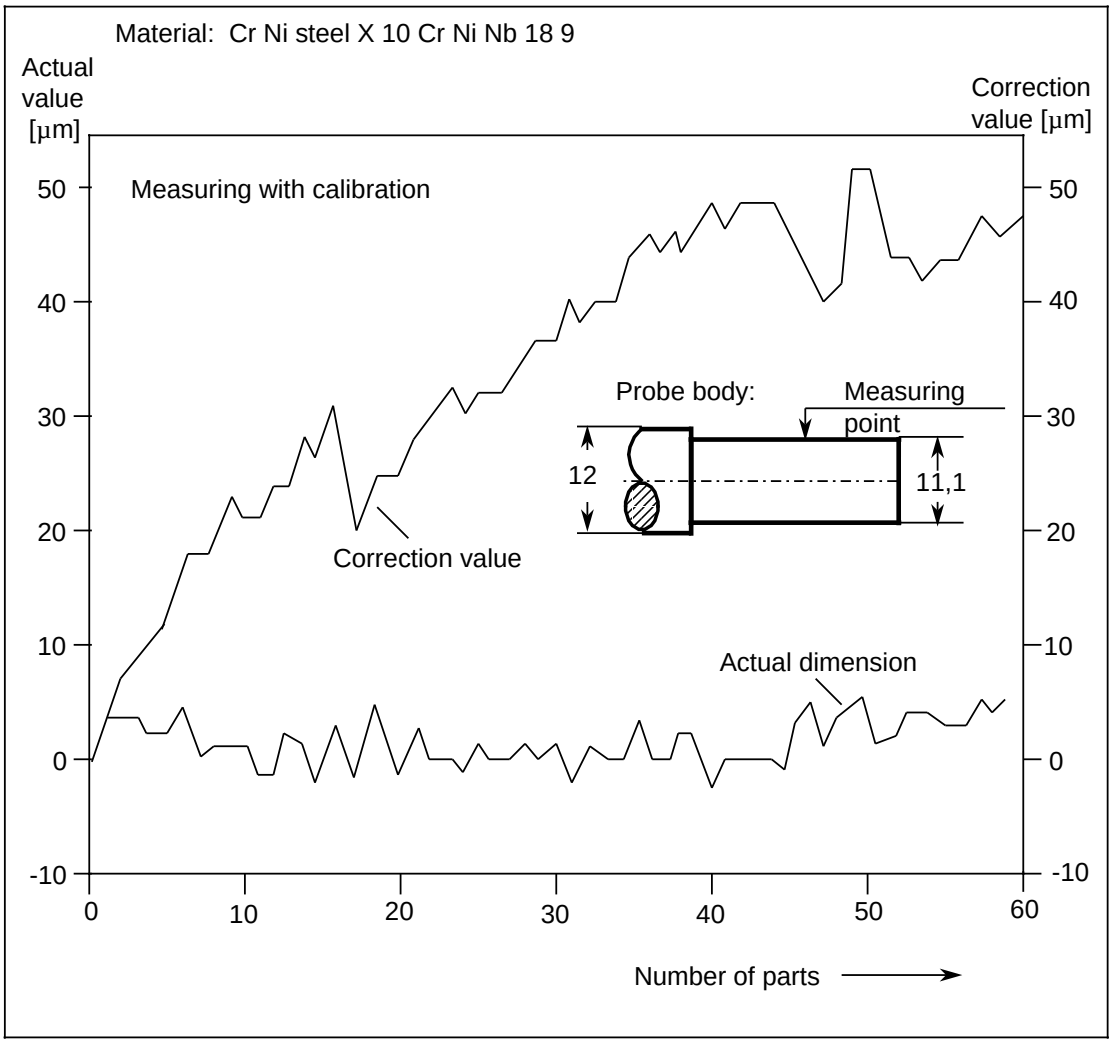
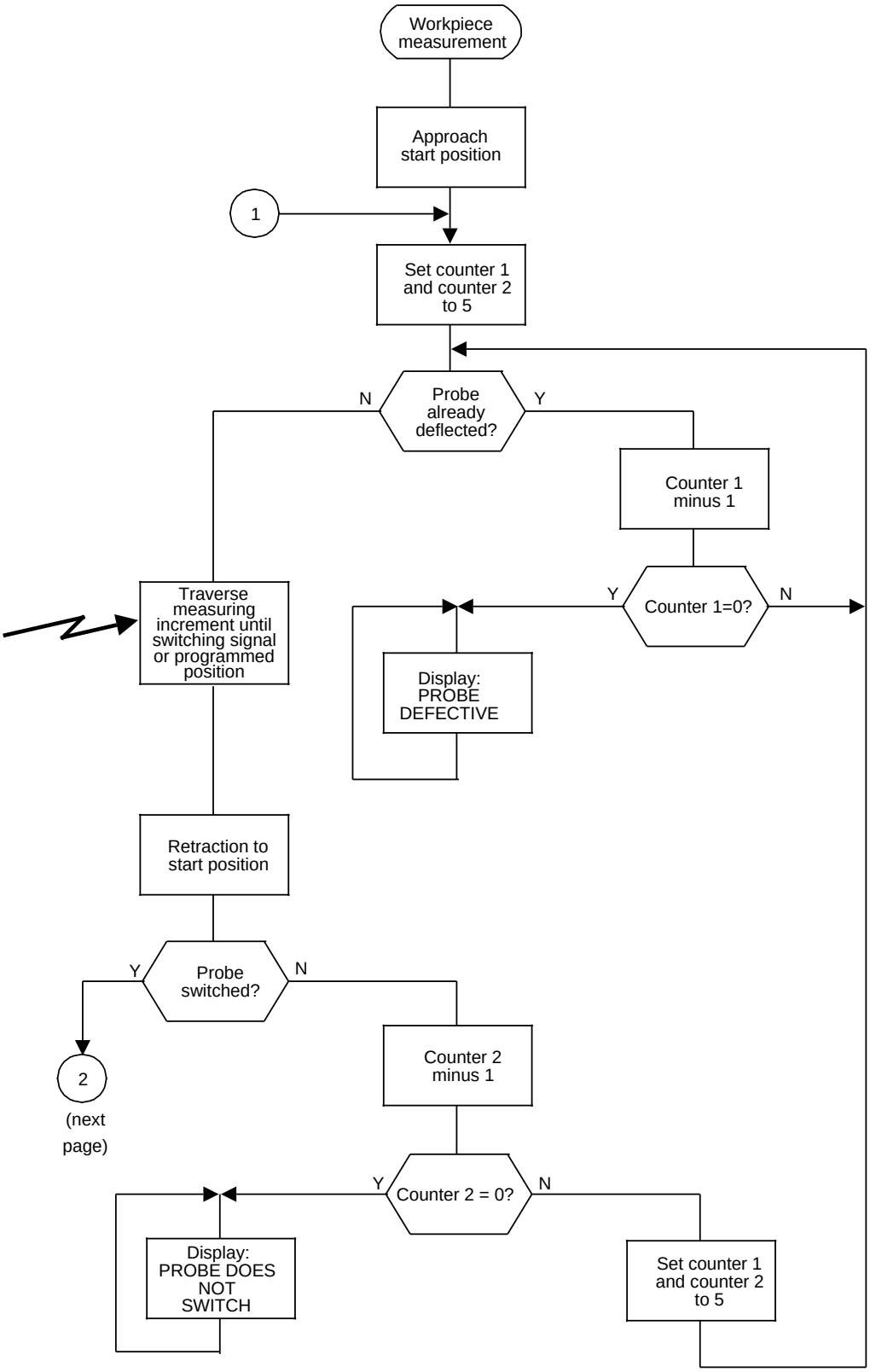


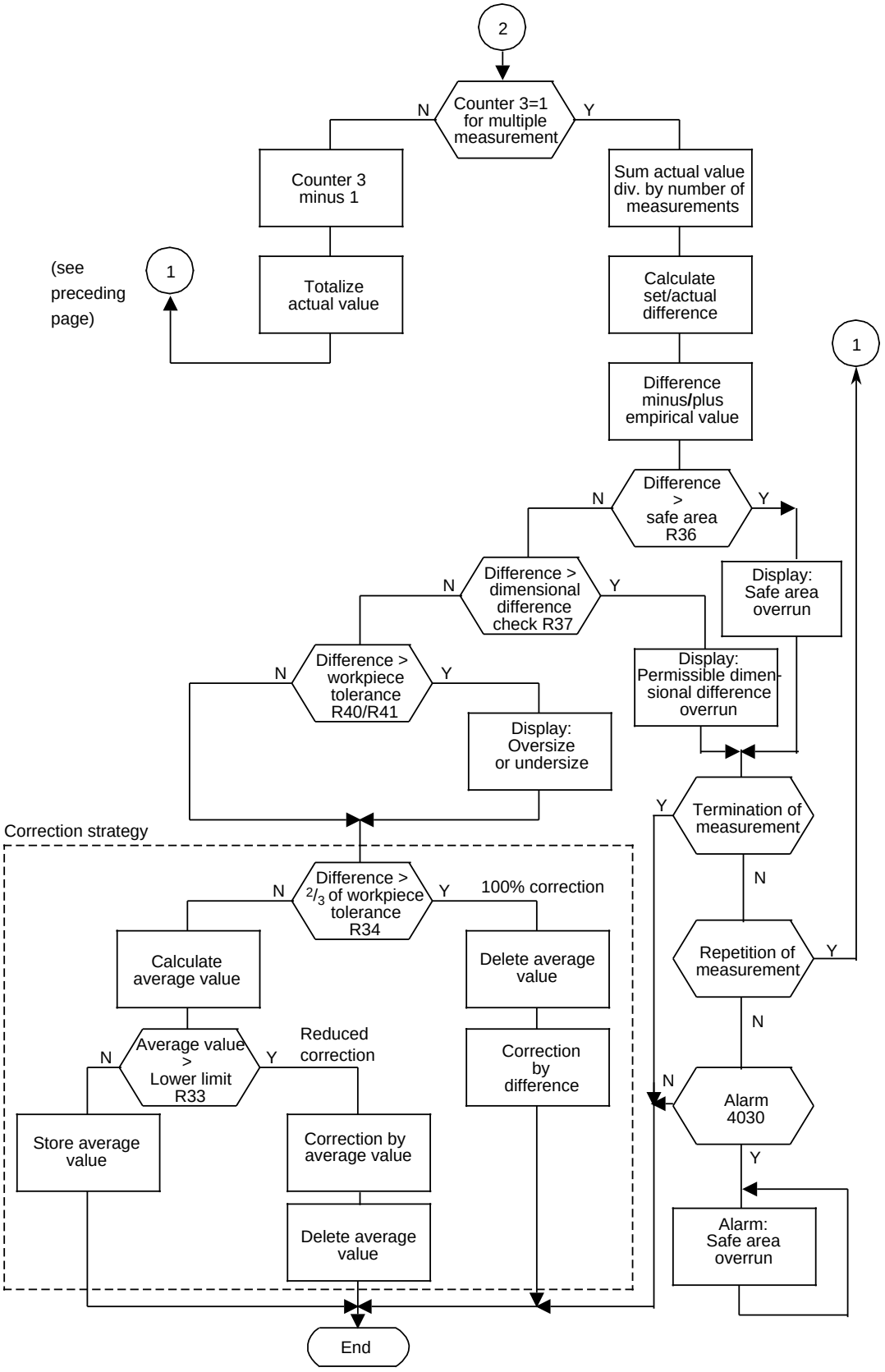
Fig. 1.18 Measuring accuracy

© Siemens AG 1990 All Rights Reserved

1.5.4 Outline flowchart for workpiece measurement



© Siemens AG 1990 All Rights Reserved



© Siemens AG 1990 All Rights Reserved

1.6 Measuring strategy and correction value determination

Precise determination of actual workpiece dimensions is required to determine the actual dimensional deviations of the workpiece and to be able to correct these during subsequent machining operations and thus to obtain a permanent high standard of machining accuracy. When measuring on the machine, the actual dimensions are derived from the path measuring systems of the position-controlled CNC feed axes. For each dimensional deviation determined from the set and actual workpiece dimensions there is a multitude of causes which essentially can be classified in 3 categories.

1.6.1 Types of dimensional deviation

- **Dimensional deviations the causes of which do n o t folllow a trend,** such as position dispersion range of the feed axes or differences in measured values between internal measurements (probe) and external measuring device (micrometer, measuring machine etc.).

  Here, the possibility is provided of using so-called empirical values (R11) which are stored in separate memories. The set/actual difference determined is corrected automatically by the empirical value. See the descriptions of the various cycles and Section 2 for more details.

- **Dimensional deviations the causes of which follow a trend,** such as tool wear or thermal expansion of the ball screw.

  These deviations are corrected by specifying fixed threshold values (e.g. R34) (see Section 1.7).

- **Random dimensional deviations,** e.g. due to temperature variations, coolant or slightly contaminated measuring points.

  Presuming the ideal case, only those dimensional deviations can be taken into account for correction value determination the courses of which follow a trend. Considering the fact, however, that it is never known to which extent and in which direction random dimensional deviations influence the measurement result, a strategy (floating average value generation) is needed that derives the compensation value from the measured set/actual difference.

© Siemens AG 1990 All Rights Reserved

1.6.2 Averaging

Averaging in combination with higher-order measurement weighting has proved a suitable means to do this.
The formula of the average-value generation chosen is:

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

Av_{new} = Average value new = amount of correction

Av_{old} = Average value of last measurement

k = Weighting factor for calculating the average value (R29)

D_i = Set/actual difference measured (minus empirical value, if any)

This formula takes into account the trend of the dimensional deviations of a series of machining operations. The weighting factor k (R29), on the basis of which the average value is generated, can be chosen.

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

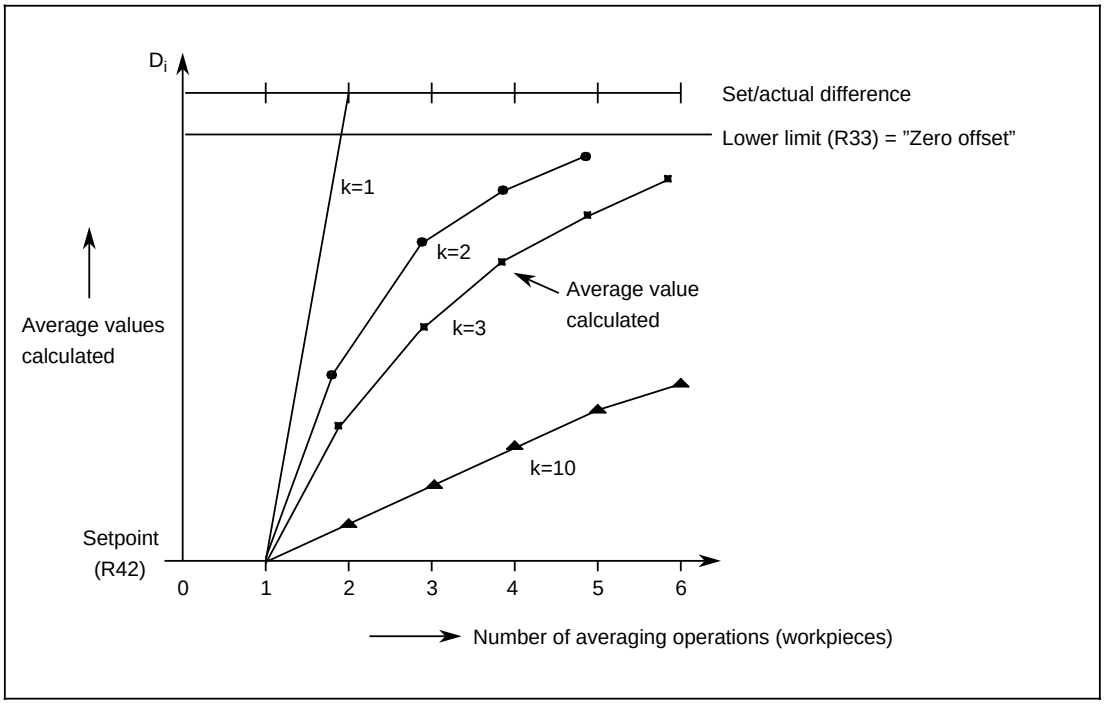


Fig. 1.19 Computational variation of the average value with different weightings k

Effects:

- The greater k, the slower the formula reacts when major deviations occur in computation or counter correction, however, at the same time random scatter is reduced with increasing k.
- The smaller k, the faster the formula reacts when major deviations occur in computation or counter correction, however, the greater the effect of random variations.
- The average value Av is calculated from 0 onwards over the number of workpieces i, until the calculated average value exceeds the range of ”Zero offset” (R33). From this limit onwards, the calculated average value is corrected.

Example: Average value generation

	Lower limit = 4 µm		
	D_i	Average value k=3	Average value k=2
1st measurement	3 µm	1 µm	1.5 µm
2nd measurement	5 µm	2.33 µm	3.25 µm
3rd measurement	6 µm	3.55 µm	③ 4.62 µm
4th measurement	2 µm	3.03 µm	1 µm
5th measurement	4 µm	3.26 µm	2.5 µm
6th measurement	5 µm	3.84 µm	3.75 µm
7th measurement	5 µm	① 4.23 µm	④ 4.375 µm
8th measurement	3 µm	1 µm	1.5 µm
9th measurement	7 µm	3 µm	⑤ 4.25 µm
10th measurement	7 µm	② 4.33 µm	3.5 µm

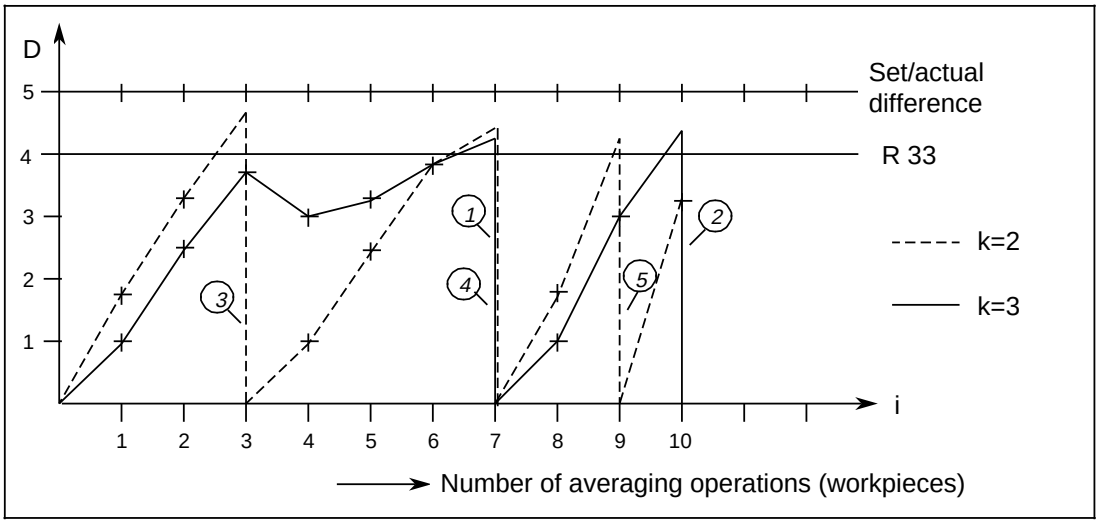


Fig. 1.20 Computational variation of average values with two different weightings k

© Siemens AG 1990 All Rights Reserved

1.7 Tolerance parameters (R33, R34, R36, R37, R40/R41)

Averaging by itself is not sufficient to ensure a permanent high standard of machining accuracy. For constant deviations (no trend), the dimensional deviation measured can be corrected by an empirical value. In addition, further corrections can be derived for deviations following a trend. For this purpose, tolerance bands must be allocated to the associated set dimension via parameters (R parameters) in the machining program prior to measuring cycle call.

The set workpiece dimension is placed in the middle of the permissible ± tolerance band for reasons of symmetry.

The tolerance bands and the reactions derived from them have been specified as follows:

Safe area (R36):

This limit has no influence on averaging. It is used for diagnostics. When this limit is reached, this means that there is a probe fault or that an incorrect setpoint position has been specified. AUTOMATIC operation is interrupted and the program cannot be continued. An alarm text appears on the display to warn the operator.

Caution:

In the case of tool measurement, the ”safe area” tolerance is derived from the difference between the old correction value (TOA memory) and the new tool length.

Dimensional difference check (R37):

Tool wear from workpiece n to workpiece n+1.

This limit has no influence on correction value generation either. When this limit is reached, the tool is probably worn down and must be replaced.

An alarm is displayed to warn the operator. The program can be continued by NC Start.

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

Caution:

For tool measurement, the tolerance is determined as in the case of the safe area.

Workpiece tolerance (R40/R41):

When measuring a dimensional deviation ranging between ”2/3 tolerance of the workpiece” and ”dimensional difference check” (this might be the case when using a new tool, for instance), this dimensional deviation is regarded 100% as tool offset.
As a further step, the previous average value is erased.

This allows fast counteraction when major dimensional deviations occur. AUTOMATIC operation is interrupted when the tolerance limit of the workpiece is exceeded. ”Oversize” or ”undersize” is indicated to the operator on the control display monitor, depending on the tolerance zone position. Machining can be continued by NC Start.

© Siemens AG 1990 All Rights Reserved

2/3 tolerance of workpiece (R34):

Calculation of an average value is carried out within the range of ”Lower limit” and ”2/3 tolerance of workpiece” according to the formula as given in Section 1.6.2 (measuring strategy).

Av_{new} is compared with R33:

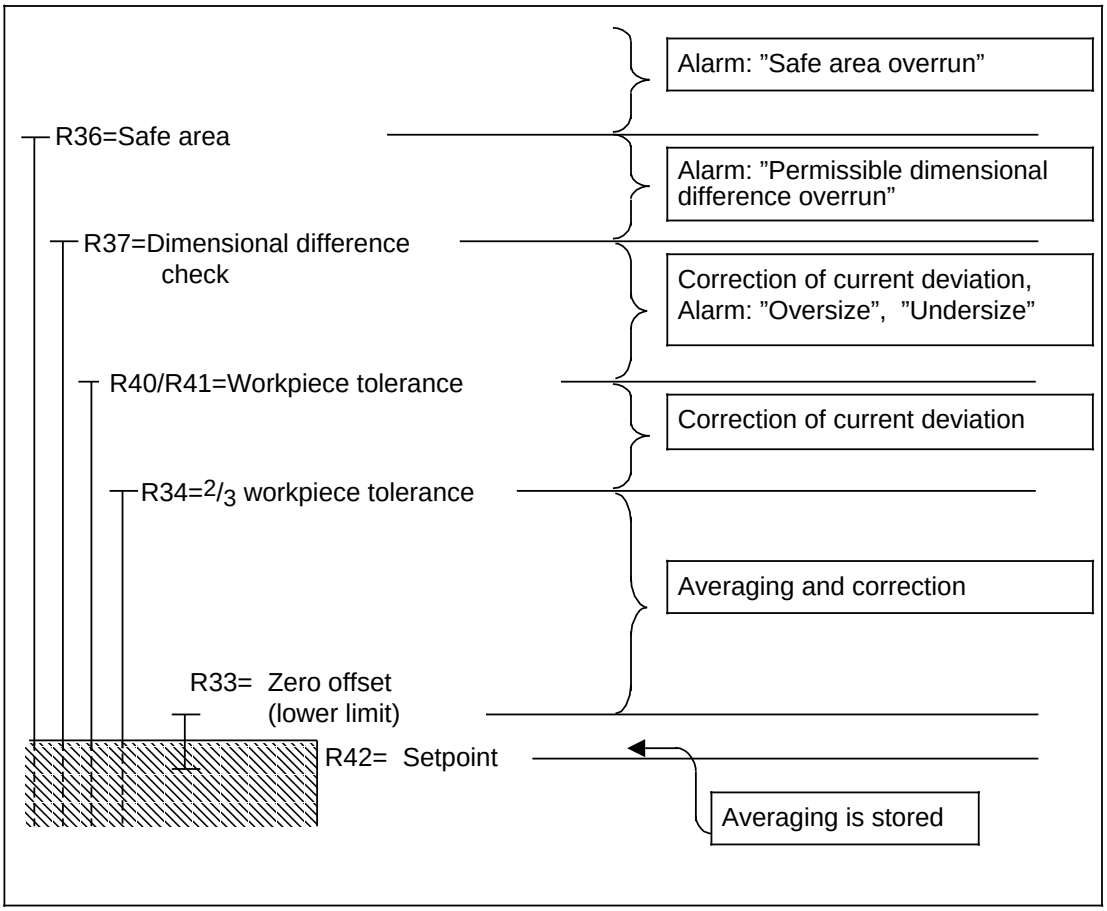
When Av_{new} is **greater than** R33, correction is carried out by Av_{new} and the associated average value memory is erased.

When Av_{new} is **less than** the value in R33, no correction is carried out. This prevents too abrupt corrections from being made (depending on weighting factor k).

Lower limit (zero offset range R33):

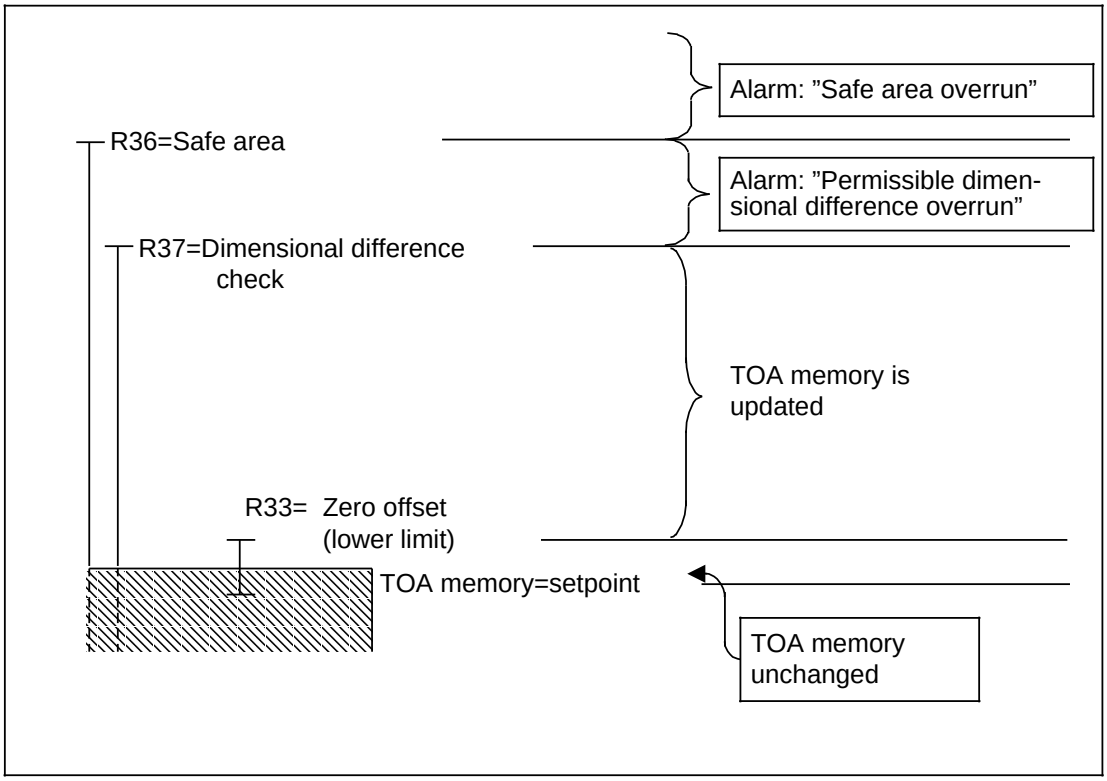
This tolerance band conforms to the amount of maximum random dimensional deviations. It must be determined for every machine. Within these limits, no tool offset applies. However, the average value of this measuring point is updated and re-stored with the set/actual difference measured corrected by an empirical value if necessary.

1.7.1 Tolerance parameters for workpiece measurement



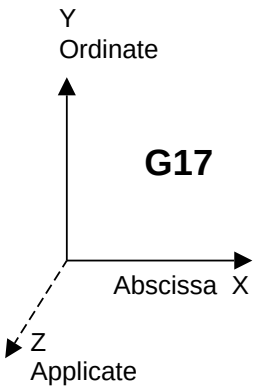
© Siemens AG 1990 All Rights Reserved

1.7.2 Tolerance parameters for tool measurement

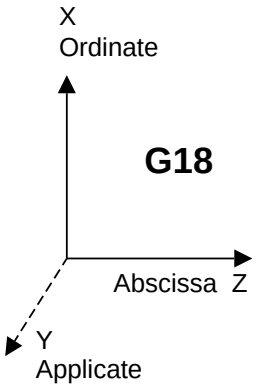


1.8 Plane definition

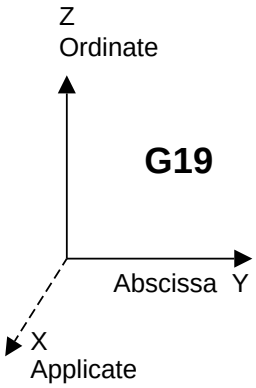
Planes can be selected with G17, G18 and G19 or arbitrarily defined with G16. Length 1 and length 2 are allocated to the axes depending on the type of tool used as follows:



Plane selection according to DIN	Arbitrary plane selection	Tool type	Length 1 effective in axis	Length 2 effective in axis
G17	G16 XY	Types 1...9 Turning tool	Y	X
	G16 XYZ	Types 10..19 Drill	Z	———
	G16 XYZ	Types 20..29 Milling tool	Z	———
	G16 XYZX G16 XYZY	Types 30..39 Angle head	Z Z	X Y



Plane selection according to DIN	Arbitrary plane selection	Tool type	Length 1 effective in axis	Length 2 effective in axis
G18	G16 ZX	Types 1...9	X	Z
	G16 ZXY	Types 10..19	Y	———
	G16 ZXY	Types 20..29	Y	———
	G16 ZXYZ G16 ZXYX	Types 30..39	Y Y	Z X



Plane selection according to DIN	Arbitrary plane selection	Tool type	Length 1 effective in axis	Length 2 effective in axis
G19	G16 YZ	Types 1...9	Z	Y
	G16 YZX	Types 10..19	X	———
	G16 YZX	Types 20..29	X	———
	G16 YZXY G16 YZXZ	Types 30..39	X X	Y Z

© Siemens AG 1990 All Rights Reserved

1.9 Logging

1.9.1 Logging via logging module

The logging module together with an NC control enables measurement results to be logged.

The logging module can be used with all SINUMERIK controls where R parameters are output area by area.

Power supply to this module is from the connections on the front panel. The bus connector provided on the module is exclusively used for mechanical fixing in the module carrier.

The module can therefore be operated in the NC subrack of the SINUMERIK control and also in the external PLC subrack. Individual module holders can also be used. For more details see Logging Module Description. This description is supplied with each module.

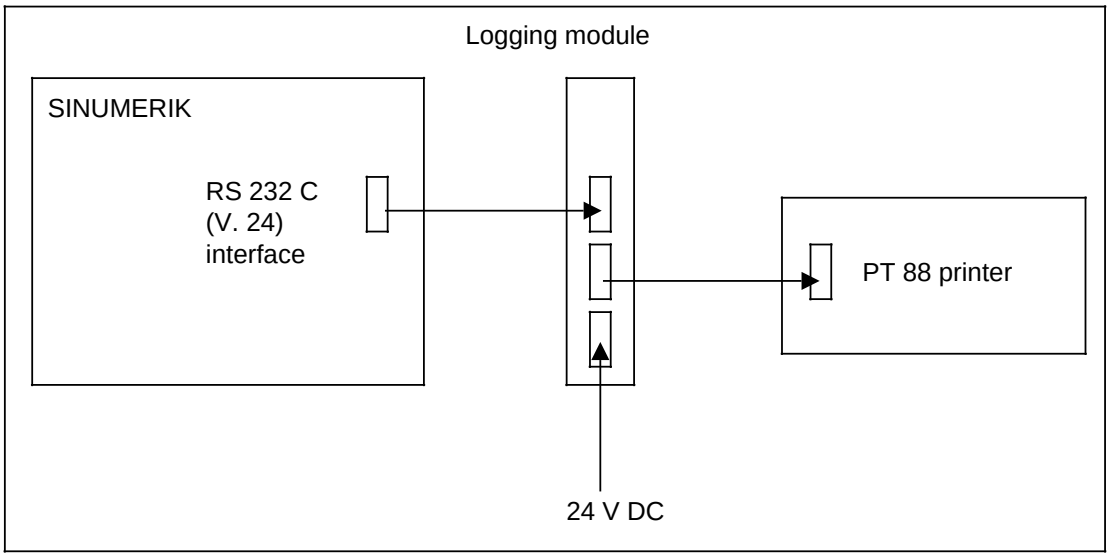


Fig. 1.21 Logging module interface

© Siemens AG 1990 All Rights Reserved

1.9.2 Logging via CP315 with the SINUMERIK 840/880

Measurement results determined by workpiece measuring cycles can be logged with the CP315 and the subfunction FB package ”Logging of measurement results”. To do this, they are transferred to the PLC using the cycle L988. The PLC prepares the measurement results for log output. The measurement results are then transferred to the printer using the subfunction FB package ”PLC controlled data output”.

1.9.2.1 General description

The workpiece measuring cycles determine the data associated with a measuring point and store them in result parameters **R200 to R219**. These R parameters are available to the user for logging.

Using the cycle L988, the FB package ”Logging of measurement results” picks those R parameters the user wants to be logged and outputs them on a printer (formatted). Data transmission is initiated by cycle L988, if the function has been activated via MDC.

The ”Logging of measurement results” function is activated via **MDC 7000.6**. The output module is defined via MDC 7000.1.

Bit 7000.1=0: Logging via P-PCB
Bit 7000.1=1: Logging via CP315

The CP315 output module is required for logging with L988.

Measurement results are output line by line.

When a workpiece has been completely machined, page feed can be effected via a control parameter to enable the log to be removed from the printer.

When a new workpiece is machined, the header and the plaintext for the measured value table is printed out for a new form. Automatic page feed takes place when the bottom of the page is reached. If some measuring points of the workpiece have not yet been recorded, the old header and plaintext are output once again, the page numbering on the following pages is incremented by 1 and the measuring point numbers are continued on the new page.

When machining of a new workpiece is started, a new protocol header is output and the page numbering of the protocol associated with the workpiece is reset to 1.

The user can influence the layout of the form. For example, the fixed texts on the form can be modified to suit the needs of the user. The data types logged on the form can also be freely chosen (within the scope of the available formats). The form is configured, i.e. texts for the form are entered via a part program.

The date and page number of the protocol pages associated with the workpiece can be output on the form.

The following procedure is used to log measurement results:

- Preparation of a part program for configuring the form and programming of initialization triggering transmission of the part program to the PLC.
- Programming of initialization triggering transmission of header data for the protocol to the PLC.
- Programming of a control parameter for protocol output and transmission to the PLC.
- Programming of initialization triggering transmission of measuring results to the PLC.

© Siemens AG 1990 All Rights Reserved

1.9.2.2 Configuring the form

The form for the protocol is configured in a part program that can be edited both at work scheduling and during manufacture.

The part program for configuring the form has the following structure:

Part 1: Description of format

Part 2: Description of fixed texts in the log header and measured value table
Description of variables in the log header

Part 3: Description of the first and last line of the table frame

Part 4: Description of the variables of the basic line of a measuring point
Description of the variables of the 1st supplementary line of a measuring point
Description of the variables of the 2nd supplementary line of a measuring point
⋮ ⋮
Description of the variables of the 8th supplementary line of a measuring point

Part 5: End identifier

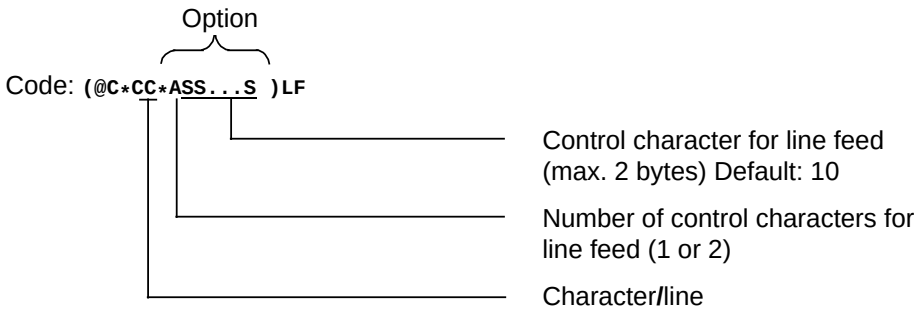
The form descriptions must be written in the part program as comments, i.e. with ”(” and ”)”.

The part program must be terminated with ”M30”.

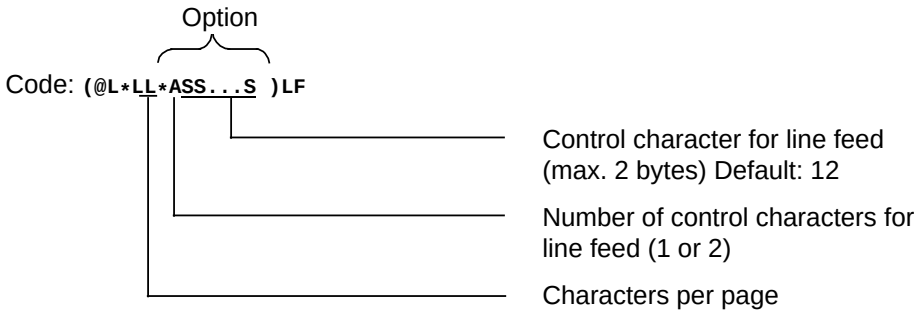
- Part 1

  In part 1, the page format is described. Part 1 consists of:

  – **Specification of character/line**



  – **Specification of lines/page**



© Siemens AG 1990 All Rights Reserved

- Part 2

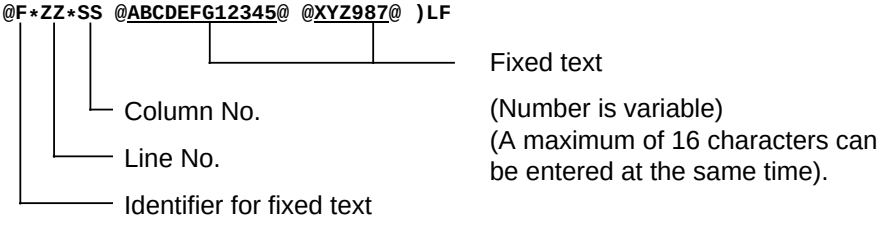
  In Part 2, the fixed texts and variables of the log header are described as well as the locations of date, time of day and page number.

  The maximum header size is 2000 characters:
  (characters per line+6) · number of lines of header 2000.

  Part 2 consists of:

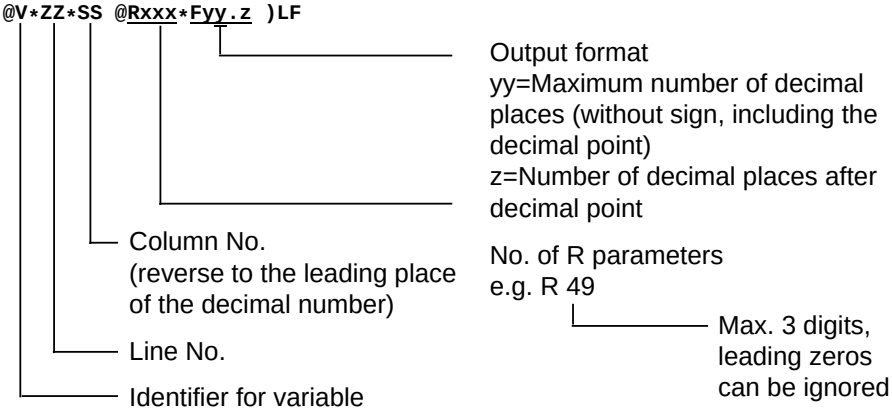
  – **Definition of fixed texts**

  Code: **(@F*ZZ*SS @ABCDEFG12345@ @XYZ987@)LF**

  – **Definition of variables**

  Code: **(@V*ZZ*SS @Rxxx*Fyy.z)LF**

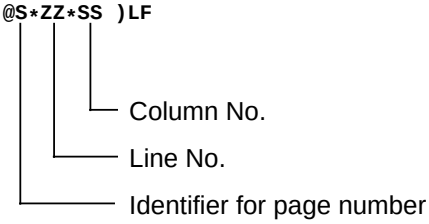
  

  The last fixed text line must not be followed by a line with a variable without fixed text. Remedy: Write additional fixed text, e.g. (@F*ZZ*SS @ @)LF in the line containing the variable.

  The R parameters for header data must be programmed as a continuous area and in ascending sequence.

  – **Definition of page numbers**

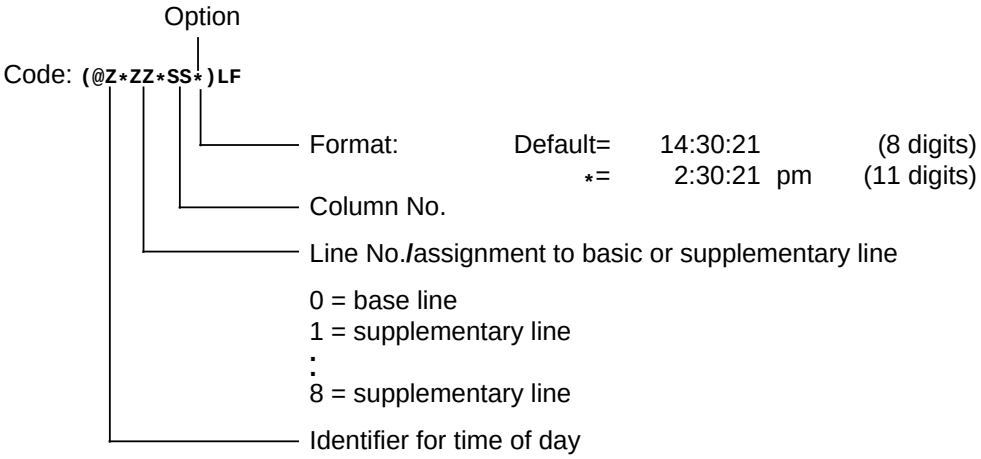
  Code: **(@S*ZZ*SS)LF**

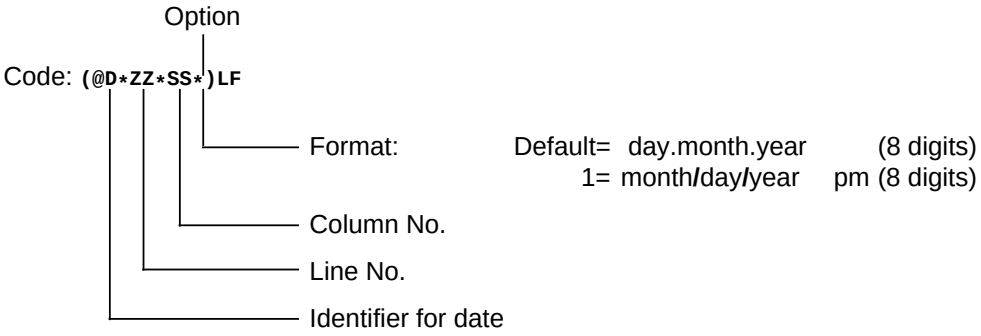
  The page number has two decimal places. The column specifications refer to the 10th place.

© Siemens AG 1990 All Rights Reserved

– **Definition of time of day**

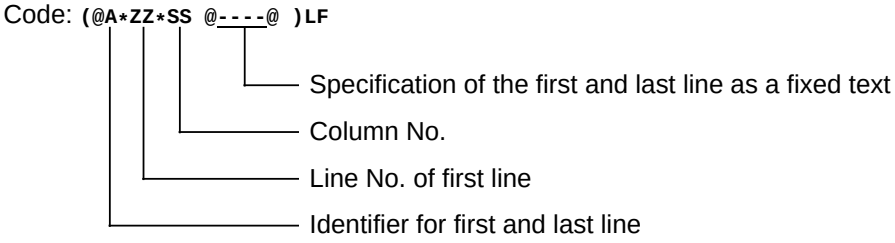


– **Definition of date**



- Part 3

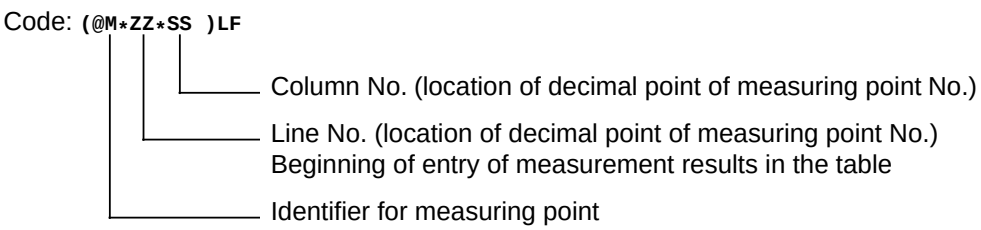
  Part 3 consists of the description of the first and last line of the table frame.



- Part 4

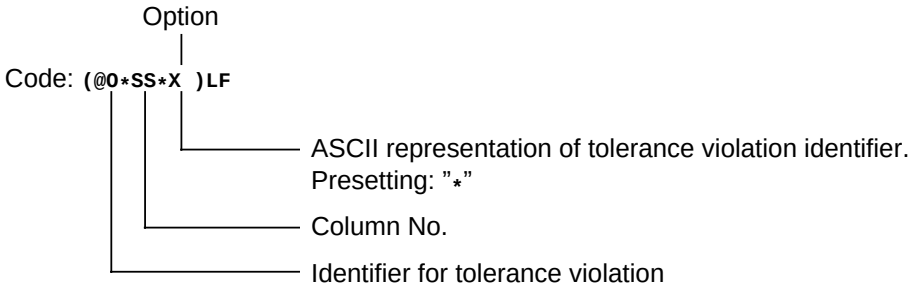
  In Part 4, a description is given of the measuring points, the table frame and the variables of the basic and supplementary lines. Part 4 consists of:

  – **Specification of locations of decimal points of the measuring points**



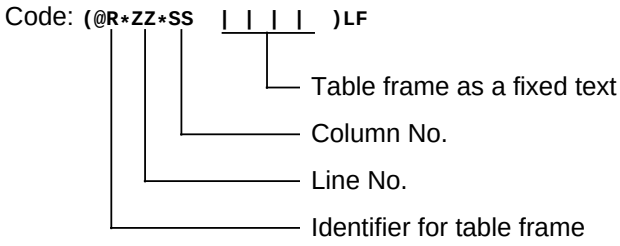
© Siemens AG 1990 All Rights Reserved

– **Specification of location of tolerance violation**

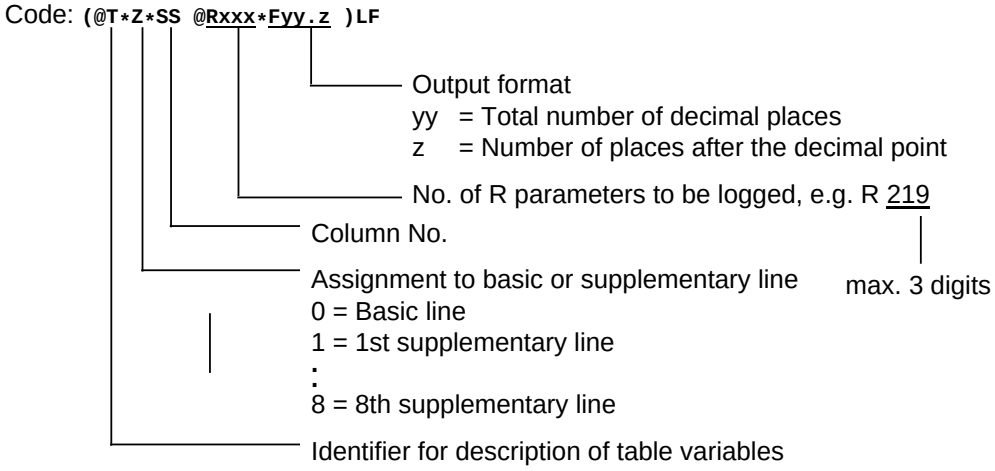


– **Definition of perpendicular lines of the table frame**

The table frame must be input including blanks



– **Description of table variables**



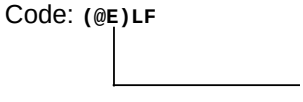
– **Definition of time of day**

Code: see above

The variables of the basic line and all possible supplementary lines must be defined one. First define all variables of the basic line, then those of the first supplementary line etc.

- Part 5

Part 5 consists of the end identifier for form description.



© Siemens AG 1990 All Rights Reserved

1.9.2.3 Data transmission between NC and PLC

Data transmission, general

The following data must be transmitted from the NC to the PLC:

- Part program for form
  The part program is transmitted to the PLC using the ”File transfer with PLC initiation” function.
- Control parameter for protocol output, data for protocol header and parameters for measuring point numbers. R parameters R39 to R45 are used for these.
- Measurement results and parameters for tolerance violations
  The measurement results are contained in the R parameters R200 to R219. The identifiers for tolerance violations are contained in the R paramenters R38 and R39.

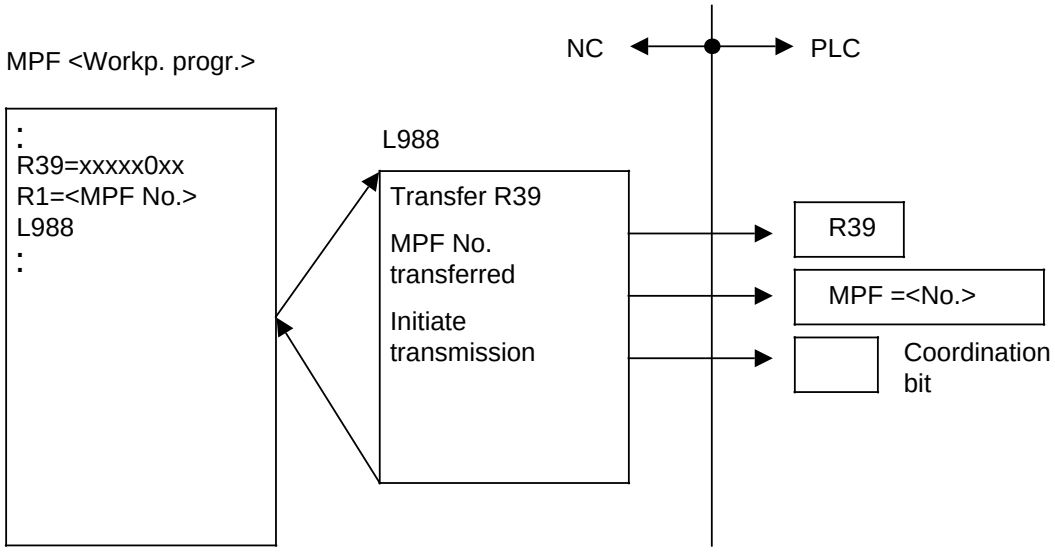
The type of data to be transmitted is defined in R39, 2nd digit:

0 = Part program for form
1 = Header data
2 = Measurement results + R39+R38 (+R45)
3 = Control function
: :
9 = Control function

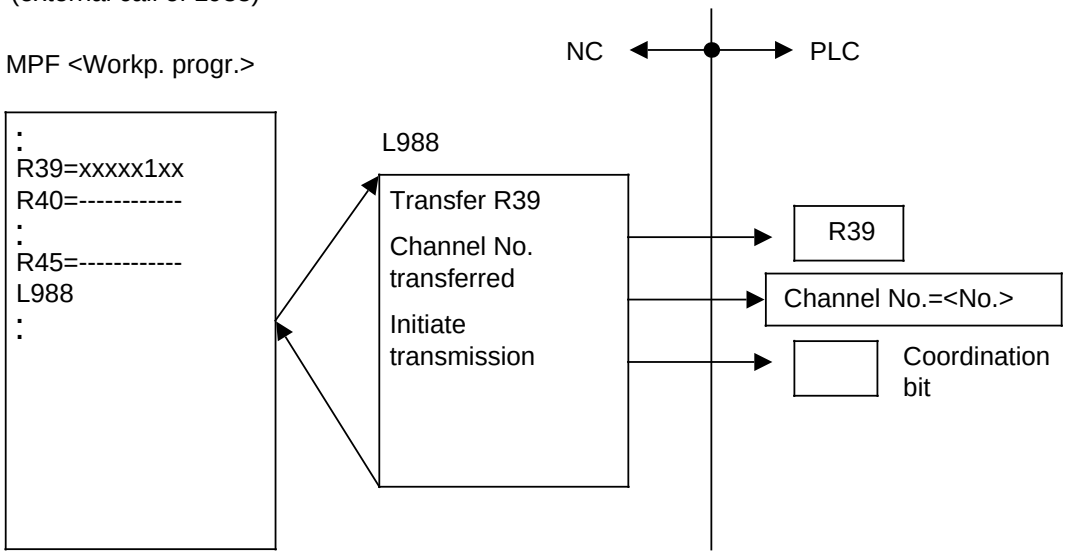
The part program number of the format description must be entered in R1. It is also possible to use a screen form to transmit the part program (for the form) and to enter the part program number.

Call of L988 (internal/external)

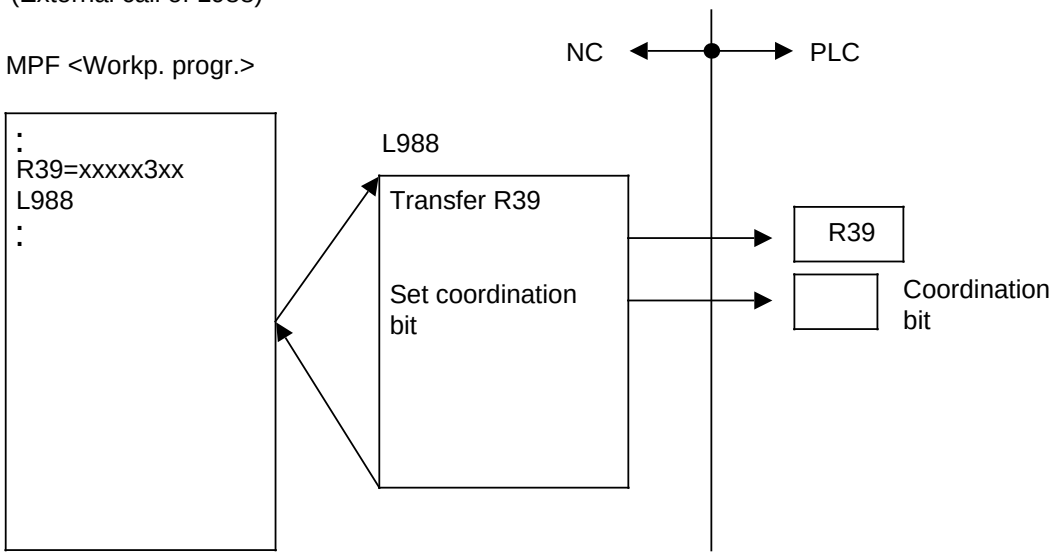
- Initiation for transmitting a part program with format description
  (external call of L988)



- Initiation for transmission for header data
  (external call of L988)

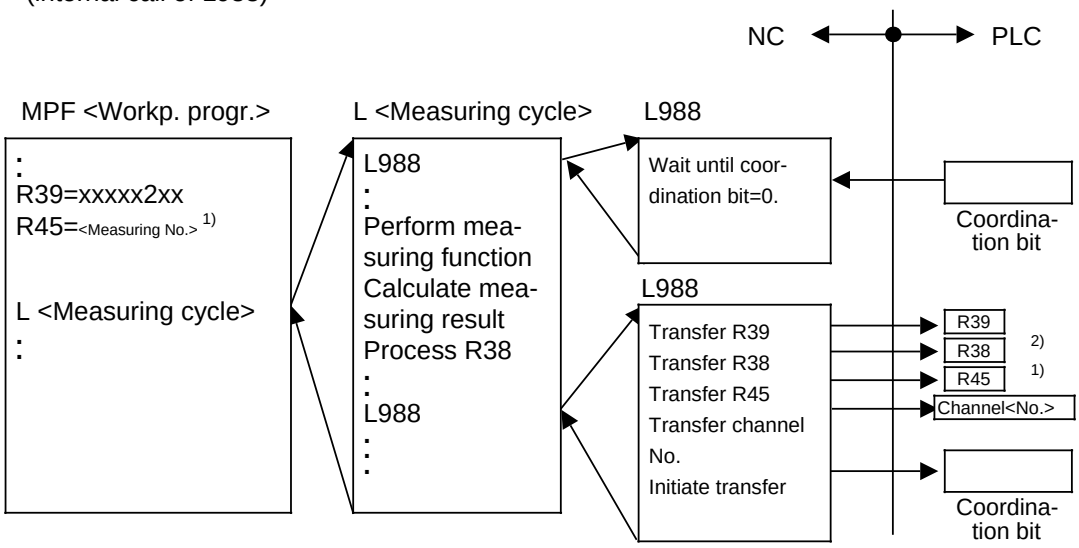


- Transmission of control parameter R39
  (External call of L988)



© Siemens AG 1990 All Rights Reserved

- Initiation for transfer of measurement results
  (internal call of L988)



1.9.2.4 Description of transfer parameters

When using Siemens measuring cycles, the R parameters R38 to R45 are used as transfer parameters for logging output, protocol header information, measuring point numbers and tolerance violations. The values stored in the R parameters are transferred and not the R parameter numbers. This enables the R parameters to be freely chosen.

The R parameters R38 and R39 are control parameters (for Siemens measuring cycles).

The parameter addresses may have a maximum of 3 digits. The R parameter values may have a maximum of 8 digits with additional sign and decimal point. Leading zeros in R parameters can be ignored. In decimal-point notation, trailing zeros need not be written either.

With values less than 1, a 0 precedes the decimal point, e.g. 0.999 or 0.98.

Only negative signs are printed, positive signs are omitted. The sign is always printed in the same column irrespective of the succeeding number of decimal places of the value to be printed.

- Control parameter R39

  R39 is an 8 digit control parameter.

  – Configuration of R39
    Digit 7 unassigned
    Digit 6 unassigned
    Digit 5 end of protocol
    Digit 4 unassigned
    Digit 3 specification of measuring point number
    Digit 2 type of data to be transferred
    Digit 1 modification of protocol
    Digit 0 number of protocol lines of measuring point or number of space lines

---

1) R45 is required only if the measuring point number is preselected by the NC.

*2) R38 is transferred only if the supplementary lines are marked with * (tolerance exceeded).*

© Siemens AG 1990 All Rights Reserved

– Description of decimal place 5
  R39=xx9xxxxx End of measurement result logging

– Description of decimal place 3
  R39=xxxx0xxx Measuring point number from the PLC
  R39=xxxx5xxx Measuring point number from R45

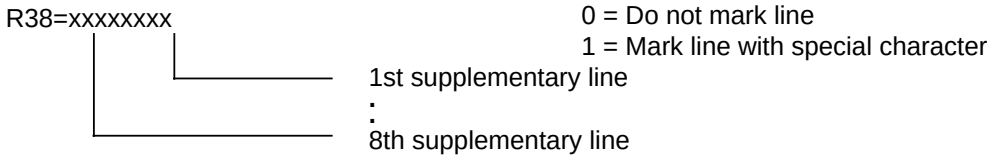
– Description of decimal place 2
  R39=xxxxx0xx Transfer of part program for a form
  R39=xxxxx1xx Transfer of workpiece-specific header data
  R39=xxxxx2xx Transfer of measurement results

– Description of decimal place 1
  R39=xxxxxx2x Do not mark the measurement result with a special character (base line)
  R39=xxxxxx3x Mark the measurement result with a special sign (base line)
  R39=xxxxxx5x Space line (with frame)
  R39=xxxxxx6x Page feed

– Description of decimal place 0
  R39=xxxxxxx 0 Number of protocol lines of a measuring point
  : or number of blanks
  9
  0 Output basic line only
  1 Output basic line + supplementary line
  2 Output basic line + 2 supplementary lines

• Control parameter R38

  R38 is an 8-digit control parameter.



  Mark measurement result with special character (1st to 8th supplementary line).
  Explanation: Special character = tolerance exceeded

• Transfer parameters for header data (R40 ... R45)

	R39=xxxxy1xx		y=0 or 5 [1]	
e.g.	R40=7 ... 0	e.g.	Part No.	
e.g.	R41=15 ... 0	e.g.	Order No.	(e.g. Digital places 8 ... 15 [2])
e.g.	R42=7 ... 0	e.g.	Order No.	(e.g. Digital places 0 ... 7 [2])
e.g.	R43=3 ... 0	e.g.	Program No.	
e.g.	R44		not evaluated	
e.g.	R45		not evaluated	

  – Example showing header data

    R39=11110100
    R40=100.4711
    R41=0
    R42=100
    R43=22

    Header data must be output by the part program before the first measuring points are measured.

---

[1] *y=0: Automatic assignment of measuring point number*
y=5: Assignment by part programmer

[2] *determined in the form description*

© Siemens AG 1990 All Rights Reserved

The following data are output in the program header after the measuring data determined above.

Part number	:	100.4711	Date: 16.08.88	Page: 1
Order number	:	100		
Program number	:	22		

- Transfer parameter for measuring point No. (4 digits) (R45)

  The measuring point number can be generated in two ways, either automatically by the PLC or from parameter R45.

Note:

If the measuring point number is defined via R45 the measuring results must only be entered in R parameters R200-R219. Bit 7 must be set in cycle machine data 7000 (only enter result in R200-R219).

Which method is used is determined in parameter R39 with the command for output of the log header before the measurement results are logged. Once the desision has been made with log header output, change between automatic measuring point specification by the PLC and manual preselection by the part programmer is no longer possible.

With manual preselection, R45 must precede each measuring circle call.

The measuring point number has a total of 6 digits. The 4 left digital places are the actual measuring point number.

– Submeasuring point No. (decimal point and 1 decimal place)

  The submeasuring point No. is used to designate suplementary lines whenever several measurements are to be made at one measuring point. How many submeasuring points of a measurement are to be logged is defined in R39. Submeasuring points must always be logged in ascending sequence and without a gap.

  The assignment of the submeasuring point No. to a particular axis or function is made by the user.

  The base line contains data of the diameter measured, the 1st supplementary line data of the horizontal axis and the 2nd supplementary line data of the particular axis when Siemens measuring cycles and submeasuring points are used. Further supplementary lines are not needed at present.

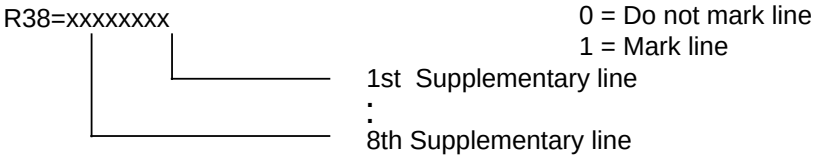
- Examples of parameter assignment

  – Log output, basic line
    R39=111102x0 Identification parameter
    x= 2 ... 3

    R39=11110220 must be assigned for logging a measuring point. If a tolerance violation of the setpoint is detected after the measurement, the relevant measuring point can be given the identifier R39=11110230. The setpoint difference is marked with a special character.

© Siemens AG 1990 All Rights Reserved

– Log output, supplementary lines



R39=111022x can be assigned for logging submeasuring points.
x=1 ... 8: Number of supplementary lines

R39=11110230 can be assigned if a tolerance violation of one or several setpoints is ascertained after the measurement. In addition, the submeasuring points concerned must be marked in R38. The relevant setpoint difference is then marked with a special character.

– End of log, logging of measurement results
R39=11911311

End of log causes page feed to be activated and deselects the ”Logging” mode in the PLC. The preceding log must be concluded with ”End of protocol” before every new log (e.g. new header data, new part program for format description).

– Line feed
R39=1111035x
x=1 ... 9

The 0th digit in identification parameter R39 specifies the number of space lines. Up to 9 space lines can be programmed. Page feed does not cause the space lines to be transferred to the next page.

– Page feed

  – Page feed selectively programmed
    R39=11100361
    Page feed can be programmed selectively, e.g. after ”End of measurement result logging”.

  – Automatic page feed
    Automatic page feed is always generated if:
    – a page is full.
    – a new log header is called.

© Siemens AG 1990 All Rights Reserved

1.9.2.5 Error messages of logging function

- Error message of auxiliary cycle L988

  ”Check MDC(N7000)”: Is generated, if MDC 7000.1 and MDC 7000.6 are not set when L988 is called.

  ”PLC interface not ready”: Is generated if the coordination bit is not reset by the PLC within 5 seconds.

- Error character when logging

Part number . . . . . : 1002458 Date: 24.12.88 Page: 1

Order number . . . . . : 1000000001

Program number . . . . . : 1234

Measuring point	Setpoint	Actual value	Tolerance limit +	Tolerance limit –	Setpoint difference	Time of day
18	–9999.333	–9999.333	–1.999	–1.999	–1.999	11:39:28
****	–99999.333	–99999.333	#####	2.500	–1.700	11:39:29
19	#####	#####	1.200	#####	–1.700	11:39:30
20	44.000	44.000	1.200	2.500	–1.700	11:39:31

Explanation:

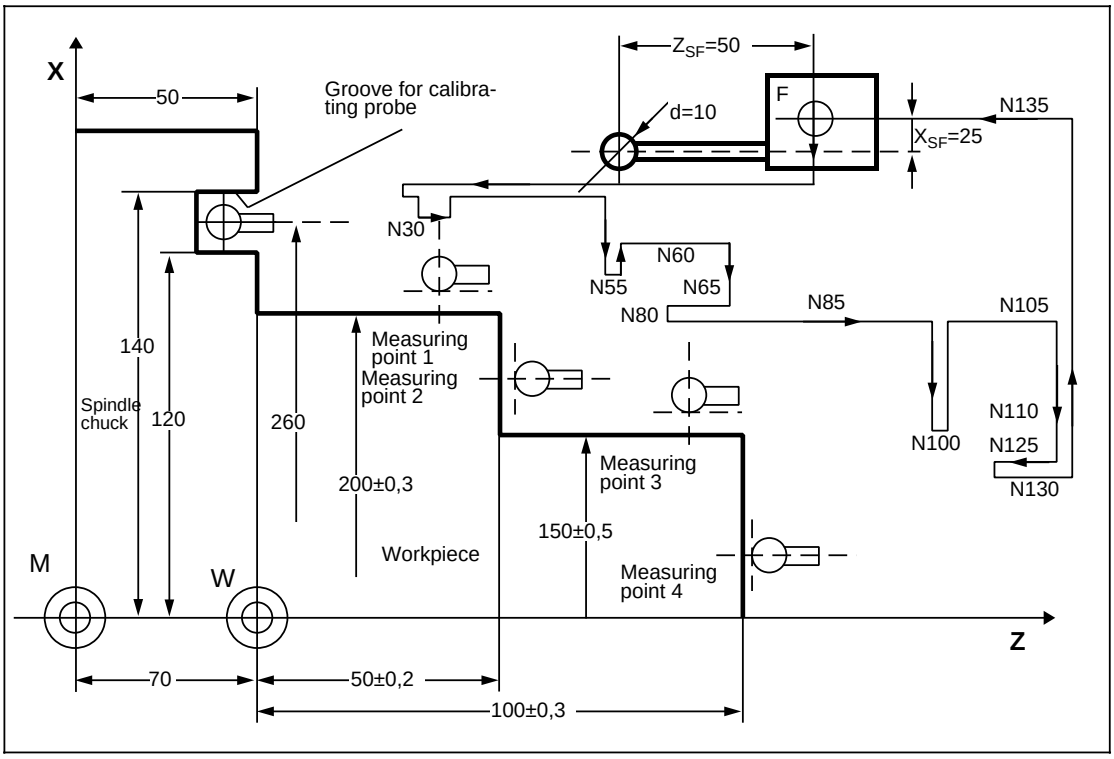
Measuring data exceeding the specified format.
**** Measuring point number in R45 too long

1.9.2.6 Example demonstrating the logging of measurement results for the SINUMERIK 840/880

- Task definition
  - Workpiece drawing

    The axially symmetric workpiece shown is to be measured with a probe after it has been machined. The measuring data are logged.

© Siemens AG 1990 All Rights Reserved



– The following test log is to be output:

Column 123456789012345678901234567890123456789012345678901234567890123456789012345678901234567890
10 20 30 40 50 60 70 80

Line

Measuring point	Setpoint	Actual value	Tolerance limit +	Tolerance limit –	Setpoint difference	Time of day
0001.0 4+1 decimal place + fixed point from the PLC	3111.180 8 places without sign with decimal point 4 places before, 3 places after decimal point (R206)	3112.303 8 places (R209)	1.123 5 places (R200)	–0.001 5 places (R203)	1.123* 5 places (R212)	09:46:02 8 places

Part number . . . . . :<23456789> (R40) Date: dd.mm.yy// / Page: No.

Order number . . . . . :<23456789> (R41)

Program number . . . :<23456789> (R43) Serial number . . . . :<23456789> (R44)

R parameters in brackets do not appear on the log. They only serve as an indication in which columns or lines the parameter contents are entered.

© Siemens AG 1990 All Rights Reserved

- Suggestion for a solution

  Prerequisites: configuration of the computer link via CP315, configuration of the FB package subfunctions ”Logging of measurement results” and ”PLC controlled data output” and activation of the logging function via MDC.

  The following procedure is then suggested:

  – Preparation of a part program for configuring the form. The part program is as follows:

Part 1

```
%MPF 200
( @C*80 )
( @L*72 )
( @F*1*1 @--------@ @-------------@ @-------------@ @-------------@
@------------@ @------------@ @--------@ )
( @F*2*1 @I@ )
( @F*2*80 @I@ )
( @F*3*1 @I part number.......:@ )
( @F*3*52 @Datum:@ )
( @F*3*71 @Seite:@ )
( @F*3*80 @I@ )
( @F*4*1 @I@ )
( @F*4*80 @I@ )
( @F*5*1 @I order number....:@ )
( @F*5*80 @I@ )
( @F*6*1 @I@ )
( @F*6*80 @I@ )
( @F*7*1 @I program number....:@ )
( @F*7*45 @ serial number......:@ )
( @F*7*80 @I@ )
( @F*8*1 @------@ @-------------@ @-------------@ @-------------@
@-------------@ @-------------@ @---------@
( @F*9*1 @I@ )
( @F*9*80 @I@ )
( @F*10*1 @I measuring@ @points@ )
( @F*10*15 @setpoint@ @     actual@ @value    toleran@ @ce limit    set@
@ point   Tim@ @e of day@ )
( @F*10*80 @I@ )
( @F*11*1 @I@ )
( @F*11*42 @+@ )
( @F*11*51 @-    @ @ difference@ )
( @F*11*80 @I@ )
( @F*12*1 @------@ @-------------@ @-------------@ @-------------@
@-------------@ @-------------@ @---------@ )
( @V*3*23 @R40*F8.0 )
( @V*5*23 @R41*F8.0 )

( @V*7*23 @R43*F8.0 )
( @V*7*65 @R44*F8.0 )

( @D*3*58 )
( @S*3*77 )
```

Part 2

```
Part 3  ( @A*13*1 @------@ @-------------@ @-------------@ @-------------@
        @-------------@ @-------------@ @---------@ )
        ( @M*15*8 )
        ( @O*64** )
        ( @R*14*1 @I     @ @      I      @ @     I      @ @     I       @
        @ I        I  @ @        I    @ @         I@ )
        ( @T*0*16 @R206*F8.3 )
        ( @T*0*28 @R209*F8.3 )
        ( @T*0*40 @R200*F5.3 )
        ( @T*0*49 @R203*F5.3 )
        ( @T*0*58 @R212*F5.3 )
        ( @Z*0*69 )
Part 4  ( @T*1*16 @R207*F8.3 )     The supplementary lines need not
        ( @T*1*28.@R210*F8.3 )     be defined for this example.
        ( @T*1*40 @R201*F5.3 )
        ( @T*1*49 @R204*F5.3 )
        ( @T*1*58 @R213*F5.3 )
        ( @Z*1*69 )
        ( @T*2*16 @R208*F8.3 )
        ( @T*2*28 @R211*F8.3 )
        ( @T*2*40 @R202*F5.3 )
        ( @T*2*49 @R205*F5.3 )
        ( @T*2*58 @R214*F5.3 )
        ( @Z*2*69 )
Part 5  ( @E )
        M30
```

– Programming of initiation triggering transfer of the part program to the PLC

```
%MPF 201
N1 R39=00000000                    (Define control parameter)
N2 R1=200                          (Specify MPF No.)
N3 L988                            (Initiate transfer)
N4 M30
```

– Programming of part program for workpiece measurement

```
%MPF 202

N1   R900=R900+1                   (Increment workpiece counter by 1)
     R39=00000100                  (Define control parameter for header data transfer)
     R40=R900                      (Header data)
     R41=12343210                  (Header data)
     R43=974                       (Header data)
     R44=10002000                  (Header data)
N4   L988                          (Transfer header data to the PLC)

N5   G54 G00 X260 Z T1 D31         (Select ZO, call T No. and TO No.)
N10  R22=1 R23=22 R25=0            (Define parameter for calibration)
     R27=1 R28=1 R30=1
N15  R31=1 R32=-20 R33=0 R36=1
N20  L973                          (Calibrate probe in minus Z direction)
N25  R30=2 R32=240                 (Define parameter for calibration)
N30  L973                          (Calibrate probe in minus X direction)

N35  G54 G00 Z40 T2 D32            (Position MP1, call T No. and MC No.)
N40  R10=8 R11=0 R22=1             (Define parameter for measurement)
     R23=21 R25=0 R27=1
     R28=1 R29=1 R30=2
N45  R33=0.002 R34=0.005
     R36=2 R37=1 R40=0.3
     R41=-0.3 R42=200              (Define control parameter for transfer of measurement
N46  R39=11110220                  results)
N55  L974                          (Measure MP1)
```


© Siemens AG 1990 All Rights Reserved

```
N60  G00 Z70 T1 D31                (Position probe to face MP2)
N65  X175
N70  R10=9 R30=1                   (Define parameter for measurement)
     R40=0.2
     R41=-0.2 R42=50
N71  R39=11110220                  (Define control parameter for transfer of measurement results)
N80  L974                          (Measure MP2)

N85  G00 Z80 T2 D32                (Position probe to face MP3)
N90  R10=10 R30=2                  (Define parameter for measurement)
     R40=0.5
     R41=-0.5 R42=150
N91  R39=11110220                  (Define control parameter for transfer of measurement results)
N100 L974                          (Measure MP3)

N105 G00 Z150 T1 D31               (Position probe to face MP4)
N110 X50
N115 R10=11 R30=1                  (Define parameter for measurement)
     R40=0.3
     R41=-0.3 R42=100
N116 R39=11110220                  (Define control parameter for transfer of measurement results)
N125 L974                          (Measure MP4)

N130 G53 G00 Z250 T0               (Retract Z axis)
N135 G53 X280                      (Retract X axis)
N140 R39=11911311                  (Define end of protocol)
N145 L988                          (and transfer to the PLC)

N150 M30
```

– Log of programming example

Part number . . . . . : 17 Date: 30.08.91 Page: 1

Order number . . . . . : 12343210

Program number . . . . . : 974 Serial number . . . . . : 10002000

Measuring point	Setpoint	Actual value	Tolerance limit +	Tolerance limit –	Setpoint difference	Time of day
1	200.000	200.104	0.300	– 0.300	– 0.104	09:36:20
2	50.000	50.797	0.200	– 0.200	–0.797 *	09:36:30
3	150.000	149.946	0.500	– 0.500	0.054	09:36:40
4	100.000	100.141	0.300	– 0.300	– 0.141	09:36:50

© Siemens AG 1990 All Rights Reserved

1.10 Notes to ensure smooth running of the measuring cycles

1. To ensure that the measuring cycles operate correctly it is imperative that the machine axes are set up according to DIN 66217.
2. Reference point approach must already have been performed.
3. GRC/CRC, (coordinate rotation) and scale modification must be deselected before a measuring cycle is called.
4. Inch/metric switchover.

   Measurements must be performed in the input system defined in MD 5002 bit 4, i.e. switchover with G70/G71 is not permitted.
5. Before the cycles are called, the axes must be positioned in such a way that they do not change direction when they move from the current position to approach the set position. Nor must the tool collide with the sensor or the workpiece probe collide with other machine parts when approaching the starting position with linear interpolation.
6. The parameters of the individual cycles must be defined before the cycles are called.
7. The cycles are automatically skipped in operating modes "Block search", "Dry run" and simulation.
8. All cycles can be exited with the following initial settings:

   T version G01, G90, G95
   M version G01, G90, G94
9. The cycle must be called no later than the 1st nesting depth.
10. Call cycle **L965**. This cycle must be started at least **once** after start-up and it must always be programmed when the measuring plane is changed.

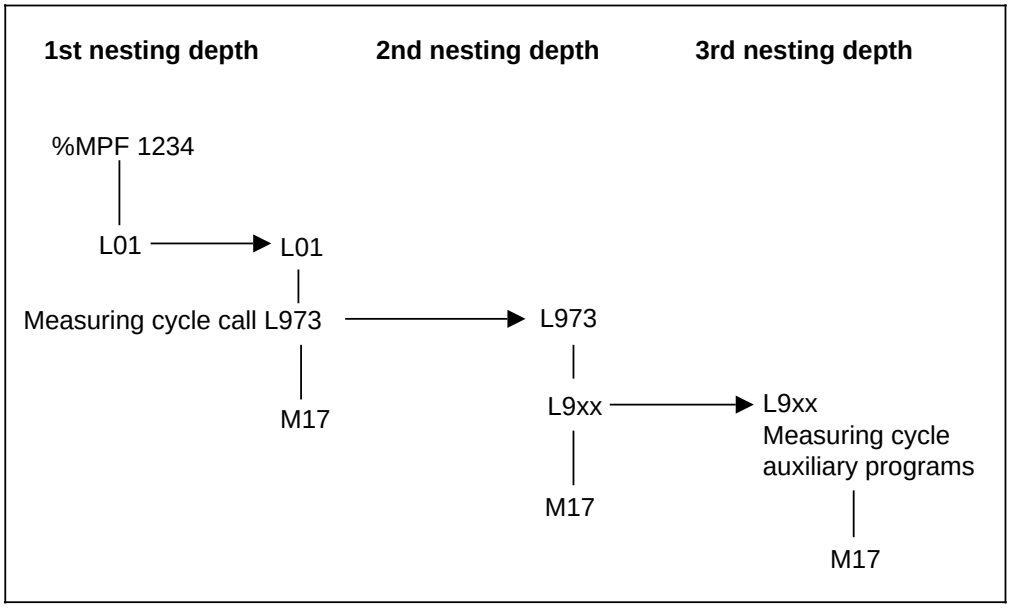


Fig. 1.22 Nesting depth when calling measuring cycles

© Siemens AG 1990 All Rights Reserved

1.11 Reference points on machine and workpiece

The actual axis values of different actual value systems must be measured depending on the kind of measurement taken. For determining the tool length, for instance, it is advisable to use the machine actual value, while for measuring workpiece dimensions and for determining tool wear compensation it is more advantageous to use the workpiece zero point (Fig. 1.23). The machine actual value is the dimension between the machine zero point and the tool reference point.

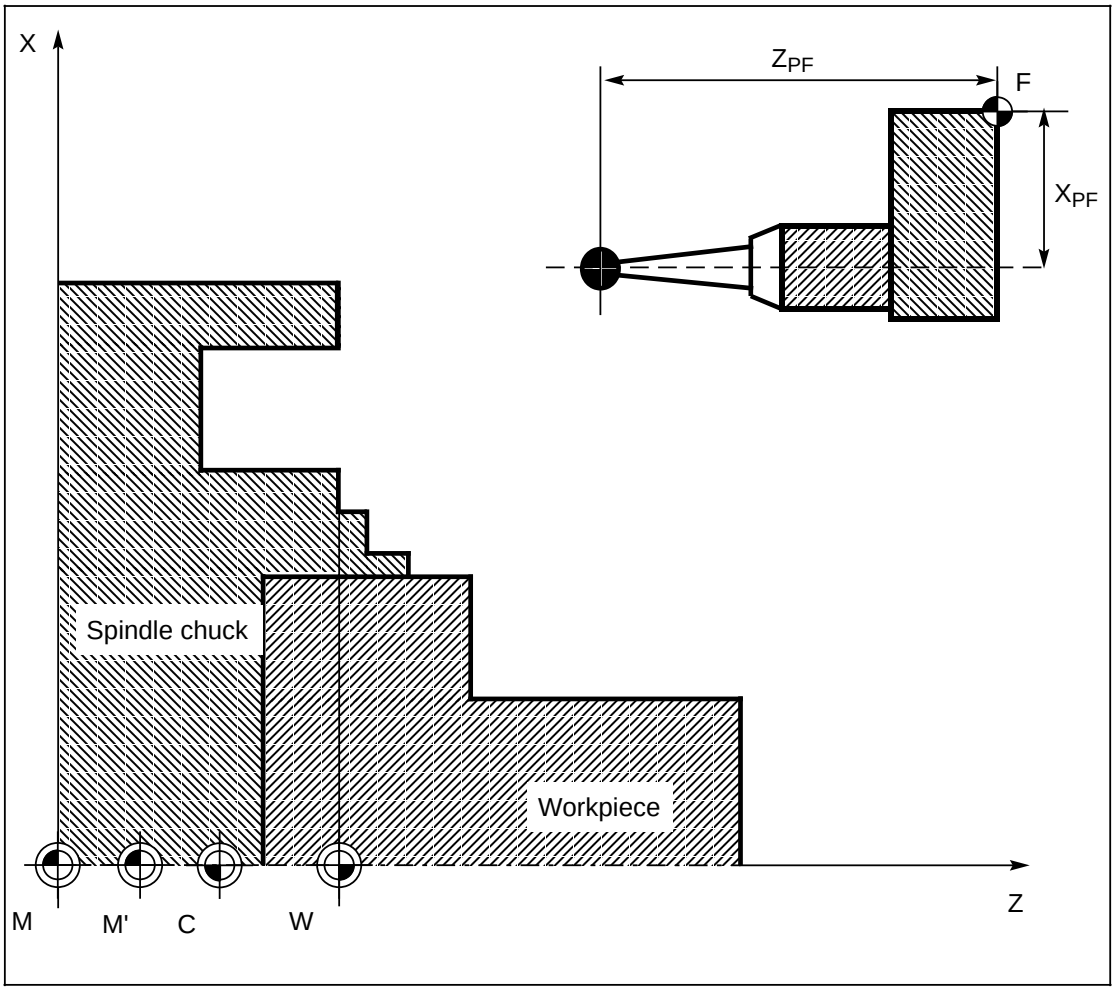


Fig. 1.23 Reference points on machine and workpiece

M = Machine zero point
M' = Machine zero point displaced by DRF
T = Control zero point by means of RESET displacement
W = Workpiece zero point
F = Tool reference point

END OF SECTION

© Siemens AG 1990 All Rights Reserved

2 Defining Parameters

2.1 R parameters used

The measuring cycles use the following R parameters:

R00	to	R42	Transfer parameters
R40	to	R49	Display parameters, measurement result,
R50			correction value
R51	to	R99	Internal; are saved
R200	to	R219	Display parameters; measurement result
R220	to	R239	Internal
R488	to	R499	Internal

All R parameters are available for other tasks on completion of the measuring cycles.

2.2 R10 Offset number

2.2.1 R10 Tool offset memory number (workpiece measurement)

MDC determine whether or not the D number is known.

(See machine tool manufacturer's specifications)

2.2.1.1 D number known: Relative D number

R10 = 0 No automatic tool offset

R10= 8 7 6 5 4 3 2 1

3-digit D number

0/1 Length compensation
2 Radius offset

TOA range 0 current TOA range

0 Normal offset when machining from above
1 Inverted offset when mach. from below

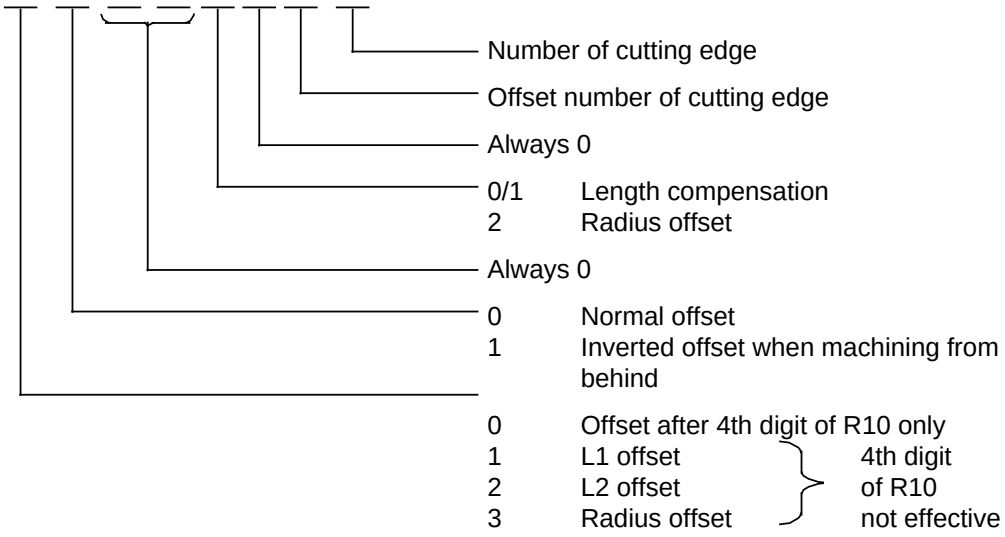
0 Offset only after 4th digit of R10
1 L1 offset
2 L2 offset
3 Radius offset

4th digit of R10 not effective

© Siemens AG 1990 All Rights Reserved

2.2.1.2 D number unknown

R8 = Extended T address
R9 = T address (tool number)
R10 = 0 No automatic tool offset
R10 = 8 7 6 5 4 3 2 1



2.2.2 R10 ZO memory number (ZO determination)

R10 =	0	ZO memory not automatically included in calculation
	1 ... 4	Automatic ZO memory inclusion in ZO G54 ... G57
	5	Automatic ZO memory inclusion in ZO G58
	6	Automatic ZO memory inclusion in the angle for coordinate rotation G58 (for angle measurement only)

The difference determined is

- included in ZO fine additive
- included in ZO coarse additive when MDC 804 bit 0 = 1 (ZO groups L960) is set.
- included in ZO coarse when measuring in JOG

2.3 R11 Empirical value / average value

MDC determine the number of empirical and average values. (See machine tool manufacturer's specifications)

Empirical values serve to suppress dimensional deviations **that do not follow a trend** (see explanation in Section 1.6.1).

R11 gives the number of the empirical value memory in the cycle setting data memory (SDC).

© Siemens AG 1990 All Rights Reserved

R11 = 0 Without empirical value, without average value memory
> 0 Number of empirical value memory

The average value memory number is likewise determined via R11 which reduces the time required for programming.

Special case:

When R11 is defined > 9999, R11 is evaluated as follows:

R11 = 8 7 6 5 4 3 2 1

Empirical value memory number (max. 0 to 99)

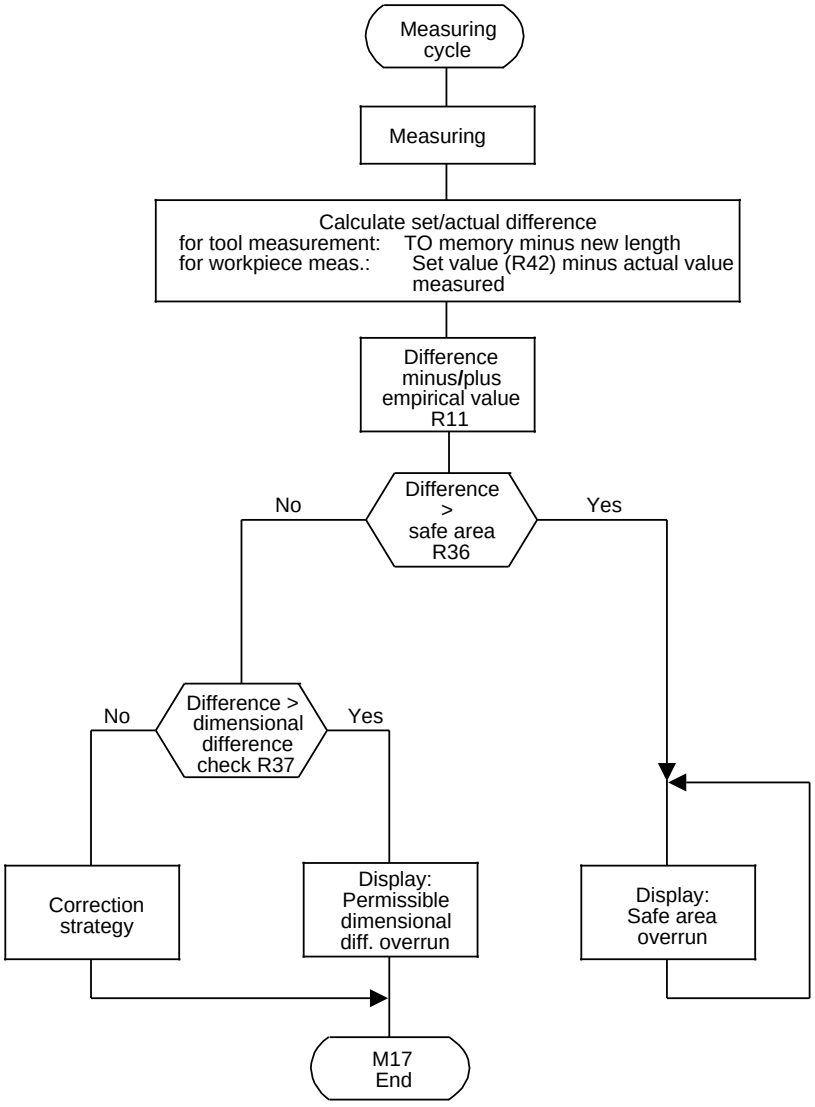
Average value memory number (max. 1 to 99)

Example: Empirical value memory number 12

Average value memory number 9

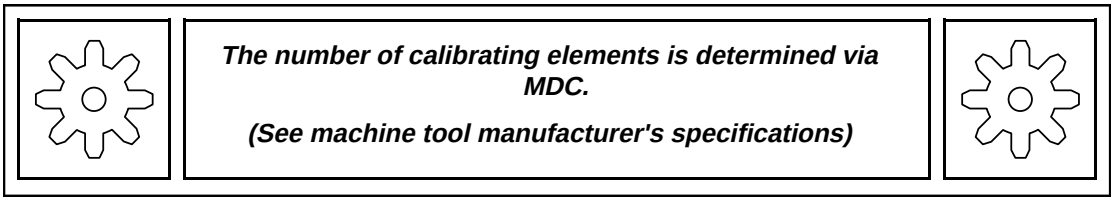
R11 = 90012

The following diagram is to illustrate the **effect of the empirical value R11** (see also outline flowchart in Section 1.5.4):



© Siemens AG 1990 All Rights Reserved

2.4 R12 Number of calibrating elements

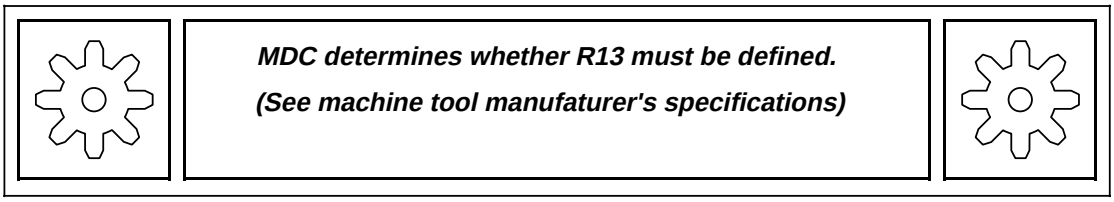


The number of calibrating elements is determined via MDC.

(See machine tool manufacturer's specifications)

The number of the calibrating element at which the workpiece probe is to be calibrated must be specified in R12.

2.5 R13 Compensating angle position for mono-directional probe, angular offset for driven tools



MDC determines whether R13 must be defined.

(See machine tool manufaturer's specifications)

Compensating angle position for mono-directional probe

When using a mono-directional probe, it may be necessary for machine-specific reasons (e.g. horizontal/vertical milling head) to compensate the position of the probe to be able to carry out the measurement. The determination in Section 1.3.3 (measuring direction/spindle position 0°) is no longer true because the milling head had been swivelled.

This wrong position can be compensated for by means of parameter R13. As a rule, R13=90° or a multiple thereof.

If the direction changes because the milling head has been swivelled, then R13 must be preset with -360° (usually 0°).

Angular offset for driven tools

For tool measurement with reversal of measurement (milling tools), the tool is automatically swivelled by 180 degrees for the 2nd measuring point.

If the 2nd measuring point cannot be measured in this angular position, for example with helical tooth T-slot mills, the angular position of the 2nd measuring point can additionally be displaced with parameter R13.

© Siemens AG 1990 All Rights Reserved

2.6 R22 Probe type/probe number

The number of probes is determined via MDC.

(See machine tool manufacturer's specification)

Workpiece measurement:

R22 = 3 2 1

Probe number
Probe type
0 = Multidirectional probe
1 = Monodirectional probe

Tool measurement:

R22 = 2 1

Probe number

2.7 R23 Measurement variant

The measurement variant of the various cycles is determined via parameter R23.

2.7.1 L972/L982 Tool measurement (T)

R23 = 4 3 2 1

0 Calibration
1 Measurement
2 Autom. measurement, depending on tool type (2nd decimal place of R23 effective only with this variant)

	Tool length in TO memory	
	before measurement	after measurement
0	PF	PF
1*)	PF	SF
2*)	SF	SF

*) *Function implemented with Version 3.2 and higher*

© Siemens AG 1990 All Rights Reserved

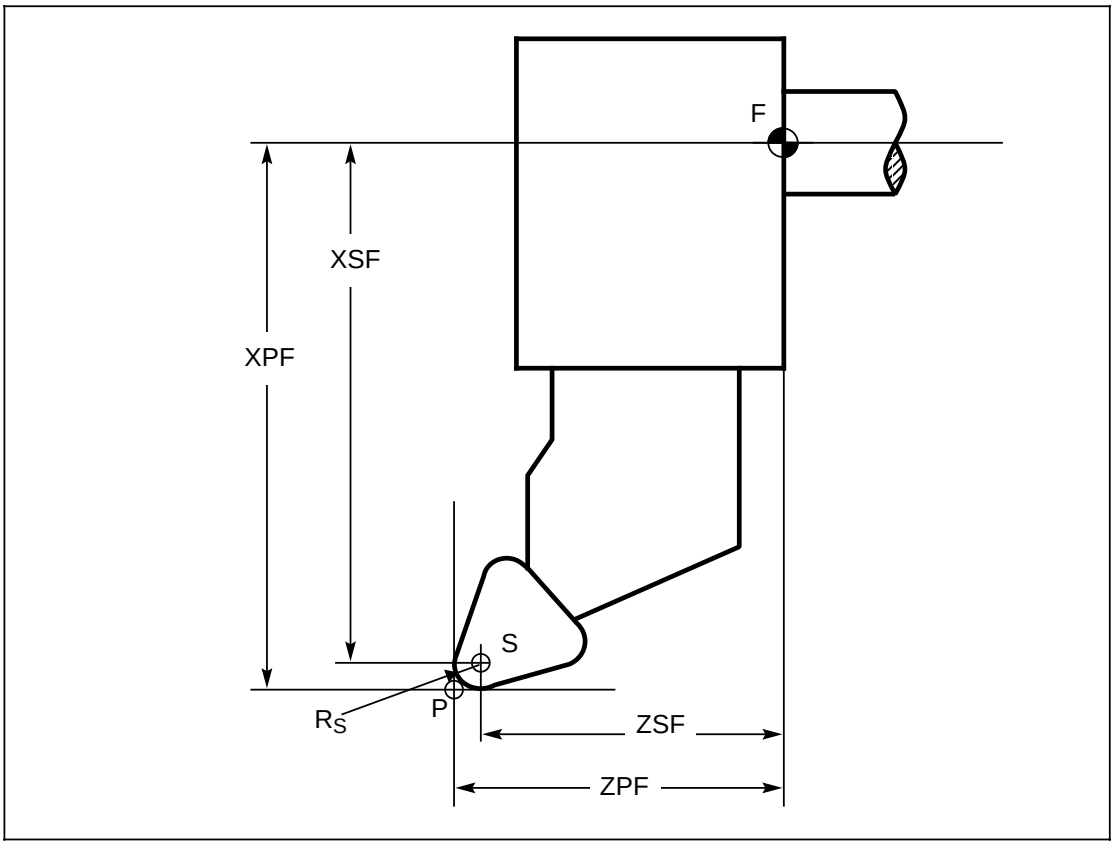


Fig. 2.1 Tool length definition

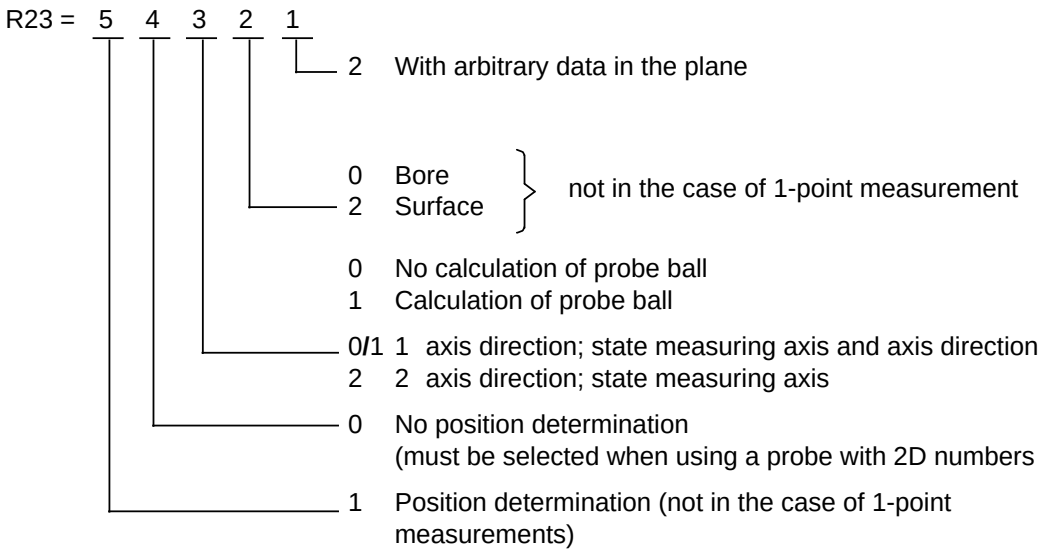
R23=4 3 2 1

		Possible with:
0	Measurement **also automatically** of L1 and/or L2 with tool types 1 ... 10, 26, 28, 31 ... 38 with one tool offset number	L972/L982
1	Measurement **automatically only** of L1 and/or L2 and R with tool types 28, 37, 38 with one tool offset number	L982 only
2	Measurement **automatically only** of L1 and/or L2 and R with tool types 28, 37, 38 with two tool offset numbers	L982 only

0	Compensate length and radius	if 3rd place of R23=1 or 2	L982 only
1	Compensate length only		L982 only
2	Compensate radius only		L982 only

© Siemens AG 1990 All Rights Reserved

2.7.2 L973 Calibrating the workpiece probe (T)



2.7.3 L974 Workpiece measurement (T)

R23 = 0 1-point measurement (ZO determination)
21 1-point measurement
22 1-point measurement with 180° reversal (workpiece)
23 2-point measurement on diameter: external measurement with R18/R19
internal measurement without R18/R19
24 2-point measurement on diameter: internal measurement **only** with R18/R19
25 Multi-point measurement on circumference
26 Multi-point measurement on cylinder

© Siemens AG 1990 All Rights Reserved

2.7.4 L976 Calibrating the workpiece probe (M)

R23 = 5 4 3 2 1

1:
- 0 With reference data in the plane (not in the case of probes with ZD numbers)
- 1 With reference data in the applicate
- 2 With arbitrary data in the plane
- 3 With arbitrary data in the applicate

2:
- 0 Bore
- 2 Surface
- 3 SLOT

} not in the case of 1-point measurement

3:
- 0 No calculation of probe ball
- 1 Calculation of probe ball

4:
- 0 4 axis directions with NC type 2
- 1 1 axis direction; state measuring axis and axis direction
- 2 2 axis directions; state measuring axis

5:
- 0 No position determination
- 1 Position determination

2.7.5 L977/L979 Workpiece measurement (M)

R23 =
- 1 Measure hole
- 2 Measure shaft
- 11 Measure slot
- 12 Measure web
- 21 ZO determination in the hole
- 22 ZO determination at the shaft
- 31 ZO determination in the slot
- 32 ZO determination at a web

2.7.6 L978 Workpiece measurement (M)

R23 =
- 0 1-point measurement (ZO determination at a surface)
- 1 1-point measurement
- 2 Multi-point measurement paraxial
- 3 Angle measurement
- 10 ZO determination at a surface with differential measurement *)
- 11 Measure surface with differential measurement *)
- 12 Multi-point measurement paraxial with differential measurement *)
- 13 Angle measurement with differential measurement *)

2.7.7 L981 Workpiece measurement (T, M)

R23=
- 1 Searching for a hole/slot on the end face
- 2 Searching for a hole/slot at the cylinder

*) *Function implemented with Version 3.2 and higher*

© Siemens AG 1990 All Rights Reserved

2.8 R25 Variable measuring speed

The measuring speed can be selected arbitrarily with R25. A feed rate of 150 mm/min is preset as standard when R25 = 0. This value increases automatically to 300 mm/min, when the measurement path a is modified via R28 (R28>1). The maximum measuring speed should be selected to be such that safe deceleration within the probe deflecting path is ensured (see Section 1.5.2, s-t diagram).

2.9 R27 Multiple measurement at same location

The number of measurements at the same location can be determined with R27. The set/actual difference D is determined arithmetically.

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

n = Number of measurements

Note:

Make sure that the permissible R parameter value of +- 99 999.999 is not exceeded, because with multiple measurement actual values are summed up.

2.10 R28 Multiplication factor for measurement path 2a

Normally, the path increment a is 1 mm. It can, however, be increased when calling measuring cycles with parameter R28. The maximum value for R28 is calculated as follows:

$$R28_{max} = \frac{\text{Axis traversing path}_{max}}{2}$$

2.11 R29 Weighting factor k for averaging

Different weighting can be given to an individual measurement with weighting factor k. Hence, the new tool compensation is only partly influenced by an new measurement result, depending on k (see Section 1.6.2).

© Siemens AG 1990 All Rights Reserved

2.12 R30 Number of measuring axis

The axis number (1 ... 3) in the coordinate system (plane) must be specified with R30 (not the hardware axis number).

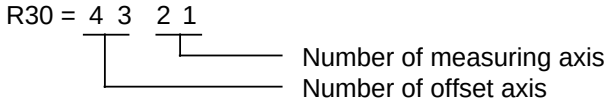
Measuring axis abscissa	R30 = 1
Measuring axis ordinate	R30 = 2
Measuring axis applicate	R30 = 3

Example:

An NC machine has the following axis configuration: X1, Y1, Z1, X2, Y2, Z2.
This results in the following programming:

```
%MPF...
:
:
G16         X2=  Y2=  Z2=                 Define measuring plane
...         R30=1...                      Measuring axis X2
...         R30=2...                      Measuring axis Y2
...         R30=3...                      Measuring axis Z2
:
G16         Z1=  X2=  X1=                 Define measuring plane
...         R30=1...                      Measuring axis Z1
...         R30=2...                      Measuring axis X2
...         R30=3...                      Measuring axis X1
            :
            :
            M30
```

R30 must be defined with offset axis**/**measuring axis for certain measurement variants.



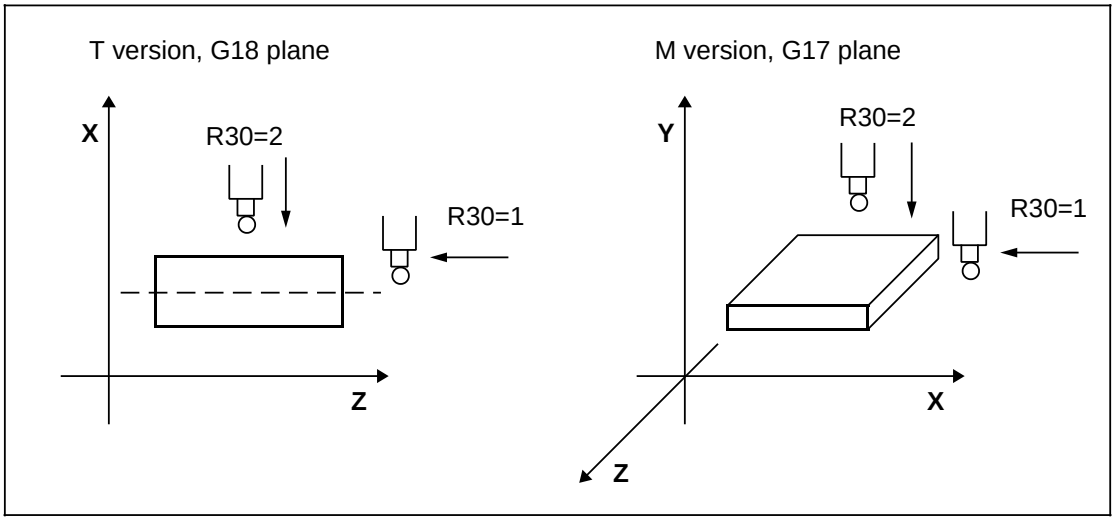
Note:

With a T version or G18 plane, the MDC 7002.1=axes not exchanged must be observed.

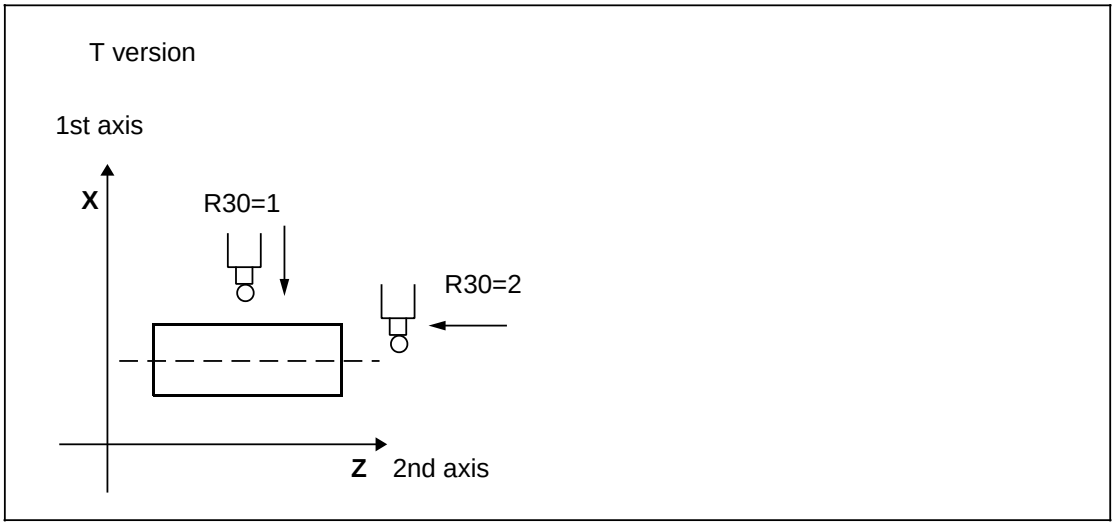
© Siemens AG 1990 All Rights Reserved

The definition of R30 can be modified via MD/MDC.
(See machine tool manufacturer's specifications)

Axis definition via R30 in accordance with DIN 66217:



Axis definition via R30 (MDC 7002.1=0, axes exchanged):



2.13 R33...R37 Tolerance parameters

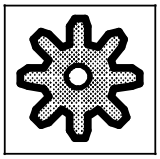
See Section 1.7

END OF SECTION

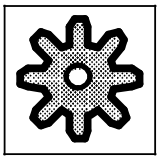
© Siemens AG 1990 All Rights Reserved

3 Measuring Cycle Auxiliary Programs

Some subroutines are called in the measuring cycles which can also be employed directly by the user.



Which programs can be used depends on the machine configuration and can also be partly determined via MDC.
(See machine tool manufacturer's specifications)



Cycle	Function
L931	Auxiliary cycle for tool or workpiece measuing cycles
L932	Check MDC and transfer parameters
L933	Auxiliary cycle for tool or workpiece measuring cycrles
L934	Auxiliary cycle for tool or workpiece measuring cycles
L935	Measuring result display selection
L936	Measurement abort
L937	Auxiliary cycle for workpiece measuring cycles
L938	Auxiliary cycle for tool or workpiece measuring cycles
L939	Auxiliary cycle for tool measuring cycles
L960	Transfer of ZO data blocks
L961	Additive input of empirical values
L962	Erase program empirical values/average values
L963	Auxiliary cycle for workpiece measuring cycle L979
L964	Auxiliary cycle for workpiece measuring cycles L973/L976
L965	Measuring plane (tool or workpiece measurement)
L966	Auxiliary program for operator guidance macro
L967	Presetting of transfer parameters
L969	Coordinate rotation
L970	Prepositioning
L971	Auxiliary cycle for tool measuring cycles
L980	Auxiliary cycle for workpiece measuring cycle L981
L988	Auxiliary cycle for workpiece measuring cycles
L989	Auxiliary cycle for workpiece measuring cycle L979

These programs **must be available**

Additional programs: Facilitate programming and operating.

© Siemens AG 1990 All Rights Reserved

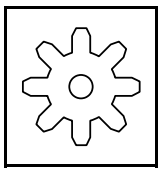
3.1 Internal measuring cycle auxiliary programs

The ”internal” auxiliary programs are called directly by the measuring cycles. They can therefore not be run when called directly by the user.

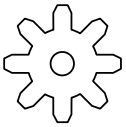
3.1.1 L931 Auxiliary cycle for tool or workpiece measuring cycles

This cycle must be available in either the part program memory or in the UMS, because it is used by all measuring cycles.

3.1.2 L932 Check MDC and transfer parameters *)



This cycle is not absolutely necessary for measurement. It may be therefore that the program is not available in the memory. (See machine tool manufacturer's specifications)



This cycle checks the MDC or the transfer parameters. It is called by the measuring cycles automatically, if SDC 800 bit 0 or 800 bit 1 is set.

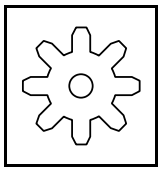
3.1.3 L933 Auxiliary cycle for tool or workpiece measuring cycles

This cycle must be available in either the part program memory or in the UMS, because it is used by all measuring cycles.

3.1.4 L934 Auxiliary cycle for tool or workpiece measuring cycles

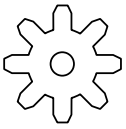
This cycle must be available in either the part program memory or in the UMS, because it is used by all measuring cycles.

3.1.5 L935 Measurement result display selection



This function depends on the machine configuration.

(See machine tool manufacturer's specifications)



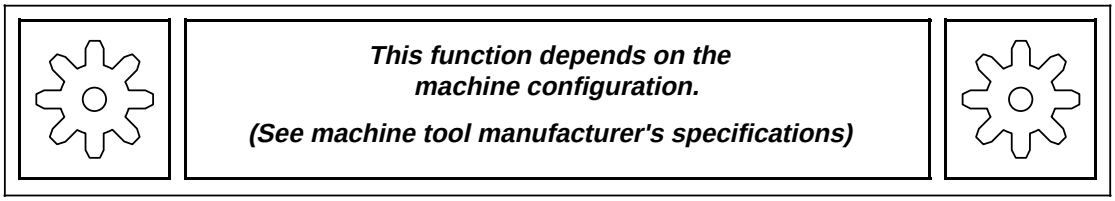
If measurement result displays are stored in the UMS, these are displayed automatically at the end of the measuring cycle.

---

*) *Version 3.0 and higher*

© Siemens AG 1990 All Rights Reserved

3.1.6 L936 Measurement abort



This cycle is required for the function ”Measurement abort signal”. This function provides the possibility of continuing the machining program when a cycle reset alarm occurs.

The measurement abort signal is normally initiated by the operator by pressing a key on the machine control panel.

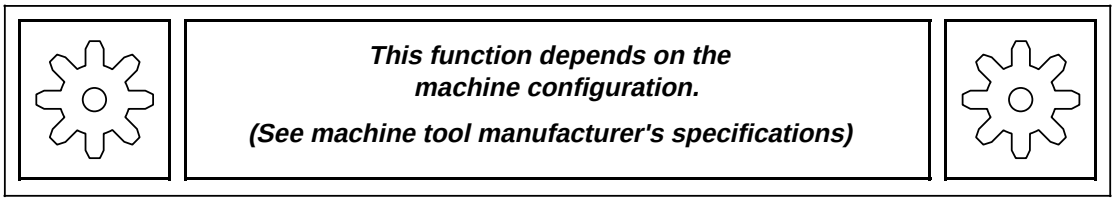
3.1.7 L937 Auxiliary cycle for workpiece measuring cycles

This cycle must be available in either the part program memory or in the UMS, because it is used by all measuring cycles.

3.1.8 L938 Auxiliary cycle for tool or workpiece measuring cycles

This cycle must be available in either the part program memory or in the UMS, because it is used by all measuring cycles.

3.1.9 L939 Auxiliary cycle for tool measuring cycles



This cycle processes the PLC interface of the tool management, when MDC 7000 bit 4 or 5 is set.

3.2 Internal/external measuring cycle auxiliary programs

Part of the ”external” cycles can also be called directly by the user.

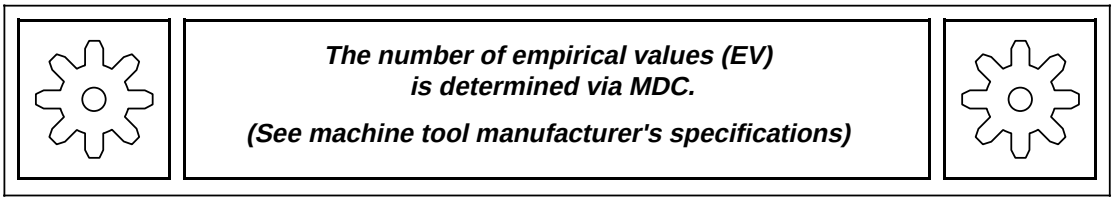
3.2.1 L960 Transfer of ZO data blocks (external)

The cycle is used to transfer ZO data from the ZO memory to the MIB or R parameter and vice versa.

It is supplied with the standard cycles and described in the associated documentation.

© Siemens AG 1990 All Rights Reserved

3.2.2 L961 Additive input of empirical values (external) *)



This subroutine enables a modification value to be entered in an ”Empirical value memory” (SDC).

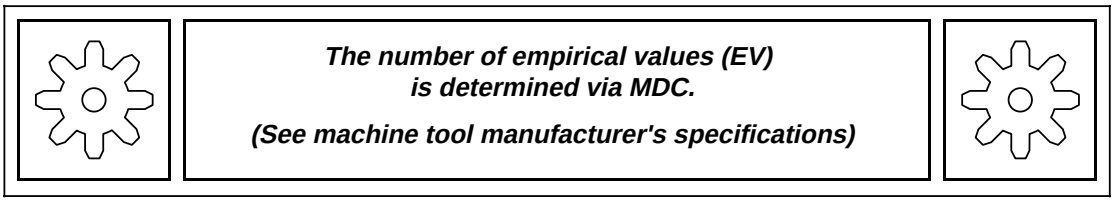
Transfer parameters for L961: R11 = EV memory number
R12 = Value

Example: EV is to be modified by 0.002.

Procedure:

- Select MDI mode
- R11=10 R12=0.002 L961 LF Press input key
- NC Start (without single block)
- Value of R12 is allowed for in EV memory number 10.

3.2.3 L962 Erase program EV/AV (external) *)



This program serves for erasing empirical and average values (AV) in the empirical value memory area or in the average value memory area (SDC). This may become necessary, for instance, when changing the part program.

Transfer parameters for L962: R40 = Start address
R41 = End address

Example: The empirical value memory numbers from 20 to 30 are to be erased.

Procedure:

- Select MDI mode
- R40=20 R41=30 L962 LF Press input key
- NC Start (without single block)
- EV 20 ... 30 are erased.
- Average values 20 ... 30 are also erased if there is an average value memory.

*) *Version 3.0 and higher*

© Siemens AG 1990 All Rights Reserved

3.2.4 L963 Auxiliary cycle for workpiece measuring cycle (external)

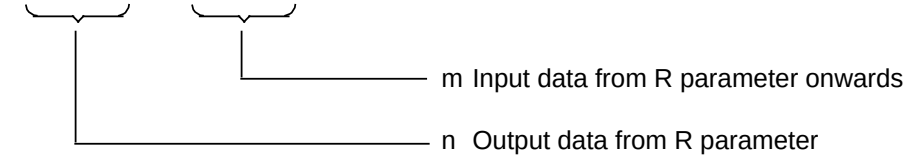
This cycle is called by measuring cycle L979.

It is used to calculate circle parameters (X_m, Y_m and r) from 3 or 4 coordinates.

It is configured in a way to allow also direct calls to be made.

Parameter definition for L963:

R0 =6 5 4 3 2 1



m Input data from R parameter onwards

n Output data from R parameter onwards

Output data:

R_n	Centrepoint abscissa
R_{n+1}	Centrepoint ordinate
R_{n+2}	Radius

Input data:

R_m	Number of circle points (3 or 4)
R_{m+1}	X1
R_{m+2}	Y1
R_{m+3}	X2
R_{m+4}	Y2
R_{m+5}	X3
R_{m+6}	Y3
R_{m+7}	X4
R_{m+8}	Y4

Execution time with 3 points depending on type of control up to approx. 3.5 s
4 points depending on type of control up to approx. 10 s

3.2.5 L964 Auxiliary cycle for workpiece measuring cycles (internal)

This cycle is called by cycles L973 and L976, when reference data are calibrated.

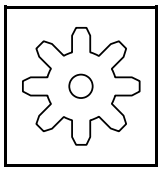
3.2.6 L965 Determination of measuring plane (external)

The cycle determines the axis data of the current plane and stores them internally. It must be started at least **once** on completion of installation.
Programming also becomes **necessary** when the measuring plane is changed or the function ”Mirroring” has been activated or deactivated.

Transfer parameters: None

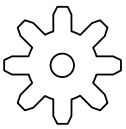
© Siemens AG 1990 All Rights Reserved

3.2.7 L966 Auxiliary program for operator guidance macro (OGM)



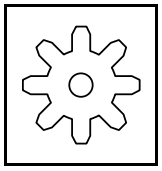
This function depends on the machine configuration.

(See machine tool manufacturer's specifications)



When editing parts of the measuring program via the input displays for measuring cycles, cycle L966 is cancelled when storing.

3.2.8 L967 Presetting of transfer parameters (internal)

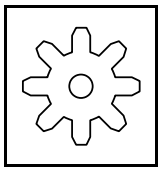


This cycle is normally provided by the machine tool manufacturer.

(See machine tool manufacturer's specifications)

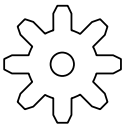
3.2.9 L969 Auxiliary cycle for coordinate rotation (internal)

This cycle must be available in either the part program memory or in the UMS, if measurement is to be carried out with coordinate rotation.



This function can be set via MDC.

(See machine tool manufacturer's specifications)



3.2.10 L970 Prepositioning cycle (external)

This cycle offers the possibility of positioning the tools around the tool probe or the workpiece probe from measuring point to measuring point with active collision monitoring.

The following parameters must be defined prior to call:

Parameters	Description
R23 = 0 = 1	Tool measurement (G53) Workpiece measurement
R25 = ... = 0	Positioning speed Rapid traverse/reduced rapid traverse (MDC4 / MDC5)
R30	Axis number of plane
R32	Approach position referred to machine zero point (R23=0) referred to workpiece zero point (R23=1) if axis is transverse axis in the diameter enter value as diameter value

© Siemens AG 1990 All Rights Reserved

Example of application:

In the examples given in Section 5.6.1, the NC blocks N10, N15, N30, N35, N40, N55, N60 and N65 must be monitored.

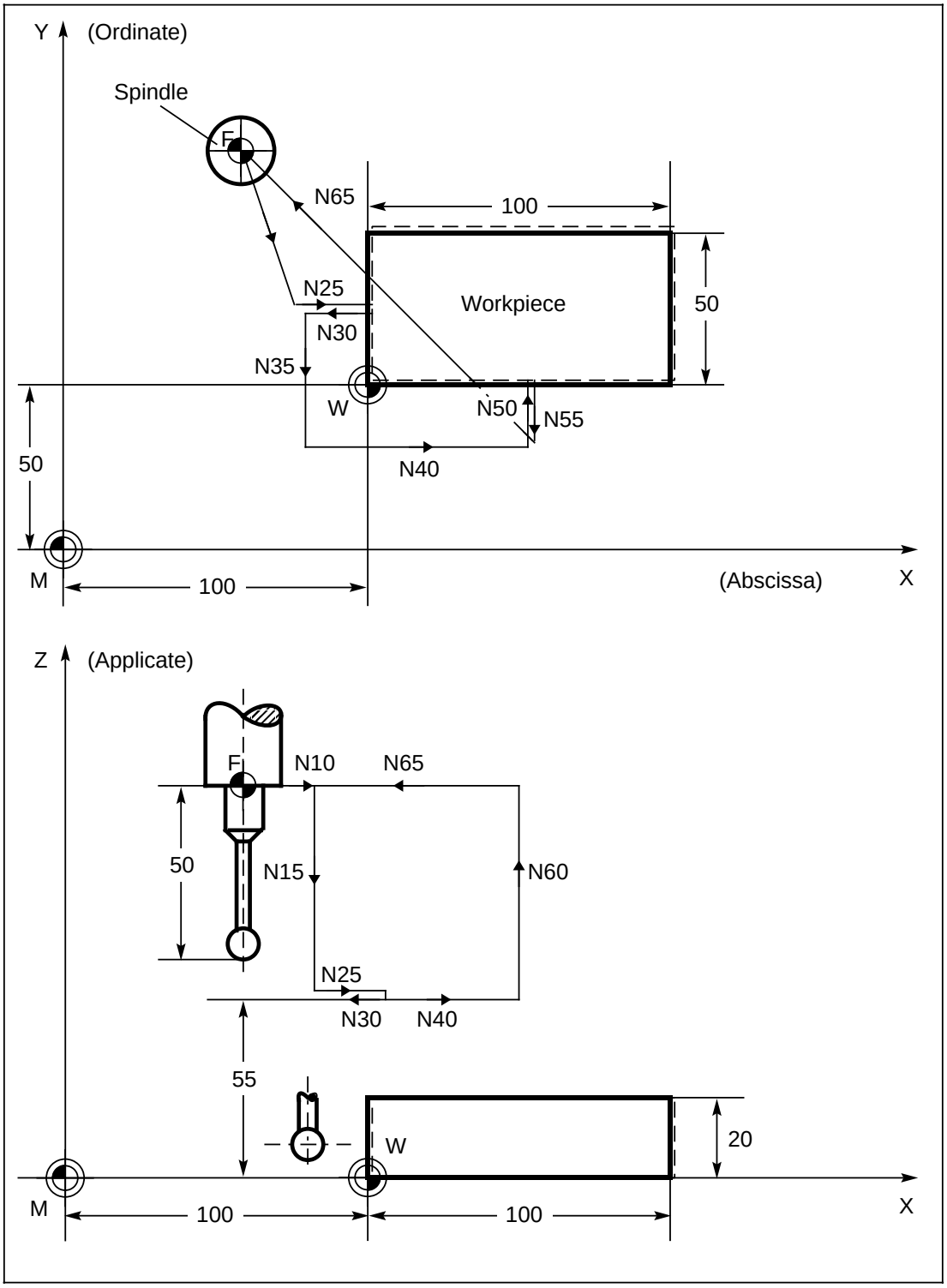


Fig. 3.1 ZO determination at surface

Example: ZO determination at a workpiece with cycle L978 in the X and Y axes without L970

Probe length (Z axis) in tool offset memory D99 (value 50)

```
%MPF    9782
N5      G54 T200                        T No. probe; ZO selection
N10     G00 G90 X-20 Y25          Position probe to face the measuring surface in X/Y
N15     Z10 D99                         Position probe in Z TO selection
N20     R10=1 R11=10 R22=1 R23=0  Define parameters for measuring cycle
        R25=0 R27=1 R28=2 R30=1
        R32=0 R36=3
N25     L978                            Cycle call for ZO determination in X axis
N30     G00 X-20                        Retraction in X axis
N35     Y-20                            Positioning in Y axis
N40     X50                             Positioning in X axis
N45     R11=11 R30=2              Define parameters for measuring cycle
N50     L978                            ZO determination in Y axis
N55     G00 Y-20                        Retraction in Y axis
N60     Z100                            Retraction in Z axis
N65     X-40 Y80                        Retraction in X/Y
:
Machining centre program
:
N900    M30
```

Example: ZO determination at a wokpiece with cycle L978 in the X and Y axes with L970

Probe length (Z axis) in the tool offset memory D99 (value 50)

```
%MPF    9783
N5      G54 T200                                          T No. probe; ZO selection
N10     R23=1 R25=0 R30 =1 R32=-20 L970                   Position probe to face
        R30=2 R32=25 L970                                 the measuring surface in X/Y
N15     R30=3 R32=10 D99 L970     Position probe in Z
                                                            TO selection
N20     R10=1 R11=10 R22=1 R23=0                          Define parameters for measuring cycle
        R25=0 R27=1 R28=2 R30=1
        R32=0 R36=3
N25     L978                                              Cycle call for ZO determination in
                                                            X axis
N30     R23=1 R30 =1 R32=-20 L970 Retraction in X axis
N35     R30=2 R32=-20 L970        Positioning in Y axis
N40     R30=1 R32=50 L970         Positioning in X axis
N45     R11=11 R23=0 R30=2 R32=0                Define parameters for measuring cycle
N50     L978                                              ZO determination in Y axis
N55     R23=1 R30 =2 R32=-20 L970 Retraction in Y axis
N60     R30=3 R32=100 L970                                Retraction in Z axis
N65     R30=1 R32=-40 L970        Retraction in X/Y
        R30=2 R32=80 L970
:
Machining centre program
:
```


© Siemens AG 1990 All Rights Reserved

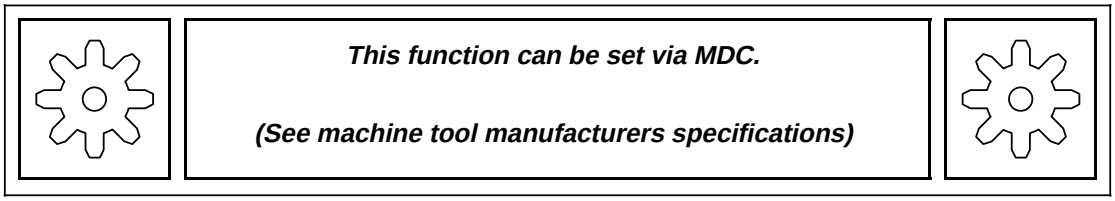
3.2.11 L971 Auxiliary cycle for tool measuring cycles (internal)

This cycle must be available in the part program memory or in the UMS for tool measurement.

3.2.12 L980 Auxiliary cycle for tool measuring cycle L981 (internal)

This cycle must be available in the part program memory or in the UMS when using measuring cycle L981 (searching for a hole/slot).

3.2.13 L988 Auxiliary cycle for workpiece measuring cycles (internal/external)



Measurement results obtained by the use of workpiece measuring cycles can be logged using the cycle L988 and the subfunction FB package ”Logging of measurement results”.

3.2.14 L989 Auxiliary cycle for workpiece measuring cycle L979 *)

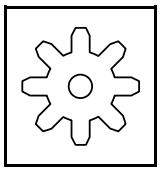
This cycle must be available either in the part program memory or in the UMS when using the workpiece measuring cycle L979.

END OF SECTION

*) *Version 3.0 and higher*

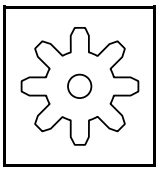
© Siemens AG 1990 All Rights Reserved

4 Measuring Cycles for Turning Machines



The possible scope of measurement depends on the machine configuration and can also be determined partly via MDC.

(See machine tool manufacturer's specifications)



Cycle	Function	
L931	Auxiliary cycle for tool or workpiece measuring cycles	See Section 3 for auxiliary cycles
L932	Check MDC and transfer parameters	
L933	Auxiliary cycle for tool or workpiece measuring cycles	
L934	Auxiliary cycle for tool or workpiece measuring cycles	
L935	Measurement result display selection	
L936	Measurement abort	
L937	Auxiliary cycle for workpiece measuring cycles	
L938	Auxiliary cycle for tool or workpiece measuring cycles	
L939	Auxiliary cycle for tool measuring cycles	
L961	Additive input of empirical values	
L962	Erase program EV/AV	
L964	Auxiliary cycle for tool measuring cycle L973	
L965	Measuring plane (tool or workpiece measurement)	
L966	Auxiliary program for operator guidance macro	
L967	Presetting of transfer parameters	
L970	Prepositioning	
L971	Auxiliary cycle for tool measurement	
L972	Tool measurement	Tool measuring cycle
L973	Calibrate workpiece probe	Workpiece measuring cycles
L974	Workpiece measurement	
L982	Tool measurement	
L988	Auxiliary cycle for workpiece measuring cycles	

These programs **must be available.**

Additional programs: facilitate programming and operating.

© Siemens AG 1990 All Rights Reserved

4.1 L972/L982 Tool measurement

Tool cycles with the following functions are available for tool measurement:

- L972 – Calibrate tool probe
  – Measure tool of L1 or L2 with tool types 1 ... 10, 26, 28, 31 ... 38 with one tool offset number
  – Automatic tool measurement of L1 and/or L2 with tool types 1 ... 10, 26, 28, 31 ... 38 with one tool offset number
- L982 – All functions of L972
  – Automatic tool measurement of L1 and/or L2 and R with tool types 28, 37, 38 with one or two tool offset numbers.
  Selection: Length and/or radius compensation

Cycle L982 needs approx. 45 % more memory space than L972 and its running time is approx. 20 % slower.

The 3rd and 4th decimal place of R23 are additionally used for coding the measurement variant of L982.

R23 = 4 3 2 1

			Possible with:
3rd decimal place	0	Measurement **also automatically** of L1and**/**or L2 with tool types 1 ... 10, 26, 28, 31 ... 38 with one tool offset number	– L972**/**L982
	1	Measurement **also automatically** of L1 and**/**or L2 and R with tool types 28, 37, 38 with one tool offset number	– L982 only
	2	Measurement **also automatically** of L1 and**/**or L2 and R with tool types 28, 37, 38 with two tool offset numbers	– L982 only
4th decimal place	0	Compensate length and radius	– L982 only
	1	Compensate length only	– L982 only
	2	Compensate radius only	– L982 only

With Version 3.2 and higher cycle L972 is available in two versions:

- L972-1 – Calibrate tool probe
  – Tool measurement of L1 or L2 with tool types 1 ... 8, PF-PF (basic function) **also automatically**
- L972-2 – Calibrate tool probe
  – Tool measurement of L1 or L2 with tool types 1 ... 10
  – also PF-SF; SF-SF for tool types 1 ... 8 **also automatically**
  – mirrored measurement
  – SF-SF for type 9 only

Cycle L982 is also available in two versions with Version 3.2 and higher:

- L982-1 – Tool measurement type 30-39 (basic functions)
- L982-2 – Tool measurement type 20-39 and mirrored measuring.

The version that you require must be loaded in the part program memory and/or linked into the user memory submodule.

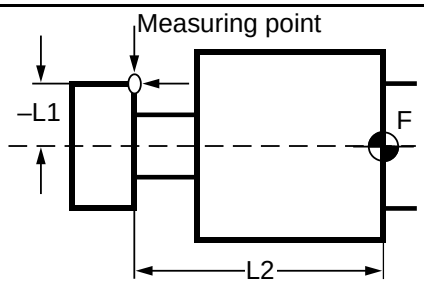
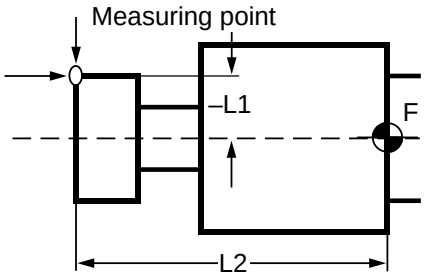
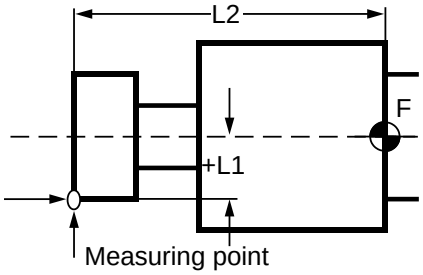
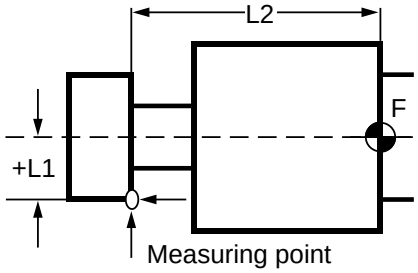
© Siemens AG 1990 All Rights Reserved

Description of tool types 26, 28, 31 ... 38

Tool types 31 ... 34, R23=00xx

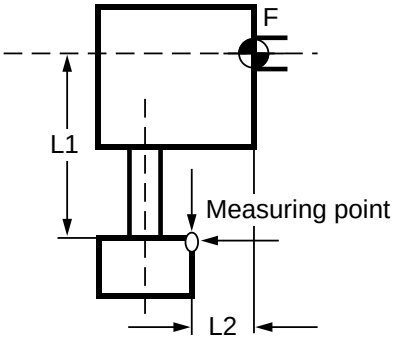
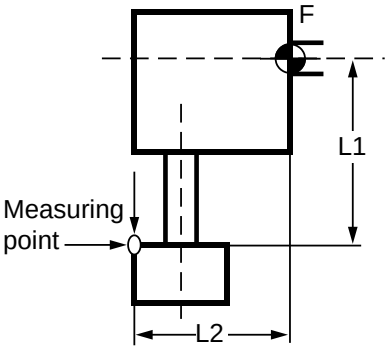
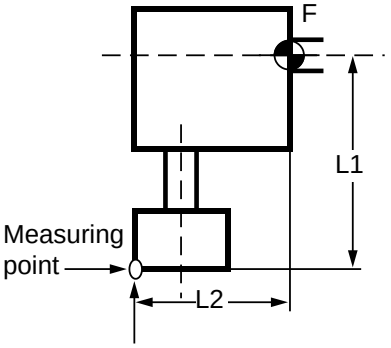
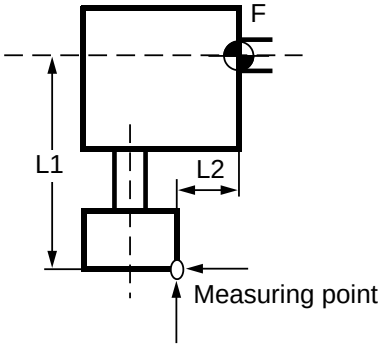
With tool types 31 ... 34, the lengths L1 and/or L2 are measured at the selected corner point of the milling cutter and compensated in the tool offset memory.

Axial tools

Tool type	Axial tool form	Tool geometry dimensions	Dimensions compensated
31		L1, L2, R=0	L1, L2
32		L1, L2, R=0	L1, L2
33		L1, L2, R=0	L1, L2
34		L1, L2, R=0	L1, L2

© Siemens AG 1990 All Rights Reserved

Radial tools

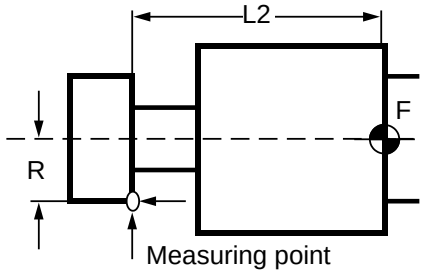
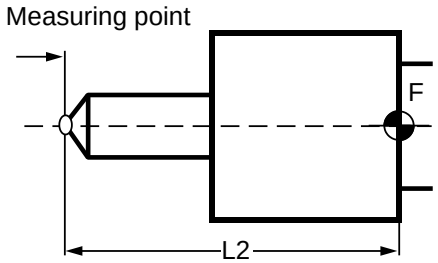
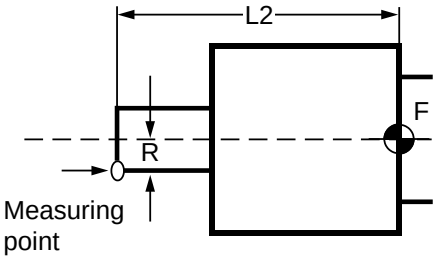
Tool type	Radial tool form	Tool geometry dimensions	Dimensions compensated
31		L1, L2, R=0	L1, L2
32		L1, L2, R=0	L1, L2
33		L1, L2, R=0	L1, L2
34		L1, L2, R=0	L1, L2

© Siemens AG 1990 All Rights Reserved

With tool types 35 and 37 only the length L2 and with tool types 26, 28, 36 and 38 only the length L1 is measured.

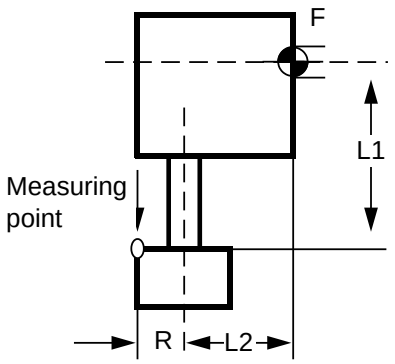
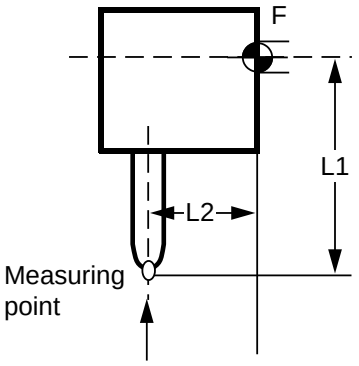
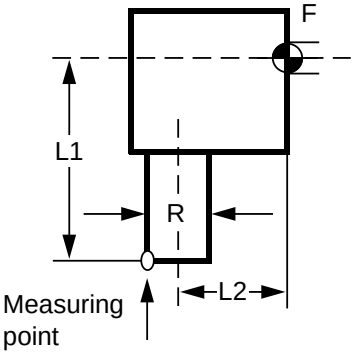
You select together with the cutter radius whether the measuring point is located on the tool centre (R=0) or displaced from the tool centre by the value R.

Axial tools

Tool type	Axial tool form	Tool geometry dimensions	Dimensions compensated
35		L1=0 L2, R	L2
37		L1=0 L2, R=0	L2
37		L1=0 L2, R	L2

© Siemens AG 1990 All Rights Reserved

Radial tools

Tool type	Radial tool form	Tool geometry dimensions	Dimensions compensated
26 36		L1, L2=0, R L1, L2, R	L1 L1
28 38		L1, L2=0, R=0 L1, L2, R=0	L1 L1
28 38		L1, L2=0, R L1, L2, R	L1 L1

© Siemens AG 1990 All Rights Reserved

Tool types 28, 37 and 38, R23=x1x2

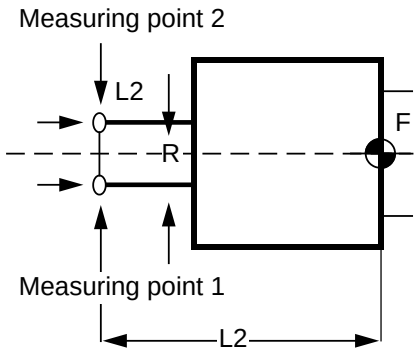
With tool type 28, L1 and R are determined.

With tool types 37 and 38, L1, L2 and R are determined.

The first measuring point is measured in the abscissa and ordinate. The tool is then turned through 180 degrees and the same cutting edge is measured again in the two axial directions.

The average values of the two measurements provide the L1 and L2 dimensions. The cutter radius is determined from half the dimensional difference between measuring point 1 and measuring point 2 in the ordinate.

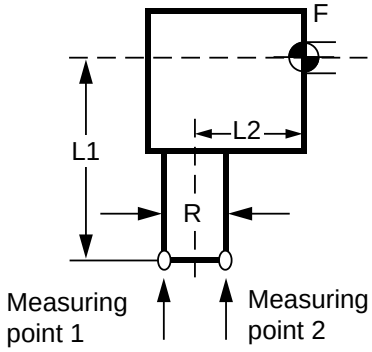
Axial tool

Tool type	Axial tool form	Tool geometry dimensions	Dimension compensations
37		L1=0 L2, R	L1, L2, R

Calculation of tool geometry:

L1=(L1,1+L1,2)/2 L2=(L2,1+L2,2)/2 R=ABS((L1,1 – L1,22)/2)

Radial tool

Tool type	Axial tool form	Tool geometry dimensions	Dimension compensations
28 38		L1, L2=0, R L1, L2, R	L1, R L1, L2, R

Calculation of tool geometry:

L1=(L1,1+L1,2)/2 L2=(L2,1+L2,2)/2 R=ABS((L2,1 – L2,2)/2)

© Siemens AG 1990 All Rights Reserved

Tool types 28, 37 and 38, R23=x2x2

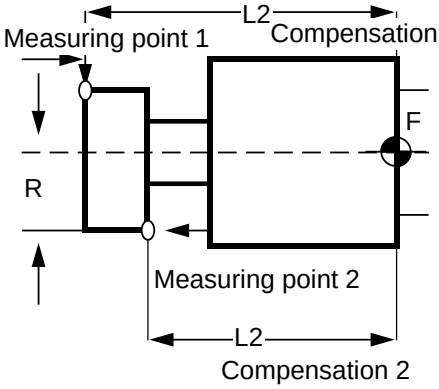
Using this varient, T slot cutters with two tool offset numbers can be measured.

With tool type 28, L1 and R of both tool offset numbers are determined.
With tool types 37 and 38, L1, L2 and R are determined.

Two consecutive offset numbers must be used. The 1st offset number is used for the greater tool length. The 2nd offset number (1st offset number+1) is called automatically by the cycle. In the tool geometry memory, the relevent tool dimensions L1, L2 and R must be stored in the two offset numbers. When tool management is used for the SINUMERIK 850 or 840/880, the 1st offset number must also be loaded first.

First, the 1st measuring point is measured in the absessor and ordinate. The tool is then swivelled by 180 degrees and the 2nd measuring point is measured at the same cutting edge in the two axial directions.

Axial tool

Tool type	Axial tool form	Tool geometry dimensions	Dimension compensation
37		Compensation 1 L1=0, L2, R Compensation 2 L1=0, L2, R	Compensation 1 L1, L2, R Compensation 2 L1, L2, R

The average value of the tool measurements in the ordinate provides tool dimension L1 for compensation 1 and 2.

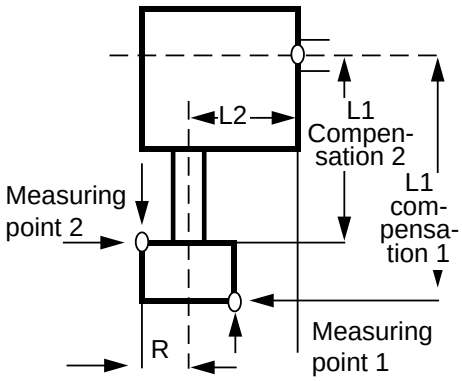
Tool dimension L2 is measured separately for each compensation and stored. The cutter radius is determined from half the dimensional difference between measuring point 1 and measuring point 2 in the ordinate.

Calculation of tool geometry:

1st offset number	2nd offset number
L1=(L1,1+L1,2)/2	L1=(L1,1+L1,2)/2
L2=L2 (compensation 1)	L2=L2(compensation 2)
R=ABS((L1,1 – L1,2)/2)	R=ABS((L1,1 – L1,2)/2)

© Siemens AG 1990 All Rights Reserved

Radial tools

Tool type	Radial tool form	Tool geometry dimensions	Dimension compensations
28 38		Compensation 1 L1, L2=0,R L1, L2, R Compensation 2 L1, L2=0, R L1, L2, R	Compensation 1 L1, R L1, L2, R Compensation 2 L1, R L1, L2, R

The tool dimension L1 is measured separately for each compensation and stored. The average value of the two measurements in the abscissa provides tool dimension L2 for the 1st and 2nd compensation.

The cutter radius is determined from half the dimensional difference between measuring point 1 and measuring point 2.

Calculation of tool geometry:

1st offset number	2nd offset number
L1=L1(compensation 1)	L1=L1(compensation 2)
L2=(L2,1+L2,2) **/** 2	L2=(L2,1+L2,2) **/** 2
R=ABS((L2,1 – L2,2) **/** 2)	R=ABS((L2,1 – L2,2) **/** 2)

R26 start angle position of driven tools.

If a cutter is to be measured at off-centre measuring points, the tool must be moved to a defined angular position to the measuring cycle. To achieve this, the tool must be preset to the correct position.

The starting angle is specified with R26.

R13 angular offset of driven tools when using reversal measurement.

When using reversal measurement, the tool is swivelled automatically by 180 degrees for the 2nd measuring point.

If measurement of the 2nd measuring point is not possible in this angular position, for example with helical-tooth T slot cutters, the angular position of the 2nd measuring point can be displaced additionally with parameter R13.

© Siemens AG 1990 All Rights Reserved

4.1.1 L972/L982 Calibrating the tool probe

Function and application

The cycle determines the current distances between machine zero point and probe trigger point with the aid of the calibrating tool and loads them automatically into the MDC area. Calculations are performed **without** empirical and average values.

The calibrating tool is located away from the measuring surface by R28 on completion of calibration.

Tool probes mirror-imaged in the ordinate or abscissa can also be calibrated with this cycle.

Mirror-imaging offers the advantage that probes can be fitted outside the machine work area. When calibrating, the calibration tool must be positioned in the turret rotated through 180 degrees under PLC control.

Preconditions:

- The side surfaces of the probe cube must be aligned parallel to the machining axes ordinate and abscissa.
- The calibrating axis must be programmed with **G53 and the TO memory number.**
- Start position as shown in Fig. 4.1. The measuring cycle calculates the start position automatically.

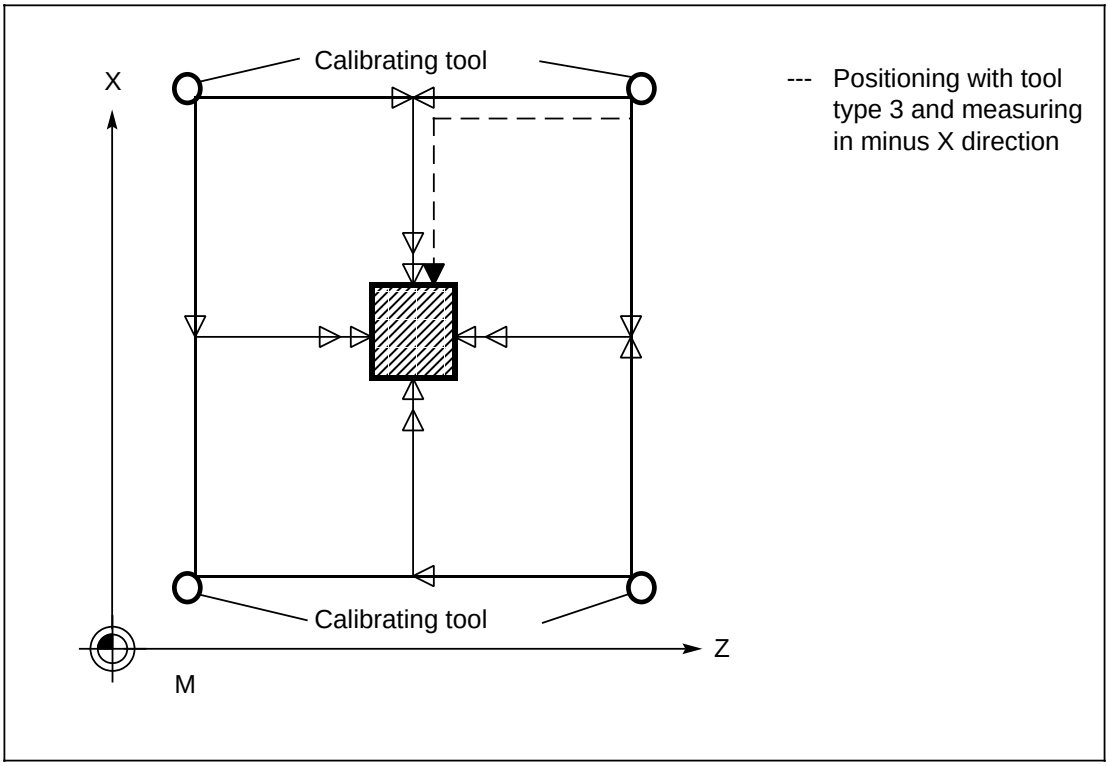


Fig. 4.1 Start position when calibrating the tool probe

The calibrating tool must be prepositioned to these positions before the calibrating cycle is called (start position).

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R22	Tool probe No. (see Section 2.6)
R23 = 0	Calibrate tool probe
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis (see Section 2.11)
R33	Zero offset range
R36	Safe area

See Section 8.2 for result display parameters.

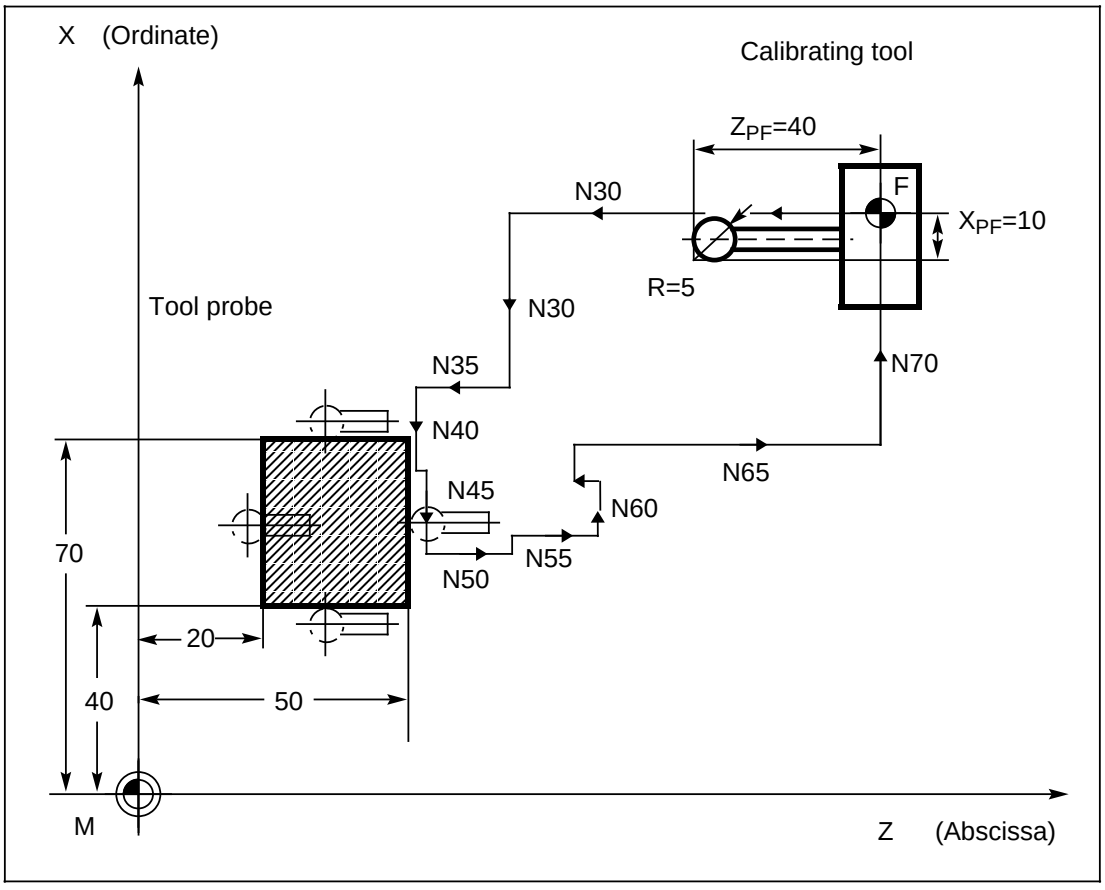


Fig. 4.2 Calibrating the tool probe

The tool probe is stationary but supplies switching signal. The calibrating tool is positioned with the turret.

© Siemens AG 1990 All Rights Reserved

Recommendation: Create a machine-specific calibrating subroutine using cycle L972/L982.

This program can be used when required either for each tool measurement using the main program or via MDI (once a day).

Example: Machine-specific calibration subroutine (see Fig. 4.2 for more details)

On turning machines, the calibration tool is treated like tool type 3 and must be entered as such in the TOA memory.

Values in the TOA memory D9:	Tool type	3
	L1 geometry	10
	L2 geometry	40
	R radius	5

The trigger points are always stored as radius values, even for a transverse axis.

Values in MDC:	MDC 1020=50	Minus direction of abscissa
	MDC 1021=20	Plus direction of abscissa
	MDC 1022=70	Minus direction of ordinate
	MDC 1023=40	Plus direction of ordinate

```
%SPF 10
N05  G00  G53  G94  Z300  G18                    Approach random change position
N10  G53  X240  T7  D9                           Random change position with calibrating
                                                 tool
N20  M71                                         Tool probe (e.g. swing-in)
N25  R22=1  R23=0  R25=0  R27=1  R28=1           Define parameters for calibration
     R30=1  R33=0,004  R36=2                     cycle
N30  L972                                        Calibrate in minus X direction
N35  G00  G53  Z52  T7  D9                       Approach new start position with G53
N40  R30=2  L972                                 Calibrate in plus Z direction
N45  G00  G53  X94                               Approach new start position with G53
N50  R30=1  L972  T7  D9                         Calibrate in plus X direction
N55  G00  G53  Z93                               Approach new start position with G53
N60  R30=2  L972  T7  D9                         Calibrate in minus Z direction
N65  G00  G53  Z300                              Approach random change position
N70  G53  X240                                   Approach random change position
N75  M17
```


© Siemens AG 1990 All Rights Reserved

4.1.2 L972/L982 Measure tool

Function and application

The cycle calculates the new tool length and checks whether the difference, after any necessary compensation by an empirical value, exceeds an amount defined in R36/R37 as compared with the old length. If this is not the case, the new tool length is loaded into the TOA memory under geometry, otherwise an alarm is triggered.

Compensation is also performed when the deviation exceeds a lower limit specified in R33.The old tool length is not modified if this is not the case. Such action serves to suppress random measuring errors.

The tools can also be measured mirror-imaged in the X or Z axis with this cycle (see also 4.1.1).

Averaging is not performed.

On completion of the cycle, the tool tip is now in a position facing the measuring surface by the amount R28.

Preconditions:

- The tool probe must have been calibrated
- Tool geometry data have been input in the TOA memory with tool nose radius and type.

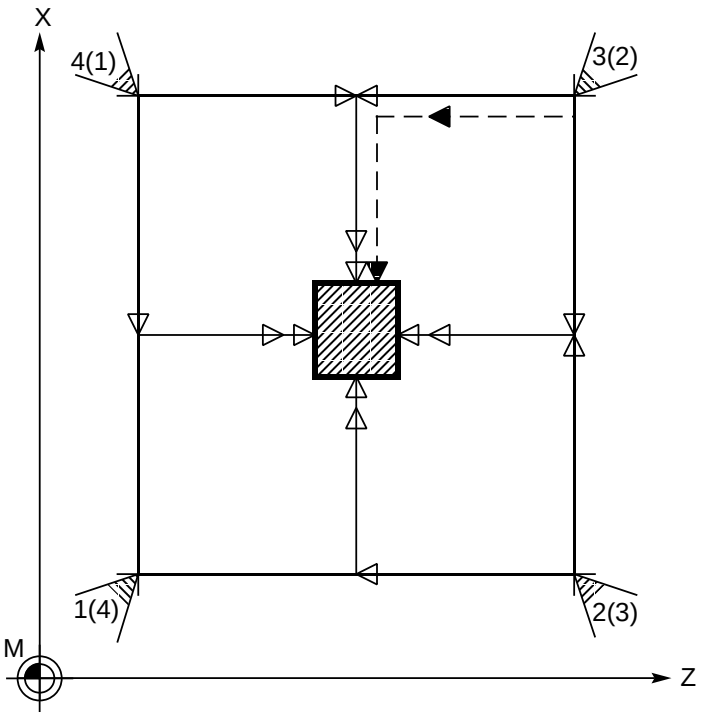
Example:

	T No.	Type	Geometry			Wear		Base (add. TO)	
	P0	P1	P2	P3	P4	P5	P6	P8	P9
D20	3	3	50.	25	1.	0	0	0	0

- The measuring axis must be programmed with G53 (deselect zero offset) and the relevant tool offset number.
- Start position as shown in Fig. 4.3. The cycle calculates the approach position automatically.
- MD 5007.6 Tool wear is not active
  = 0 Tool wear active
  The wear memory is cleared by the cycle, the derived offset value is added to geometry L1 or L2.

  = 1 Tool wear not active
  The wear is not deleted.
- If tool management is active (MDC 7000.4 = 1, 7000.5 = 1), the calculated difference is added to the TOA memory the first time the tool is used and the wear value is deleted, otherwise the calculated difference is added to the wear memory if it is active.

© Siemens AG 1990 All Rights Reserved



Example:
--- Positioning for tool type 3 and measuring in minus X direction

1, 2, 3, 4:
Tool type:
Machining **behind** turning centre

(1), (2), (3), (4):
Tool type:
Machining **in front of** turning centre

Fig. 4.3 Start positions for tool measurement

The tool must be moved to these positions before the measuring cycle is called.

The following parameters must be defined prior to call:

Parameters	Description
R11 = 0 > 0	Without empirical value Empirical value memory number (see Section 2.3)
R22	Tool probe No.
R23 = 1	Measure tool
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R26	Starting angle (with tool types 26)
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R36	Safe area
R37	Dimensional deviation check

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

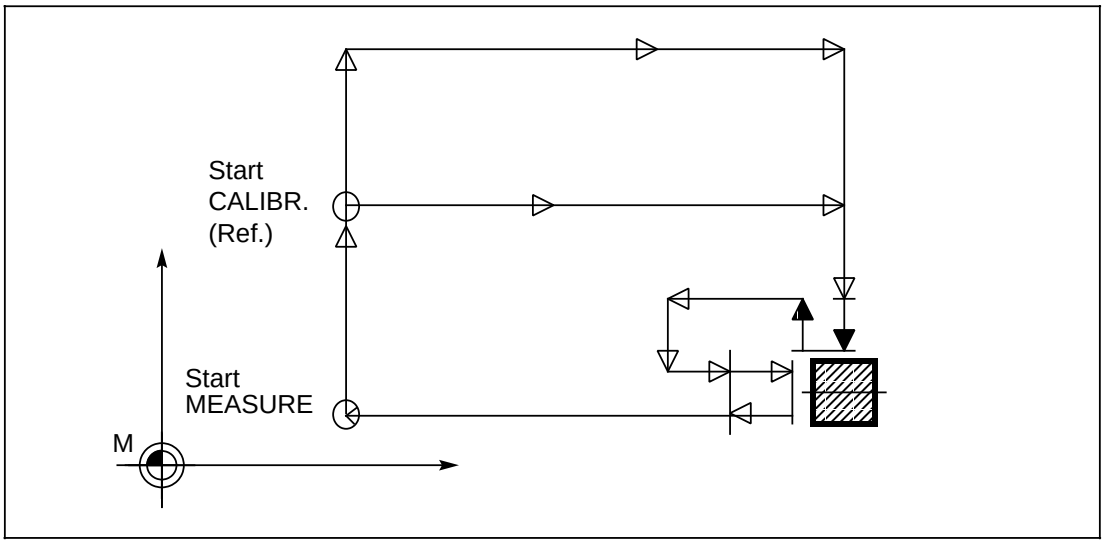


Fig. 4.4 Calibrate tool probe, measure tool (Z axis mirror-imaged)

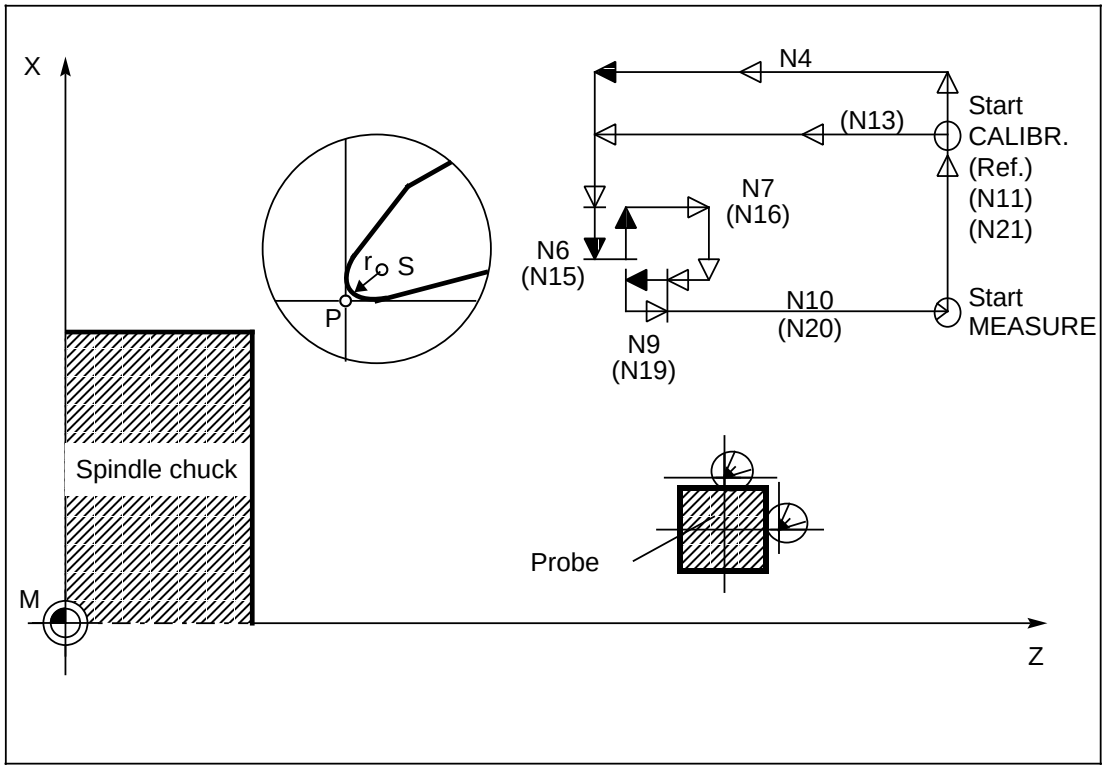


Fig. 4.5 Measure tool

© Siemens AG 1990 All Rights Reserved

Example: Calibrate tool probe and then measure a tool T3 as shown in Fig. 4.5

```
%MPF  9721
:
:
N3   G53  X595  G18
N4   G53  Z250  D9                                    Call calibrating tool

N5      R22=1  R23=0  R25=0  R27=2  R28=1
        R30=2  R33=0.004  R36=1                       Parameter definition for L972
N6      L972                                          1st call calibration cycle X axis
N7      G53  G0  Z200
N8      R30=1                                         Parameter definition
N9      L972                                          2nd call calibration cycle Z axis

                              L10

N10  G00  G53  Z520
N11  G53  X575
N12  T3                                               Call the tool to be measured
N13  Z250  D20
N14  R11=0  R22=1 R23=1 R25=0  R27=1  R28=1           Parameter definition
     R30=2  R33=0.004  R36=1  R37=0.8
N15  L972                                             Measure tool in minus X direction
N16  G00  Z400
N17  R30=1                                            Parameter definition
N19  L972                                             Measure tool  in minus Z direction

N20  G00  G53  Z250  D0
N21  G53  X560
M30
```

Subroutine L10 can be used instead of blocks N3 - N9 (machine-specific calibration program, see Section 4.1.1).

For further simplification, the tool measurements can be performed with the measurement variant ”Automatic tool measurement”.

This measurement variant is described in the following Section 4.1.3.

© Siemens AG 1990 All Rights Reserved

4.1.3 L972/L982 Automatic tool measurement

Function and application

The cycle positions the tool to the tool probe depending on the tool type used.
To begin with, the ”Tool change position” F, stored in the MDC or programmed under R18/19, is approached. The tool is then measured in the abscissa and subsequently in the ordinate (see Fig. 4.7).

When empirical values are used (R11 0), the specified empirical value memory number R11= m is used for the abscissa (see Section 2.1).

The cycle defines R11 = m+1 for the ordinate. Two consecutive memories are used.

$R11_{max-1}$ is specified for the constant when tool types 1-4 are used.

R11 = m+2 must be specified for the next tool.

Tool types 0, 5, 6, 7, 8, 10, 26, 28, 35 - 38 are measured in one axis only, if R23 = 00x2.
Tool type 9 cannot be measured with this variant.

The tool lengths are calculated according to ”PN” or ”SN”, i.e. referred to the tool tip or tool nose radius centre, depending on the definition of R23.

”N” is located at the position stored in the MDC on completion of measurement and between the measuring points 1 and 2 (with R23 102) if R18/19 has been defined with the value 0. In the case of R18/19 0, ”N” is positioned each time at a distance corresponding to the amount of R18/19 away from the tool probe.

Preconditions :

- The MDC (tool change position) must have been loaded.

  With these values, a position **”Tool change position” N** must be defined at which all tools which can possibly be used on this machine can be changed without collision at the tool probe.
- It must be possible to approach the change position without collision.
- The measuring axis must be programmed **with G53** and the associated **TO memory No.**

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R11 = 0 > 0	Without empirical value Empirical value memory number m
R13	Angular offset (with R23 102)
R18 = 0 0	Change position X axis (from MDC) (with reference to machine zero) Variable change position X axis (with reference to trigger points tool probe)
R19 = 0 0	Change position Z axis (from MDC) (with reference to machine zero) Variable change position Z axis (with reference to trigger points tool probe)
R22	Tool probe No.
R23	Measurement variant xxx 02 xxx 12 xxx 22 } see Section 2.7.1
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R26	Starting angle (with tool type 26)
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R33	Zero offset range
R36	Safe area

© Siemens AG 1990 All Rights Reserved

Example: Tool measuring program

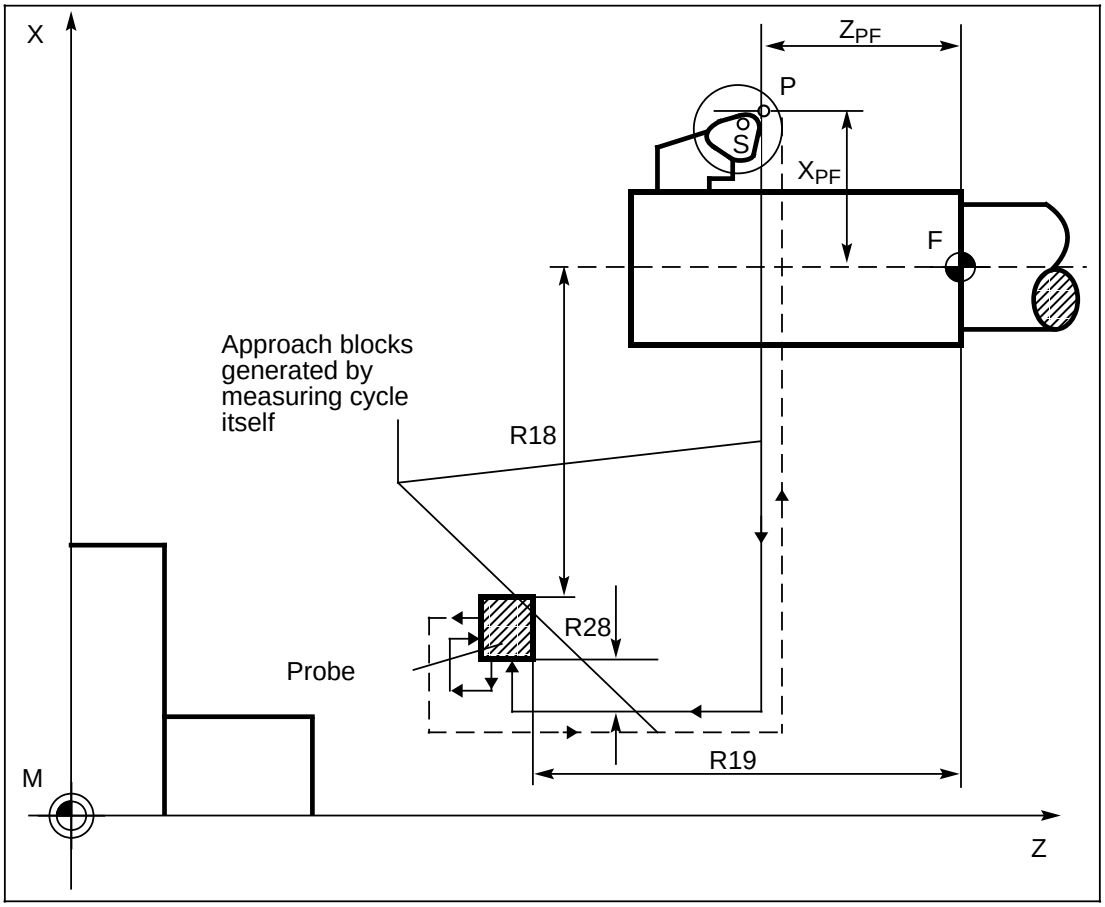


Fig. 4.6 Tool measurement towards spindle tool type 1

```
%MPF 9722
N5   L10  G18
N10  T9  D1
N15  R11=11  R18=0  R19=0  R22=1  R23=2
     R25=0  R27=1  R28=1
     R33=0.002  R36=1  R37=0.8
N20  L972

N25  T11
N30  R11=13  L972

N90  M30
```

Probe is calibrated (see Section 4.1.1)
Tool 9

R11=11: EV mem. no. 11 Z axis
EV mem. no. 12 X axis
R18=0: Change position X axis (MDC)
R19=0: Change position Z axis (MDC)
L972: Tool T9 is measured in the Z and X axes

Tool 11
R11=13: EV mem. no. 13 Z axis
EV mem. no. 14 X axis
L972: Tool T11 is measured in the Z and X axes

4.2 L973 Calibrating the workpiece probe

This cycle is used to calibrate the workpiece probe. Selection is made by definition of R23.

If double probes are used, L973 must always be called with the first of the two succesive D numbers.

The measuring variant is selected by defining R23:

Definition of R23	Calibrate workpiece probe
R23 = xxx02 R23 = xxx22	In any hole (plane) On any cube**/**surface (plane)

Probe types that can be used: (see Section 1.3)

- For turning machines, probe type 3, 5 - 8 must be entered in the TOA memory
- Multidirectional probe
- Bidirectional probe

© Siemens AG 1990 All Rights Reserved

4.2.1 L973 Calibrating on any surface

Function and application

This measuring cycle can be used to calibrate the probe at any surface, e.g. the workpiece. As from Version 3.5 the function "Mirroring" can also be active in the axes concerned.

After calibration has been completed, the probe is at a distance ”a” facing the calibration surface.

Preconditions :

- The probe is called with tool offset and **without** G53 to face the calibration surface.
- When calibrating in a hole (groove**/**slot) the probe must be prepositioned to the centre point.
- When calibrating beneath the turning centre in plus-X direction or left of the workpiece zero point W in plus-Z direction, the setpoint (R32) must be specified as a **negative** value.
- When calibrating in a random hole (groove**/**slot) the probe must be prepositioned to the centre point.

The following parameters must be defined prior to call:

Parameters	Description
R22	Workpiece probe number
R23 = 22	Calibrating the workpiece probe on any surface (see Section 2.7.2)
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis
R31 = 0 R31 = 1	Positive axis direction Negative axis direction
R32	Setpoint referred to workpiece zero point, for transverse axis in the diameter. When calibrating beneath the turning centre in plus-X direction or left of the workpiece zero point in plus-Z direction, the setpoint must be specified as a **negative** value. Where the measurement variant is any hole, R32 means diameter of hole, the approach position is the centre point.
R33	Zero offset range
R36	Safe area

© Siemens AG 1990 All Rights Reserved

Example: Calibrating a probe on any surface in minus-Z direction

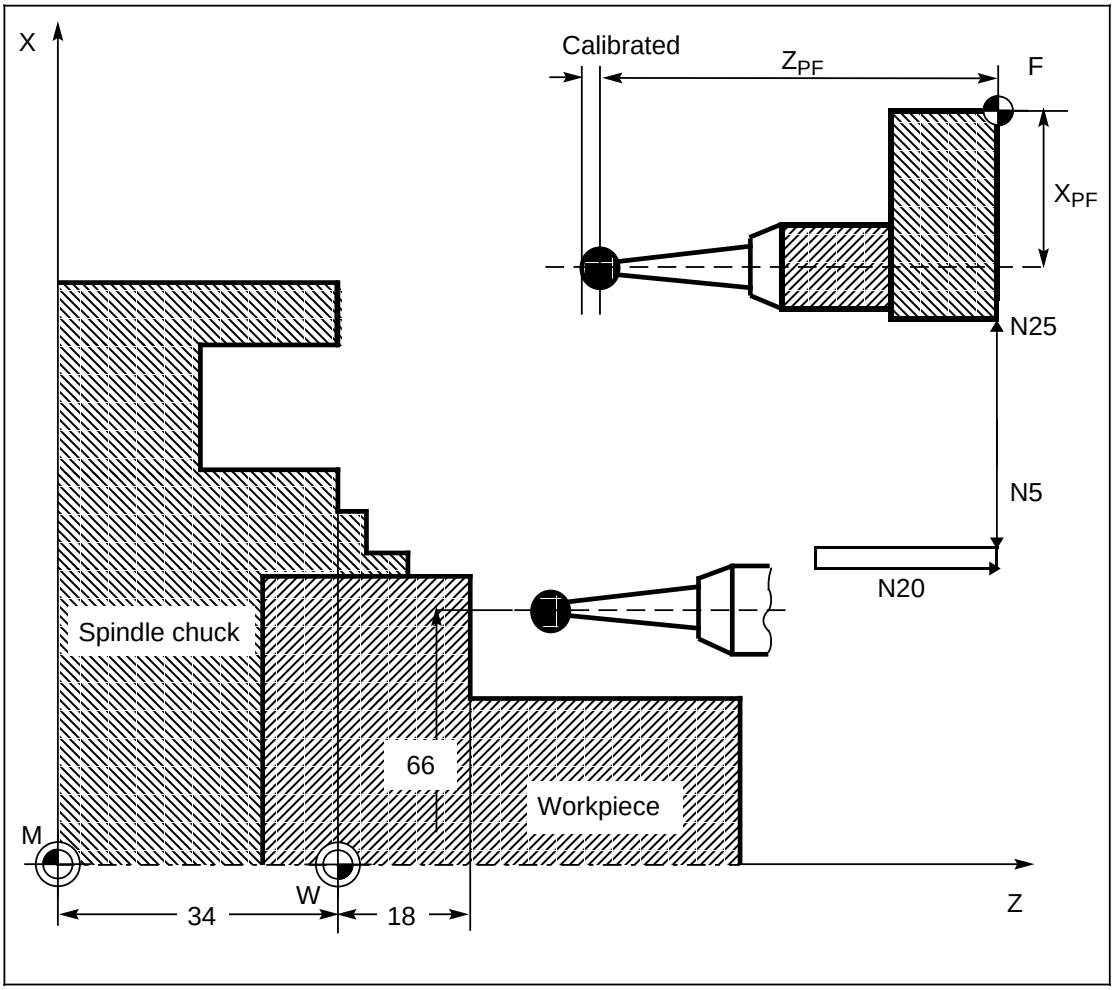


Fig. 4.7 Calibration on any surface

```
%MPF    9732
N1      T8                                          Probe call
N5      G00 G54 X66 D22 Z25 G18     Call probe type 7
N10     R22=1 R23=22                                Parameters for calibrating cycle
        R25=0 R27=1 R28=3 R30=2                 Calibrating in Z axis
        R31=1 R32=18 R33=0 R36=1
N15     L973
N20     G0 Z90 T0                                   Retraction
N25     X146
N30     M30
```


© Siemens AG 1990 All Rights Reserved

4.3 L974 Workpiece measurement

Function and application

The measuring cycle ascertains the actual value of the workpiece in the selected measuring axis with respect to the workpiece zero point and the set/actual difference. As from Version 3.5 the function "Mirroring" can also be active in the axes concerned.

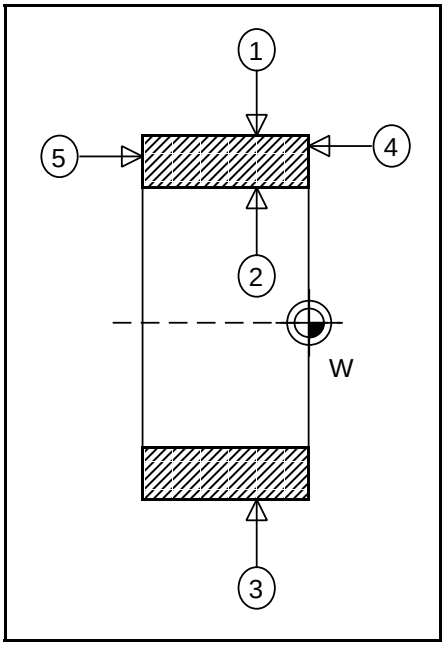
An empirical value stored in the SDC is then allowed for with the correct sign. The average value is formed over several parts and the tolerance bands are checked as described in Section 1.7.

The cycle corrects automatically the zero offset memory/compensation memory selected via R10 in the measuring axis (define tool type).

Measurements can be performed in all axial directions without restriction (inside/outside diameters, holes, slots).

Selection of the measurement variant is carried out via definition of R23:

R23 = 0	1-point measurement ZO determination
R23 = 21	1-point measurement
R23 = 22	1-point measurement with reversal
R23 = 23 R23 = 24	2-point measurement on diameter; inside without protection zone 2-point measurement on diameter; inside with protection zone
R23 = 25	Multi-point measurement on circumference
R23 = 26	Multi-point measurement on cylinder



Start position

1: - X Outside diameter
2: +X Inside diameter
3: +X Outside diameter
(Calibrating and measuring beneath turning centre:
Set value negative)
4: - Z Measure length
5: +Z Measure length:
(left of workpiece zero point in +Z direction:
Set value negative)

Fig. 4.8 Start positions

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

4.3.1 L974 1-point measurement ZO determination

Function and application

The measuring cycle ascertains the actual value of a blank in the selected measurement axis in relation to the workpiece zero point.

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

The probe is located at a distance ”a” facing the measuring surface.

No automatic entry is carried out or additive entry of the difference of the measuring axis is made in the specified ZO memory, depending on the definition of R10.

Preconditions:

- The workpiece must be positioned with M19 prior to cycle call when necessary.
- The probe must have been calibrated in the direction of measurement.
- The probe must be positioned with tool offset and without G53 facing the surface to be measured (Fig. 4.9).
- The maximum diameter that can be measured depends on the traversing range of the turret slide in +X.

Outside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} + 2a$$

Inside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} - 2a$$

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO entry Automatic ZO entry in ZO G54...G57 Automatic ZO entry in ZO G58
R11 = 0 > 0	Without empirical value Empirical value memory number (see Section 2.3)
R22	Probe number (see Section 2.6)
R23 = 0	ZO determination
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis (see Section 2.12)
R32	Set value referred to the workpiece zero point
R36	Safe area

© Siemens AG 1990 All Rights Reserved

Example: ZO determination (probe calibrated)

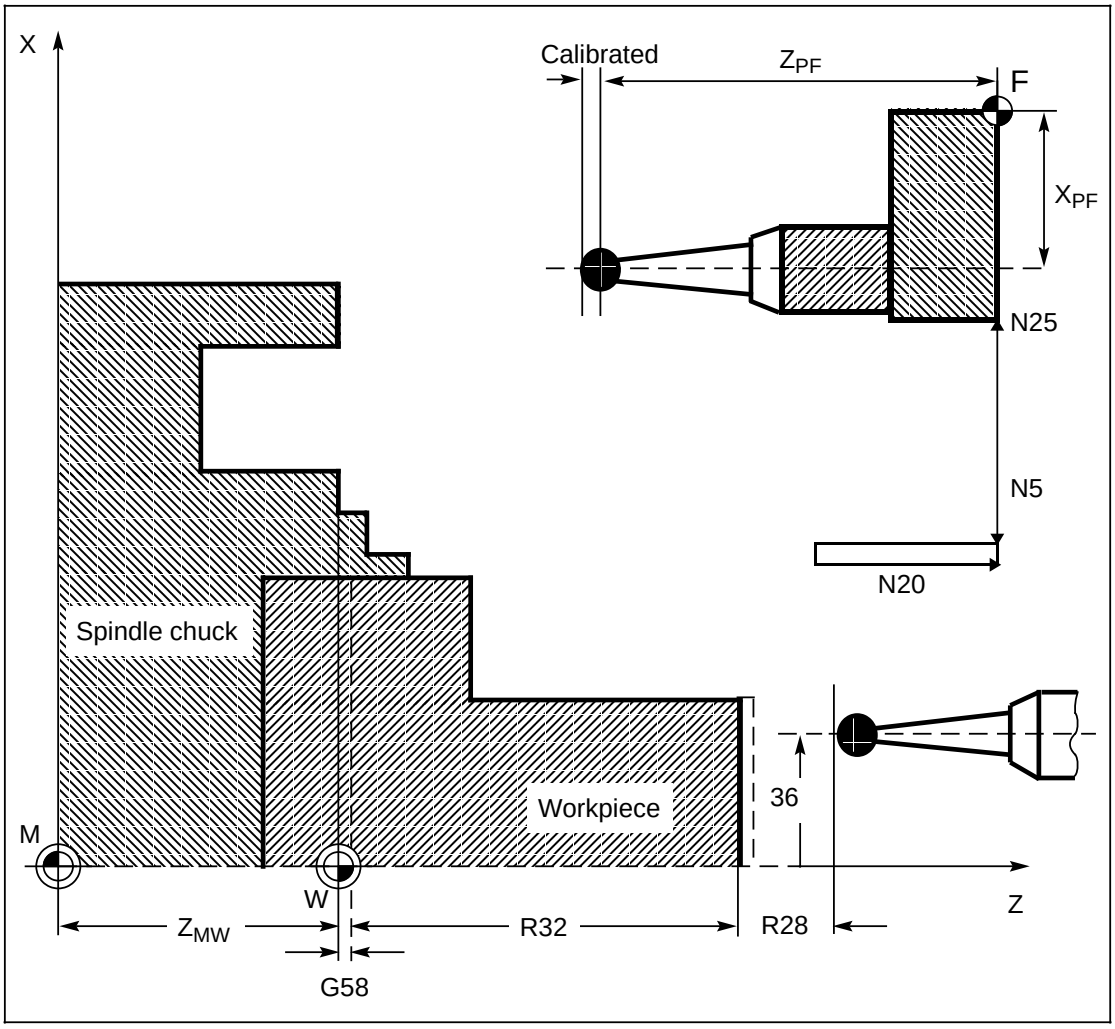


Fig. 4.9 ZO determination

```
%MPF    9733
N1      T8    G18                          Probe call
N5      G00 G54 X36 D22
N10     R10=5 R11=0 R22=1 R23=0            Parameters for measuring cycle
        R25=0 R27=1 R28=1 R30=1
        R32=60 R36=1
N15     L974                               Measuring in minus Z direction
N20     G0 Z110 D0                         Retraction
N25     X114
N30     M30
```

© Siemens AG 1990 All Rights Reserved

4.3.2 L974 1-point measurement

Function and application

The measuring cycle ascertains the actual value of the workpiece in the selected measuring axis in relation to the workpiece zero point and the set/actual difference.

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, the average value is formed over several parts and the tolerance bands are checked (Section 1.7.1).

The probe is located at a distance ”a” facing the measuring surface on completion of measurement.

No automatic compensation or a length compensation is carried out, depending on the definition of R10.

Preconditions:

- The workpiece must be positioned with M19 prior to cycle call when necessary.
- The probe must have been calibrated in the direction of measurement.
- The probe must be positioned with tool offset and without G53 to face the surface to be measured (Fig. 4.9).
- The maximum diameter that can be measured depends on the traversing range of the turret slide in +X.

Outside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} + 2a$$

Inside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} - 2a$$

© Siemens AG 1990 All Rights Reserved

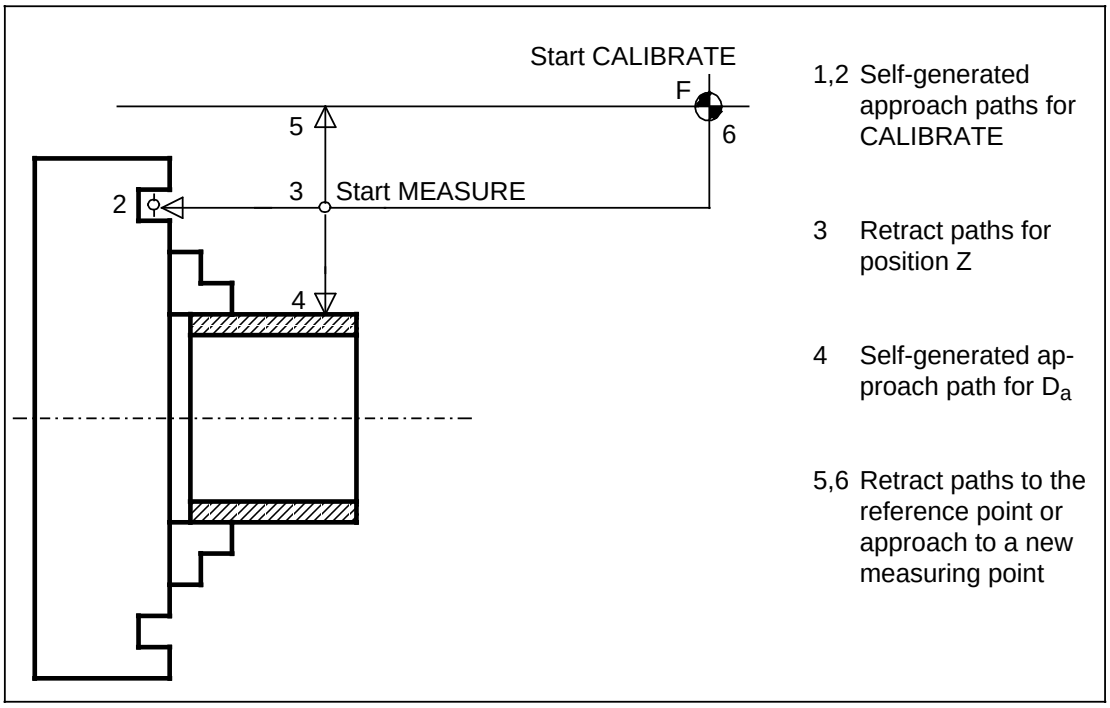


Fig. 4.10 1-point measurement on outside diameter with CALIBRATE

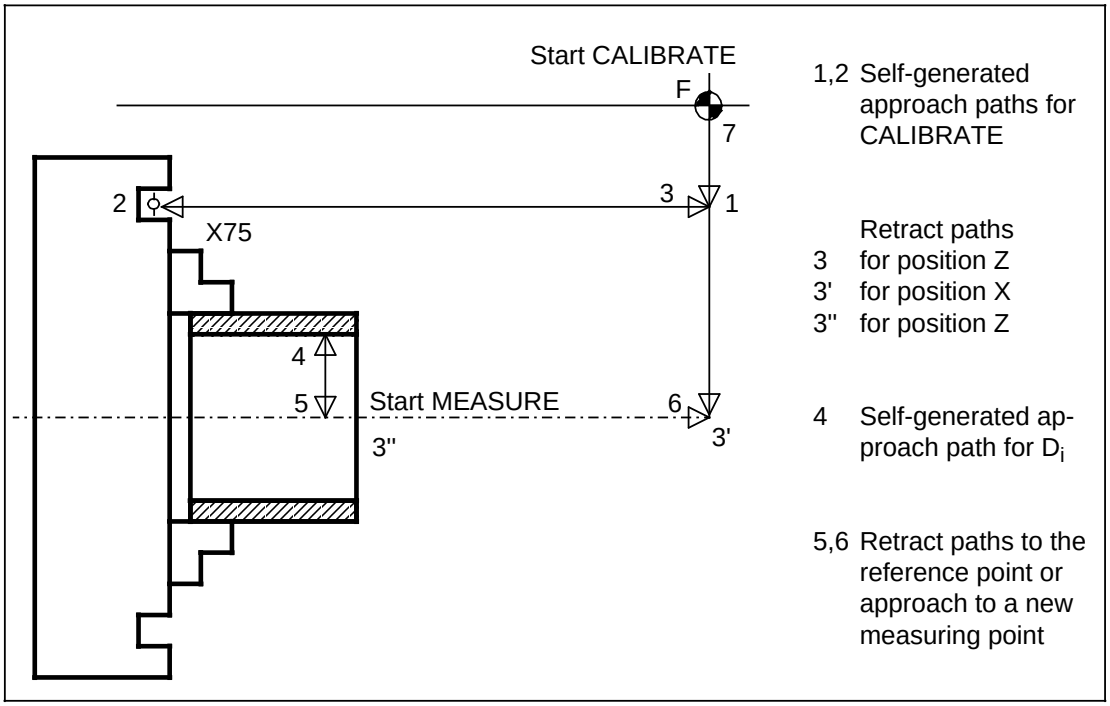


Fig. 4.11 1-point measurement on inside diameter with CALIBRATE

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory No./average value memory No. (see Section 2.3)
R22	Probe number (see Section 2.6)
R23 = 21	1-point measurement
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Average value compensation
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Calibrate workpiece probe, 1-point measurement on outside and inside diameter

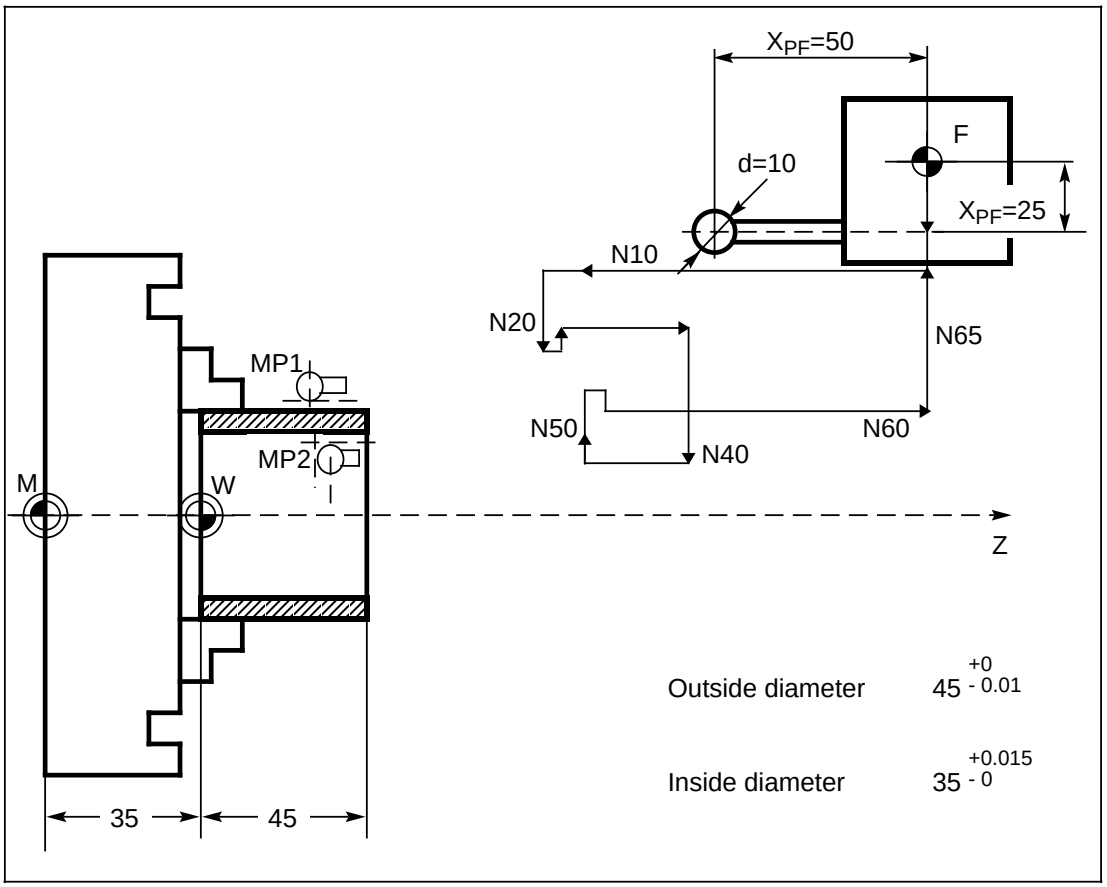


Fig. 4.12 1-point measurement on outside and inside diameter

```
%MPF    9741
N5      T1 G18                                          Call T No. with TO No.
N10     G54 G0 Z30 D31 X                                Preposition probe
N15     R10=8 R11=13 R22=1 R23=21 R25=0                 Define parameters for measurement
        R27=1 R28=1 R29=2 R30=2
        R33=0.002 R34=0.005 R36=0.5 R37=0.04
        R40=0 R41=-0.01 R42=45
N25     L974                                            Measure outside diameter
N30     G0 G54 Z60                                      Position probe to face MP2
N35     X0
N40     R10=9 R11=14 R22=1 R23=21 R25=0                 Define parameters for measurement
        R27=1 R28=1 R29=2 R30=2
        R33=0.002 R34=0.005 R36=0.5 R37=0.04
        R40=0.015 R41=0 R42=35
N50     L974                                            Measure inside diameter
N55     G0 G53 Z110  D0                                 Retraction
N60     G53 X90
N65     M30
```

© Siemens AG 1990 All Rights Reserved

4.3.3 L974 1-point measurement with reversal

Function and application

The measuring cycle ascertains the actual value of the workpiece in the selected measuring axis in relation to the workpiece zero point and the set/actual difference.

The workpiece is positioned to the value of R26 (start angle) prior to the first measurement using M19. The second measurement is preceded by a reversal through 180°. The measurement with reversal compensates any possibly existing eccentricity of the workpiece caused by a chuck with three jaws. The actual value and the set/actual difference are determined on the basis of the two measurements.

An empirical value stored in SDC is subsequently allowed for with the correct sign.

In addition, averaging is performed over several parts and the tolerance bands are checked (Section 1.7.1).

The probe is located at a distance ”a” facing the measuring surface on completion of measurement.

No automatic compensation or a length compensation is carried out, depending on the definition of R10.

Preconditions :

- The probe must have been calibrated in the direction of measurement.
- The probe must be positioned with **tool offset** and **G53** to face the surface to be measured (Fig. 4.9).
- The maximum diameter that can be measured depends on the traversing range of the turret slide in +X.
- M19 oriented spindle stop with axis movement must be defined.

Outside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} + 2a$$

Inside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} - 2a$$

© Siemens AG 1990 All Rights Reserved

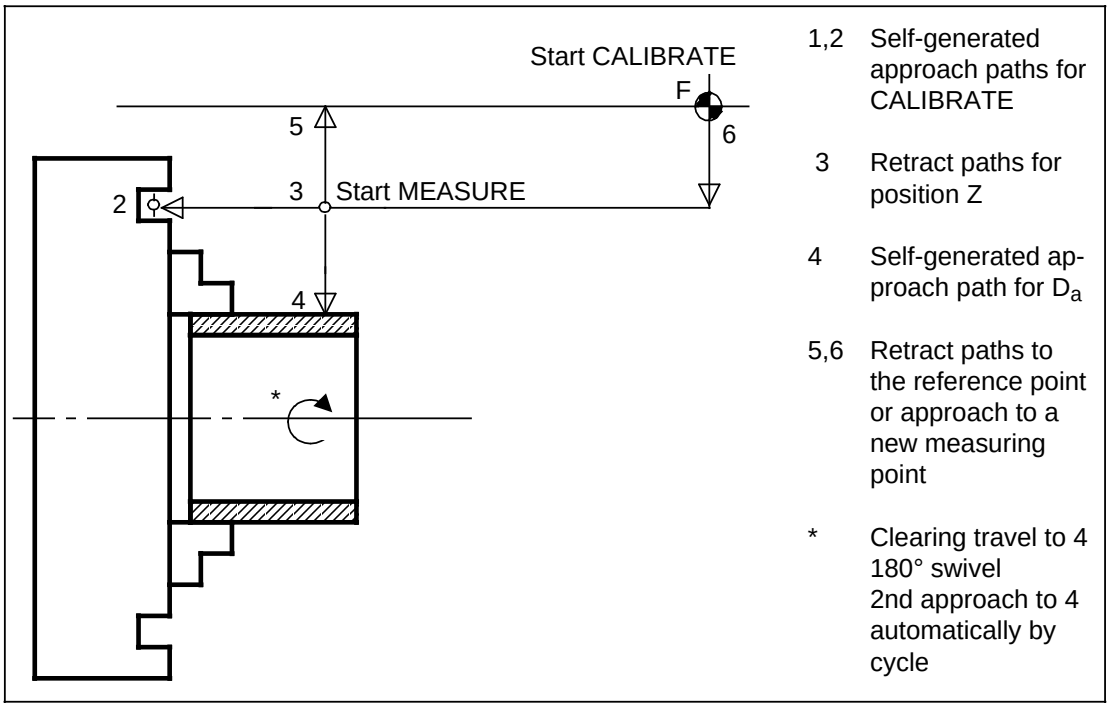


Fig. 4.13 1-point measurement with reversal on outside diameter with CALIBRATE

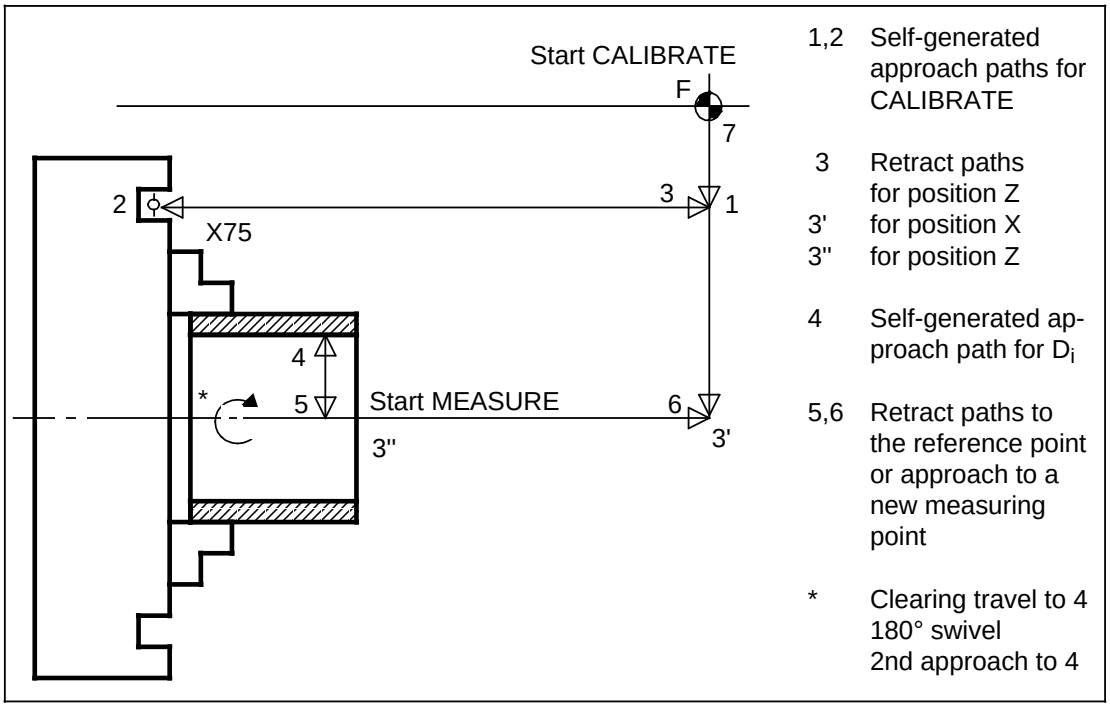


Fig. 4.14 1-point measurement with reversal on inside diameter with CALIBRATE

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory No./average value memory No. (see Section 2.3)
R22	Probe number (see Section 2.6)
R23 = 22	1-point measurement with reversal
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R26= 0-359.5	Start angle (only positive)
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Average value compensation
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: 1-point measurement with reversal on outside diameter (probe calibrated)

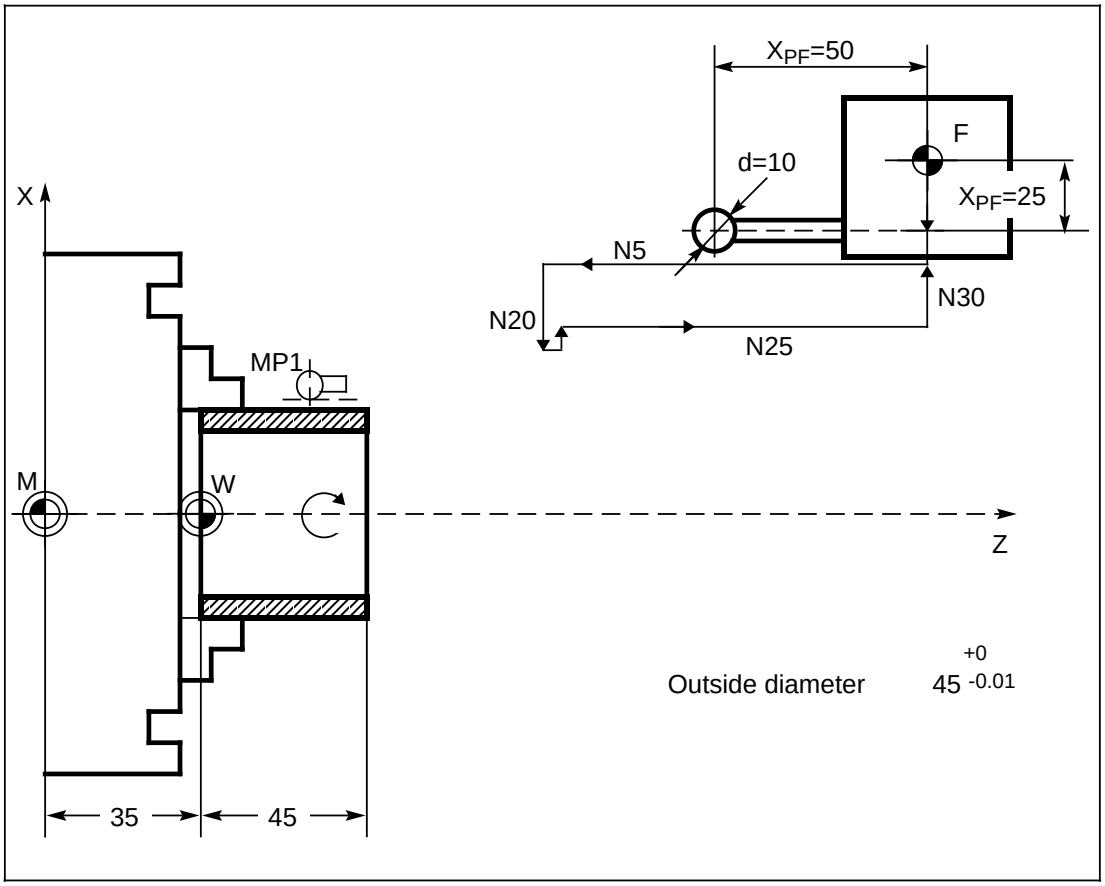


Fig. 4.15 1-point measurement with reversal on outside diameter

```
%MPF    9742
N1      T8 G18
N5      G54 G00 Z30 D30
N10     R10=10 R11=20 R22=1 R23=22 R25=0          Parameters for measuring cycle
        R26=0 R27=1 R28=3 R29=1
        R30=2 R33=0 R34=0.2 R36=1
        R37=0.04 R40=0 R41=-0.01 R42=45
N20     L974                                      Call for measuring cycle
N25     G00 G53 Z110 D0                           Retraction
N30     G53   X90
N35     M30
```

© Siemens AG 1990 All Rights Reserved

4.3.4 L974 2-point measurement on diameter

The measuring cycle ascertains the actual value of the workpiece in relation to the workpiece zero point and the set/actual difference.

The cycle measures the diameter by measuring the upper (1st measurement) and the lower (2nd measurement) point without turning the part. In this case, the probe need not be calibrated anymore. Measuring must not be carried out in any other order as otherwise the protection zone will not have any effect!

A protection zone must be specified with R18 and R19 because the cycle positions automatically with G00 from the 1st to the 2nd measuring point. A negative setpoint is not possible.

If double sensors are used with 2D numbers, the protection zone always refers to the active D number. Therefore the protection zone dimensions required in the construction along the measuring axis must be increased by the amount 2r (outside distance between the 2 probe balls).

The maximum diameter that can be measured depends on the traversing range of the turret slide in negative direction and on the geometry dimension of the probe in X.

The actual value and the set/actual difference are determined on the basis of the two measurements.

An empirical value stored in SDC is subsequently allowed for with the correct sign.

In addition, averaging is performed over several parts and the tolerance bands are checked (Section 1.7.1).

The probe is located outside the protection zone ”X” and ”Z” on completion of measurement.

No automatic compensation or a length compensation is carried out, depending on the definition of R10.

Preconditions:

- The workpiece must be positioned with M19 prior to cycle call when necessary.
- The probe must be positioned with **tool offset** and **G53** to face the surface to be measured (Fig. 4.9).

If double sensors are used with 2D numbers, the protection zone always refers to the active D number. Therefore the protection zone dimensions required in the construction along the measuring axis must be increased by the amount 2r (outside distance between the 2 probe balls).

© Siemens AG 1990 All Rights Reserved

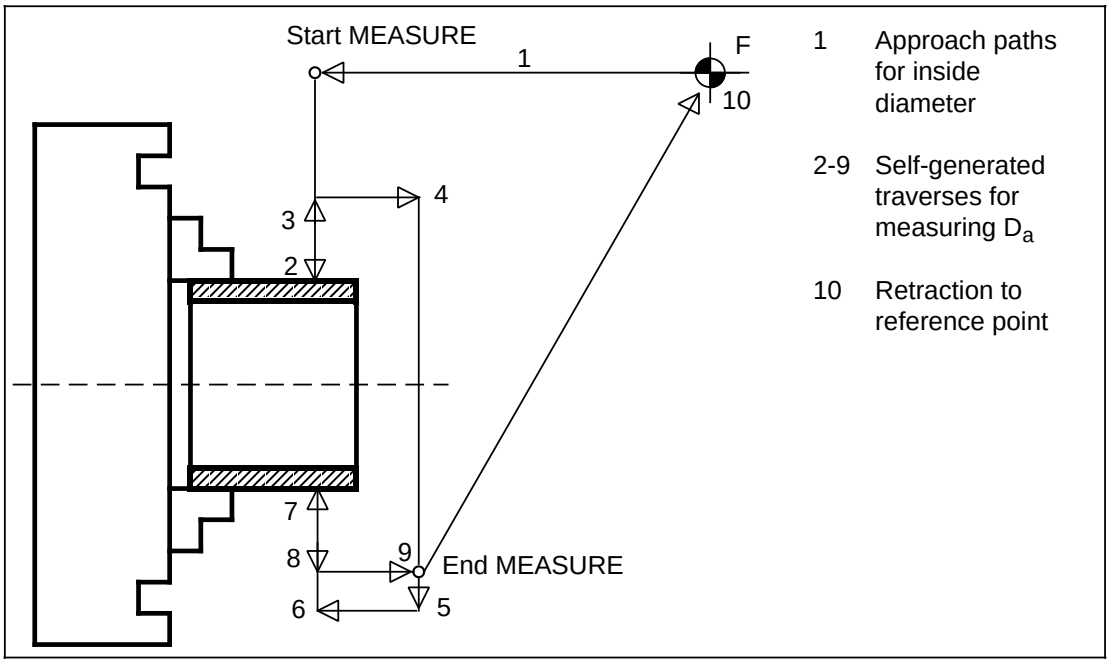


*Fig. 4.16 2-point measurement on outside diameter **without CALIBRATE***

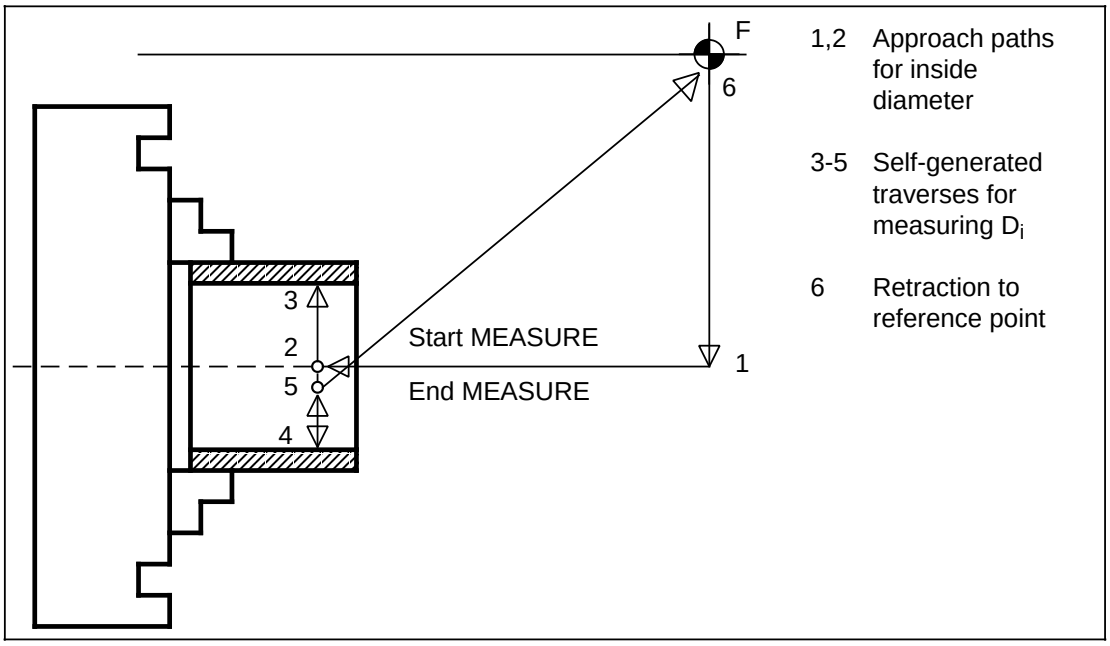


*Fig. 4.17 2-point measurement on inside diameter **without CALIBRATE***

© Siemens AG 1990 All Rights Reserved

The following diameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory No./average value memory No. (see Section 2.3)
R18	Protection zone around workpiece ordinate (radius)
R19	Protection zone around workpiece abscissa
R22	Probe number (see Section 2.6)
R23 = 23 = 24	2-point measurement on diameter (for outside measurement with R18/R19; for inside measurement without R18/R19) 2-point measurement on diameter (for inside measurement only: with R18/R19)
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Average value compensation
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing)

See Section 10.2 for result display parameters

© Siemens AG 1990 All Rights Reserved

Example: 2-point measurement outside and inside

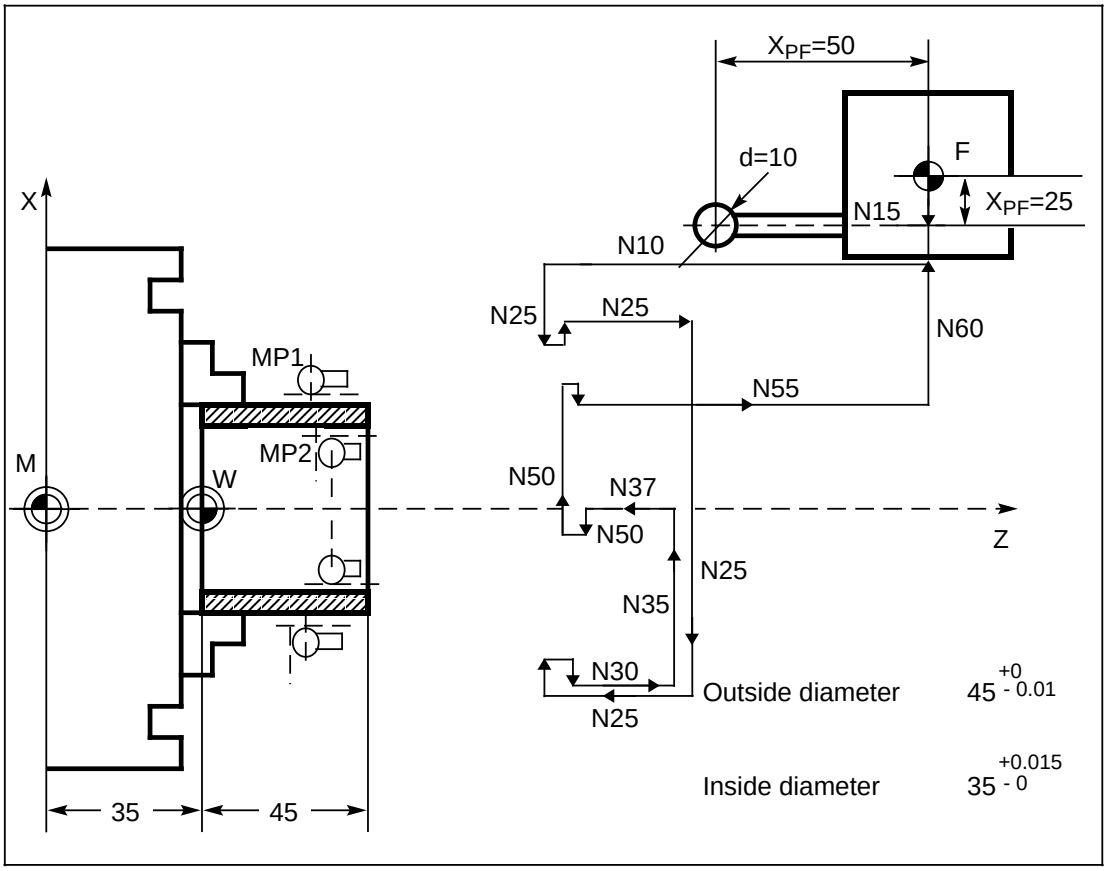


Fig. 4.18 Workpiece measurement, 2-point measurement outside and inside

```
%MPF    9743
N5      T1 G18                                          Call probe
N10     G00 G54 Z30 D31 X60                             ZO selection, position probe
                                                          to face MP1
N15     R10=3 R11=3 R18=30 R19=55 R22=1                 Define parameters for
        R23=23 R25=0 R27=1 R28=1 R29=3 R30=2    measurement

        R33=0.002 R34=0.005 R36=0.5 R37=0.04
        R40=0 R41=-0.01 R42=45
N25     L974                                            Measure MP1
N30     G00 Z35                                         Position probe to face MP2
N35     X-5
N37     Z30
N40     R10=4 R11=4 R22=1  R23=23 R25=0                 Define parameters for measurement
        R27=1 R28=1 R29=3 R30=2
        R33=0.002 R34=0.005 R36=0.5 R37=0.04
        R40=0.015 R41=0 R42=35
N50     L974                                            Measure MP2
N55     G53 G00 Z110 T0
N60     G53 X90
N65     M30
```

© Siemens AG 1990 All Rights Reserved

4.3.5 L974 Multi-point measurement on circumference

Function and application

The measuring cycle ascertains the actual value of the workpiece in relation to the workpiece zero point and the set/actual difference.

The number of measurements and the indexing angle (=start angle) are determined on the circumference via 2 R parameters.

The actual value and the set/actual difference are determined on the basis of the arithmetic mean of the measuring point.

An empirical value stored in SDC is subsequently allowed for with the correct sign.

In addition, averaging is carried out over several parts and the tolerance bands are checked (Section 1.7.1).

The probe is located at a distance ”a” facing the last measuring point on completion of measurement.

No automatic compensation or a length compensation is carried out, depending on the definition of R10.

Preconditions:

- The probe must have been calibrated in the direction of measurement.
- The probe must be positioned with **tool offset** and **G53** to face the surface to be measured (Fig. 4.9).
- The maximum diameter that can be measured depends on the traversing range of the turret slide in +X.

Outside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} + 2a$$

Inside diameter:

$$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} - 2a$$

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory No./mean value memory No. (see Section 2.3)
R22	Probe number (see Section 2.6)
R23 = 25	Multi-point measurement on circumference
R24	Number of measuring points on circumference
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R26= 0-359.5	Indexing angle = Start angle (only positive)
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Average value compensation
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing) * The set value must be specified as a **negative** value when measuring below the turning centre in plus X direction.

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Multi-point measurement on circumference (probe calibrated)

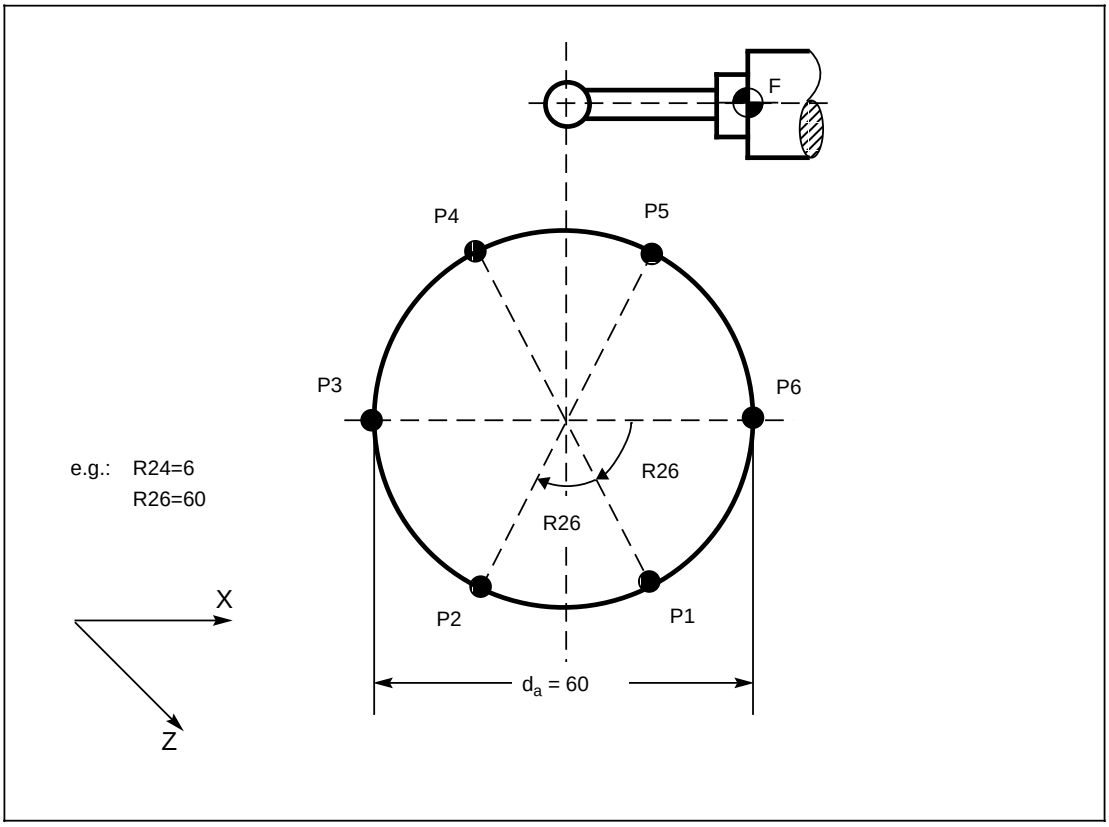


Fig. 4.19 Multi-point measurement on circumference (outside diameter)

```
%MPF    9744
N1      T8 G18                                  Select probe
N5      G00 G54 Z25 D30                         Approach start position
N10     X70
N15     R10=10 R11=20 R22=1 R23=25 R24=6        Parameters for measuring cycle
        R25=0 R26=60 R27=1 R28=3
        R29=1 R30=2 R33=0 R34=0.2
        R36=1 R37=0.06 R40=0.03 R41=0.03
        R42=60
N25     L974                                    Call for measuring cycle
N30     G00 G53 Z110                            Approach reference point
N35     X160
N40     M30
```


© Siemens AG 1990 All Rights Reserved

4.3.6 L974 Multi-point measurement on cylinder

Function and application

The measuring cycle ascertains the actual value of the workpiece in relation to the workpiece zero point and the set/actual difference.

The number of measurements and the offset on the cylinder are determined via 2 R parameters.

The actual value and the set/actual difference are determined on the basis of the arithmetic mean of the measuring points.

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, averaging is carried out over several parts and the tolerance bands are checked (Section 1.7.1).

The probe is located at a distance ”a” facing the last measuring point on completion of measurement.

No automatic compensation or a length compensation is carried out, depending on the definition of R10.

Preconditions :

- The probe must have been calibrated in the direction of measurement.
- The probe must be positioned with **tool offset** and **without G53** to face the surface to be measured (Fig. 4.9).
- The maximum diameter that can be measured depends on the traversing range of the turret slide in +X.

  Outside diameter:

  $$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} + 2a$$

  Inside diameter:

  $$D_{max} = 2\times(X_{MFmax} - X_F) - {}_{ball} - 2a$$

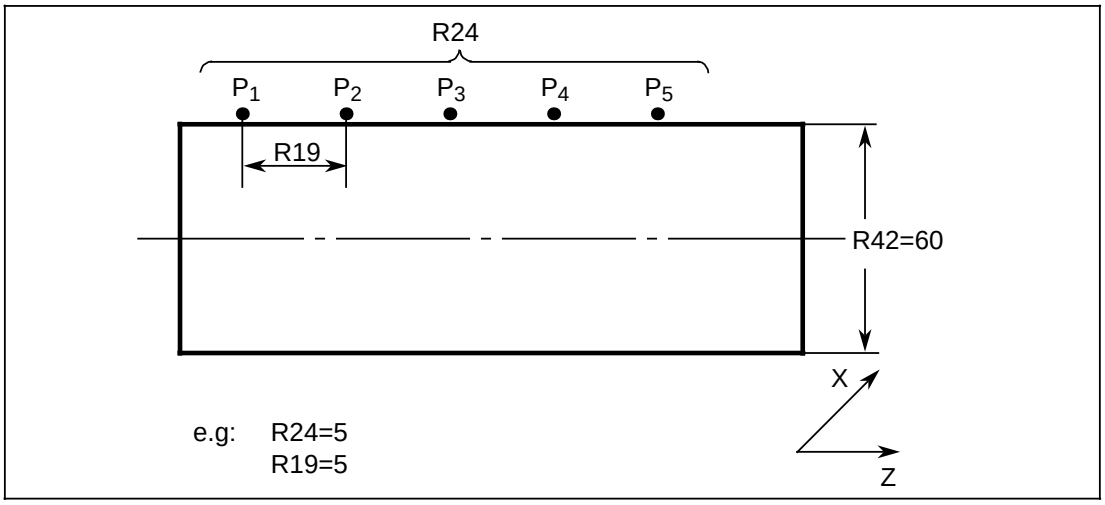


Fig. 4.20 Multi-point measurement on cylinder (outside diameter)

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory No./average value memory No. (see Section 2.3)
R19	Offset of abscissa (incremental)
R22	Probe number (see Section 2.6)
R23 = 26	Multi-point measurement on cylinder
R24	Number of measuring points on cylinder
R25 R25=0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Average value compensation
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Multi-point measurement on cylinder (probe calibrated)

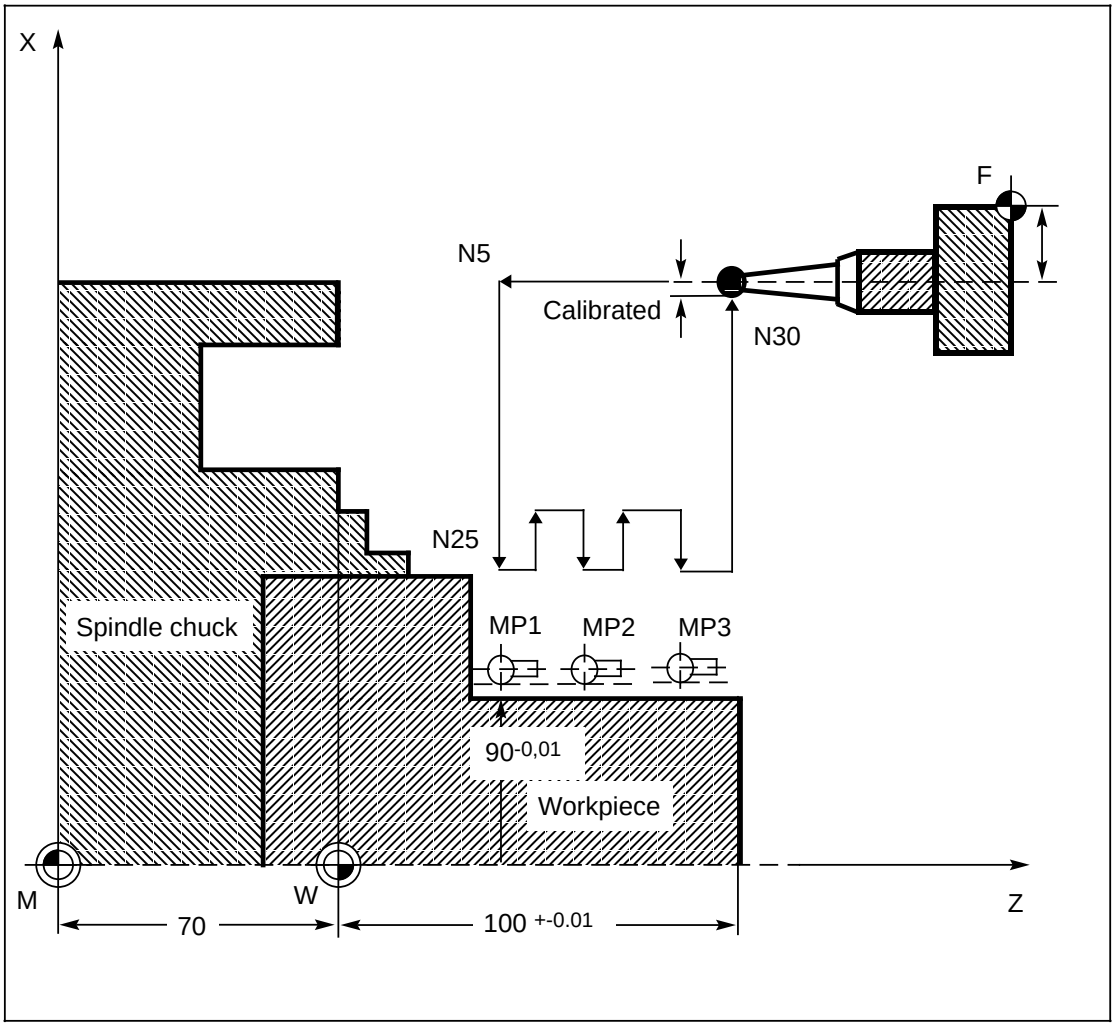


Fig. 4.21 Multi-point measurement on cylinder

```
%MPF    9745
N1      T8 G18                                  Select probe
N5      G00 G54 Z30 D30                         Approach start position
N15     R10=10 R11=20 R19=20 R22=1 R23=26       Parameters for measuring cycles
        R24=3 R25=0 R27=1 R28=3
        R29=1 R30=2 R33=0 R34=0.2
        R36=1 R37=0.06 R40=0 R41=-0.01
        R42=90
N25     L974                                    Call for measuring cycle
N30     G00 G53 X152 D0                         Approach measuring point
N35     G53 Z103
N40     M30
```

© Siemens AG 1990 All Rights Reserved

4.4 Examples of application for workpiece measurement (L973, L974)

The workpiece shown below is to be measured with the aid of a probe.

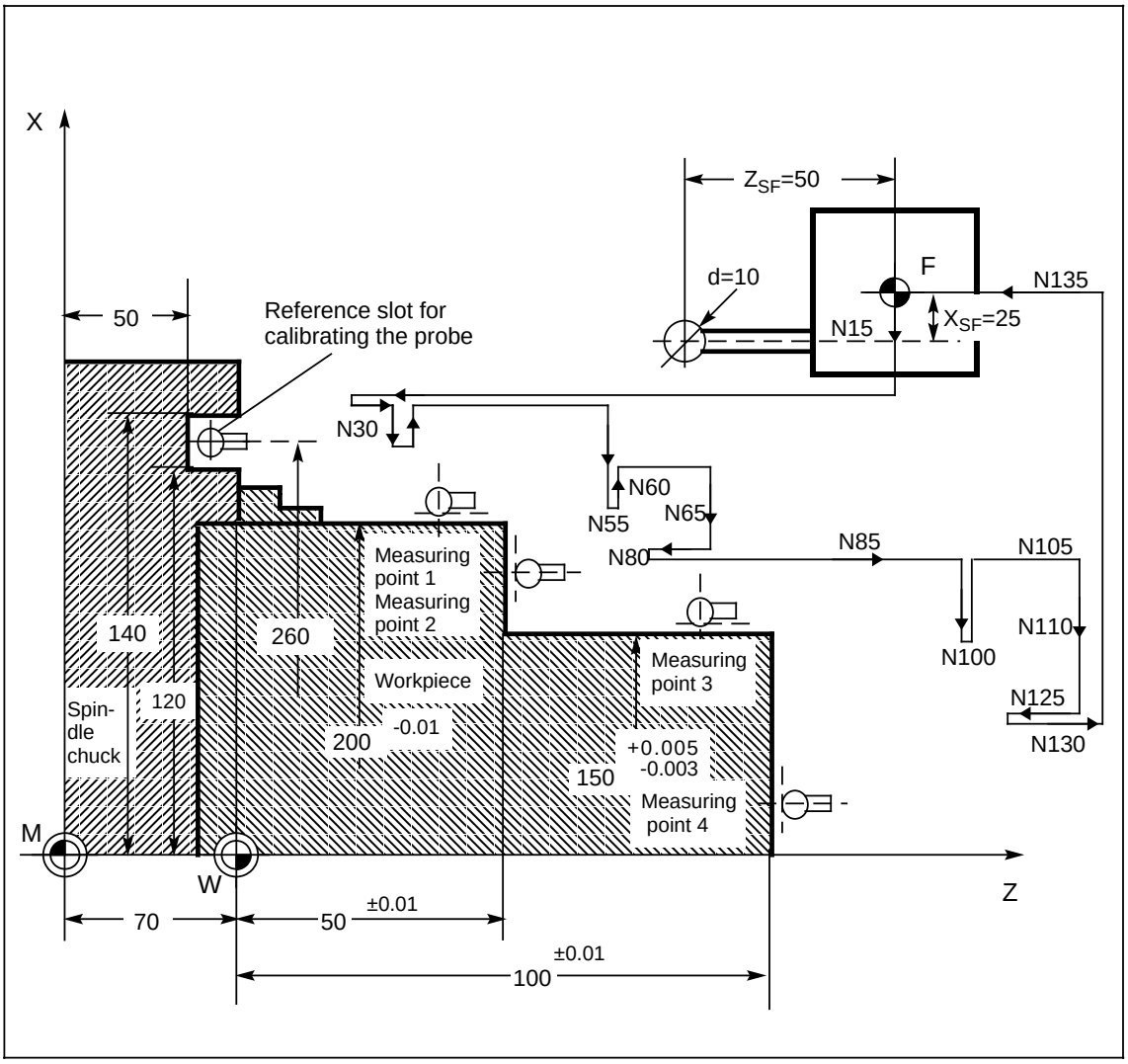


Fig. 4.22 Workpiece measurement

© Siemens AG 1990 All Rights Reserved

Example: Calibrate workpiece probe, workpiece measurement with L973 and L974
(Data as in Fig. 4.22)

```
%931
N5      G18  G53  T1  D31  X130                  Call probe type 7
N10     R22=1 R23=2002 R25=0 R27=1               Calibrate 2 axis direction -X,+X
        R28=1 R30=2  R32=20  R36=1
        R33=0
N20     L973
N25     R23=22  R30=1  R32=0                        Calibrate in Z axis
N30     L973                                        Calibrate probe in minus Z direction
N35     G54 G00 Z40                              Select ZO; position Z axis to face measuring
                                                 point 1
N40     R10=8 R11=3 R22=1 R23=21 R25=0              Define parameters for measurement
        R27=1 R28=1 R29=2 R30=2
N45     R33=0.002 R34=0.005 R36=0.3
        R37=0.2 R40=0 R41=-0.01 R42=200
N55     L974                                        Measure MP1
N60     G00 Z70                                     Position probe to face MP2
N65     X175
N70     R10=9 R11=4 R30=1 R40=0.01               Define parameters for measurement
        R41=-0.01 R42=50
N80     L974                                        Measure MP2
N85     G00 Z180                                    Position probe to face MP3
N90     R10=10 R11=5 R30=2 R40=0.005                Define parameters for measurement
        R41=-0.003 R42=150
N100    L974                                        Measure MP3
N105    G00 Z150                                    Position probe to face MP4
N110    X50
N115    R10=11 R11=6 R30=1 R40=0.01                 Define parameters for measurement
        R41=-0.01 R42=100
N125    L974                                        Measure MP4
N130    G00 Z250 D0 G53                             Retraction Z
N135    G53 X280 M30                             Retraction X
```

4.5 Parameter recommendations for L973, L974

Parameter recommendations for reliable program run

Measure workpiece

a) Calibrate L973

Zero offset range	R33=0.001	
Compensation range with averaging	R34=0.010	
Safe area	R36=1	(Continuous mode)
Safe area	R36=3	(Setup mode)

b) Measure L974

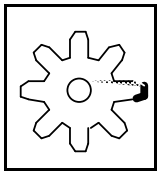
Zero offset range	R33=0.001	
Compensation range with averaging	R34=0.010	
Safe area	R36=1	(Continuous mode)
Safe area	R36=3	(Setup mode)
Dimensional difference check	R37=0.3	(Continuous mode)
Dimensional difference check	R37=3	(Setup mode)

All X-axis values are related to the diameter.

END OF SECTION

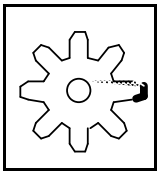
© Siemens AG 1990 All Rights Reserved

5 Measuring Cycles for Milling Machines and Machining Centres



The possible scope of measurement depends on the machine configuration and can also be partly determined via MDC.

(See machine tool manufacturer's specifications)



Cycle	Function	
L931	Auxiliary cycle for tool or workpiece measuring cycles	See Section 3 for auxiliary cycles
L932	Check MDC and transfer parameters	
L933	Auxiliary cycle for tool or workpiece cycles	
L934	Auxiliary cycle for tool or workpiece cycles	
L935	Measurement result display selection	
L936	Measurement abort	
L937	Auxiliary cycle for tool or workpiece measuring cycles	
L938	Auxiliary cycle for tool or workpiece measuring cycles	
L939	Auxiliary cycle for tool measuring cycle	
L961	Additive input of empirical values	
L962	Erase program empirical values/average values	
L963	Auxiliary cycle for workpiece measuring cycle L979	
L964	Auxiliary cycle for workpiece measuring cycle L976	
L965	Measuring plane	
L966	Auxiliary program for operator guidance macro	
L967	Presetting of transfer parameters	
L969	Coordinate rotation	
L970	Pre-positioning	
L976	Calibrate workpiece probe	Workpiece measuring cycles
L977	Workpiece measurement BWN	
L978	Workpiece measurement	
L979	Workpiece measurement 2D BWN	
L988	Auxiliary cycles for workpiece measuring cycles	
L989	Auxiliary cycle for workpiece measuring cycle L979	

These programs **must be available**

Additional programs: Facilitate programming and operating.

© Siemens AG 1990 All Rights Reserved

5.1 L976 Calibrating the workpiece probe

On milling machines and machining centres, the probe is usually loaded into the spindle from a tool magazine. This may lead to errors in further measurements caused by clamping tolerances of the probe in the spindle.

In addition, the probe trigger point must be precisely determined in relation to the spindle centre. This is achieved by the calibration cycle which makes it possible to calibrate the probe either in a hole (plane) or at a surface (applicate).

The measuring variant is selected by defining R23:

Definition of R23	Workpiece probe type 30 calibration
R23 = xxx00 R23 = xxx10 R23 = xxx20 R23 = xxx01	In reference hole (plane) On reference ball (plane) On reference cube, surface (plane) In reference hole (applicate)
R23 = xxx02 R23 = xxx23	In random hole (plane) On any cube, surface (applicate)

Applicable types of probe: (see Section 1.3)

- In the case of milling and machining centres the probe must be entered as type 30 in the TOA memory.
- Multidirectional probe
- Monodirectional probe
- Bidirectional probe

The various probe types are differentiated by defining R22 (see Section 2.6):

The probe ball diameter must only be calculated when the following cycles (measurement variants) are used:

L978 With differential measurement
L979 Workpiece measurement hole**/**shaft**/**slot**/**web at random angle

© Siemens AG 1990 All Rights Reserved

5.1.1 L976 Calibrating the workpiece probe in reference hole (plane)

Function and application

The measuring cycle calibrates the probe clamped in the spindle in a reference hole. The trigger points determined are loaded into the relevant MDC area.

The cycle positions the probe automatically into the selected reference hole. Positioning is paraxial with linear interpolation over the shortest distance.

The probe is at the centre of the hole on completion of calibration.

Preconditions:

- The probe must be called with **tool offset** and **G53**.
- The dimensions of the reference hole in relation to the machine zero point must have been input in the MDC area.
- The valid reference hole is specified with R12.
- Random start point
- The probe ball must be located above the upper edge of the reference hole.

The following parameters must be defined prior to call:

Parameters	Description
R12	Selection of reference hole
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type**/**probe number (see Section 2.6)
R23 = 0	Calibrating in reference hole (plane) (see Section 2.7.4)
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30	Measuring axis (definition depending on R23)
R31 = 0 = 1	Axial direction (definition depending on R23) Positive axial direction Negative axial direction
R33	Zero offset range
R36	Safe area

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

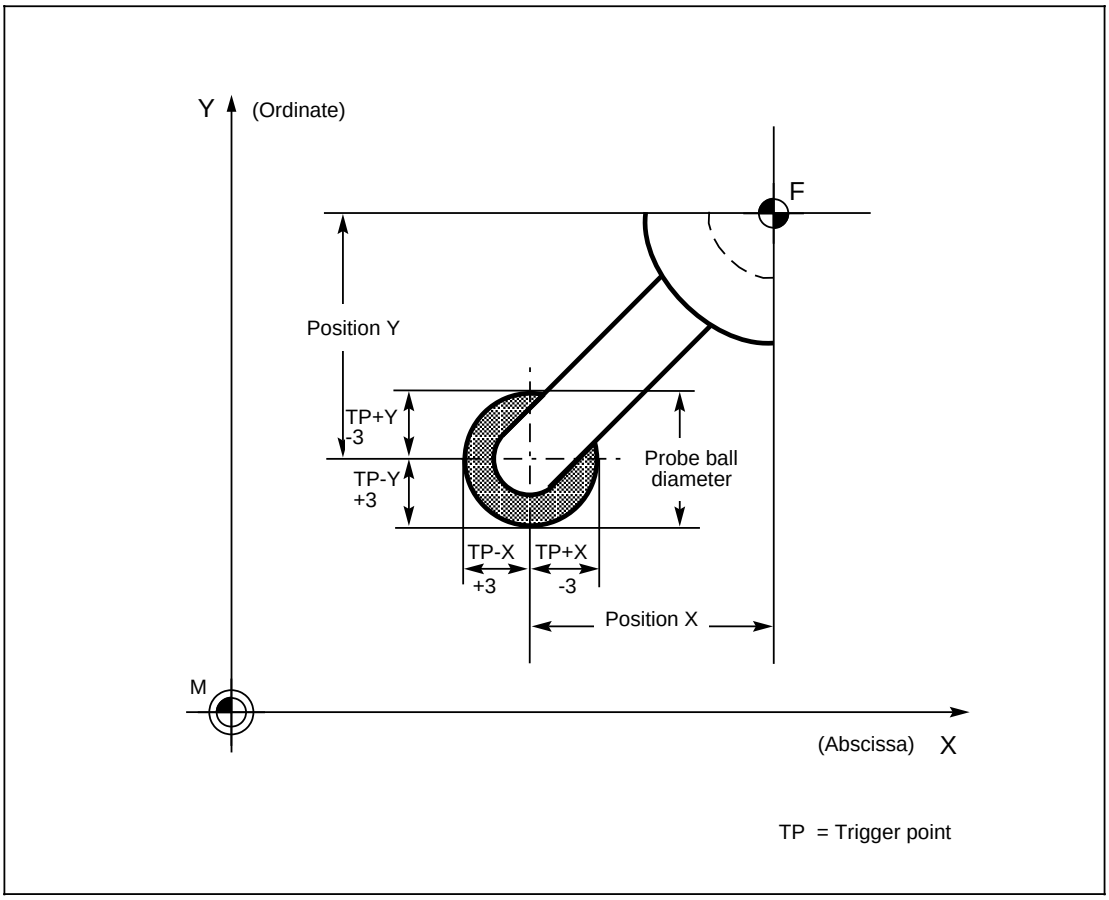


Fig. 5.1 Workpiece probe

Example: Calibrating workpiece probe 1 in reference hole 1 in the X-Y plane
(Fig. 5.4)

```
%MPF    9761
N1      T200                                                    T No. probe
N5      G53   G17   G00   D99   Z100                          Select length compensation without ZO
N10     R12=1       R22=1         R23=0       R25=0         Parameters for calibrating cycle
        R27=1       R28=1         R33=0       R36=1
N15     L976                                                  Cycle call for calibrating
                                                                in the X-Y plane
N20     M30
```

The new trigger values are stored in the relevant MDC area.

© Siemens AG 1990 All Rights Reserved

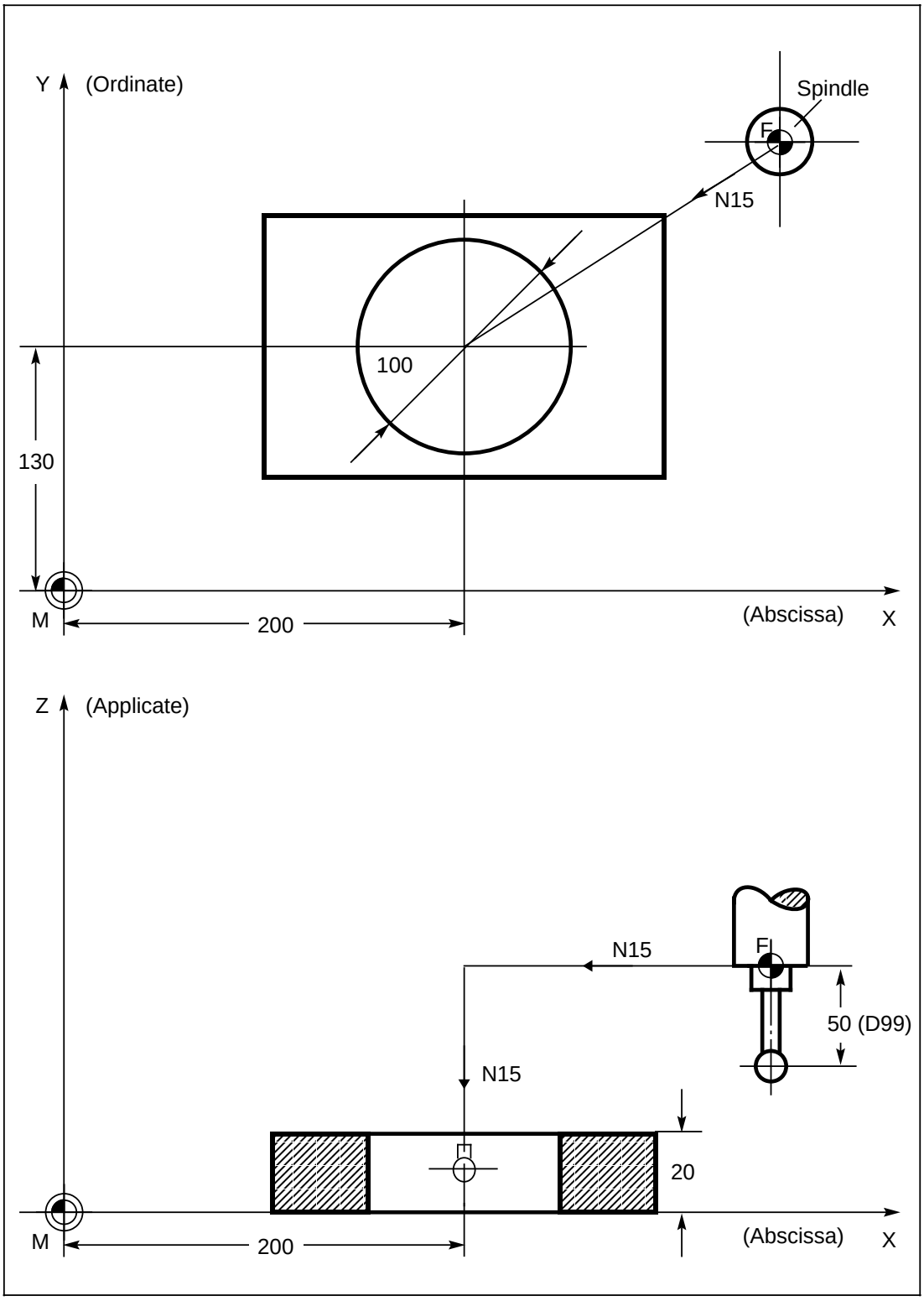


Fig. 5.2 Reference hole

© Siemens AG 1990 All Rights Reserved

5.1.2 L976 Calibrating workpiece probe in reference hole (applicate)

Function and application

The measuring cycle makes it possible to calibrate the probe in the applicate in a reference hole, and thus to determine the trigger length. The cycle positions the probe automatically into the reference hole selected, positioning being paraxial with linear interpolation over the shortest distance.

As from Version 3.5, the length in the TOA memory can refer to the probe end (MDC 7004.5=1), otherwise it refers to the ball centre (see also Section 1.4.3).

The probe is located above the calibration surface by the amount ”a” on completion of calibration.

Preconditions:

- The probe must be called with **tool offset** and **G53**.
- The dimensions of the reference hole in relation to the machine zero point must have been input in the MDC area.
- The valid reference hole is specified with R12.
- Random start point
- The probe ball must be located above the upper edge of the reference hole.

The following parameters must be defined prior to call:

Parameters	Description
R12	Selection of reference hole
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type**/**probe number (see Section 2.6)
R23 = 1	Calibrating in reference hole (applicate) (see Section 2.7.4)
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30	Measuring axis (definition depending on R23)
R31 = 0 = 1	Axial direction (definition depending on R23) Positive axial direction Negative axial direction
R33	Zero offset range
R36	Safe area

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

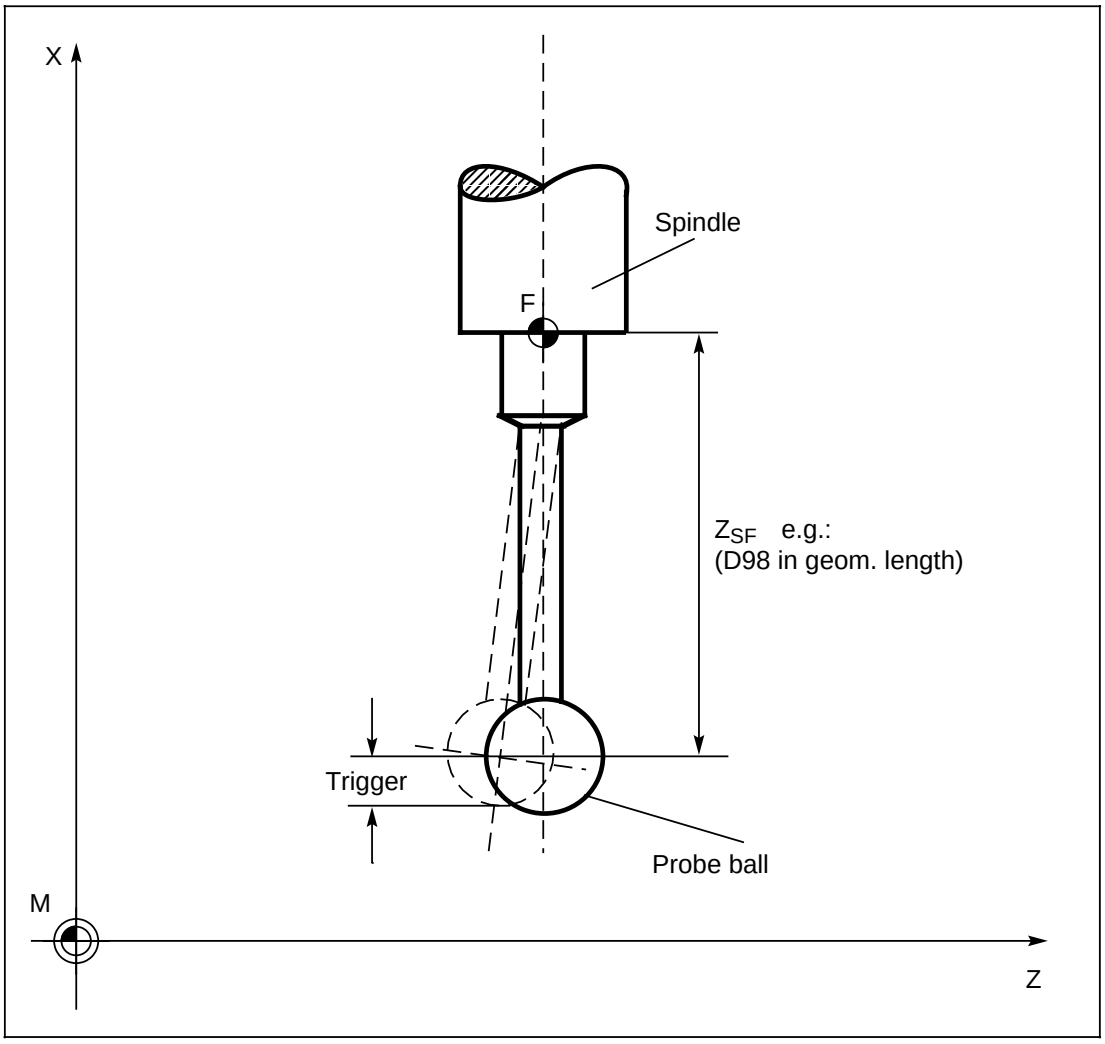


Fig. 5.3 Workpiece probe

Example: Calibrating workpiece probe 2 in reference hole 3 in the Z axis (Data as in Fig. 5.6)

The probe length (Z axis) must have been input prior to cycle call in the TO memory, e.g. D98 (in this case value 50).

```
%MPF    9763
N1      T200                                          T No. probe
N5      G53 G17 G00 D98 Z100                          Select length compensation, without ZO
N10     R12=3     R22=2    R23=1    R25=0             Parameters for calibrating cycle
        R27=1     R28=1    R33=0    R36=1
N15     L976                                          Cycle call for calibrating in the
                                                      Z axis
N20     M30
```

The new trigger point is entered in accordance with the MDC.

© Siemens AG 1990 All Rights Reserved

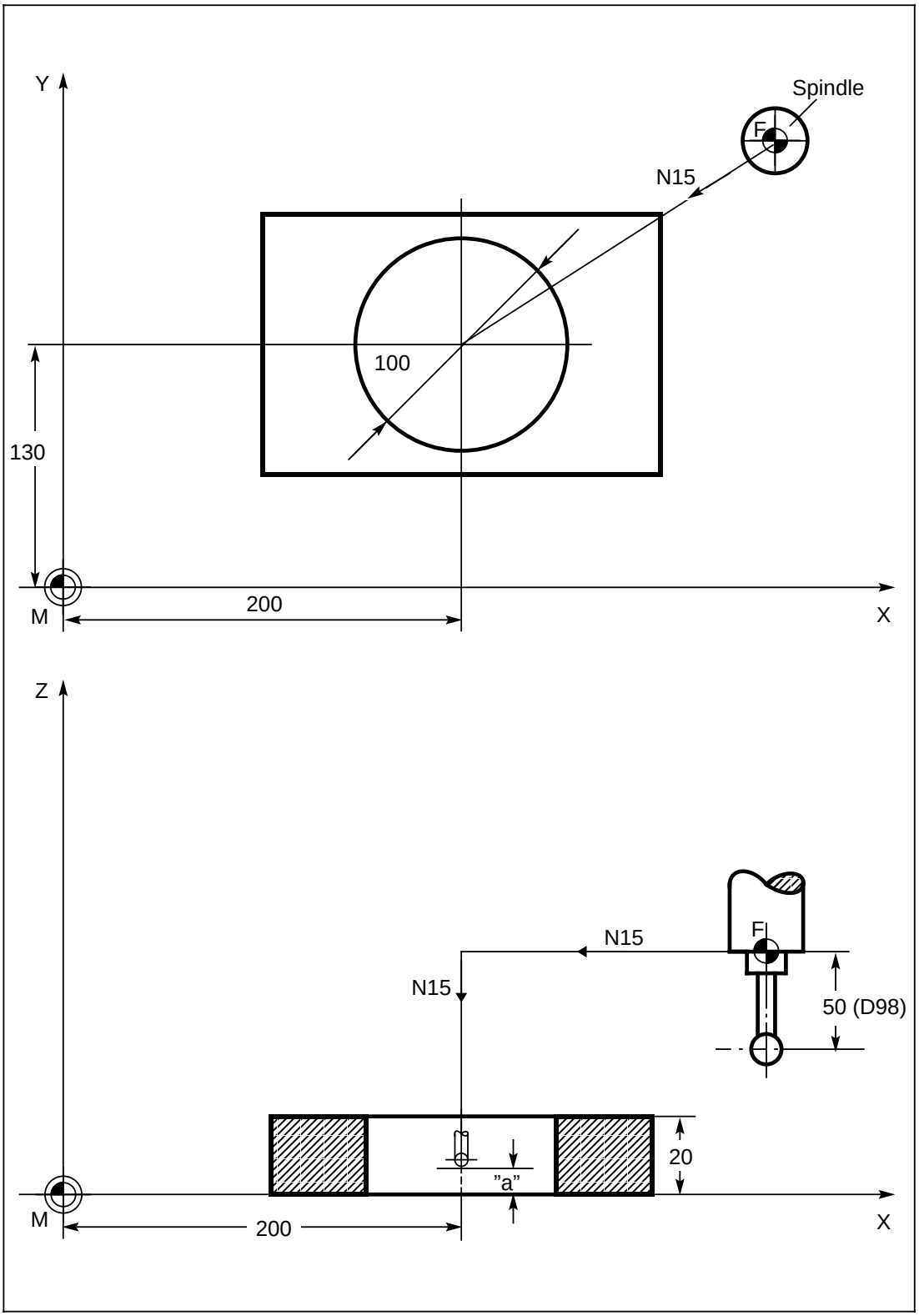


Fig. 5.4 Reference hole

© Siemens AG 1990 All Rights Reserved

5.1.3 L976 Calibrating workpiece probe in any hole (plane)

Function and application

The measuring cycle enables the probe to be calibrated in any hole, e.g. on the workpiece.

The trigger points determined are loaded automatically into the relevant MDC area.

The probe is located at the centre of the hole on completion of calibration. Where one point is measured, the probe is located above the calibration surface by the amount ”a”.

Preconditions:

- The probe must be called with **tool offset** and without **G53**.
- The probe must be positioned to the hole centre point in the abscissa and ordinate of the selected measuring plane and to the calibrating depth within the hole.

The following parameters must be defined prior to call:

Parameters	Description
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type**/**probe number (see Section 2.6)
R23 = 2	Calibrating in any hole (plane) (see Section 2.7.5)
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30	Measuring axis (definition depending on R23)
R31 = 0 = 1	Axial direction (definition depending on R23) Positive axial direction Negative axial direction
R32	Set value for calibration=diameter of hole
R33	Zero offset range
R36	Safe area

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Calibrating workpiece probe 3 in the X-Y plane

The probe length (Z axis) must have been input in the TO memory prior to cycle call, e.g. D98 (in this case value 50).

```
%MPF    9764
N1      T200                                                    T No. probe
N5      G59   X100  Y50   Z0                                    Load ZO values
N10     G54   G17   G00   X100  Y80                             Probe at centre point and
                                                                  ZO selection
N15     D98   Z10                                               Selection of length compensation
                                                                  Position probe in hole
N20     R22=3       R23=2         R25=0       R27=1             Parameters for calibrating cycle
        R28=1       R32=100       R33=0       R36=1
N25     L976                                                    Cycle call for calibrating in the
                                                                  X-Y-plane
N20     M30
```

The new trigger values are stored in the relevant MDC areas.

© Siemens AG 1990 All Rights Reserved

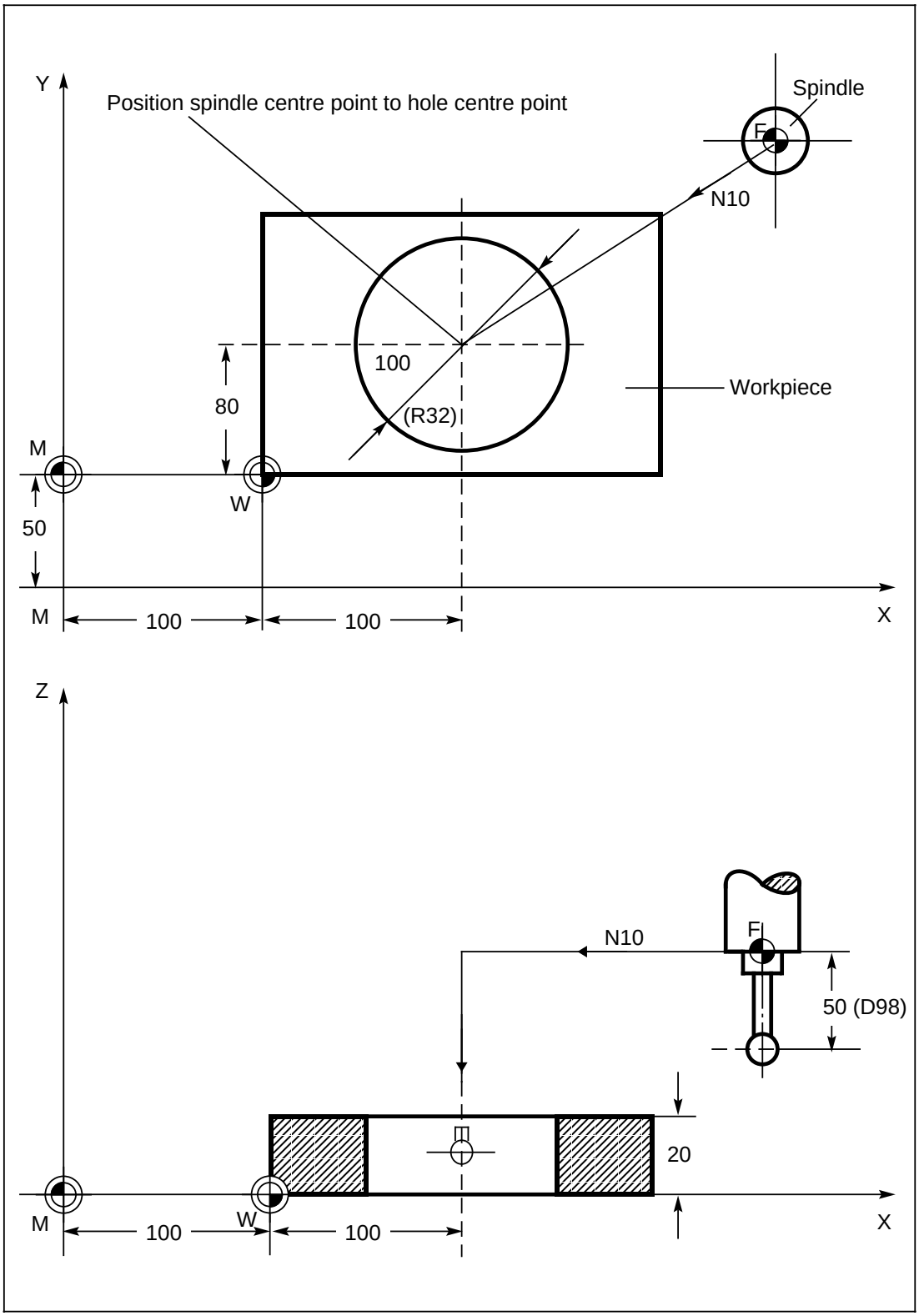


Fig. 5.5 Any calibration hole

5.1.4 L976 Calibrating workpiece probe on any surface (applicate)

Function and application

The measuring cycle enables the probe to be calibrated in the applicate on any surface, e.g. at the workpiece, and thus to determine the length.

As from Version 3.5, the length can refer to the probe end (MDC 7004.5=1), otherwise it refers to the ball centre (see also Section 1.4.3).

The probe is located above the calibration surface by the amount ”a” on completion of calibration.

Precondition:

The probe must be positioned **with** tool offset and **without G53** to face the calibration surface.

The following parameters must be defined prior to call:

Parameters	Description
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type**/**probe number (see Section 2.6)
R23 = 3	Calibrating on any surface (applicate) (see Section 2.7.4)
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30	Measuring axis (definition depending on R23)
R31 = 0 = 1	Axial direction (definition depending on R23) Positive axial direction Negative axial direction
R32	Set value for calibration
R33	Zero offset range
R36	Safe area

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Calibrating workpiece probe 1 in the Z axis at the workpiece

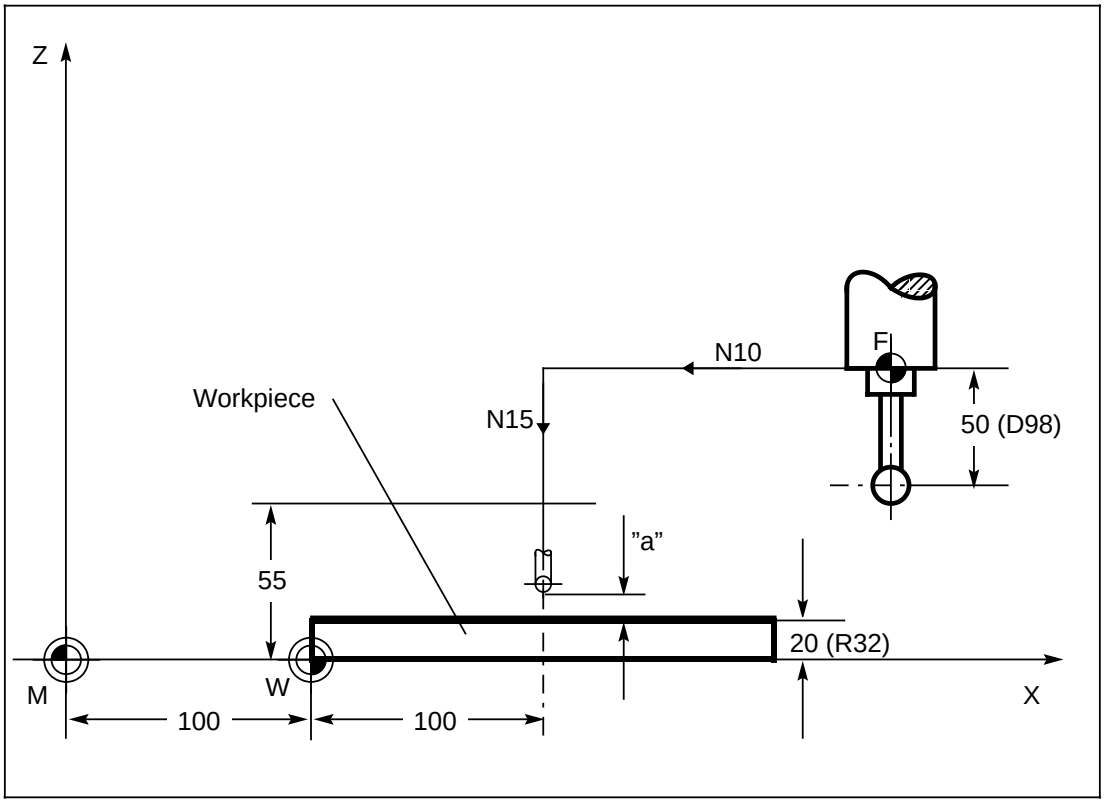


Fig. 5.6 Any calibration surface

The probe length (Z axis) must be input in the TO memory prior to cycle call, e.g. D98 (in this case value 50).

```
%MPF    9766
N5      T200                                          T No. probe
N10     G54   G17   G00   X100   Y80                  Position probe above calibration point
N15     D98       Z55                                 Select length compensation
N20     R22=1     R23=23    R25=0    R27=1            Parameters for calibrating cycle
        R28=1     R32=20    R33=0    R36=1
                  R30=3     R31=1
N25     L976                                          Cycle call for calibrating in the
                                                        Z axis
N30     M30
```

The new trigger value is entered in the relevant MDC.

© Siemens AG 1990 All Rights Reserved

5.2 L977 Workpiece measurement hole/shaft/slots/ZO determination (paraxial)

The cycle determines the dimensions of holes, shafts and slots (H-S-S measuring cycle) and performs automatic tool offset when necessary.

It also determines the zero offset (ZO) between the centre point of a hole, shaft or slot and the workpiece zero point.

For measuring variants hole and shaft, measurements can be performed with active coordinate rotation (see MDC 7004.1).

Selection is performed on the basis of the definition of R23:

Definition of R23			Prepositioning
R23=	1	Measure hole with tool offset	C. p., applicate for depth
R23=	2	Measure shaft with tool offset	C. p., applicate approx. 1 mm above shaft
R23=	11	Measure slot with tool offset	C. p., applicate for depth
R23=	12	Measure web with tool offset	C. p., applicate approx. 1 mm above web
R23=	21	ZO determination in hole with correction of the zero offset	C. p., applicate for depth
R23=	22	ZO determination at shaft with correction of the zero offset	C. p., applicate approx. 1 mm above shaft
R23=	31	ZO determination in slot with correction of the zero offset	C. p., applicate for depth
R23=	32	ZO determination at web with correction of the zero offset	C. p., applicate approx. 1 mm above web

Applicable types of probe: (see Section 1.3)

- In the case of milling and machining centres the probe must be entered as type 30 in the TOA memory.
- Multidirectional probe
- Monodirectional probe
- Bidirectional probe

A distinction is made between the probe types by defining R22 (see Section 2.6).

© Siemens AG 1990 All Rights Reserved

5.2.1 L977 Measure hole (paraxial)

Function and application

The measuring cycle measures points P1, P2, P3 and P4 in the abscissa and ordinate. These four measured values are used to calculate the actual value of the hole diameter and the position of the hole centre point in the abscissa and ordinate relative to the workpiece zero point (see Fig. 5.9).

The centre point of the abscissa is calculated from points P1 and P2. The probe is then positioned to the calculated centre point and points P3 and P4 are measured. These two points are used to calculate the hole centre point of the ordinate and the hole diameter.

The probe is at the determined hole centre on completion of measurement.

Refers to hole diameter only:

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, averaging is performed over a number of parts and the tolerance bands are checked (see Section 1.7.1).

No automatic offset is carried out or alternatively length compensation or radius compensation (difference halved) is performed, depending on the definition of R10.

Preconditions :

- The probe must be called **with** tool offset and **without G53**.
- The probe must be positioned to the hole centre point in the abscissa and ordinate and the probe ball must be positioned within the hole to the measuring height.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type/probe number (see Section 2.6)
R23 = 1	Measure hole
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value diameter (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure hole with measuring cycle L977
(Data as in Fig. 5.9)

Probe length (Z axis) in TO memory D99 (value 50).

```
%MPF
:
Machining program
:
N500   G54 T200                                                  T No. probe; select ZO
N505   G00 X180 Y130                                            Position probe in X and Y axes
                                                                to hole centre point
N510   Z20 D99                                                  Position Z axis in hole
N515   R10=2030    R11= 10     R22=1       R23=1               Define parameters for measuring cycle
       R25=0       R27= 1      R28=1       R29=3
       R33=0.002   R34= 0.03   R36=1       R37=0.06
       R40=0.03    R41=-0.03   R42=130
N525   L977                                                     Cycle call for hole
                                                                measurement in X/Y
N530   G00 Z160                                                 Withdraw Z axis from hole
N535   M30
```

© Siemens AG 1990 All Rights Reserved

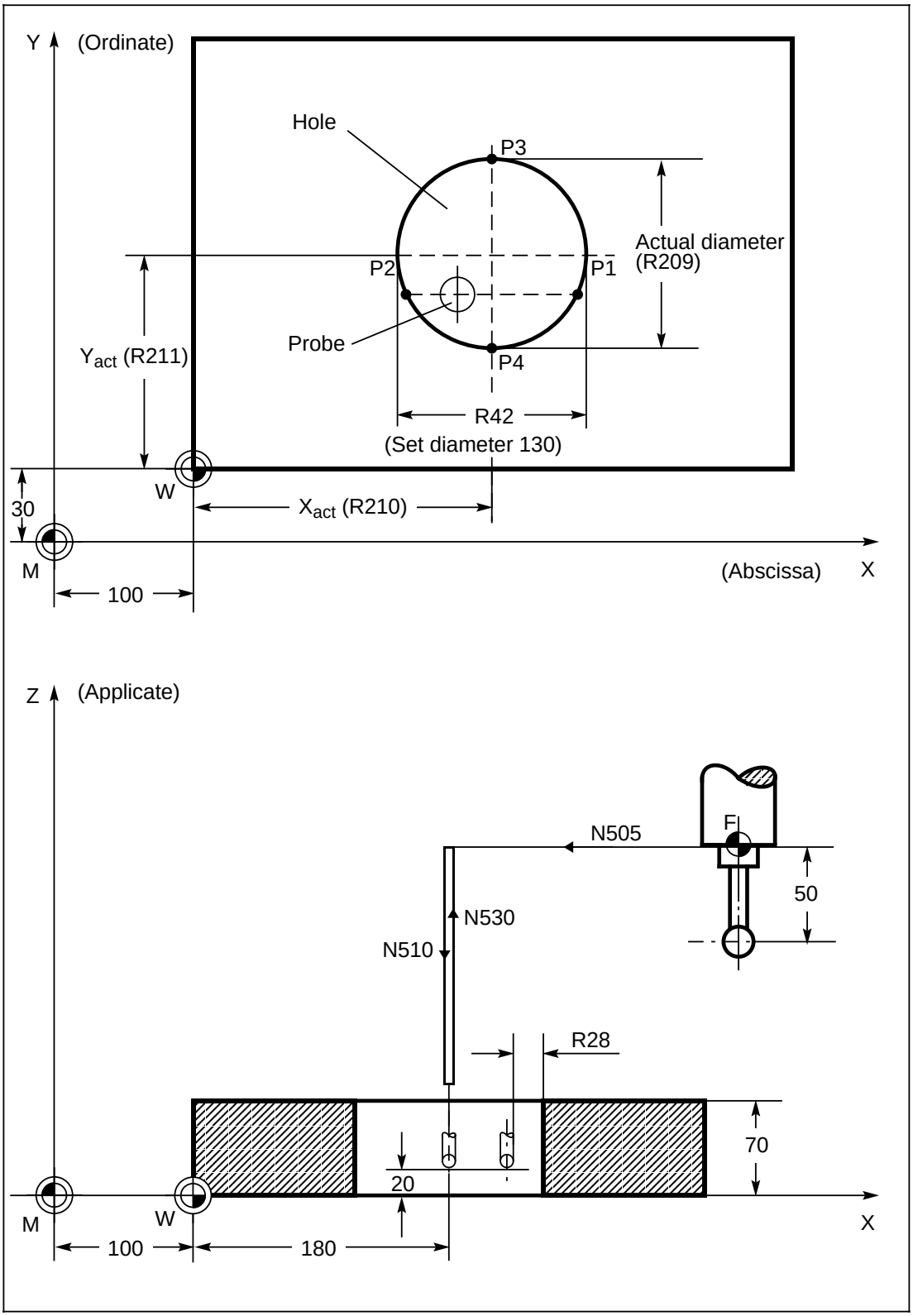


Fig. 5.7 Measure hole

© Siemens AG 1990 All Rights Reserved

5.2.2 L977 Measure shaft (paraxial)

Function and application

The measuring cycle measures points P1, P2, P3 and P4 at a shaft in the abscissa and ordinate. These four measured values are used to calculate the actual shaft diameter and the position of the shaft centre point in the abscissa and ordinate relative to the workpiece zero point (see Fig. 5.10).

The centre point of the abscissa is calculated from points P1 and P2. The probe is then positioned at the calculated centre point and points P3 and P4 are measured. The shaft centre point of the ordinate and shaft diameter are calculated on the basis of these points.

The probe is located above the determined shaft centre on completion of measurement.

Refers to shaft diameter only:

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, averaging is formed over several parts and the tolerance bands are checked (see Section 1.7.1).

No automatic offset is performed or alternatively length compensation or radius offset (difference halved) is carried out, depending on the definition of R10.

Preconditions:

- The probe must be called **with** tool offset and **without G53**.
- The probe must be positioned at the shaft centre point in the abscissa and ordinate, the probe ball being positioned approx. 1 mm **above** the shaft in the applicate.
- R19 is calculated incrementally from the starting position.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R19	Incremental infeed of applicate with sign (travel over the shaft)
R22 _	Probe type/probe number (see Section 2.6)
R23 = 2	Measure shaft
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value diameter (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure shaft with measuring cycle L977
(Data as in Fig. 5.10)

```
%MPF   9773
:
Machining program
:
N500   G54   T200                                          T No. probe; select ZO
N505   G00   X180  Y130                                    Position probe in X and Y axes at
                                                            shaft centre point
N510   Z101  D99                                           Position Z axis above shaft
N515   R10=2030     R11=10       R19=-30      R22=1       Define parameters for measuring cycle
       R23=2        R25=0        R27= 1       R28=1
       R29=3        R33=0.002    R34= 0.03    R36=1
       R37=0.06     R40=0.03     R41=-0.03    R42=130

N530   L977                                                Cycle call for shaft measurement
                                                            in X/Y
N535   G00   Z160                                          Run up Z axis
N540   M30
```

© Siemens AG 1990 All Rights Reserved

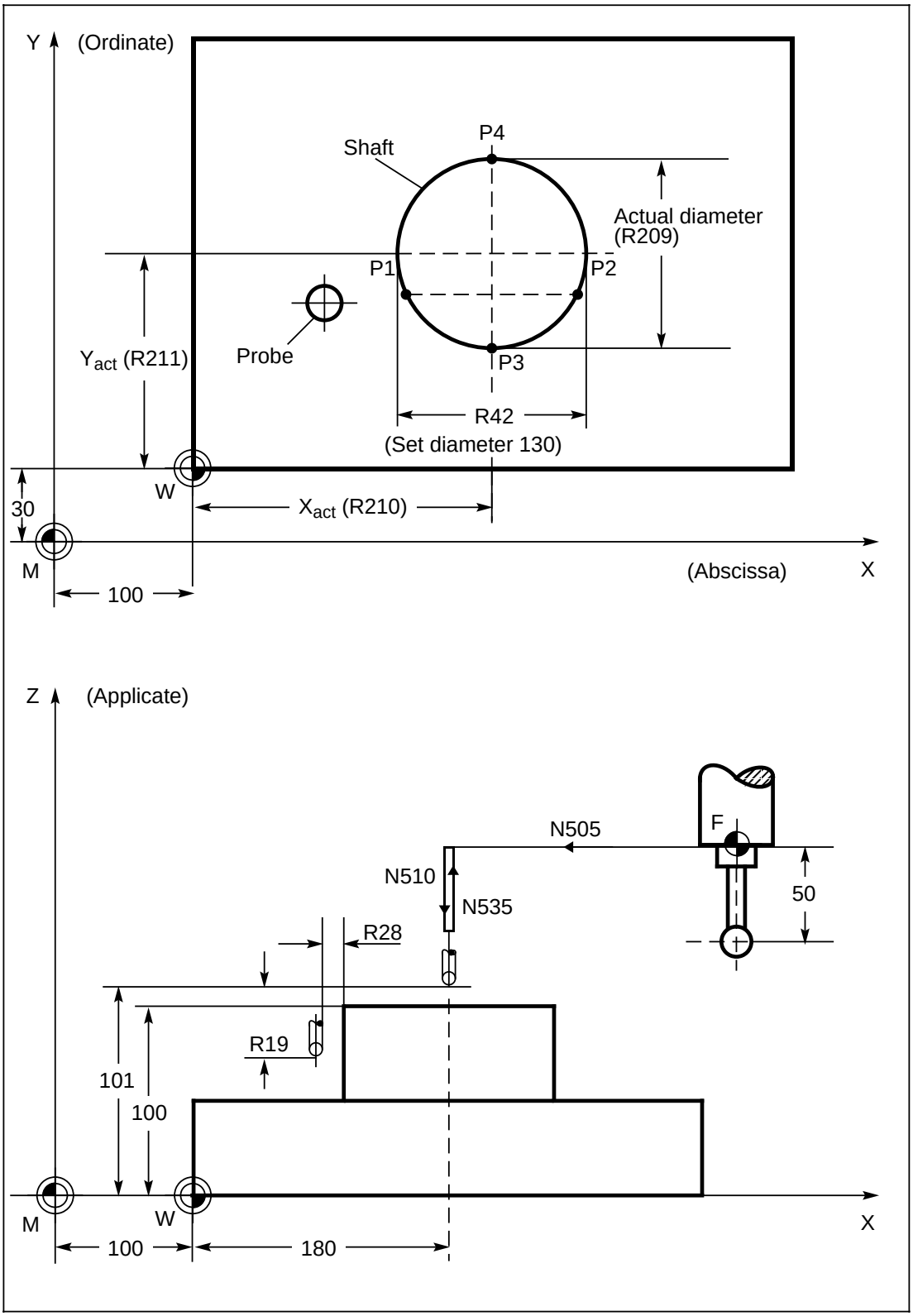


Fig. 5.8 Measure shaft

© Siemens AG 1990 All Rights Reserved

5.2.3 L977 Measure slot (paraxial)

Function and application

The measuring cycle measures points P1 and P2 inside the slot in the measuring axis. These two measured values are used to calculate the actual value of the slot and the position of the slot centre point in the measuring axis in relation to the workpiece zero point (see Fig. 5.11).

The probe is located at the determined slot centre point on completion of measurement.

Refers to the slot width only:

An empirical value stored in the SDC is allowed for with the correct sign.

In addition, averaging is performed over several parts and the tolerance bands are checked, (see Section 1.7.1).

No automatic offset is carried out or alternatively length compensation or radius offset (difference halved) is performed, depending on the definition of R10.

Preconditions:

- The probe must be called **with** tool offset and **without G53**.
- The probe must be positioned at the slot centre point in the measuring axis, the probe ball being positioned in the applicate inside the slot.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type**/**probe number (see Section 2.6)
R23 = 11	Measure slot
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measuring path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value slot (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure slot with measuring cycle L977
(Data as in Fig. 5.11)

```
%MPF   9774
:
Machining program
:
N500   G54 T200                                              T number probe; select ZO
N505   G00 X150 Y130                                        Position probe in Y axis at slot
                                                              centre point and in the X axis
                                                              at measuring position
N510   Z40 D99                                              Position Z axis inside slot
N515   R10=2030    R11=10      R22= 1      R23=11           Define parameters for measuring cycle
       R25=0       R27=1       R28= 1      R29=3
       R30=2       R33=0.002   R34= 0.03   R36=1
       R37=0.06    R40=0.03    R41=-0.03   R42=100
N525   L977                                                 Cycle call for slot measurement in Y
N530   G00 Z160                                             Withdraw Z axis from slot
N535   M30
```

© Siemens AG 1990 All Rights Reserved

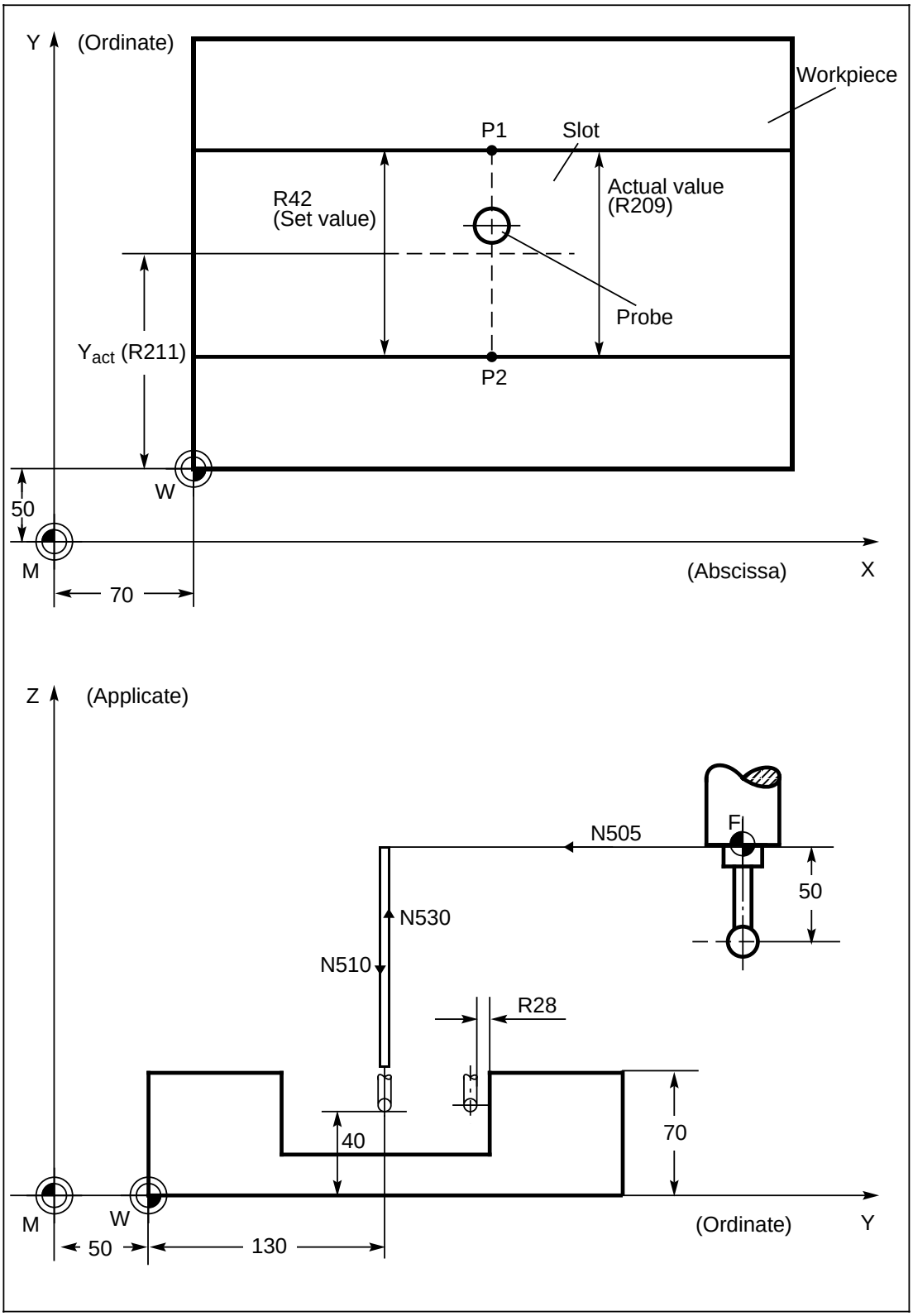


Fig. 5.9 Measure slot

© Siemens AG 1990 All Rights Reserved

5.2.4 L977 Measure web (paraxial)

Function and application

The measuring cycle measures two parallel surfaces (web) at points P1 and P2 in the measuring axis. These two measured values are used to calculate the actual distance of the parallel surfaces and the position of the centre point in the measuring axis (Fig. 5.12).

The probe is located above the determined centre point on completion of measurement.

Refers to web width only:

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, averaging is performed over a number of parts and the tolerance bands are checked (see Section 1.7.1).

No automatic compensation is carried out or alternatively length compensation or a radius offset (difference is halved) is performed, depending on the definition of R10.

Preconditions:

- The probe must be called **with** tool offset and **without G53**.
- The probe must be positioned on the centre point of the parallel surfaces and the probe ball approx. 1 mm above the parallel surfaces in the applicate.
- R19 is calculated incrementally from the starting position.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R19	Incremental infeed of applicate with sign (travel over the web)
R22	Probe type/probe number (see Section 2.6)
R23 = 12	Measure web
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value web (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure web with measuring cycle L977
(Data as in Fig. 5.12)

```
%MPF   9775
:
Machining program
:
N500   G54         T200                                    T number probe; select ZO
N505   G00         X220          Y130                      Position probe in X axis above
                                                           slot centre point and in the
                                                           Y axis at measuring position
N510   Z101 D99                                           Position Z axis above web
N515   R10=2030    R11=10        R19=-40      R22= 1       Define parameters for measuring
                                                           cycles
       R23=12      R25=0         R27= 1       R28= 1
       R29=3       R30=1         R33= 0.002   R34= 0.03
       R36=1       R37=0.06      R40= 0.03    R41=-0.03
       R42=60
N525   L977                                               Cycle call for web measurement
                                                           in X direction
N530   G00 Z160                                           Run up Z axis
N535   M30
```

© Siemens AG 1990 All Rights Reserved

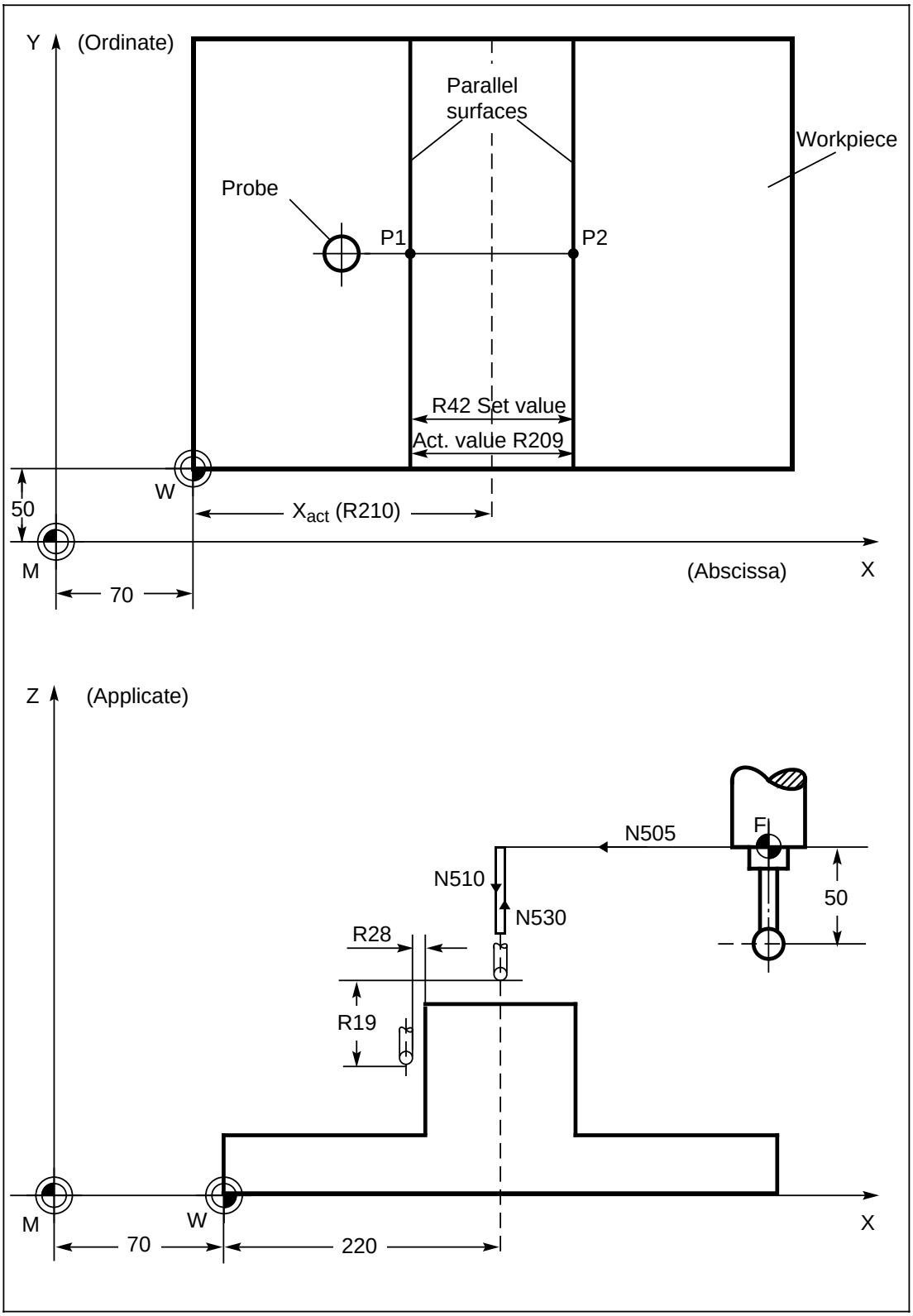


Fig. 5.10 Measure web

© Siemens AG 1990 All Rights Reserved

5.2.5 L977 ZO determination in hole (paraxial)

Function and application

The measuring cycle measures points P1, P2, P3 and P4 in the abscissa and ordinate. These four measured values are used to calculate the position of the hole centre point in the abscissa and ordinate in relation to the workpiece zero point (see Fig. 5.13).

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

The difference is determined from the set centre point (start position) and the centre point determined.

The multiplication factor for measurement path ”2a” makes it possible to take into account the variation range of the blanks (set value).

The probe is located at the centre of the hole when measurement is completed.

No automatic ZO entry is carried out or alternatively additive input of the difference between the two measuring axes in the ZO memory specified is performed, depending on the definition of R10.

Preconditions:

- The probe must be called **with** tool offset and **without G53**.
- The spindle centre point must be positioned at the hole centre point in the abscissa and ordinate, the probe ball being positioned inside the hole in the applicate.

The following must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO determination Automatic ZO entry in ZO G54 ... G57 Automatic ZO entry in ZO G58
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type/probe number (see Section 2.6)
R23 = 21	ZO determination in hole
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R32	Set value diameter
R36	Safe area (centre point coordinates)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: ZO determination in hole with measuring cycle L977
(Data as in Fig. 5.13)

```
%MPF    9776
N5      G54 T200                                                    T number probe; select ZO
N10     G00 X180 Y130                                               Position probe in X and Y axes
                                                                      at hole centre point
N15     Z20 D99                                                     Position Z axis in hole
N20     R10=1       R22=1       R23=21      R25=0                   Define parameters for measuring cycle
        R27=1       R28=1       R32=130     R36=1
N25     L977                                                        Cycle call for ZO determination in X/Y
N30     G00 Z160                                                    Withdraw Z axis from hole
:
Machining program
:
:
N...    M30
```


© Siemens AG 1990 All Rights Reserved

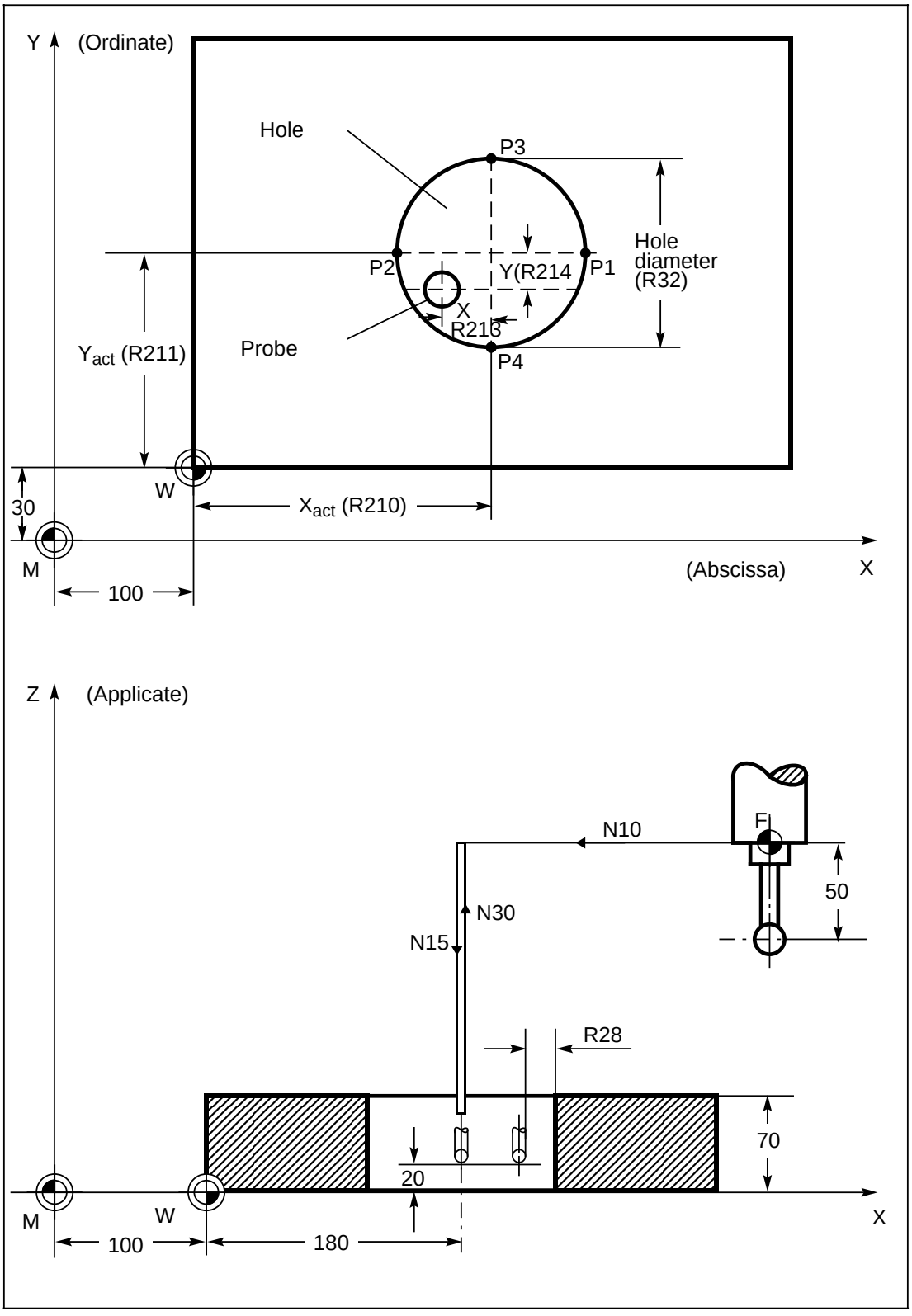


Fig. 5.11 ZO determination in hole

5.2.6 L977 ZO determination on shaft

Function and application

The measuring cycle measures points P1, P2, P3 and P4 in the abscissa and ordinate. These four measured values are used to calculate the position of the shaft centre point in the abscissa and ordinate in relation to the workpiece zero point (see Fig. 5.14).

Points P1 and P2 are used to calculate the centre point of the abscissa. The probe is then positioned at the centre point calculated and points P3 and P4 are measured. These two points provide the shaft centre point of the ordinate.

The difference is determined from the set centre point (start position) and the centre point determined.

The multiplication factor for measurement path ”2a” makes it possible to take into account the variation range of the blanks (set value).

The probe is located above the shaft centre point when measurement is completed.

No automatic ZO entry is carried out or alternatively additive input of the difference of the two measuring axes is performed in the ZO memory specified, depending on the definition of R10.

Preconditions:

- The probe must be called **with** tool offset and **without G53**.
- The probe must be positioned at the shaft centre point in the abscissa and ordinate, the probe ball being positioned approx. 1 mm above the shaft in the applicate.
- R19 is calculated incrementally from the starting position.

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO determination Automatic ZO entry in ZO G54 ... G57 Automatic ZO entry in ZO G58
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R19	Incremental infeed of applicate with sign (travel over the shaft)
R22	Probe type/probe number (see Section 2.6)
R23 = 22	ZO determination at shaft
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R32	Set value diameter
R36	Safe area (centre point coordinates)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: ZO determination on shaft with measuring cycle L977
(Data as in Fig. 5.14)

```
%MPF    9777
N5      G54 T200                                                    T number probe; select ZO
N10     G00 X180 Y130                                               Position probe in X and Y axes
                                                                      at shaft centre point
N15     Z101 D99                                                    Position Z axis above shaft
N20     R10=1       R19=-30     R22=1       R23=22                  Define parameters for measuring cycle
        R25=0       R27= 1      R28=1       R32=130
        R36=1
N25     L977                                                        Cycle call for ZO determination in X/Y
N30     G00 Z160                                                    Run up Z axis
:
Machining program
:
:
N...    M30
```

© Siemens AG 1990 All Rights Reserved

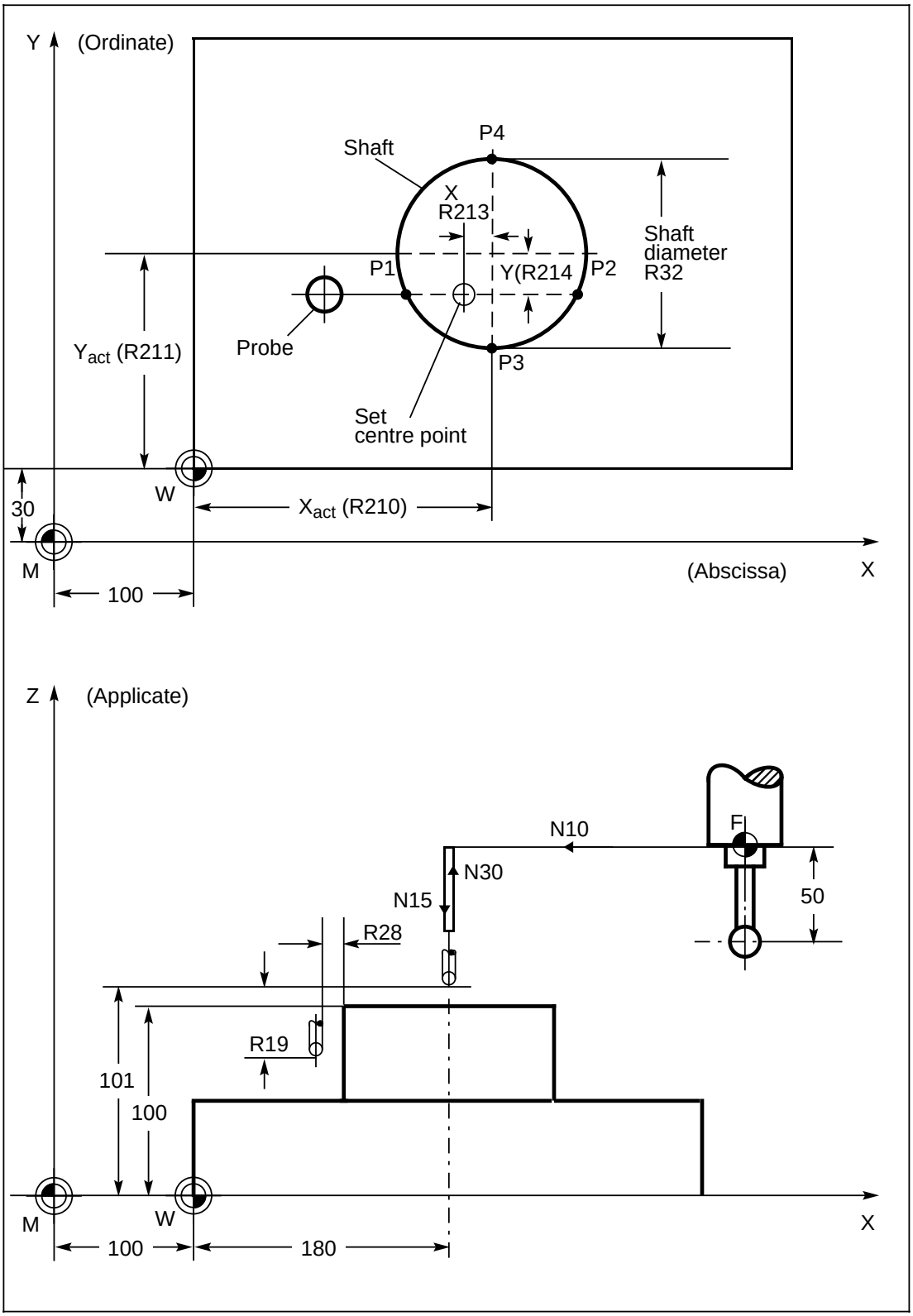


Fig. 5.12 ZO determination at shaft

© Siemens AG 1990 All Rights Reserved

5.2.7 L977 ZO determination in slot (paraxial)

Function and application

The measuring cycle measures points P1 and P2 in the measuring axis inside the slot. The two measured values are used to calculate the position of the slot centre point in the measuring axis in relation to the workpiece zero point (see Fig. 5.15).

The difference is determined from the set centre point (start position) and the centre point determined.

The multiplication factor for measurement path ”2a” makes it possible to take into account the variation range of the blanks (set value).

The probe is located above the slot centre determined when measurement is completed.

No automatic ZO entry is carried out or alternatively additive input of the measuring axis difference is performed in the ZO memory specified, depending on the definition of R10.

Preconditions :

- The probe must be called **with** tool offset and **without G53**.
- The probe must be positioned at the slot centre point in the measuring axis, the probe ball being positioned inside the slot in the applicate.

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO determination Automatic ZO entry in ZO G54 ... G57 Automatic ZO entry in ZO G58
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe type/probe number (see Section 2.6)
R23 = 31	ZO determination in slot
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis (see Section 2.12)
R32	Set value slot width
R36	Safe area (centre point coordinates)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: ZO determination in slot with measuring cycle L977
(Data as in Fig. 5.15)

```
%MPF    9778
N5      G54 T200                                                    T number probe; select ZO
N10     G00 X150 Y130                                               Position probe in Y axis at
                                                                      slot centre point and
                                                                      in X axis at measuring position
N15     Z40 D99                                                     Position Z axis in slot
N20     R10=1       R22=1       R23=31      R25=0                   Define parameters for measuring cycle
        R27=1       R28=1       R30=2       R32=100
        R36=1
N25     L977                                                        Cycle call for ZO determination in Y
N30     G00 Z160                                                    Withdraw Z axis from slot
:
Machining program
:
:
N...    M30
```


© Siemens AG 1990 All Rights Reserved

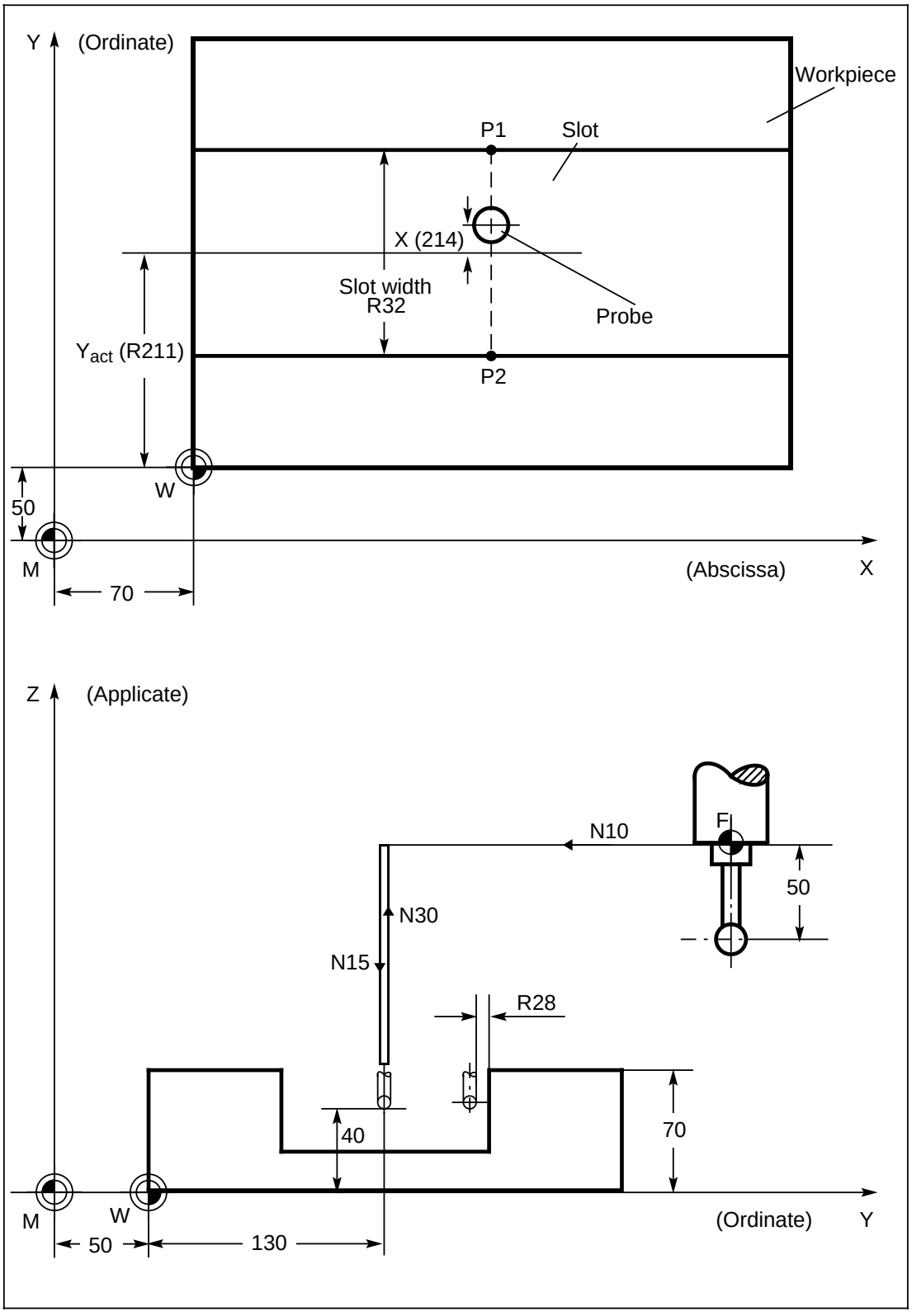


Fig. 5.13 ZO determination in slot

© Siemens AG 1990 All Rights Reserved

5.2.8 L977 ZO determination on a web (paraxial)

Function and application

The measuring cycle measures points P1 and P2 in the measuring axis on two parallel surfaces (web). The two measured values are used to calculate the position of the centre point in the measuring axis in relation to the workpiece zero point (see Fig. 5.16).

The difference is calculated from the set centre point (start position) and the centre point determined.

The multiplication factor for measurement path ”2a” makes it possible to take into account the variation range of the blanks (set value).

The probe is located above the centre point of the parallel surfaces on completion of measurement.

No automatic ZO entry is carried out or alternatively additive input of the measuring axis difference is performed in the ZO memory specified, depending on the definition of R10.

Preconditions:

- The probe must be called **with** tool offset and **without G53**.
- The spindle centre point must be positioned at the centre point of the parallel surfaces in the measuring axis, the probe ball being positioned above the parallel surfaces in the applicate.
- R19 is calculated incrementally from the starting position.

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO determination Automatic ZO entry in ZO G54 ... G57 Automatic ZO entry in ZO G58
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R19	Incremental infeed of applicate with sign (travel over the web)
R22	Probe type/probe number (see Section 2.6)
R23 = 32	Measure web
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1...3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis (see Section 2.12)
R32	Set value web width
R36	Safe area (centre point coordinate)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: ZO determination on a web with measuring cycle L977
(Data as in Fig. 5.16)

```
%MPF    9779
N5      G54 T200                                                    T number probe; select ZO
N10     G00 X220 Y130                                               Position probe in X axis
                                                                      above web centre point
                                                                      and in Y axis at measuring position
N15     Z101 D99                                                    Position Z axis above web
N20     R10=1       R19=-40     R22=1       R23=32                  Define parameters for measuring cycle
        R25=0       R27= 1      R28=1       R30=1
        R32=130     R36=1
N25     L977                                                        Cycle call for ZO determination in X
N30     G00 Z160                                                    Run up Z axis
:
Machining program
:
:
N ...   M30
```

© Siemens AG 1990 All Rights Reserved

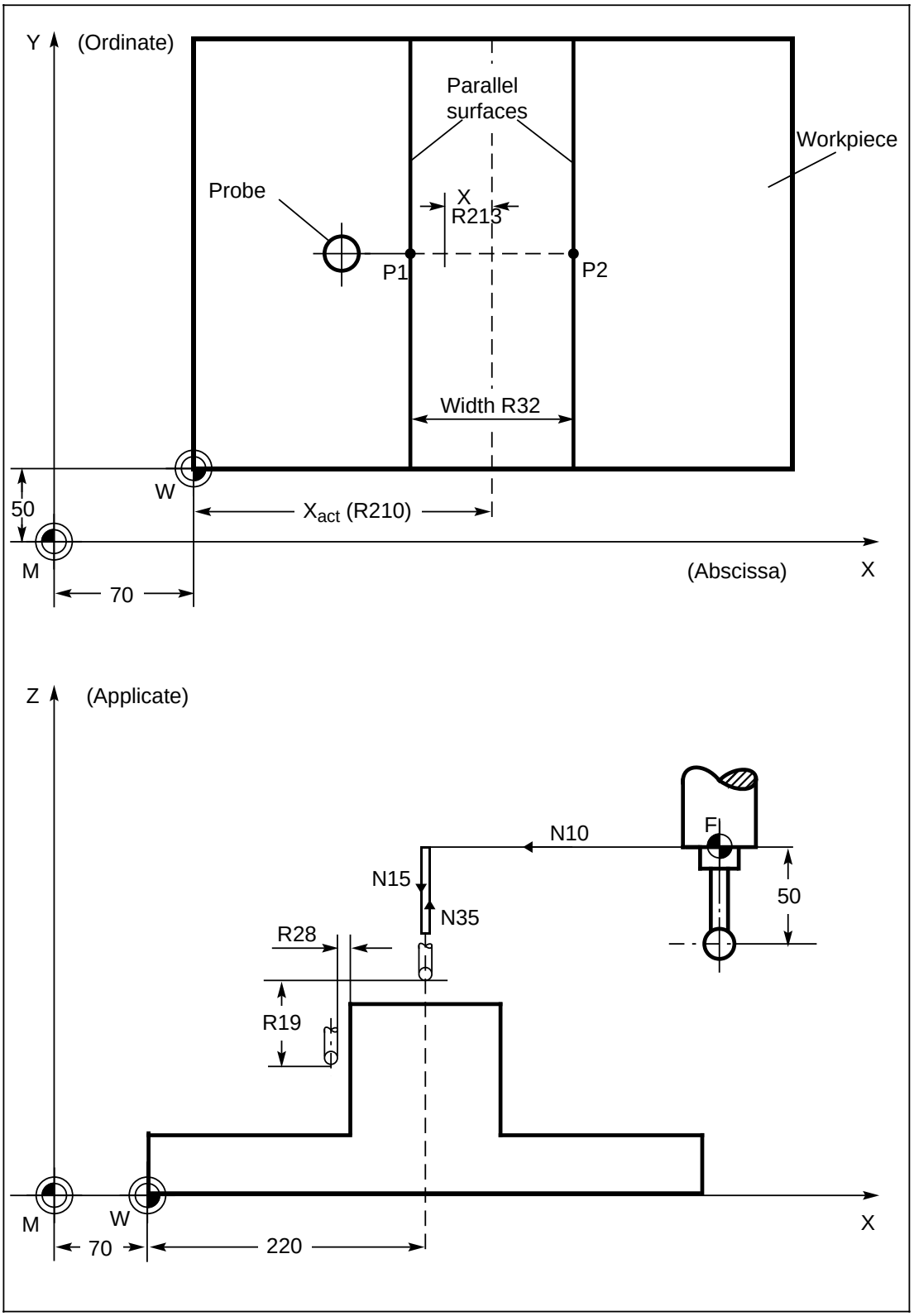


Fig. 5.14 ZO determination on a web

© Siemens AG 1990 All Rights Reserved

5.3 L979 Workpiece measurement hole/shaft/slot/web/ZO determination (at random angles)

Preconditions :

- Circle radius programming
- Polar coordinate programming
- Oriented spindle stop (M19) via NC with axis movement
- Optional positioning of probe in spindle between 0 and 360 degrees (all-round coverage)

The cycle has the same range of functions and the same preconditions as L977.

The hole or shaft is determined by this cycle by means of 3-point or 4-point measurement. It is thus possible to measure **circle segments** whose centre points are located outside the machine (see Fig. 5.17).

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

The probe is positioned from P1 to P2, from P2 to P3 and from P3 to P4 with circular interpolation. The R28 distance between the probe and the contour is maintained. The probe is positioned at the relevant angle of measurements with M19 to ensure that P1 - P4 are always measured with the **same** probe point.

The probe must be calibrated in the plane. If a multidirectional probe is used, calibration must include determination of the position deviation (e.g. measuring variant R23=10002). The spindle must be positioned with 0°.

If cycle machine data MDC 7004.0=1 (spindle offset), parameter R13 takes effect even if a multidirectional probe is in use and can be used to adapt the positioning direction of the spindle.

- R13=0 Positioning direction of the spindle corresponds to M3
- R13=-360 Opposite positioning direction

At the end of the cycle, the probe is facing P3 (P4) by the amount of R28 (facing P2 in the case of slot measurement).

The measurement variant is selected by defining R23:

R23 = 1	Measure hole
R23 = 2	Measure shaft
R23 = 11	Measure slot
R23 = 12	Measure web
R23 = 21	ZO determination in hole
R23 = 22	ZO determination at shaft
R23 = 31	ZO determination in slot
R23 = 32	ZO determination at web

Prepositioning: Always facing and level with P1

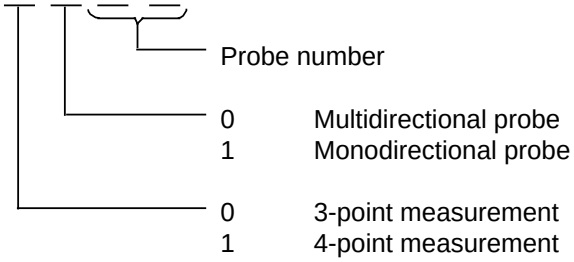
© Siemens AG 1990 All Rights Reserved

Applicable types of probe (see Section 1.3)

- In the case of milling and machining centres the probe must be entered as type 30 in the TOA memory.
- Multidirectional probe
- Bidirectional probe
- Monodirectional probe

The various types of probe and the number of measuring points are differentiated by defining R22 (see Section 2.6)

R22 = 4 3 2 1



An internal call of L963 is made by the cycle for calculation, parameter R00 is used for defining L963. Depending on the type of control, cycle L963 takes up to approx. 0.5 to 10 seconds for the 4-point measurement!

© Siemens AG 1990 All Rights Reserved

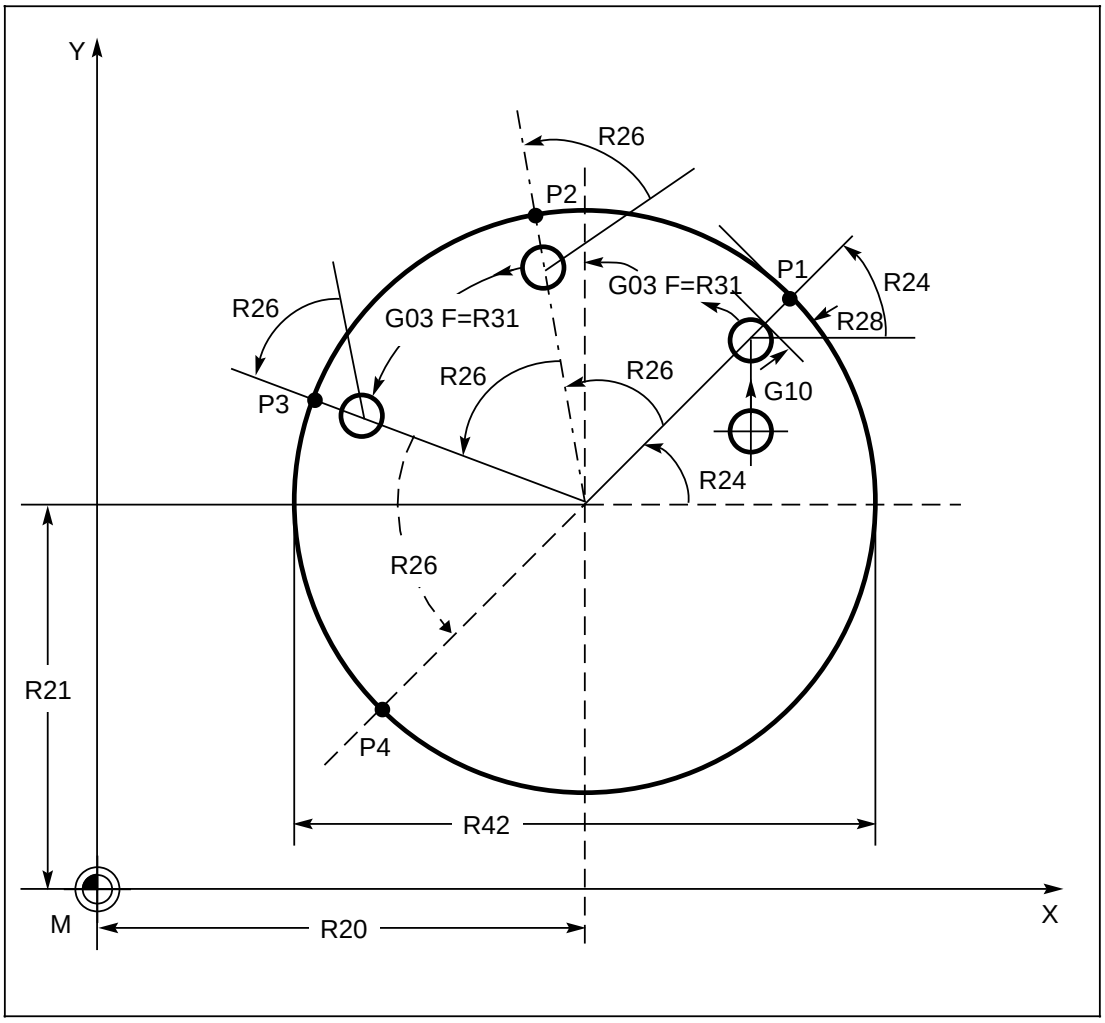


Fig. 5.15 Measure circle segment

© Siemens AG 1990 All Rights Reserved

5.3.1 L979 Measure hole (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Number of measuring points**/**probe type**/**probe number
R23 = 1	Measure hole
R24 = 0...360	Start angle (with reference to abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm**/**min Standard cycle value
R26 = 0...360	Indexing angle
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R31	Velocity for circular interpolation
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value diameter (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure hole with measuring cycle L979
(Data as in Fig. 5.18)

```
%MPF   9791
:
Machining centre program
:
N500   G54 T200                                                    T number probe; select ZO
N505   G00 X240 Y130                                              Position probe in X and Y axes
                                                                    near P1
N510   Z20 D99                                                     Position Z axis level with P1
N515  R10=2030       R11=10  R13=0   R20= 180  R21=130           Define parameters for measuring cycle
      R22=1   R23=1  R24= 10 R25=0
      R26=90  R27=1  R28= 1  R29=3
      R31=1000       R33=0.002      R34= 0.03 R36=1
      R37=0.06       R40=0.03       R41=-0.03 R42=130
N525   L979                                                        Cycle call for hole measurement
                                                                    in X/Y
N530   G00 Z160                                                   Withdraw Z axis from hole
N535   M30
```

© Siemens AG 1990 All Rights Reserved

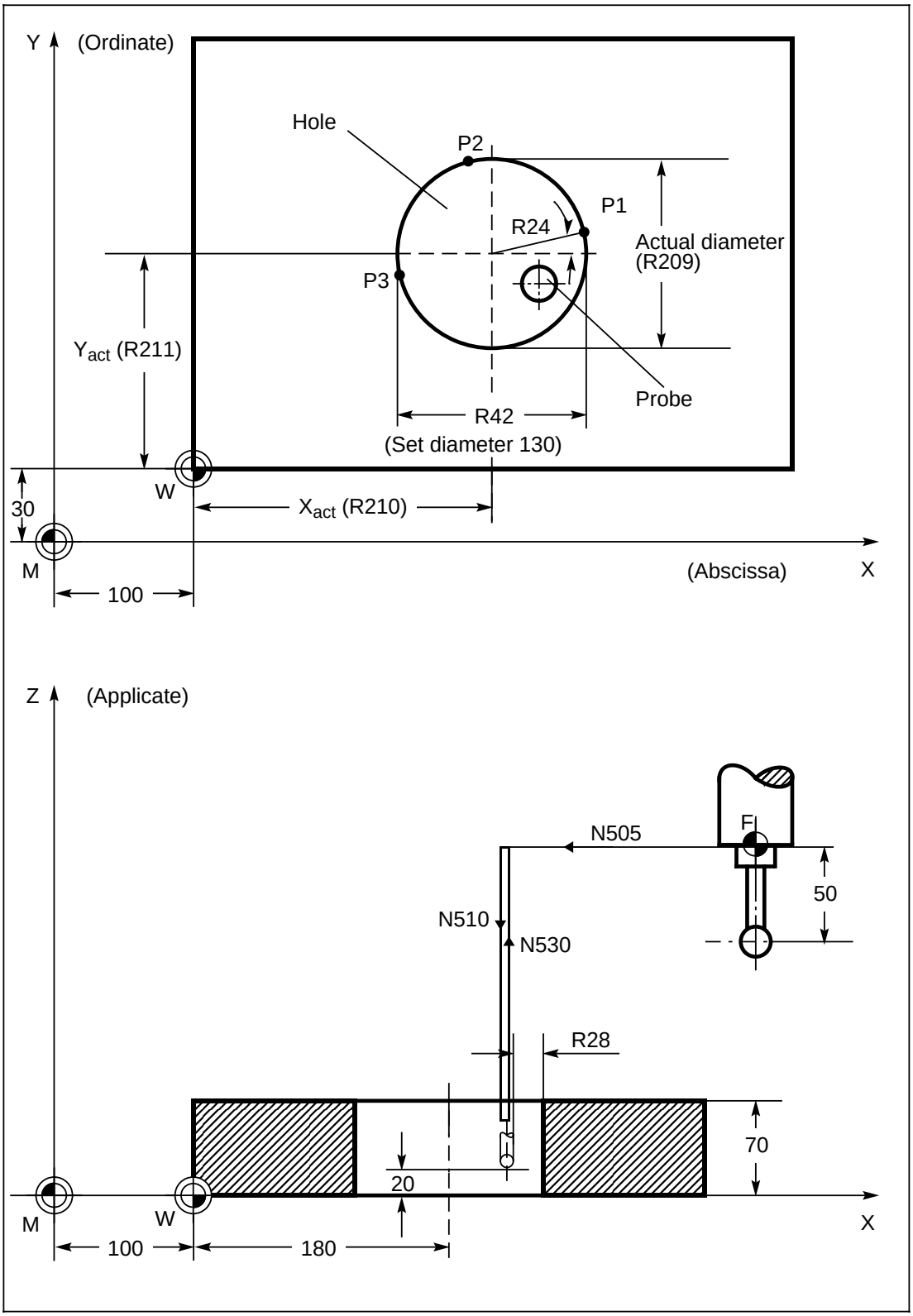


Fig. 5.16 Measure hole

© Siemens AG 1990 All Rights Reserved

5.3.2 L979 Measure shaft (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory number for diameter) (see Section 2.3)
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Number of measuring points**/**probe type**/**probe number
R23 = 2	Measure shaft
R24 = 0...359.5	Start angle (with reference to abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm**/**min Standard cycle value
R26 = 0...359.5	Indexing angle
R27 = 1...R_{max}	Number of measurements at same locations (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R31	Velocity for circular interpolation
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value diameter (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure shaft with measuring cycle L979

(Data as in Fig. 5.19)

```
%MPF    9792
:
Machining centre program
:
N500    G54 T200                                                    T number probe; select ZO
N505    G00 X255 Y130                                               Position probe in X and
                                                                      Y axes near P1
N510    Z20 D99                                                     Position Z axis level with P1
N515    R10=2030     R11=10    R13=0     R20= 180  R21=130         Parameter für Meßzyklus definieren
        R22=1 R23=2  R24= 10   R25=0
        R26=90R27=1  R28= 1    R29=3
        R31=1000     R33=0.002 R34= 0.03 R36=1
        R37=0.06     R40=0.03  R41=-0.03 R42=130

N525    L979                                                        Cycle call for shaft measurement
                                                                      in X/Y
N530    G00 Z160                                                    Run up Z axis

N535    M30
```


© Siemens AG 1990 All Rights Reserved

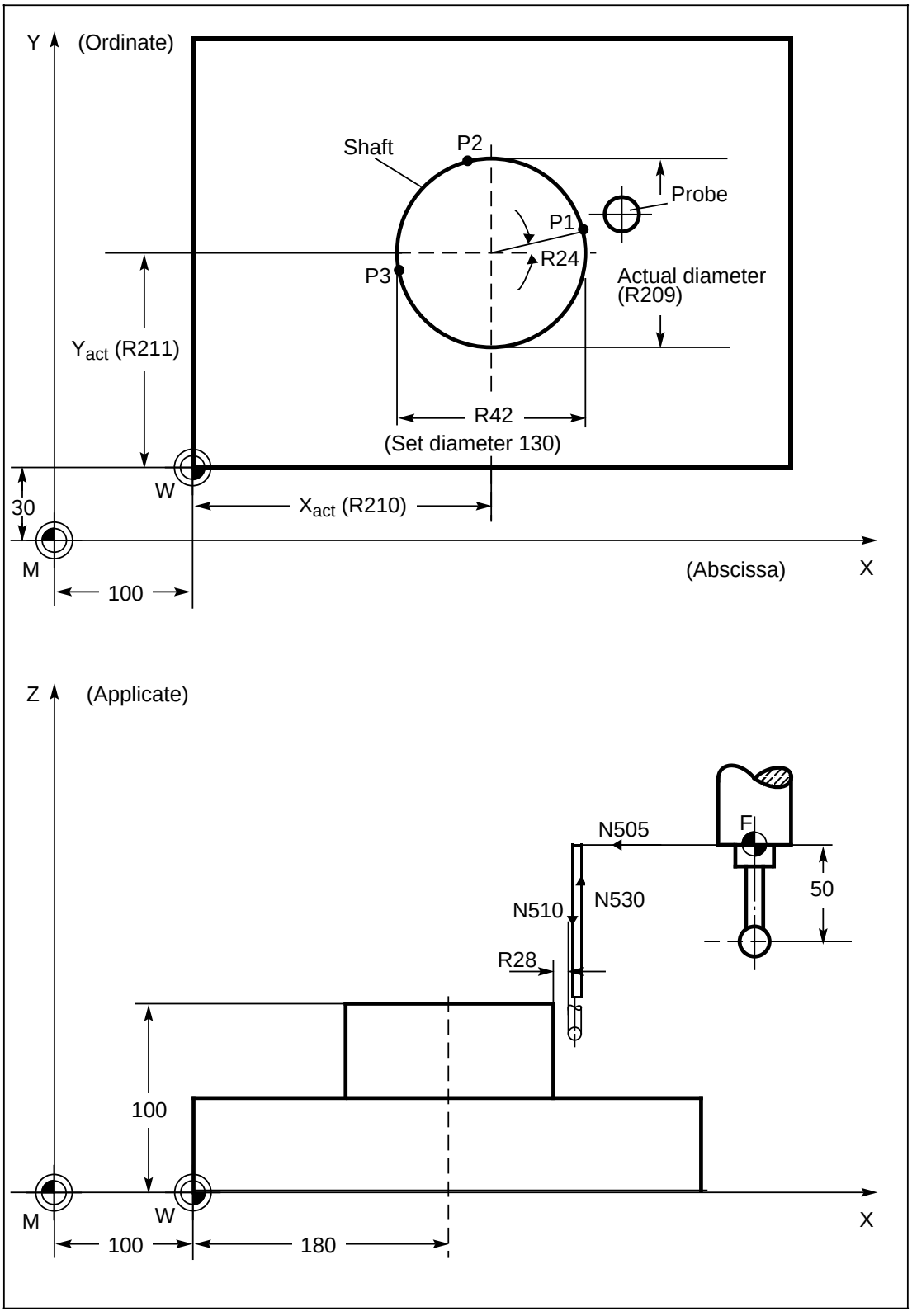


Fig. 5.17 Measure shaft

5.3.3 L979 Measure slot (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Probe type**/**probe number
R23 = 11	Measure slot
R24 = 0...360	Start angle (with reference to abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm**/**min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set point slot (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure slot with measuring cycle L979
(Data as in Fig. 5.20)

```
%MPF   9793
:
Machining centre program
:
N500   G54 T200                                          T number probe; select ZO
N505   G00 X150 Y170                                     Position probe in X and
                                                          Y axes near P1
N510   Z40 D99                                           Position Z axis level with P1
N515   R10=2030    R11= 10     R20=150     R21=130      Define parameters for measuring cycle
       R22=1       R23= 11     R24=70      R25=0
       R27=1       R28= 1      R29=3
       R33=0.002   R34= 0.03   R36=1       R37=0.06
       R40=0.03    R41=-0.03   R42=100
N525   L979                                              Cycle call for slot measurement
                                                          in X/Y
N530   G00 Z160                                          Run up Z axis
N535   M30
```

© Siemens AG 1990 All Rights Reserved

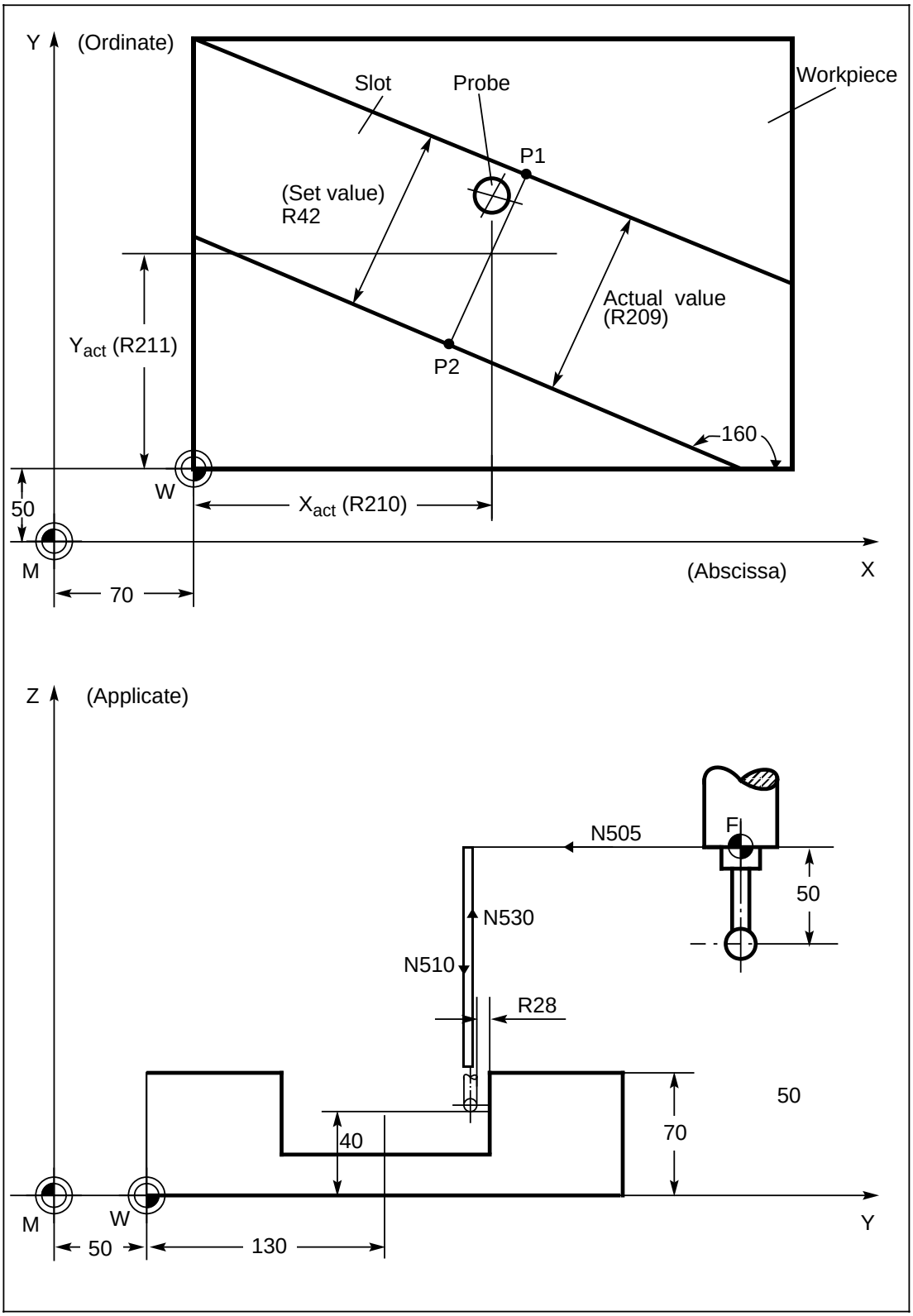


Fig. 5.18 Measure slot

© Siemens AG 1990 All Rights Reserved

5.3.4 L979 Measure web (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R19	Incremental infeed of hole axis with sign (travel over web)
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Probe type**/**probe number
R23 = 12	Measure web
R24 = 0...359.5	Start angle (with reference to abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm**/**min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R33	Zero offset range
R34	Compensation area with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set point web (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Measure web with measuring cycle L979
(Data as in Fig. 5.21)

```
%MPF    9794
:
Machining centre program
:
N500    G54   T200                                          T number probe; select ZO
N505    G00   X260 Y130                                     Position probe in X and
                                                              Y axes near P1
N510    Z70 D99                                             Position Z axis level with P1
N515    R10=2030     R11=10       R19= 35      R20=220      Define parameters for measuring cycle
        R21=130      R22=1        R23= 12      R24=10
        R25=0        R27=1        R28= 1       R29=3
                     R33=0.002    R34= 0.03    R36=1
        R37=0.06     R40=0.03     R41=-0.03    R42=100
N525    L979                                                Cycle call for web measurement
                                                              in X/Y
N530    G00 Z160                                            Run up Z axis
N535    M30
```


© Siemens AG 1990 All Rights Reserved

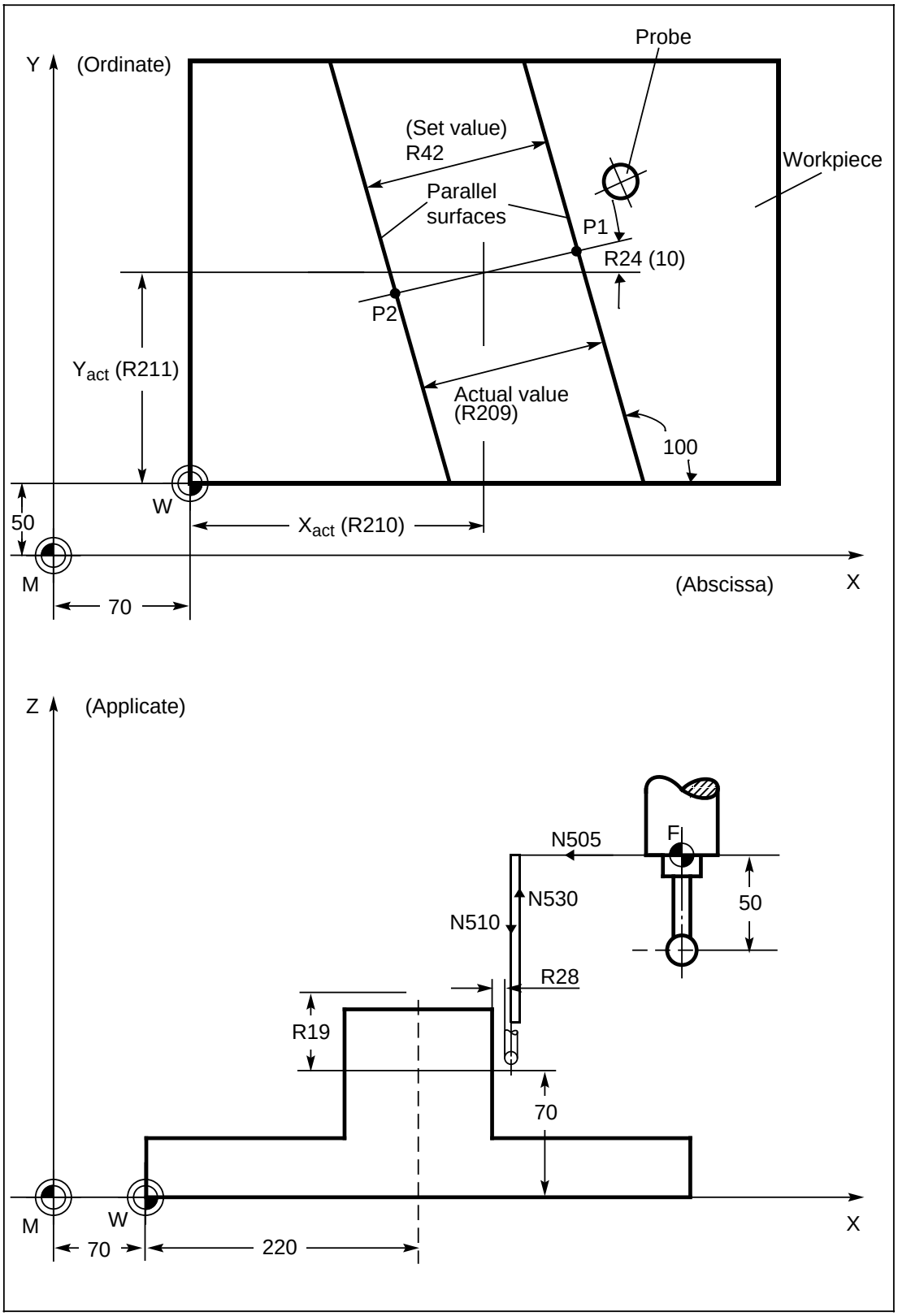


Fig. 5.19 Measure web

5.3.5 L979 ZO determination in hole (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO entry Automatic ZO entry in ZO G54 ... G57 Automatic ZO entry in ZO G58
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22 _	Number of measuring points/probe type/probe number
R23 = 21	ZO determination in hole
R24 = 0...359.5	Start angle
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R26 = 0...359.5	Indexing angle
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R31	Velocity for circular interpolation
R32	Set value diameter
R36	Safe area (centre point coordinates)

See Section 10.2 for result display parameters.

Example: ZO determination in hole with measuring cycle L979 (Data as in Fig. 5.22)

```
%MPF    9795
N5      G54 T200                                                    T number probe; select ZO
N10     G00 X240 Y130                                               Position probe in X and
                                                                      Y axes near P1
N15     Z20 D99                                                     Position Z axis level with P1
N20     R10=1       R20=180     R21=130     R22=1                   Define parameters for measuring cycles
        R23=21      R24=10      R25=0       R26=90
        R27=1       R28=1       R31=1000    R32=130
        R36=1
N25     L979                                                        Cycle call for ZO determination in X/Y
N30     G00 Z160                                                    Withdraw Z axis from hole
:
Machining centre program
:
:
N...    M30
```


© Siemens AG 1990 All Rights Reserved

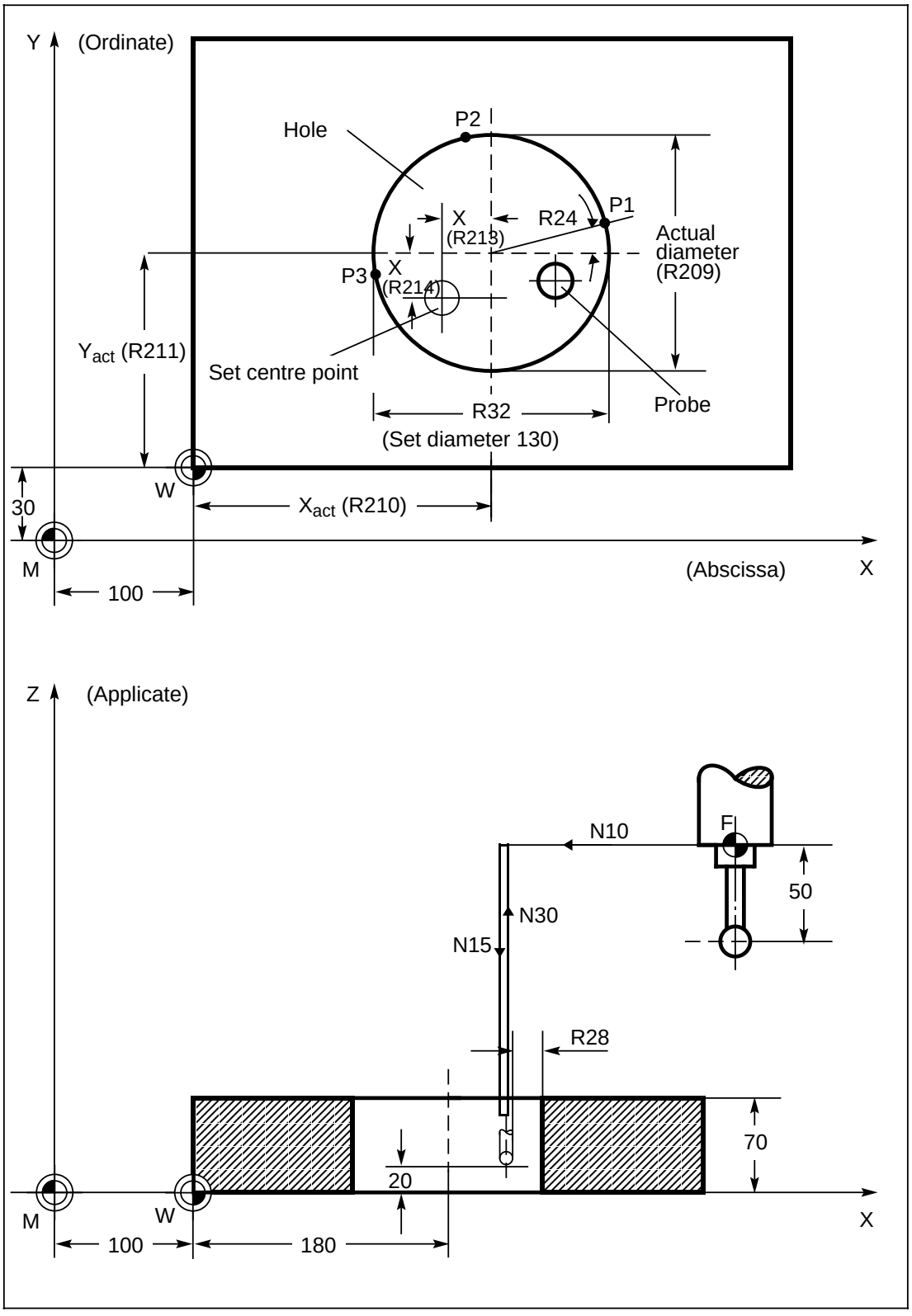


Fig. 5.20 ZO determiantion in hole

5.3.6 L979 ZO determination at shaft (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO entry Automatic ZO entry in ZO G54 ... G57 Automatic ZO entry in ZO G58
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Number of measuring points/probe type/probe number
R23 = 22	ZO determination at shaft
R24 = 0...359.5	Start angle (from the abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R26 = 0...359.5	Indexing angle
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R31	Velocity for circular interpolation
R32	Set value diameter
R36	Safe area (centre point coordinates)

See Section 10.2 for result display parameters.

Example: ZO determination at shaft with measuring cycle L979
(Data as in Fig. 5.23)

```
%MPF    9796
N5      G54 T200                                          T number probe; select ZO
N10     G00 X250 Y130                                     Position probe in X and
                                                            Y axes near P1
N15     Z70 D99                                           Position Z axis level with P1
N20     R10=1       R20=180     R21=130     R22=1         Define parameters for measuring cycle
        R23=22      R23=21      R24=10      R25=0
        R26=90      R27=1       R28=1       R31=1000
        R32=130     R36=1
N25     L979                                              Cycle call for ZO determination in X/Y
N30     G00 Z160                                          Withdraw Z axis
:
Machining centre program
:
:
N...    M30
```


© Siemens AG 1990 All Rights Reserved

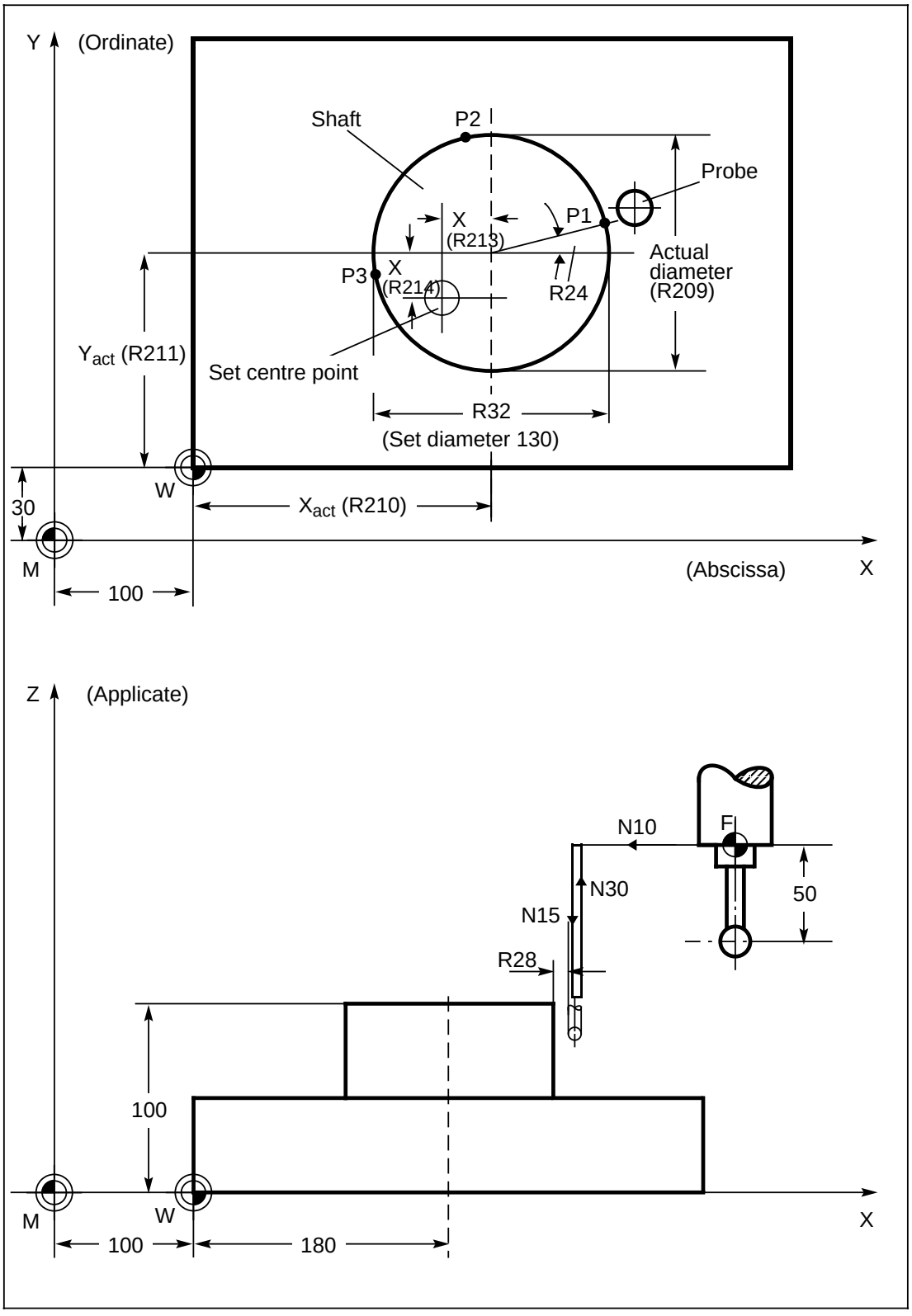


Fig. 5.21 ZO determination at shaft

5.3.7 L979 ZO determination in slot (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO entry Automatic ZO entry in ZO G54...G57 Automatic ZO entry in ZO G58
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Probe type/probe number
R23 = 31	ZO determination in slot
R24 = 0...359.5	Start angle (from the abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R32	Set value slot width
R36	Safe area (centre point coordinate)

See Section 10.2 for result display parameters.

Example: ZO determination in slot with measuring cycle L979
(Data as in Fig. 5.24)

```
%MPF    9797
N5      G54 T200                                          T number probe; select ZO
N10     G00 X150 Y175                                     Position probe in X and
                                                            Y axes near P1
N15     Z40 D99                                           Position Z axis level with P1
N20     R10=1       R20=150     R21=130     R22=1         Define parameters for measuring cycle
        R23=31      R24=70      R25=0       R27=1
        R28=1       R32=100     R36=1
N25     L979                                              Cycle call for ZO determination in X/Y
N30     G00 Z160                                          Withdraw Z axis
:
Machining centre program
:
:
N...    M30
```


© Siemens AG 1990 All Rights Reserved

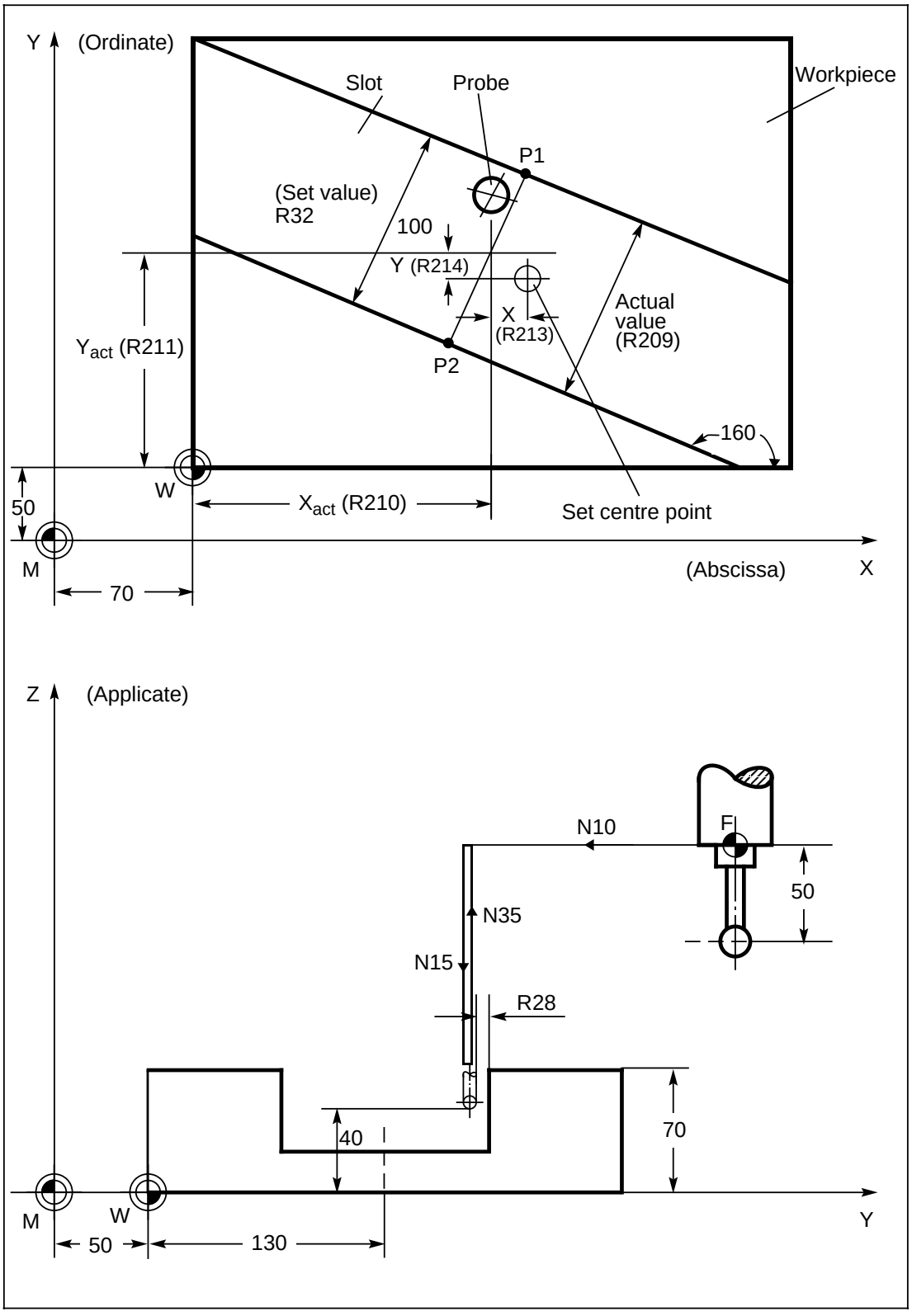


Fig. 5.22 ZO determination in internal slot

5.3.8 L979 ZO determination at web (at random angles)

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO entry Automatic ZO entry in ZO G54...G57 Automatic ZO entry in ZO G58
R13 = 0...359.5 R13 = 0/-360	Compensation angle position for monodirectional probe (see Section 2.5) Positioning direction of spindle with multidirectional probe and MDC 7004.0=1
R19	Incremental infeed of hole axis with sign (travel over web)
R20	Centre point abscissa (in relation to workpiece zero point)
R21	Centre point ordinate (in relation to workpiece zero point)
R22	Probe type/probe number
R23 = 32	ZO determination at web
R24 = 0...359.5	Start angle (from the abscissa (horizontal axis))
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R32	Set value web width
R36	Safe area (centre point coordinates)

See Section 10.2 for result display parameters.

Example: ZO determination at web with measuring cycle L979 (Data as in Fig. 5.25)

```
%MPF    97978
N5      G54 T200                                          T number probe; select ZO
N10     G00 X290 Y130                                     Position probe in X and
                                                            Y axes near P1
N15     D99                                               Position Z axis level with P1
N20     R10=1       R19=-35     R20=220                   Define parameters for measuring cycle
        R21=130     R22=1       R23=32      R24=10
        R25=0       R27=1       R28=1       R32=130
        R36=1
N25     L979                                              Cycle call for ZO determinationin X/Y
N30     G00 Z160                                          Withdraw Z axis
:
Machining centre program
:
:
N...    M30
```


© Siemens AG 1990 All Rights Reserved

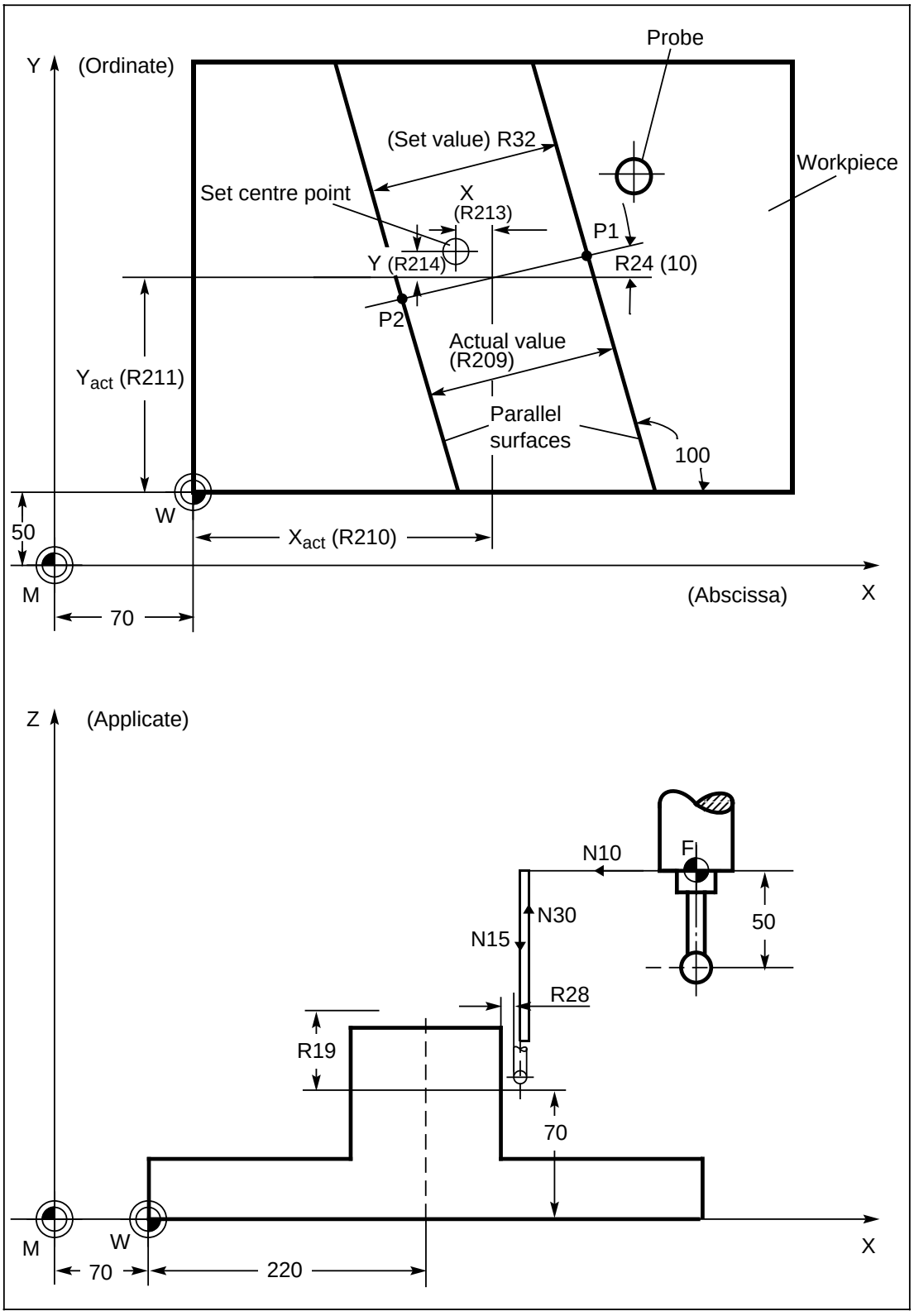


Fig. 5.23 ZO determination at web

© Siemens AG 1990 All Rights Reserved

5.4 L978 Workpiece measurement surface/angle/ZO determination (with and without differential measurement)

The measurement cycle makes it possible to measure blanks and zero offsets can be determined automatically relative to the workpiece zero point.

Zero offset can be determined either at a surface by single point measurement or by means of angular measurement between the blank (e.g. on a rotary table) and the main machine axes.

In addition, the cycle determines the dimensions of surfaces in relation to the workpiece zero point by single point measurement or paraxial multipoint measurement and performs automatic tool offset when necessary.

Differential measurements are also possible with this cycle.

Differential measurement means:

The measuring point (only the first measuring point in the case of paraxial multipoint measurement and angular measurement) is measured twice with the spindle being rotated through 180°, i.e. the probe is turned by 180° between the two measurements. In this way, the trigger points are determined. The trigger point is stored in the MDC for the relevant axis direction. An uncalibrated probe can therefore be used for measurement.

Preconditions for differential measurement:

- Oriented spindle stop (M19) by the NC
- Bidirectional/multidirectional probe
- Random positioning of probe in spindle between 0 and 360° (min. every 90°) (all-round coverage).
- **Monodirectional probe must be calibrated. Differential measurements cannot be performed with this probe.**

Selection is made by defining R23:

R23 = 0	ZO determination at surface	
R23 = 1	Measure surface	
R23 = 2	Paraxial multipoint measurement	
R23 = 3	Angular measurement	
R23 = 10	ZO determination at surface	with diff. measurement
R23 = 11	Measure surface	with diff. measurement
R23 = 12	Paraxial multipoint measurement	with diff. measurement
R23 = 13	Angular measurement	with diff. measurement

Applicable types of probe: (see Section 1.3)

- In the case of milling and machining centres the probe must be entered as type 30 in the TOA memory.
- Multidirectional probe
- Bidirectional probe
- Monodirectional probe

© Siemens AG 1990 All Rights Reserved

5.4.1 L978 ZO determination on surface (1-point measuring cycle)

Function and application

The measuring cycle determines the actual value of a blank relative to the machine zero point.

An empirical value stored in the MDC is subsequently allowed for with the correct sign.

The multiplication factor for measurement path ”2a” makes it possible to take into account the variation range of the blanks (set value).

The probe faces the measuring surface at a distance of ”a” on completion of measurement.

No automatic ZO entry is carried out or alternatively additive input of the measuring axis difference is performed in the ZO memory specified, depending on the definition of R10.

Preconditions:

The probe must be positioned **with** tool offset and **without G53** to face the surface to be measured.

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5	No automatic ZO entry Automatic ZO entry in ZO G54...G57 Automatic ZO entry in ZO G58
R11 = 0 > 0	Without empirical value Empirical value memory number (see Section 2.3)
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe number (see Section 2.6)
R23 = 0 = 10	ZO determination at surface ZO determination at surface with differential measurement
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R26 = 0...359.5	Start angle (with differential measurement only)
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R30 = 1...3	Number of measuring axis (see Section 2.12)
R32	Set value in relation to workpiece zero point
R36	Safe area

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: ZO determination on workpiece with cycle L978 in the X and Y axes
(Data as in Fig. 5.26)

ZO is to be checked on a workpiece as shown in Fig. 5.26. Any deviation from the selected ZO caused by clamping tolerances is to be compensated for automatically by additive ZO to enable machining of the workpiece to be started.

Probe length (Z axis) in TO memory D99 (value 50)

```
%MPF    9781
N5      G54 T200                                          T number probe; select ZO
N10     G00 G90 X-20 Y25                                  Position probe to face the
                                                            measuring surface in X/Y
N15     Z10 D99                                           Position probe in Z
                                                            Select TO
N20     R10=1       R11=10      R22=1       R23=0         Define parameters for measuring cycle
        R25=0       R27=1       R28=2       R30=1
        R32=0       R36=3
N25     L978                                              Cycle call for ZO determination
                                                            in X axis
N30     G00 X-20                                          Retract in X axis
N35     Y-20                                              Position in Y axis
N40     X50                                               Position in X axis
N45     R11=11 R30=2                                  Define parameters for measuring cycle
N50     L978                                              ZO determination in Y axis
N55     G00 Y-20                                          Retract in Y axis
N60     Z100                                              Retract in Z axis
N65     X-40 Y80                                          Retract in X/Y axis
:
:
Machining centre program
:
:
N900    M30
```


© Siemens AG 1990 All Rights Reserved

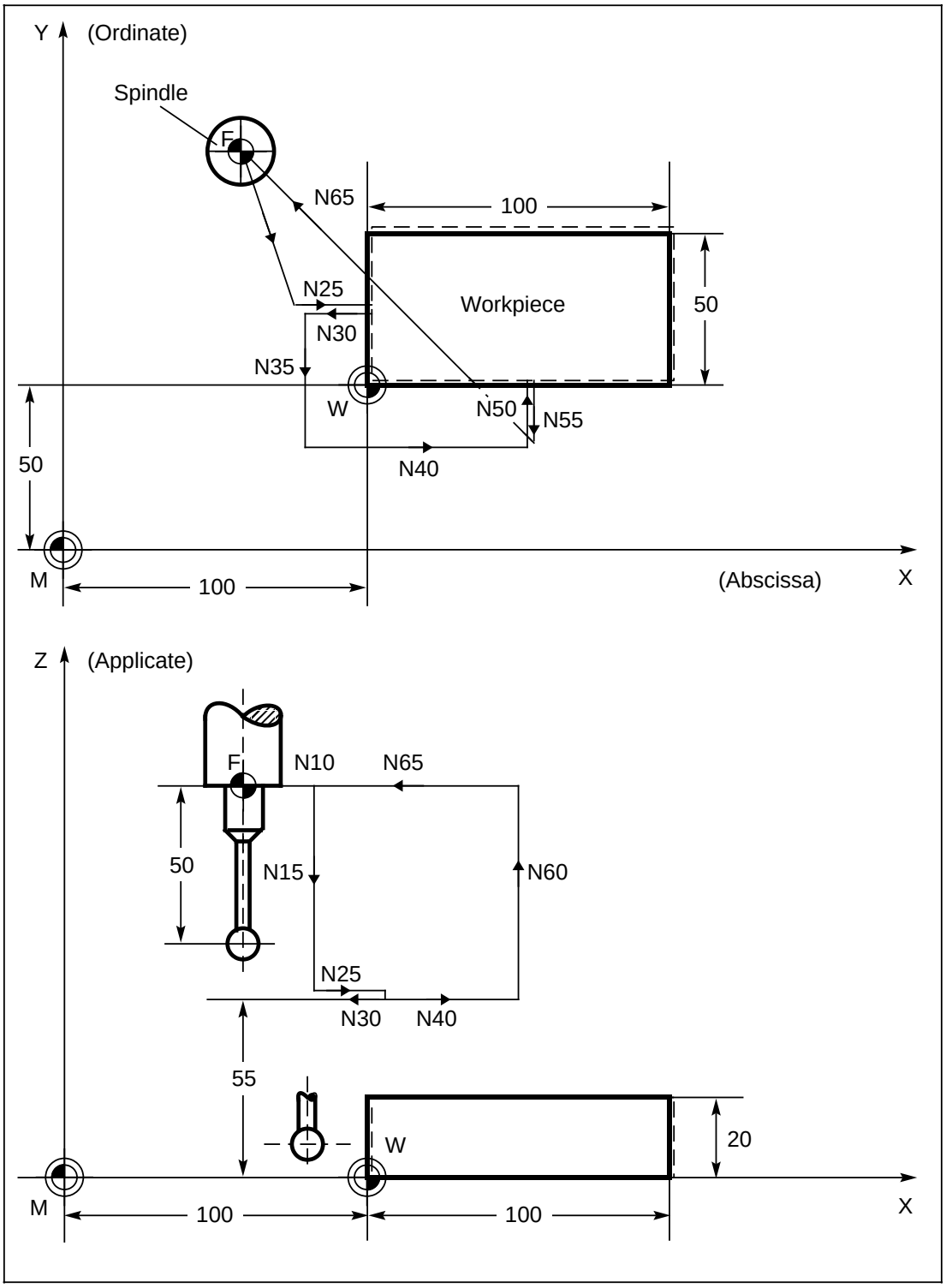


Fig. 5.24 ZO determination on surface

5.4.2 L978 1-point measurement

Function and application

The measuring cycle ascertains the actual value of the workpiece in the selected measuring axis in relation to the zero point of the workpiece and the set/actual difference.

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, averaging is performed over a number of parts and the tolerance bands are checked (Section 1.7.1).

The probe faces the measuring surface at a distance of ”a” on completion of measurement.

No automatic offset is carried out or alternatively length compensation or radius offset (difference halved) is performed, depending on the definition of R10.

Precondition:

The probe must be positioned **with** tool offset and **without G53** to face the surface to be measured.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical and average value Empirical value memory number (average value memory for diameter) (see Section 2.3)
R13 = 0...359.5	Compensation angle position for monodirectional probe (see Section 2.5)
R22	Probe number (see Section 2.6)
R23 = 1 = 11	1-point measurement 1-point measurement with differential measurement
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R26 = 0...359.5	Start angle (with differential measurement only)
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor a for averaging (typically 1...3)
R30 = 1...3	Number of measuring axis (see Section 2.12)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: 1-point measurement in X axis with cycle L978

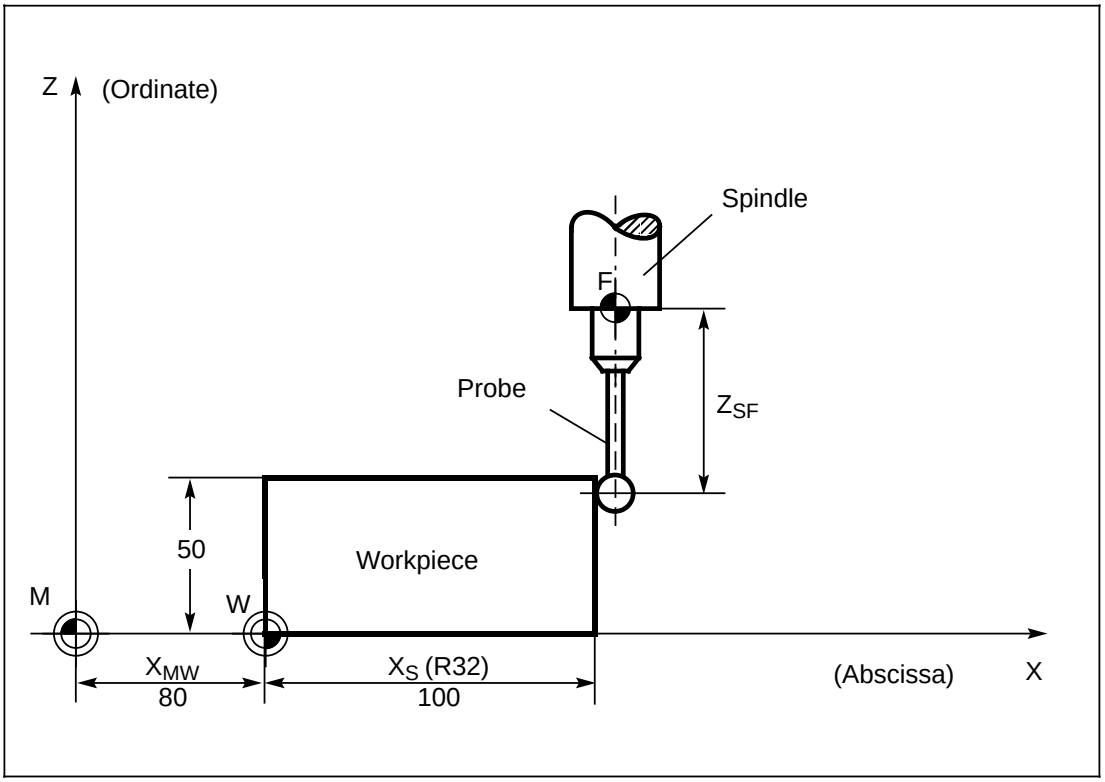


Fig. 5.25 1-point measurement

Probe length (Z axis) in TO memory D99 (value 50)

```
%MPF    9782
:
Machining centre program
:
N500    G54 T200                                          T number probe; select ZO
N505    G00 X120 Y150                                     Position probe in X and
                                                            Y axes to face the measuring point
N510    Z40 D99                                           Position Z axis level with
                                                            measuring point
N515    R10=1021    R11=10      R22= 1      R23=1         Define parameters for measuring cycle
        R25=0       R27=1       R28= 1      R29=3
        R30=1       R33=0.002   R34= 0.015  R36=1
        R37=0.06    R40=0.03    R41=-0.03   R42=100
N525    L978                                              Cycle call for 1-point measurement
                                                            in X axis
N530    G00 Z160                                          Run up Z axis
N535    M30
```


© Siemens AG 1990 All Rights Reserved

5.4.3 L978 Paraxial multipoint measurement

Function and application

The measuring cycle ascertains the actual value of the workpiece in the selected measuring axis in relation to the workpiece zero point and the set*/*actual difference.

In this cycle, the actual value is calculated on the basis of the arithmetic mean of ”n” values offset by R19 along the measuring axis (R30).

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

In addition, averaging is performed over a number of parts and the tolerance bands are checked (see Section 1.7.1).

The probe faces the measuring surface at a distance of ”a” on completion of measurement.

No automatic offset is carried out or alternatively length compensation or radius offset (difference halved) is performed, depending on the definition of R10.

When differential measurement is selected (R23 = 12) **only** MP1 is measured twice.

Preconditions:

- The probe must be positioned **with** tool offset and **without G53** to face the surface to be measured and measuring point number 1 (MP1) (allow for offset distance).
- Parameter R19 specifies the distance between two measuring points along the measuring axis.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R08	Extended T address (see Section 2.2)
R09	T number (tool number) (see Section 2.2)
R10 = 0 > 0	No automatic tool offset Automatic tool offset (see Section 2.2)
R11 = 0 > 0	Without empirical value Empirical value memory number (see Section 2.3)
R13 = 0...359,5	Compensation angle position for monodirectional probe (see Section 2.5)
R19	Infeed offset axis (incremental)
R22	Probe number (see Section 2.6)
R23 = 2 = 12	Paraxial multipoint measurement Paraxial multipoint measurement with differential measurement
R24	Number of measuring points
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R26 = 0...359.5	Start angle (with differential measurement only)
R27 = 1...R_{max}	Number of measurement at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor for measurement path ”2a”
R29 = 1...R_{max}	Weighting factor k for averaging (typically 1...3)
R30 = 102..302	Number of offset axis (see Section 2.12)
R33	Zero offset range
R34	Compensation range with averaging
R36	Safe area
R37	Dimensional difference check
R40	Upper tolerance limit (according to drawing)
R41	Lower tolerance limit (according to drawing)
R42	Set value (according to drawing)

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Multipoint measurement in X axis with cycle L978

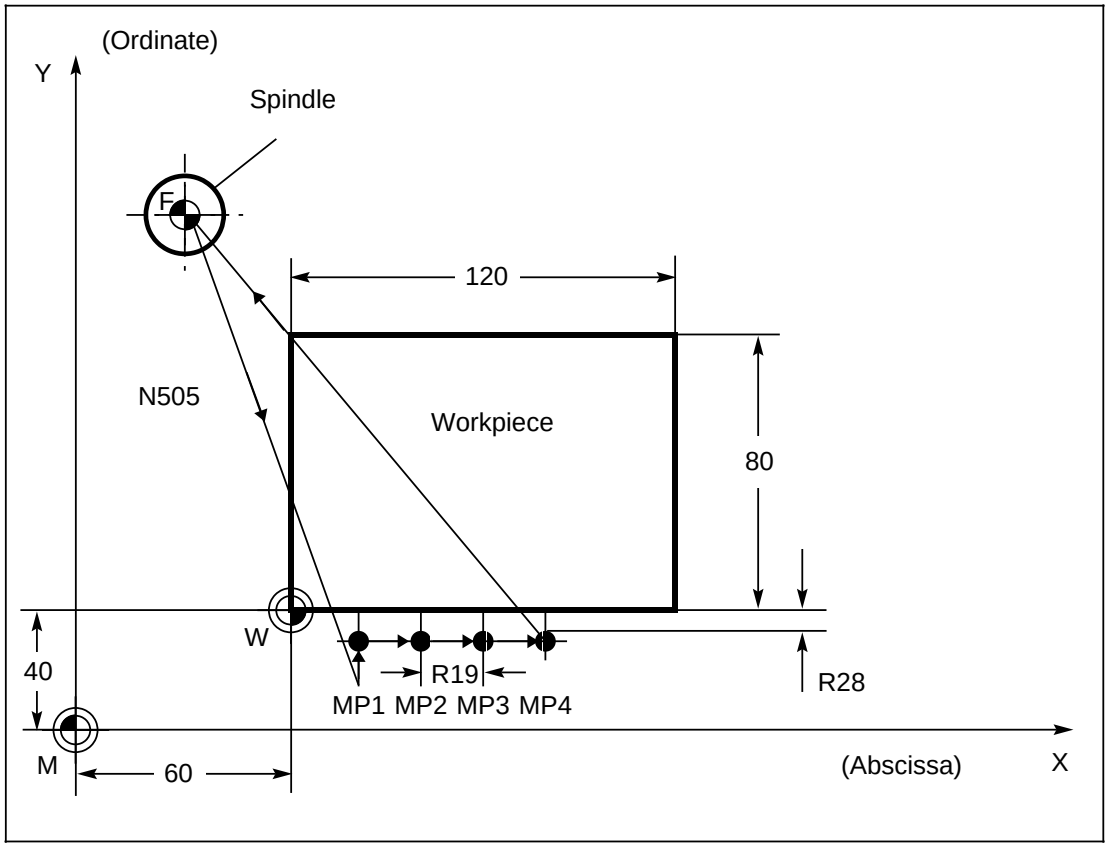


Fig. 5.26 Paraxial multipoint measurement

Probe length (Z axis) in TO memory D99 (value 50)

```
%MPF 9783
:
Machining centre program
:
N500   G54 T200                                          T number probe; select ZO
N505   G00 X70 Y30                                       Position probe in X and Y
                                                          axes to face the measuring point
N510   Z40 D99                                           Position Z axis level with
                                                          measuring point
N515   R10=1021    R11=10      R19=10      R22=1        Define parameters for measuring
                                                          cycle
       R23= 2      R24=4       R25=0       R27=1
       R28= 1      R29=3       R30=102     R33=0.002
       R34= 0.015  R36=1       R37=0.06    R40=0.03
       R41=-0.03   R42=40
N525   L978                                              Cycle call for multipoint
                                                          measurement in Y axis
N530   G00 Z160                                          Run up Z axis
N535   X20 Y70
N540   M30
```

© Siemens AG 1990 All Rights Reserved

5.4.4 L978 Angular measurement (ZO determination)

Function and application

The cycle makes it possible to determine the angular position or clamping position of a workpiece in relation to the set angle value R24 in relation to the offset axis (Fig. 5.27).

An empirical value stored in the SDC is subsequently allowed for with the correct sign.

The multiplication factor for measurement path ”2a” makes it possible to take into account the variation range of the blanks (set value).

The probe faces the measuring surface by the amount ”a” on completion of measurement.

No automatic ZO entry is carried out or alternatively additive input of the **rotary axis difference** (R31) is performed in the specified ZO memory, depending on the definition of R10.

When differential measurement is selected (R23 = 13) **only** MP1 is measured twice.

Preconditions:

- The probe must be positioned **with** tool offset and **without G53** to face the first measuring point (MP1, see Fig. 5.26).
- Parameter R19 is used to specify the distance in the offset axis between MP1 and MP2 (positive values only).
- Measurement path ”m” in the measuring axis must be less than the offset path (R19) (see Fig. 5.26).

  It is therefore only possible to measure an angle of max. 44.999 degrees. However, the measurement can be performed from all sides.

  The angle setpoint is the angle between the offset axis and the workpiece edge.

© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R10 = 0 = 1...4 = 5 = 6	No automatic ZO entry Automatic entry in ZO G54...G57 of the rotary axis defined in R31 Automatic entry in ZO G58 of the rotary axis defined in R31 Automatic entry in angle for coordinate rotation (G58)
R11 = 0 > 0	Without empirical value Empirical value memory number (see Section 2.3)
R13 = 0...359,5	Compensation angle position for monodirectional probe (see Section 2.5)
R19	Incremental infeed offset axis (positive only)
R22	Probe number (see Section 2.6)
R23 = 3 = 13	Paraxial angle point measurement Paraxial angle point measurement with differential measurement
R24	Set value angle (max. +/- 44.99 degrees)
R25 R25 = 0	Variable measuring speed in mm/min Standard cycle value
R26 = 0...359.5	Start angle (for differential measurement only)
R27 = 1...R_{max}	Number of measurements at same location (typically 1..3)
R28 = 1...R_{max}	Multiplication factor measurement path ”2a”
R30 = 102..302	Number of offset axis/measuring axis (see Section 2.12)
R31	Number of rotary axis
R32	Set value approach position with reference to workpiece zero and measuring axis
R36	Safe area

See Section 10.2 for result display parameters.

© Siemens AG 1990 All Rights Reserved

Example: Angular measurement with cycle L978 (Data as in Fig. 5.29)

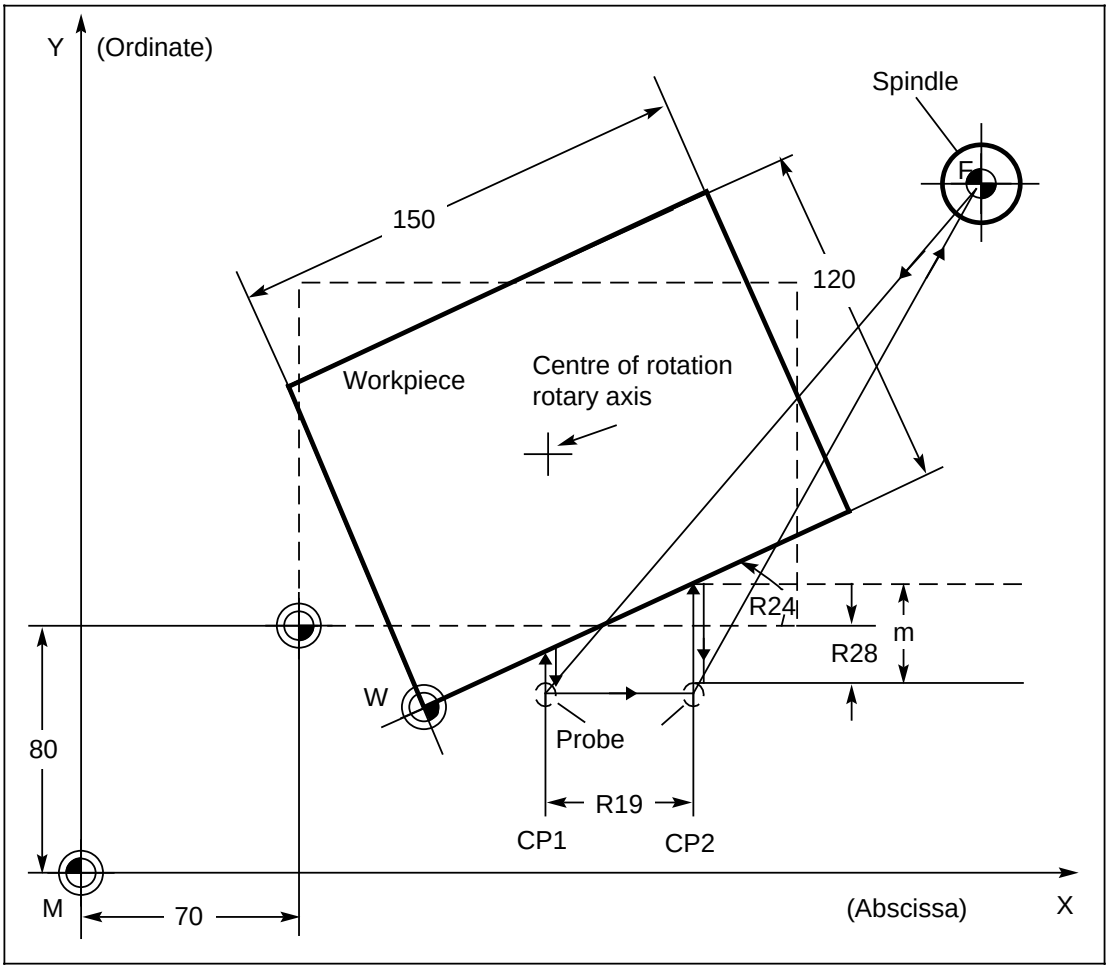


Fig. 5.27 Angular measurement

Probe length (Z axis) in TO memory D99 (value 50)

```
%MPF    9784
N5      G54 T200                                        T number probe; select ZO
N10     G00 X140 Y70                                    Position probe in X and Y
                                                          axes to face the measuring point
N15     Z40 D99                                         Position Z axis level with
                                                          measuring point
N20     R10=6       R11=10      R19=40      R22=1       Define parameters for measuring cycle
        R23=3       R24=0       R25=0       R27=1
        R28=5       R30=102     R31=4       R32=0
        R36=5
N25     L978                                            Cycle call for angular measurement
N30     G00 Z160                                        Run up Z axis
:
Machining centre program
:
:
N...    M30
```

END OF SECTION

© Siemens AG 1990 All Rights Reserved

6 Measuring Cycles for Turning and Milling Work

6.1 L981 Searching for a hole/slot

Function and application

Cycle L981 searches for the position of a hole/slot on a face or at a cylinder (outside or inside) with the aid of a probe on a rotary axis (C axis).

To begin with, the cycle uses the linear axis to position the ball center point of the probe on the center point circle of the hole/slot specified by R18. Following this, the reference plane defined by R2 is approached. Now, sensing (searching) of the workpiece can be started. A hole/slot is assumed to be found if the set approach value (R32) could be approached without a measuring signal. The search grid (in degrees) and the measuring speed can be preselected via R parameters.

After the hole/slot has been found, position determination is carried out in the rotary axis. In addition, the rotary axis position programmed prior to cycle call is used to calculate the deviation. The deviation is allowed for in a ZO memory if the deviation ascertained is less than the specified positional tolerance (R36), depending on the definition of R10. An alarm is triggered when the positional tolerance is exceeded.

The probe is in the hole or slot centre on completion of measurement.

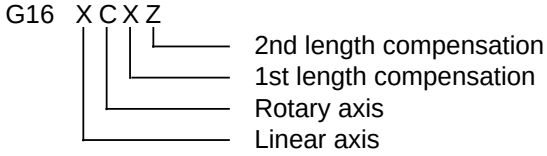
Precondition:

The workpiece probe must be defined as type 30 (see Section 1.4.4).

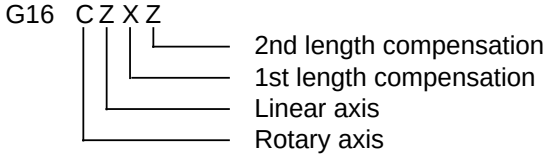
Note:

Position determination in the linear axis can be carried out subsequently using cycle L977 (previously L982).

Example: Plane selection for searching for a hole/slot on the face:



Example: Plane selection for searching for a hole/slot on the cylinder:



© Siemens AG 1990 All Rights Reserved

The following parameters must be defined prior to call:

Parameters	Description
R1	Setpoint diameter hole/slot
R2	Reference plane
R10 = 0 = 1-4 = 5	No automatic ZO input Automatic ZO input in G54 - G57 Automatic ZO input in G58
R18	Centre point linear axis (in radius for transverse axis)
R22	Probe number
R23 = 1 = 2	On face At cylinder
R25 R25 = 0	Variable measuring speed Standard cycle velocity
R26	Search increment in degrees (positive only)
R27 = 1...R_{max}	Number of measurements at same location
R32	Approach setpoint
R36	Safe area (positional tolerance)

© Siemens AG 1990 All Rights Reserved

Example: Determine positional deviation of a hole on a face

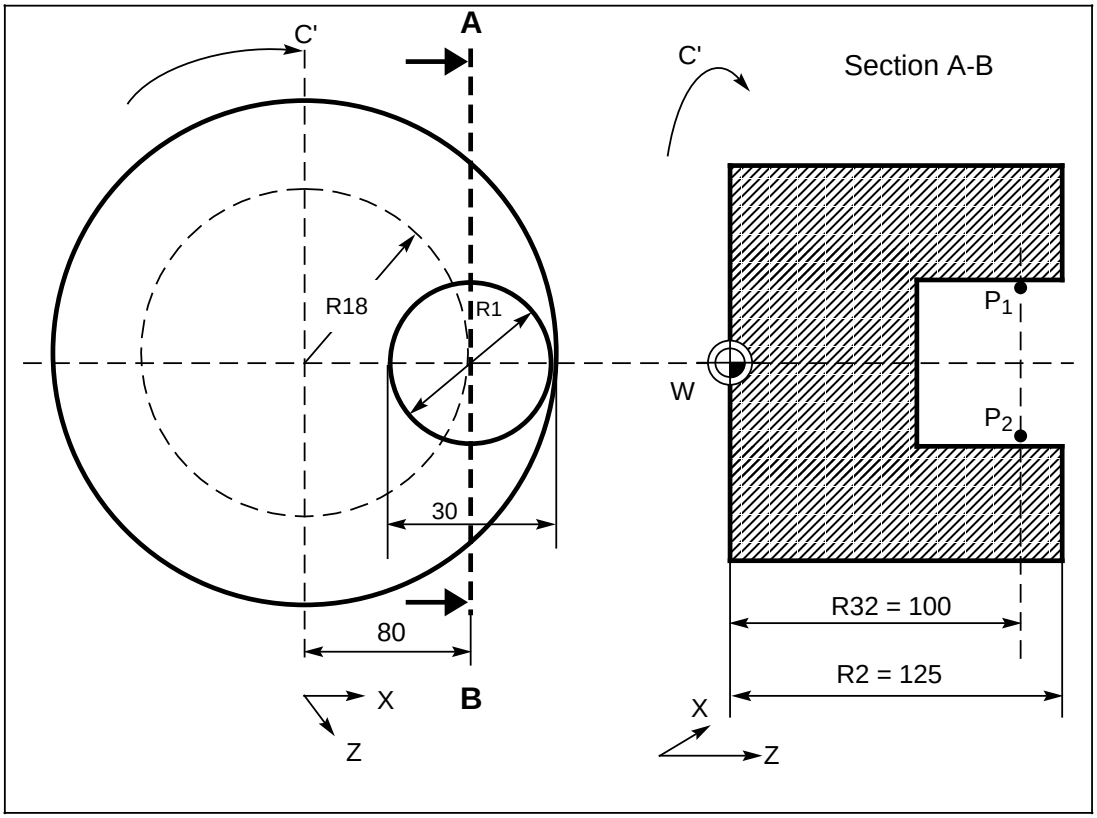


Fig. 6.1 Searching for a hole on the face

```
%MPF    981
N5      G16   X     C     X     Z                          Plane selection
N10     G     X154  D33                                    Position probe near the
                                                           centre point circle of the hole
N15     C0                                                 Position rotary axis
N20     R1=30       R2=125       R10=5       R18=80        Parameters for L981
        R22=1       R23=1        R25=0       R26=5
        R27=1       R32=100      R36=10
N25     L981                                               Search for hole
N30     M30
```

END OF SECTION

© Siemens AG 1990 All Rights Reserved

7 Measuring in JOG Mode (SINUMERIK 880/880 GA2)

The function MEASURING IN JOG mode contains the following functions:

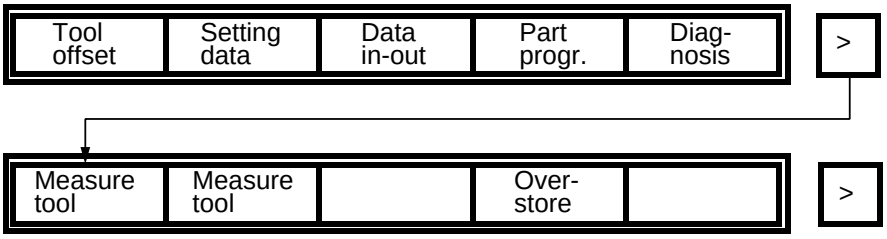
- Semi-automatic determining of tool lengths and storing of values in tool offset memory,
- semi-automatic determining of workpiece zero and storing of value in zero offset memory.

The functions are operated via softkeys and input displays. Incremental traversing (INC) is disabled in MEASURING IN JOG mode. The measuring process is aborted with RESET or on mode change.

Caution!

Please make sure that you select the correct channel as the function MEASURING IN JOG mode is channel-dependent. If the incorrect channel is selected and the measuring process is activated the probe may be damaged.

The measuring function is selected from the row of softkeys in the JOG basic display.



7.1 Tool measurement

With this function it is possible to measure tools which are in the machine without previously having to enter the geometry data in a tool offset memory. This means that tools with unknown lengths can be measured.

The tool lengths can be entered in a tool offset memory automatically and are then immediately available for machining of the workpiece.

Prerequisites:

- The reference points have already been approached.
- The starting conditions for automatic mode have been fulfilled.
- The tool probe is swivelled in or retracted.
- The tool probe has been calibrated with measuring cycle L972/L975.
- Plane correctly defined, initial setting.

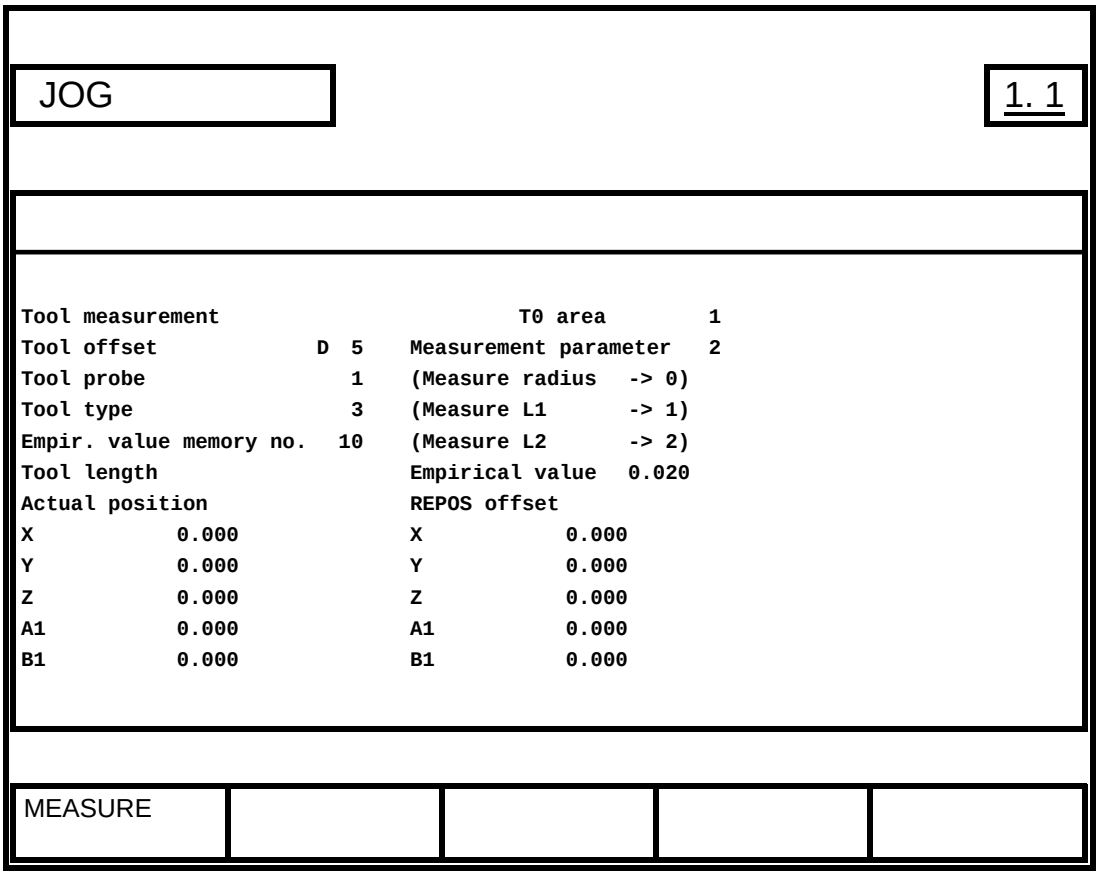
© Siemens AG 1990 All Rights Reserved

7.1.1 Operating and function sequence

1. The tool is in position or has been put in position manually.

Measure Tool

2. The following screen form appears when you press the softkey MEASURE TOOL:



Input display for Tool measurement

• • •

3. You enter the values in the different input fields using the alphanumeric keyboard

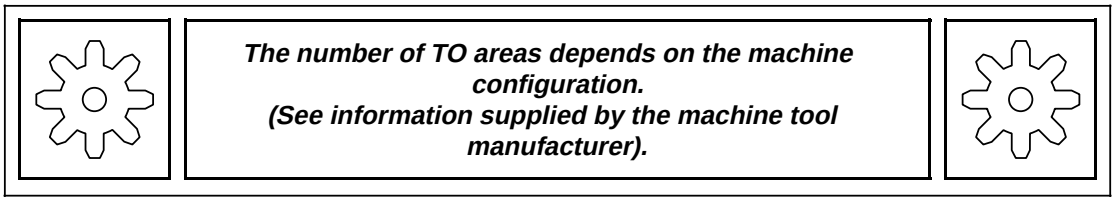
© Siemens AG 1990 All Rights Reserved

Explanation of the input values:

TO area

Number of the TO area in which the determined tool data for the tool being measured are to be stored.

The TO area is preselected via the channel and is displayed here.



Tool offset

Tool offset number D... The determined tool data are stored under this tool number.

Tool type (1-39)

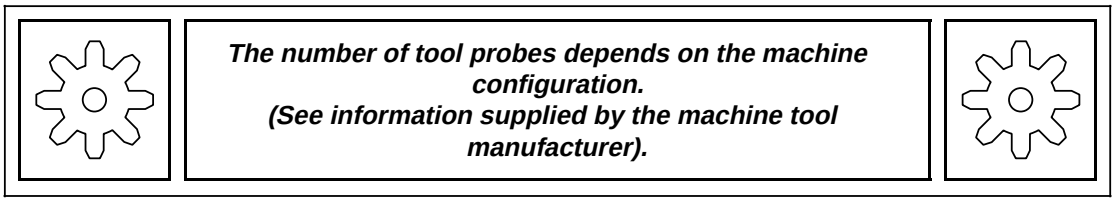
Tool type of the tool to be measured.

1 - 9 : Calculation of the tool geometry via the set plane

10 - 39 : Additional information radius/L1/L2 required! See description below (measurement parameters).

Tool probe

Number of tool probe.



Empirical value memory no.

The number of the empirical value memory to be used must be entered here.

If the value "0" is entered the tool length required is calculated without using an empirical value.

© Siemens AG 1990 All Rights Reserved

If a value other than "0" is entered the contents of the chosen empirical value memory are displayed. This value is taken into account when the tool geometry is calculated. The empirical values must be entered/stored in the cycle setting data.

The number of empirical value memories depends on the machine configuration. (See information supplied by the machine tool manufacturer).

Measurement parameter (0 - 2)

If a tool type 1 ... 9 is entered the tool geometry is calculated using the defined plane and the selected traversing axis, i.e. any entries in the input display at this point are of no significance.

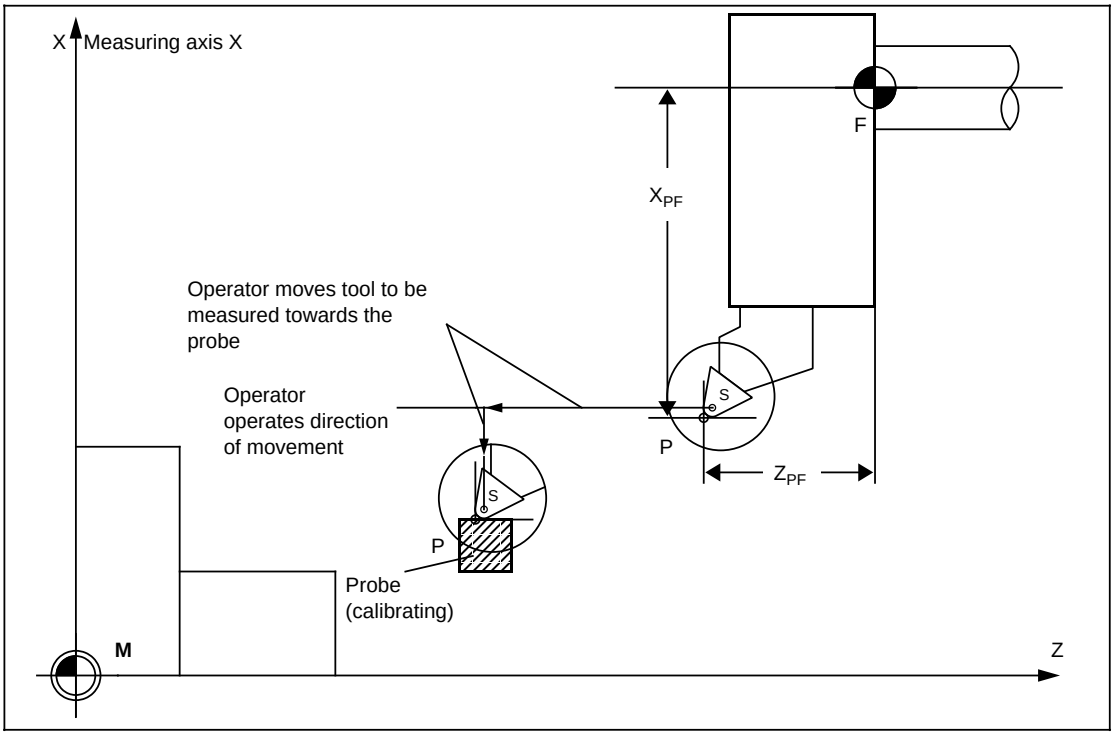
If a tool type 10 ...39 is entered, the geometrical value which is to be used (radius, length L1 or L2) must be entered under measurement parameter.

Input	”0”	Measure radius
	”1”	Measure length L1
	”2”	Measure length L2

X

4. You position the tool near the tool probe by operating the JOG direction keys.

Z



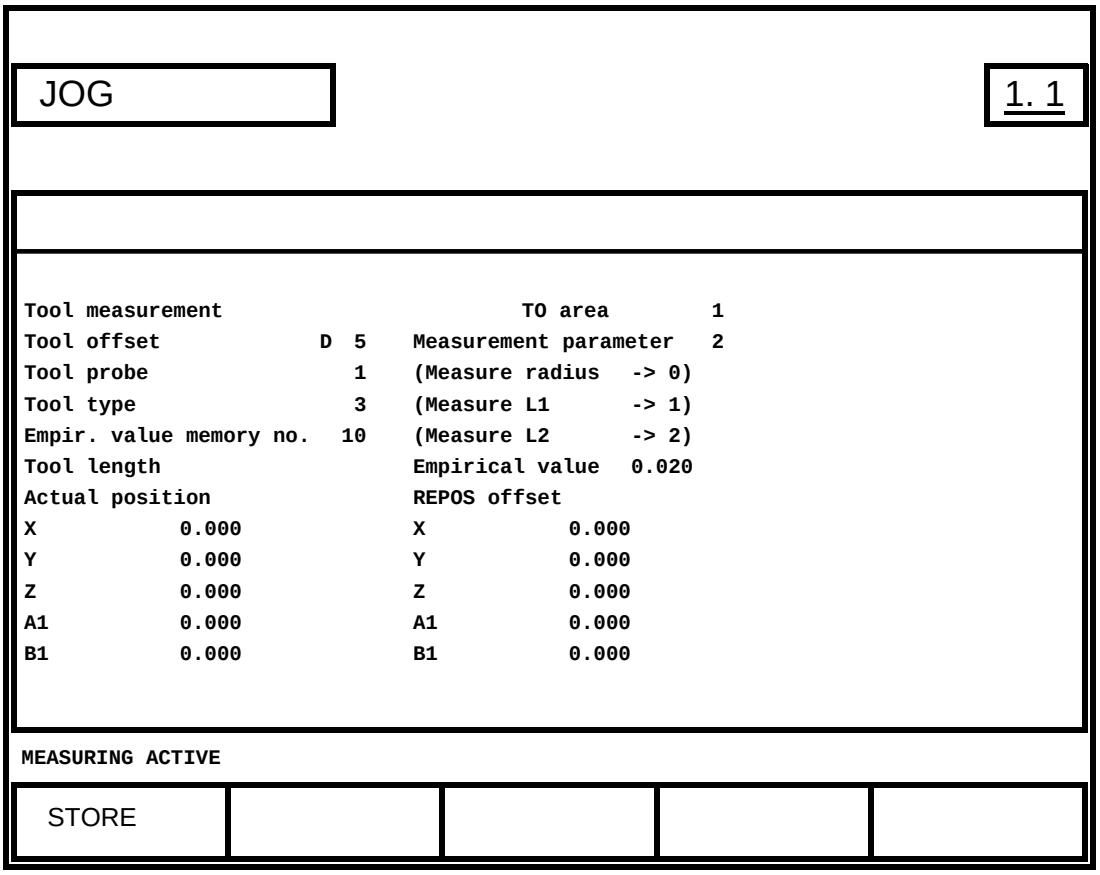
Tool measurement in JOG mode

© Siemens AG 1990 All Rights Reserved

MEASURE

5. The measuring function is activated once the softkey MEASURE has been operated. If incorrect values are entered, alarm 3050, "WRONG INPUT" is displayed. The dialog text MEASURING ACTIVE is displayed.

   The displayed softkey function STORE is not enabled until the probe has been deflected.



Display after the softkey MEASURE has been pressed

6. The axis which is first traversed from zero speed and the direction define the direction of measurement and only with this value is it possible to traverse to the probe with the set measuring speed as in JOG mode.

   Until the probe has been deflected, it is only possible to traverse along this axis in this direction. All other axes, directions and rapid traverse remain disabled.

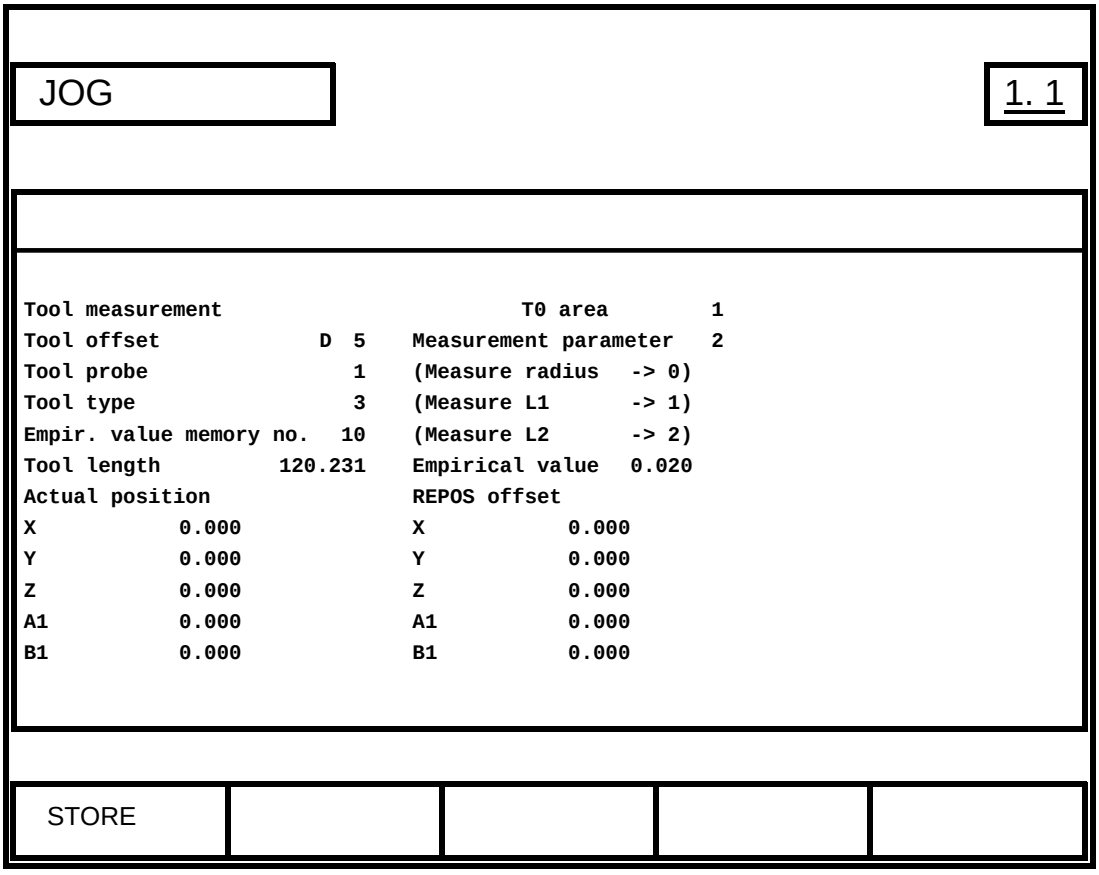
   It is possible to reduce the set measuring speed with feedrate override.

The measuring speed depends on the machine configuration.
(See information supplied by the machine tool manufacturer).

© Siemens AG 1990 All Rights Reserved

7. As soon as the probe has been deflected, the traversing axis (measuring axis) is stopped and the selected tool geometry is calculated. The derived tool length is displayed (tool length).

   The dialog text MEASURING ACTIVE is deleted and the STORE softkey is enabled.



Display after probe has been deflected

STORE

8. When you press the softkey STORE, the measured tool length is stored in the selected tool offset memory. The P no. within the selected tool offset memory in which the determined value is entered is derived from the measurement plane, the measuring axis and the tool type.

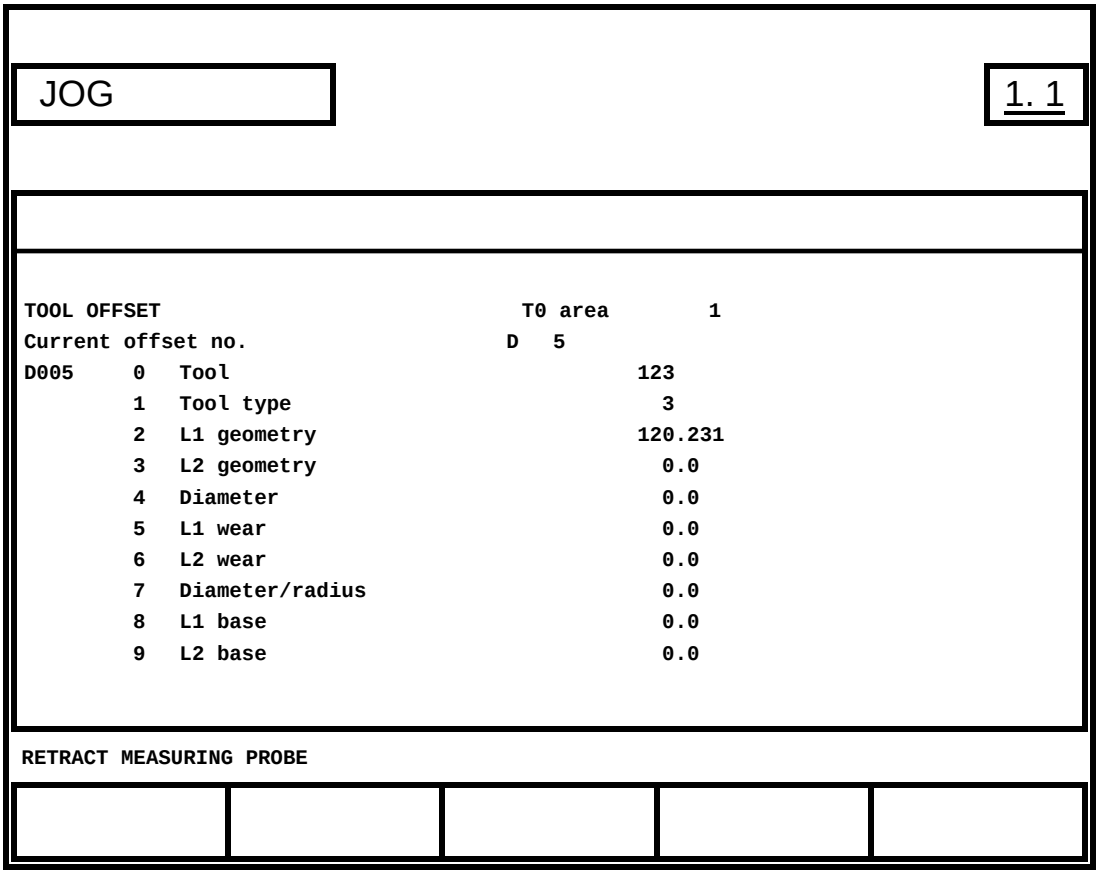
   Which measuring axis wear memory is deleted depends on the machine configuration.

The machine data defines whether the wear memory is deleted or not.
(See information supplied by the machine tool manufacturer).

© Siemens AG 1990 All Rights Reserved

After the value has been stored, the standard display TOOL OFFSET is automatically selected for the previously entered D no.

The dialog text RETRACT MEASURING PROBE appears.



Tool offset memory display after retraction of probe

X

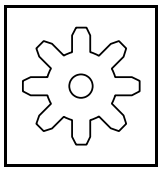
Z

9. You can retract the probe with the appropriate direction keys.

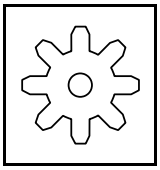
© Siemens AG 1990 All Rights Reserved

The following applies when retracting the measuring probe:

- The probe can only be retracted in the opposite direction to that of the measuring direction.
- All other axes, axis directions and rapid traverse are disabled.
- When the retraction path has been completed it again becomes possible to move along every axis and the basic display JOG appears.



The length of the retraction path depends on the machine configuration.
(See information supplied by the machine tool manufacturer).



If the measured value is not to be stored the function can be aborted with RESET or a mode change. The dialog text "JOG-MEASUREMENT ABORTED" appears on the screen. In this case retraction is not limited to a predefined distance, i.e. any axis can be traversed without limitation.

© Siemens AG 1990 All Rights Reserved

7.2 Workpiece measurement

With this function it is possible to determine the workpiece zero offsets using a workpiece probe in the machine. The resulting zero offsets are automatically loaded in the zero offset memories G54 to G57.

The zero offset memory G58/G59 and the external zero offset cannot be loaded with this function.

Prerequisites

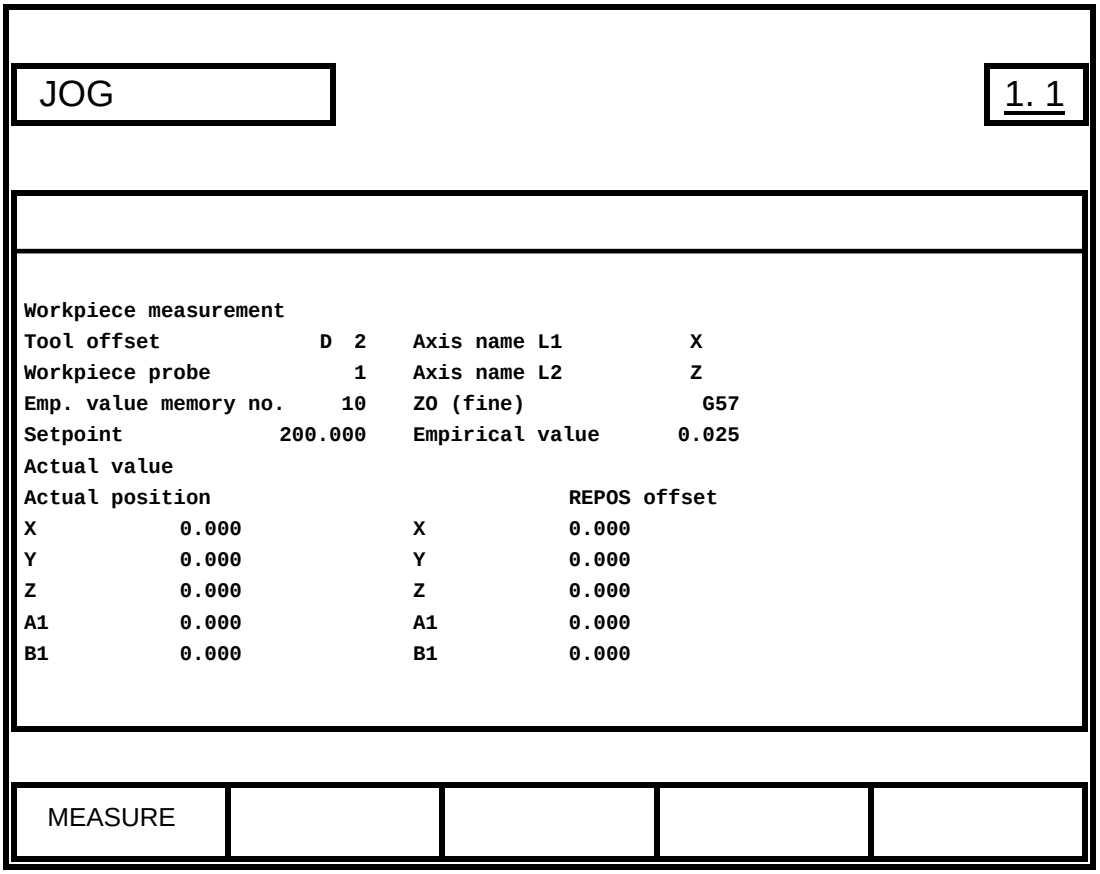
- The reference points have been approached.
- The starting conditions for automatic mode are fulfilled.
- The workpiece probe is in the turret or the spindle.
- The workpiece probe has been calibrated with measuring cycle L973/L976.
- Plane correctly defined, initial setting.

7.2.1 Operating and function sequence

1. The workpiece is clamped in position.

MEASURE WORKP.

2. The following screen form appears when you press the softkey MEASURE WORKPIECE:



Input display for workpiece measurement

•••

3. You enter the values in the different input fields using the alphanumeric keyboard.

© Siemens AG 1990 All Rights Reserved

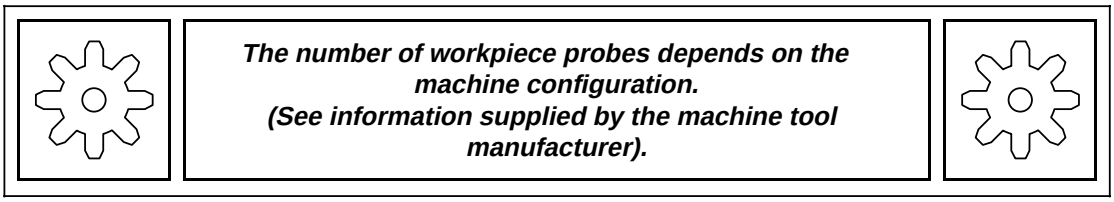
Explanation of the input values:

Tool offset

The tool offset number D...The geometry data of the workpiece probe are stored under this tool offset number.

Workpiece probe

Number of the workpiece probe.



The number of workpiece probes depends on the machine configuration.
(See information supplied by the machine tool manufacturer).

Zero offset (fine)

Here you define in which zero offset memory the results of the measurement are to be stored.

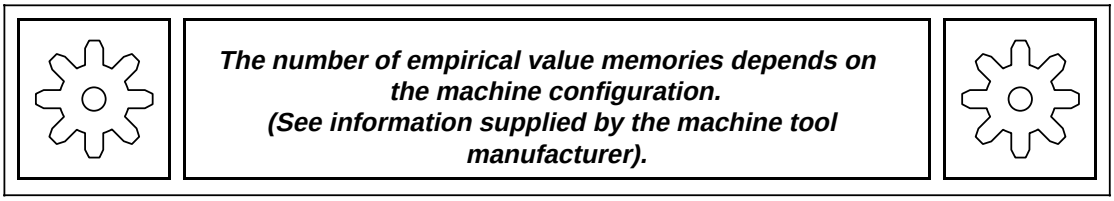
Empirical value memory no.

The number of the empirical value memory to be used must be entered here.

If the value ”0” is entered the zero offset required is calculated without using an empirical value.

If a value other than ”0” is entered the contents of the chosen empirical value memory are displayed. This value is taken into account when the zero offset at the workpiece is calculated.

The empirical values must be entered/stored in the cycle setting data.



The number of empirical value memories depends on the machine configuration.
(See information supplied by the machine tool manufacturer).

Setpoint

The zero offset is determined taking the preset workpiece setpoint into consideration.

Axis names for L1/L2

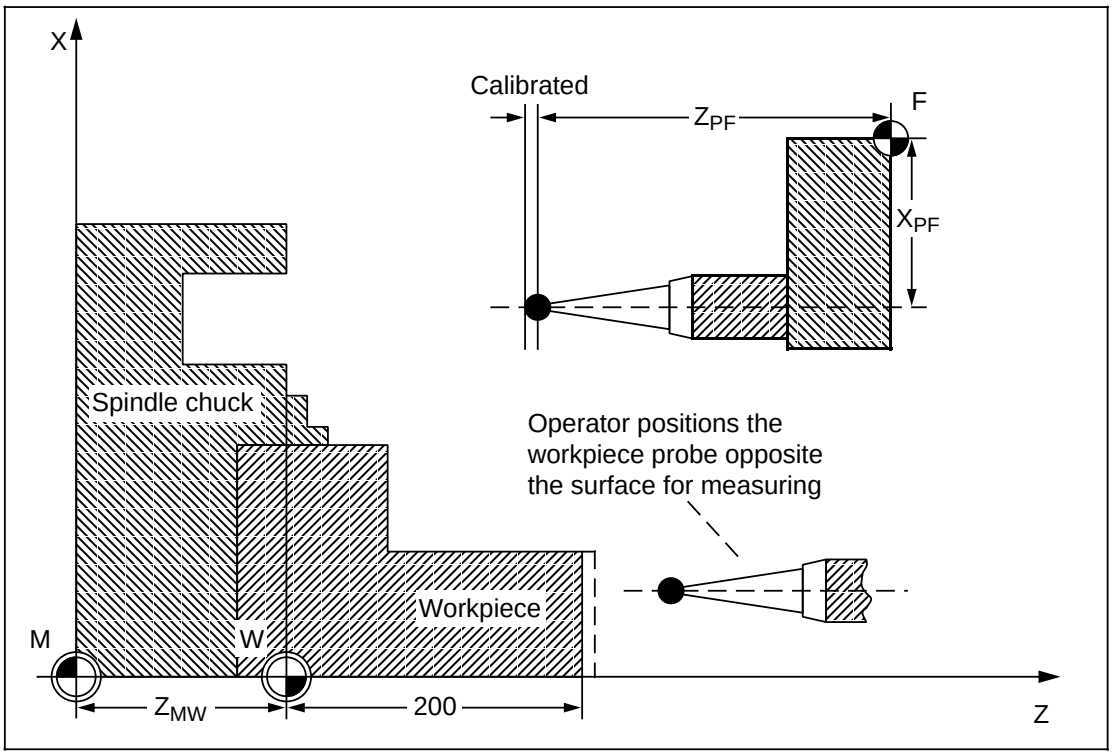
The data for L1 and L2 must be assigned to axes so that the tool offset data of the workpiece probe can be used to determine the zero offset.

© Siemens AG 1990 All Rights Reserved

X

Z

4. You position the workpiece probe close to and opposite the defined setpoint surface by operating the JOG direction keys.



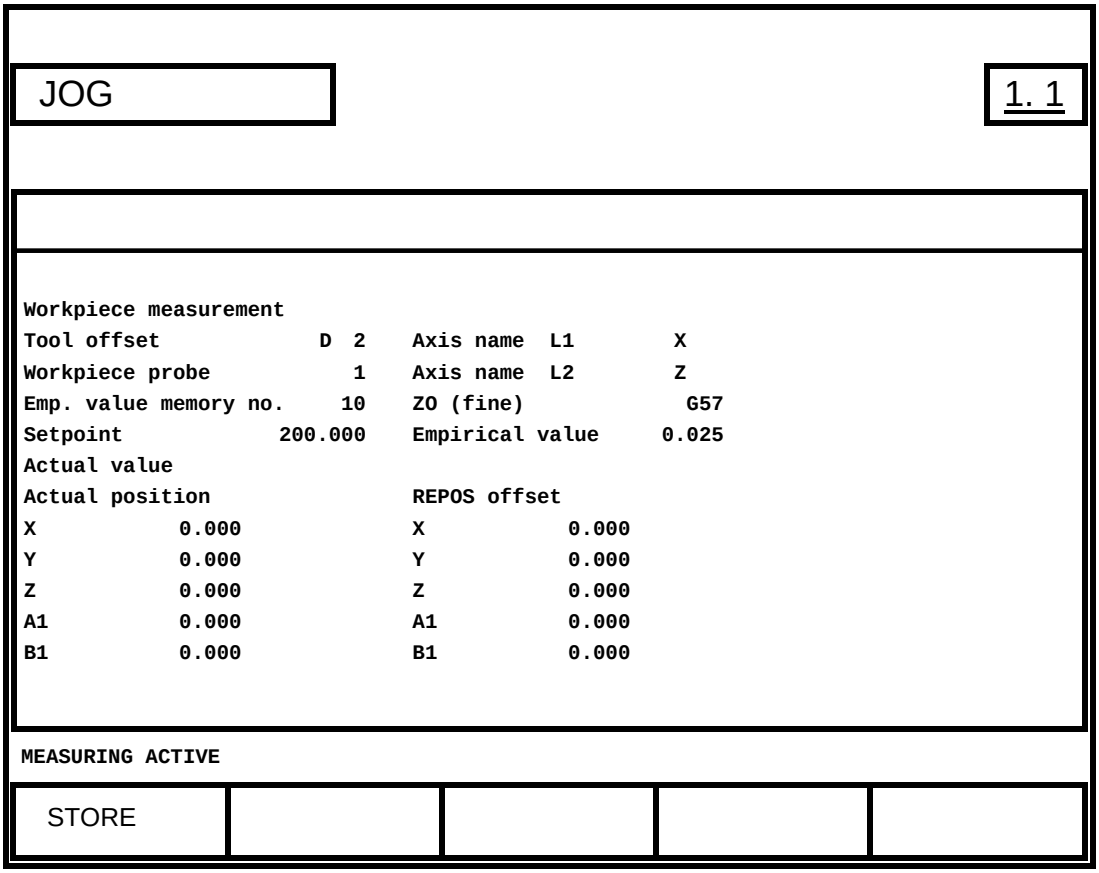
Measuring the workpiece in JOG mode

MEASURE

5. The measuring function is activated once the softkey MEASURE has been operated. If incorrect values are entered, alarm 3050, "WRONG INPUT" is displayed. The dialog text MEASURING ACTIVE is displayed.

   The displayed softkey function STORE is not enabled until the workpiece probe has been deflected

© Siemens AG 1990 All Rights Reserved



Display after the softkey MEASURE has been pressed

6. The axis which is first traversed from zero speed and the direction define the direction of measurement and only with this value is it possible to traverse the workpiece probe with the set measuring speed as in JOG mode.

   Until the probe has been deflected, it is only possible to traverse along this axis in this direction. All other axes, directions and rapid traverse remain disabled.

   It is possible to reduce the set measuring speed with feedrate override.

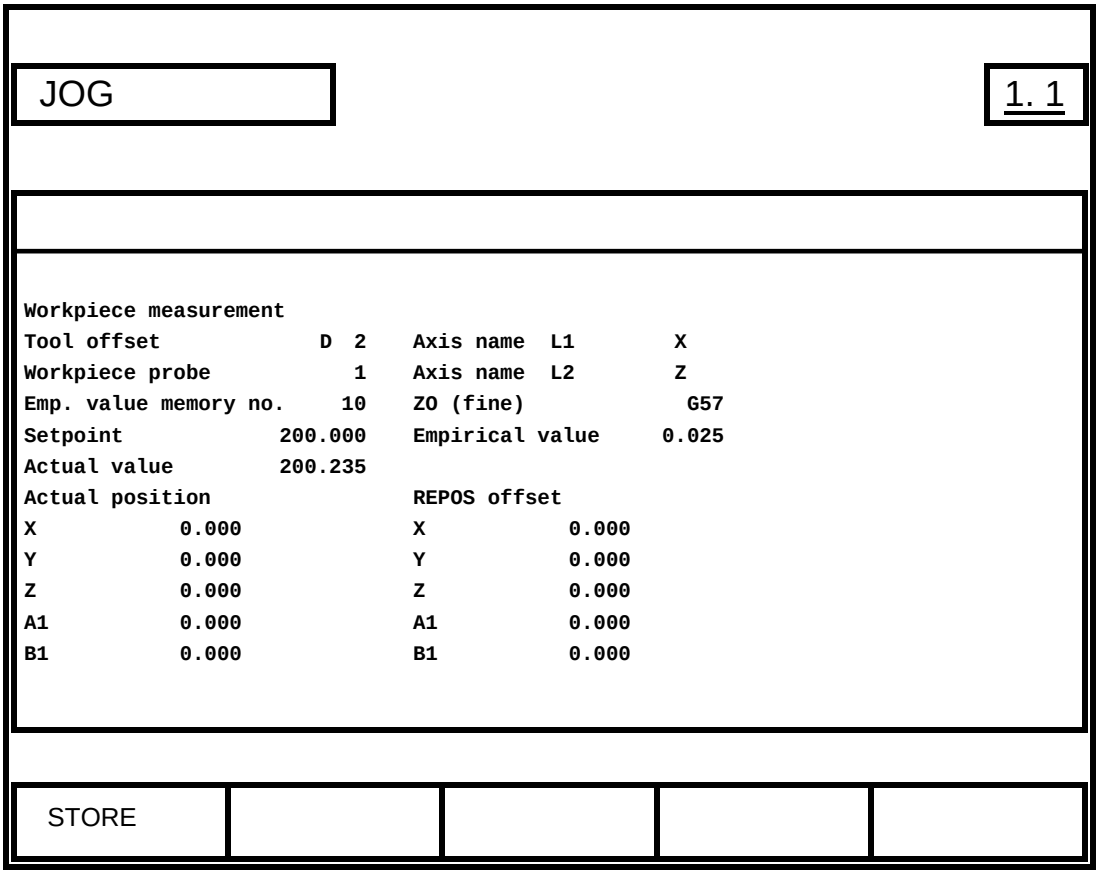
The measuring speed depends on the machine configuration.
(See information supplied by the machine tool manufacturer).

© Siemens AG 1990 All Rights Reserved

7. As soon as the probe gives a signal the traversing axis (measuring axis) is stopped. The workpiece actual value and the workpiece zero offset are calculated.

   The derived actual value is displayed (actual value).

   The dialog text MEASURING ACTIVE is deleted and the STORE softkey is enabled.



Display after workpiece probe has been retracted

STORE

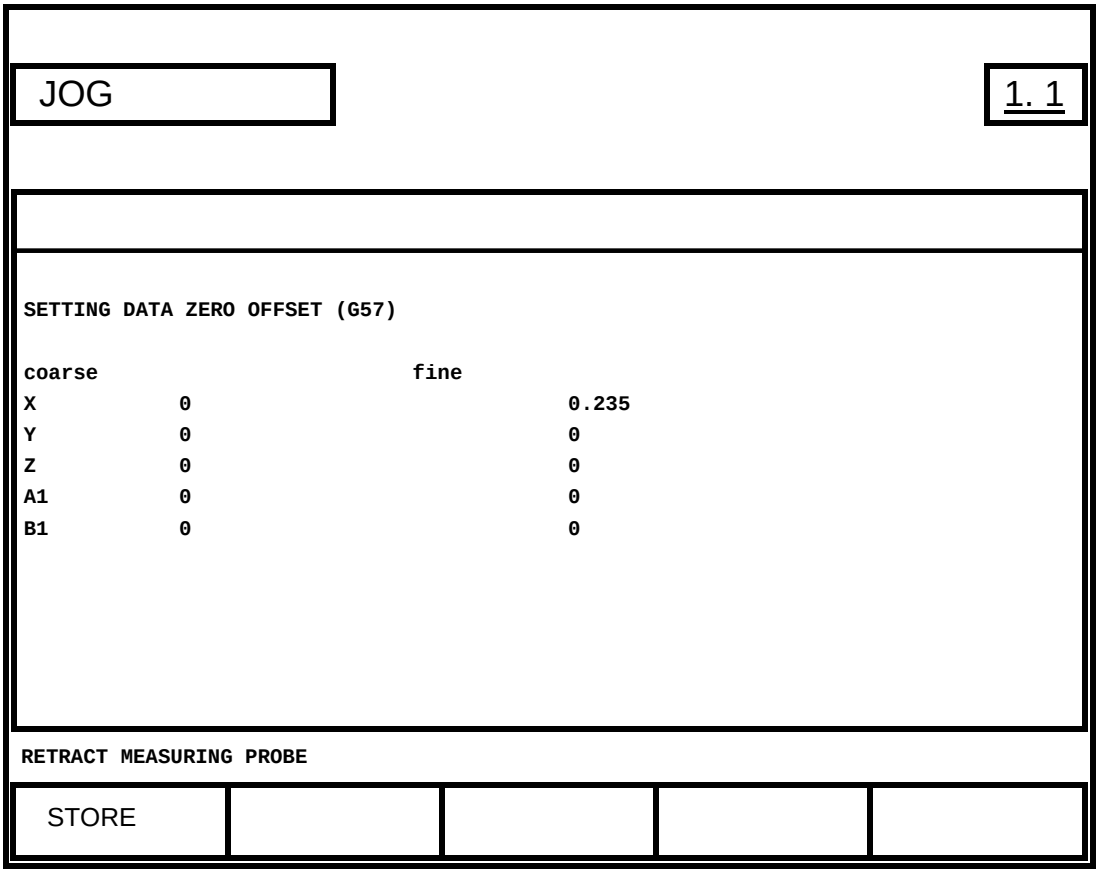
8. When you press the STORE softkey the determined ZO value is stored in the selected zero offset memory (fine).

   After the value has been stored, the standard input display zero offset of the previously entered group is entered. The dialog text RETRACT MEASURING PROBE appears.

   Special case:

   If G53 is entered (into the input display before the measuring is started), neither is the zero offset value stored nor the zero offset input display called up, but the input display "MEASURE WORKPIECE" is displayed directly.

© Siemens AG 1990 All Rights Reserved



Tool offset memory display after retraction of probe

X

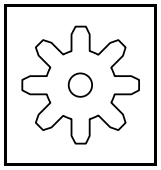
Z

9. You can retract the probe with the appropriate direction keys.

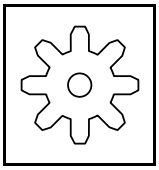
© Siemens AG 1990 All Rights Reserved

The following applies when retracting the measuring probe:

- The probe can only be retracted in the opposite direction to that of the measuring direction.
- All other axes, axis directions and rapid traverse are disabled.
- When the retraction path has been completed it again becomes possible to move along every axis and the basic display JOG appears.



The length of the retraction path is depends on the machine configuration.
(See information supplied by the machine tool manufacturer).



If the measured value is not to be stored the function can be aborted with RESET or a mode change. The dialog text ”JOG-MEASUREMENT ABORTED” appears on the screen. In this case retraction is not limited to a predefined distance, i.e. any axis can be traversed without limitation.

END OF SECTION

© Siemens AG 1990 All Rights Reserved

8 Automatic Workpiece Measurement in JOG Mode (SINUMERIK 840 SW2)

8.1 General

Automatic workpiece measurement in JOG mode is an extended version of the Siemens "Inprocess Measurement" cycles for SINUMERIK. With the JOG measuring function the distance between machine zero and the reference point (workpiece zero) can be determined and then stored with the softkeys "STORE ZO" and "STORE PRESET".

The difference between these cycles and conventional cycles lies in the way they are operated and parameterized.

Emphasis has been placed on implementing simple cycles which are easy to operate. Almost every type of workpiece found in practice is available to the operator in the various selection menus. Workpieces can be measured in the machining planes XY, ZX or YZ defined by G17, G18, G19.

Only the most essential parameters have to be set, such as:

- Drilling axis selection
- Number of probe
- Number of D offset in which probe geometry is entered
- Measuring speed
- Number of measurement repetitions
- Measurement path
- Number of zero offset
- Setpoint or estimated setpoint of several workpiece dimensions.

All intermediate positions are approached at the entered max. axis velocity in percent (NC MD 280*) (if MDC 800.0=0 collision monitoring deactivated) or, if collision monitoring is active (MDC 800.0=1), at the positioning velocity entered in MDC 5.

© Siemens AG 1990 All Rights Reserved

The whole process is carried out in "JOG" mode. Once the measurement variants have been selected and the cycle parameters set, the probe must be positioned, for example, at the centre of the drill hole in "JOG" mode.

When the softkeys "MEASURING START" AND "CALIBRATE START" have been pressed, the control switches to "AUTOMATIC" mode and measuring cycle L887 is activated.

This process runs in the channel defined by the machine tool manufacturer and is initiated by the PLC if FX 34 is linked into the PLC.

Once "AUTOMATIC" mode is active, NC Start must be executed via the machine control panel.

If NC Start is not executed via the machine control panel within 30 seconds, the control returns to the start menu and switches back to "JOG" mode.

After NC Start and program end the control switches back to "JOG" mode and the selection display showing the results of the workpiece measurement is displayed.

The following results are shown:

- Actual value: e.g. diameter of drill hole
- Difference: Setpoint - actual value
- Axis position: 1st axis (abscissa) and 2nd axis (ordinate), e.g. the centre point coordinates of the hole.

The axis-specific measuring results can be displayed both as machine coordinates or as workpiece coordinates, although this display refers to the ZO memory G5x entered in the input display.
The display is selected via bit 3 of the cycle setting data SDC 800 (channel-oriented bits).
SDC 800.3 = 0 Display in machine coordinates
SDC 800.3 = 1 Display in workpiece coordinates

The user defines the memory in which the measured values are to be stored.

There are two possibilities:

When the measuring cycle has been processed the axis position is stored

- as a preset offset
  ⟶ softkey "STORE PRESET"
- as a zero offset in zero offset coarse
  ⟶ softkey "STORE ZO"

The user is now able process the measured values (centre point coordinates or coordinates of workpiece reference point) in the next NC part program.

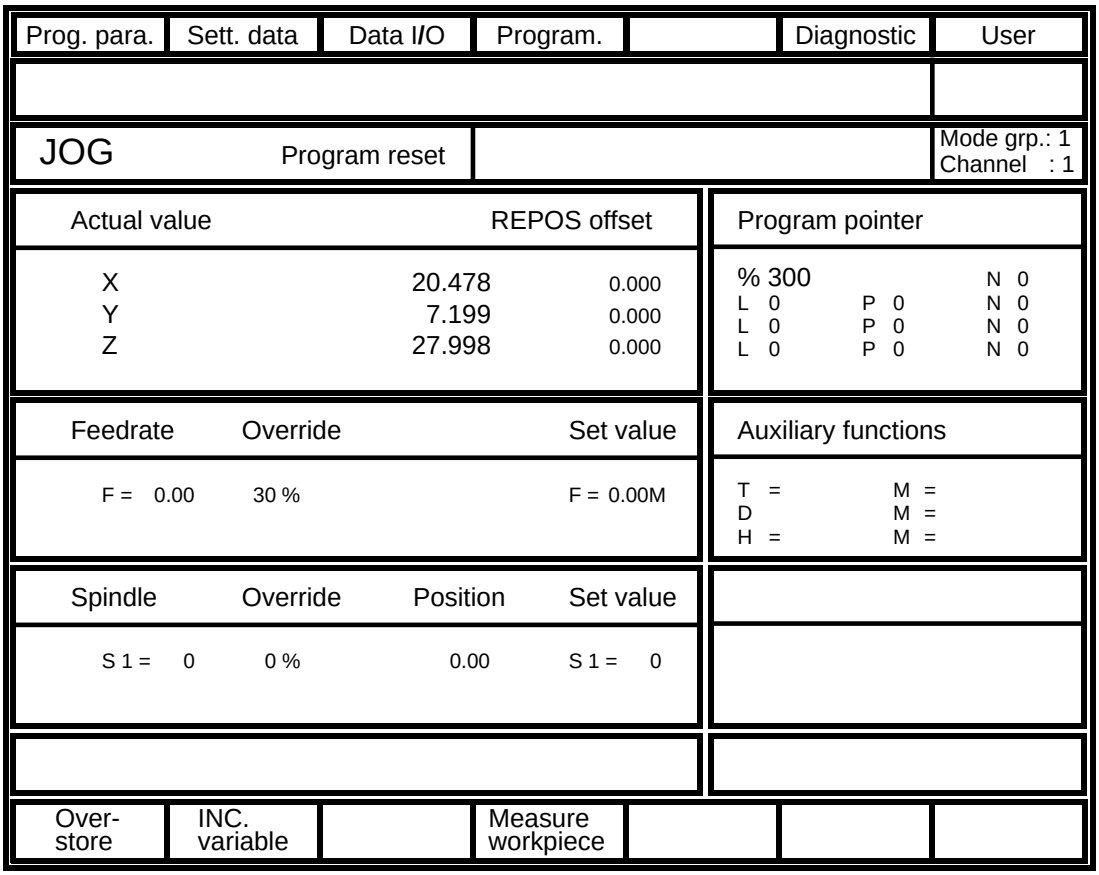
You will discover from reading the User Guide that the cycles have to be set with many more parameters than are described here.

These other parameters are stored in measuring cycle L887.

© Siemens AG 1990 All Rights Reserved

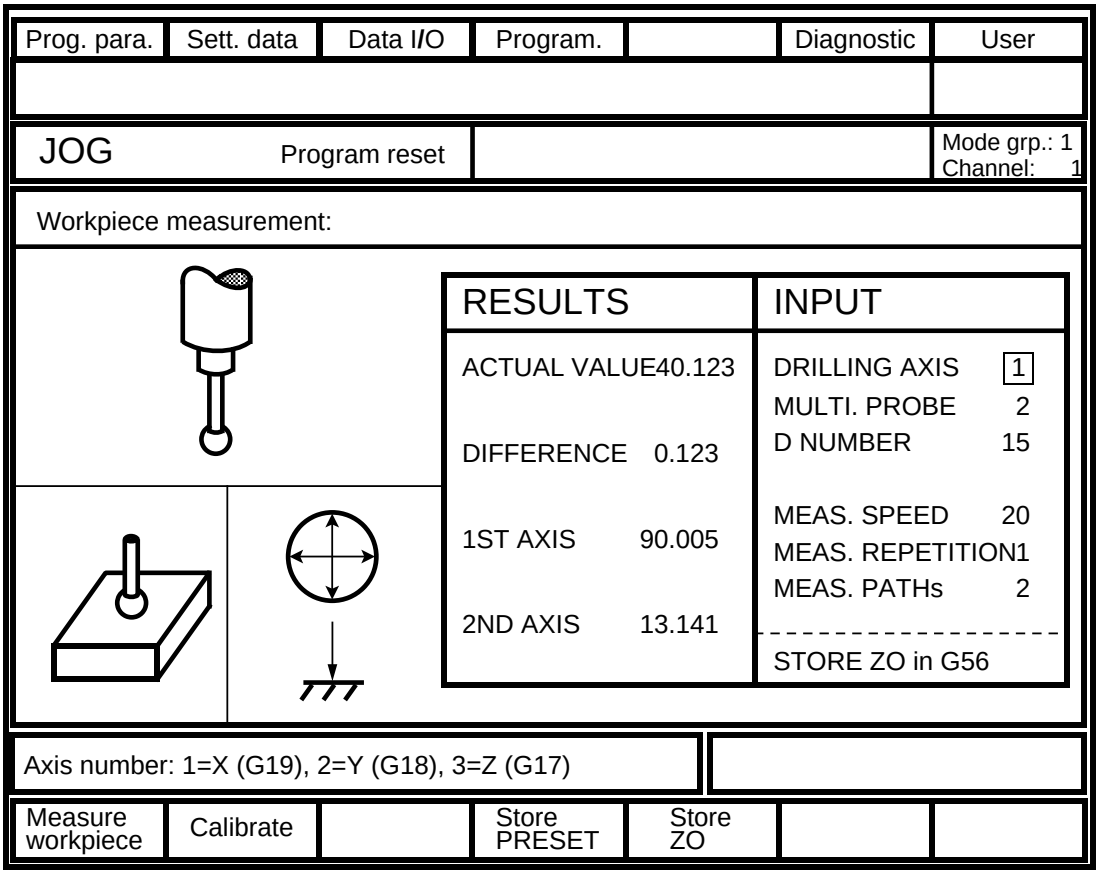
Selecting the measuring cycles:

Select ”JOG” mode



When the softkey "MEASURE WORKPIECE" is pressed the selection display "WORKPIECE MEASUREMENT" appears in which the parameters are entered for the measuring cycles and the functions, measure workpiece, calibrate, store PRESET and store ZO can be selected.

© Siemens AG 1990 All Rights Reserved



8.2 Setting the parameters for workpiece measurement and calibration

The necessary entries are input with operator guidance in the selection display, "WORKPIECE MEASUREMENT". The following parameters must be set.

- Drilling axis

  Permissible input values are 1 for X axis (machining plane YZ)
  2 for Y axis (machining plane ZX)
  3 for Z axis (machining plane XY).

- Number of probe

  Depending on the number of workpiece probes defined in installation, any value between 1 and 14 can be entered.

- D number

  Here you enter the D number under which the geometry data of the probe are stored.

- Measuring speed

  Path increment a is usually 1mm but can be increased or decreased when calling measuring cycles with parameter R28. If a > 1 is defined, the approach velocity is automatically increased from 150mm/min to 300 mm/min. This value was selected to ensure that the braking distance is less than the permissible deflection of the probe.

© Siemens AG 1990 All Rights Reserved

- Measurement repetition

  Here you enter the number of measurements at a certain measurement point.

- Measurement path s

  Here you enter the multiplication factor for the measurement path 2a, where a = 1 mm. If, for example, 3 is entered, this corresponds to a measurement path of 3 x 2a = 6 mm where, within the range 3 mm before the expected set position and 3 mm beyond it, the probe is expected to switch on.

- Store ZO in G5..

  Here any input value from 4 ≙ G54
  to 7 ≙ G57 is permissible.

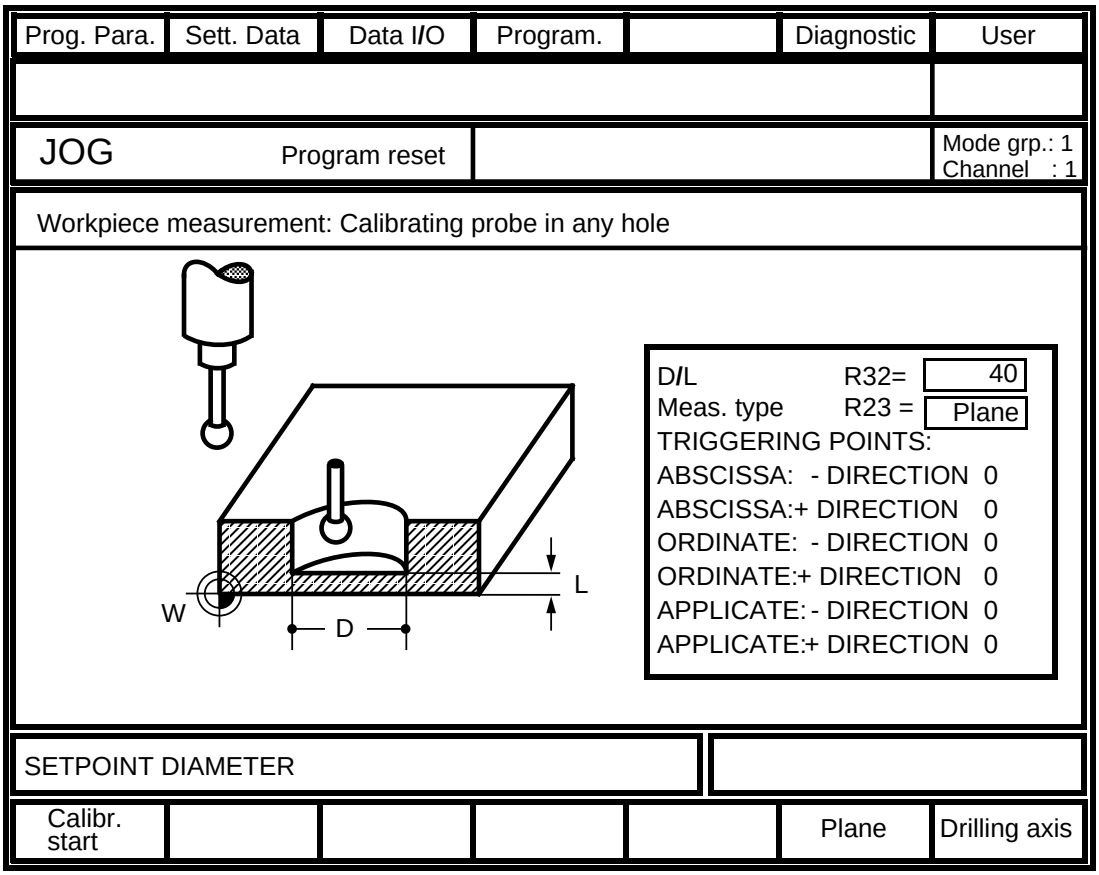
  The results obtained from the workpiece measurement (axis coordinates) are stored in the zero offset memories defined G54 - G57 when you press the softkey "STORE ZO".

The active input field in inverted. In addition, a text appears in the comment line bottom left.

When you have entered the required values and have pressed the key , the value is accepted and displayed.

8.3 Calibrating probe in any hole

This function is selected by pressing the softkey "CALIBRATE" in the "WORKPIECE MEASUREMENT" selection display.

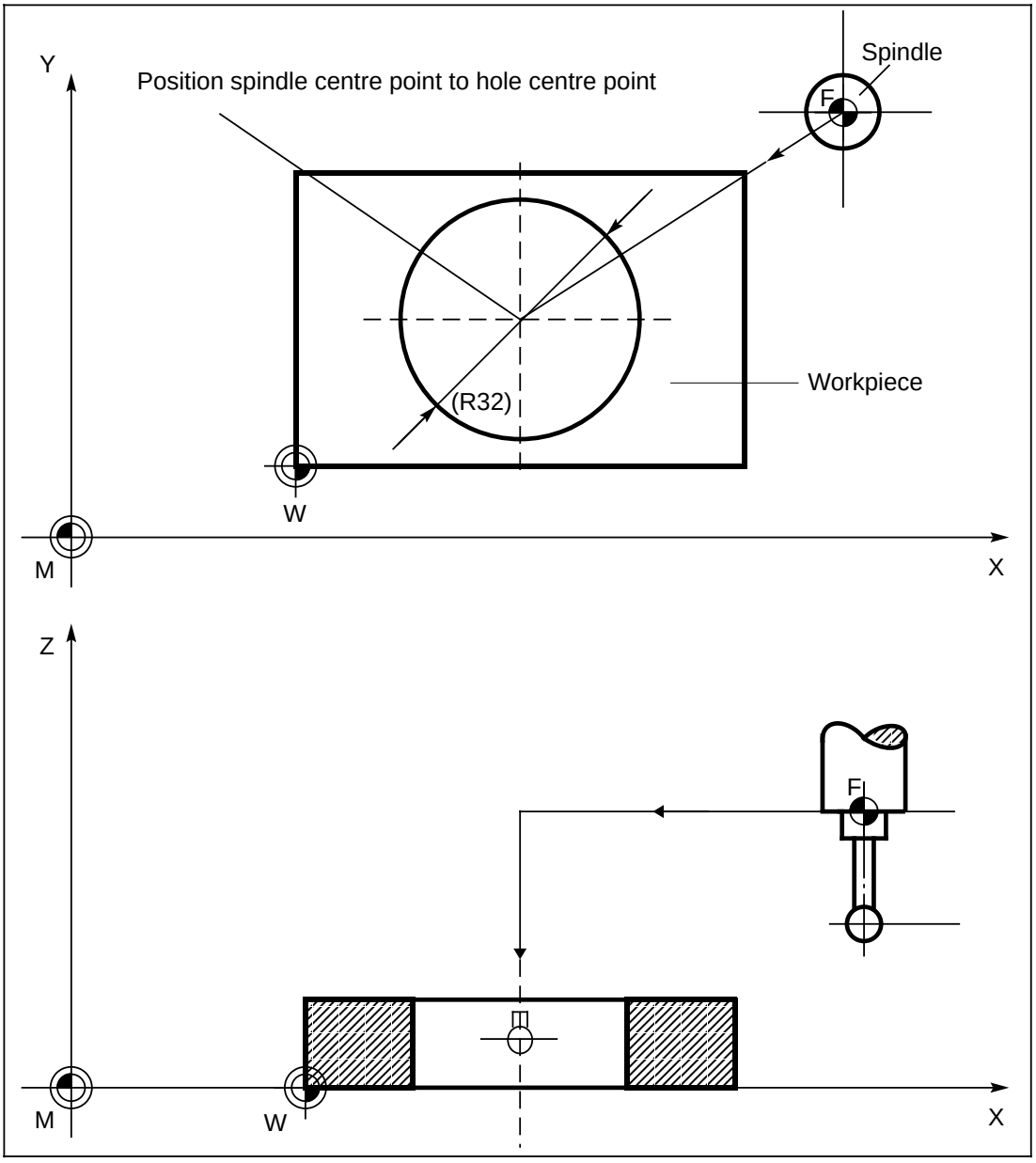


© Siemens AG 1990 All Rights Reserved

Once calibration type plane or drilling axis (Version 3.5 and higher) has been selected the only parameter that has to be entered is the setpoint diameter or the floor of the hole. The probe must be positioned in the centre of the hole at the calibrating depth.

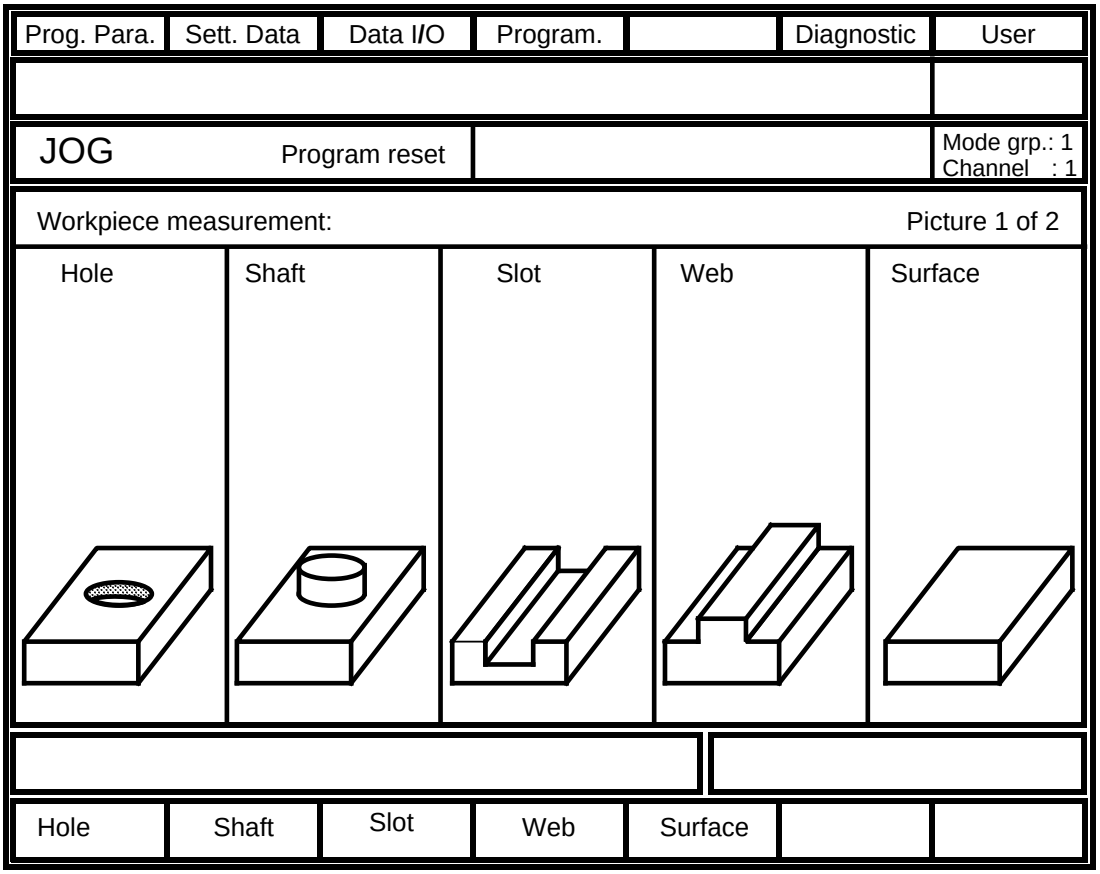
The hole centre or the hole floor must be known. If this is not the case or the determined TRIGGERING points are very different, the centre point of the hole must be determined with measuring variant "Measure hole" or the hole floor must be determined with the measuring version "Measure surface" in the applicate. Calibration then has to be repeated.

When you press the softkey "CALIBRATION START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. The measuring cycle is processed when "NC START" is operated. The determined trigger points are automatically stored in the relevant MDC area. The probe is located at the centre of the hole on completion of calibration. The control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen. The determined trigger points are displayed when the softkey "CALIBRATION" is pressed.



© Siemens AG 1990 All Rights Reserved

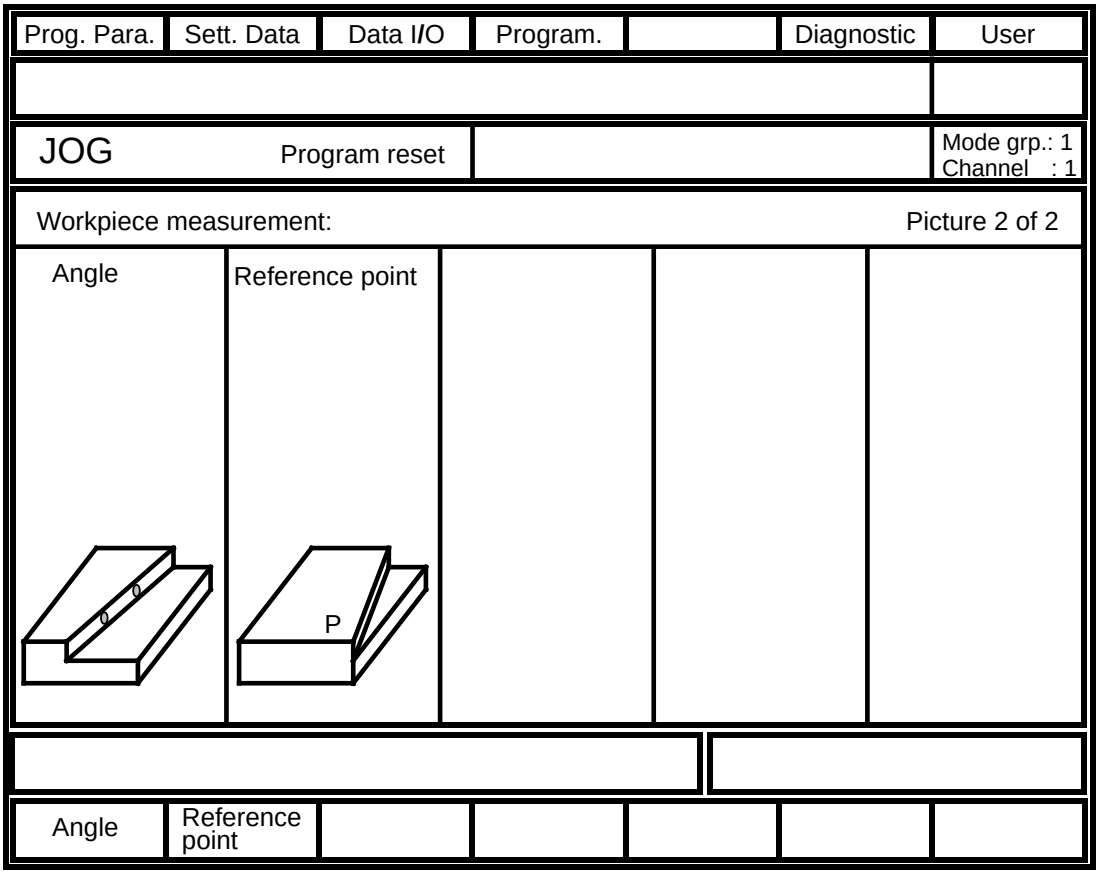
8.4 Workpiece measurement



The following measurement variants are available:

- Measurement of a hole
- Measurement of a shaft
- Measurement of a slot
- Measurement of a web
- Measurement of a surface
- Measurement of an angle
- Measurement of a reference point on the workpiece

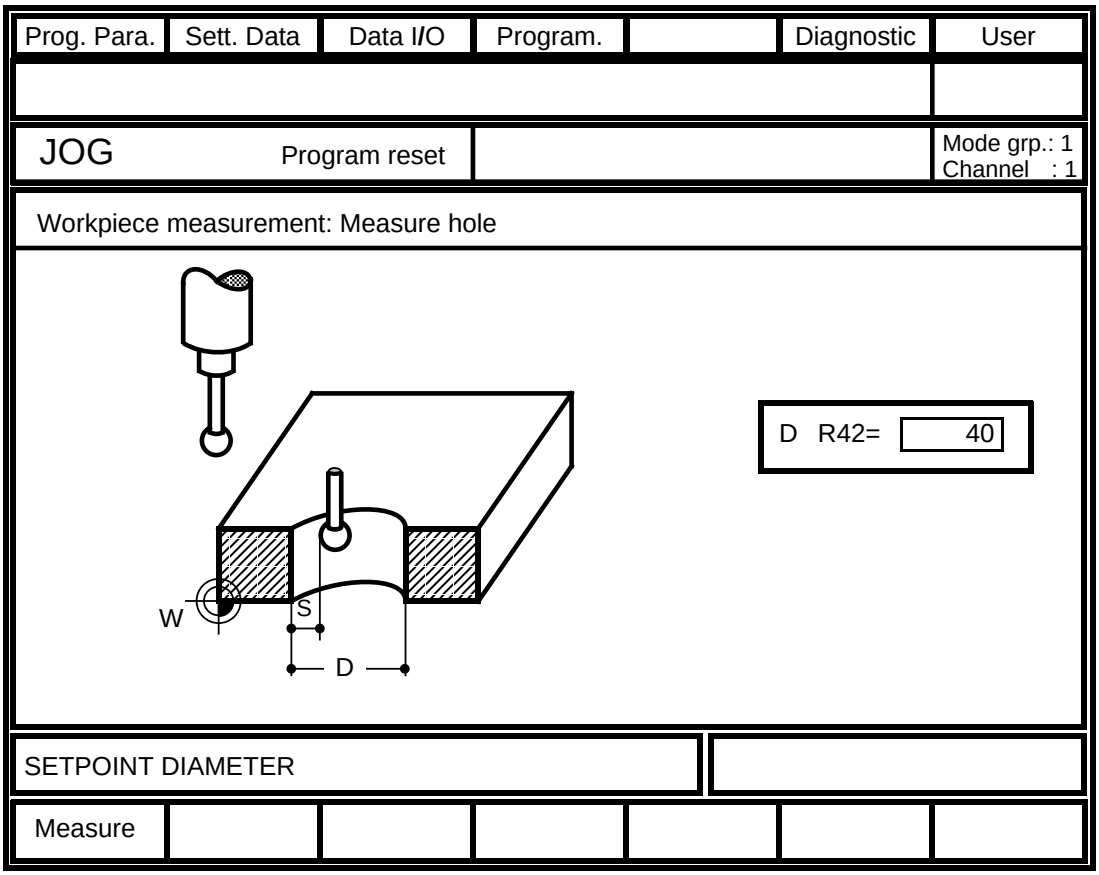
© Siemens AG 1990 All Rights Reserved



When you press the softkey to select the desired measurement variant the corresponding input display appears in which several more parameters have to be set.

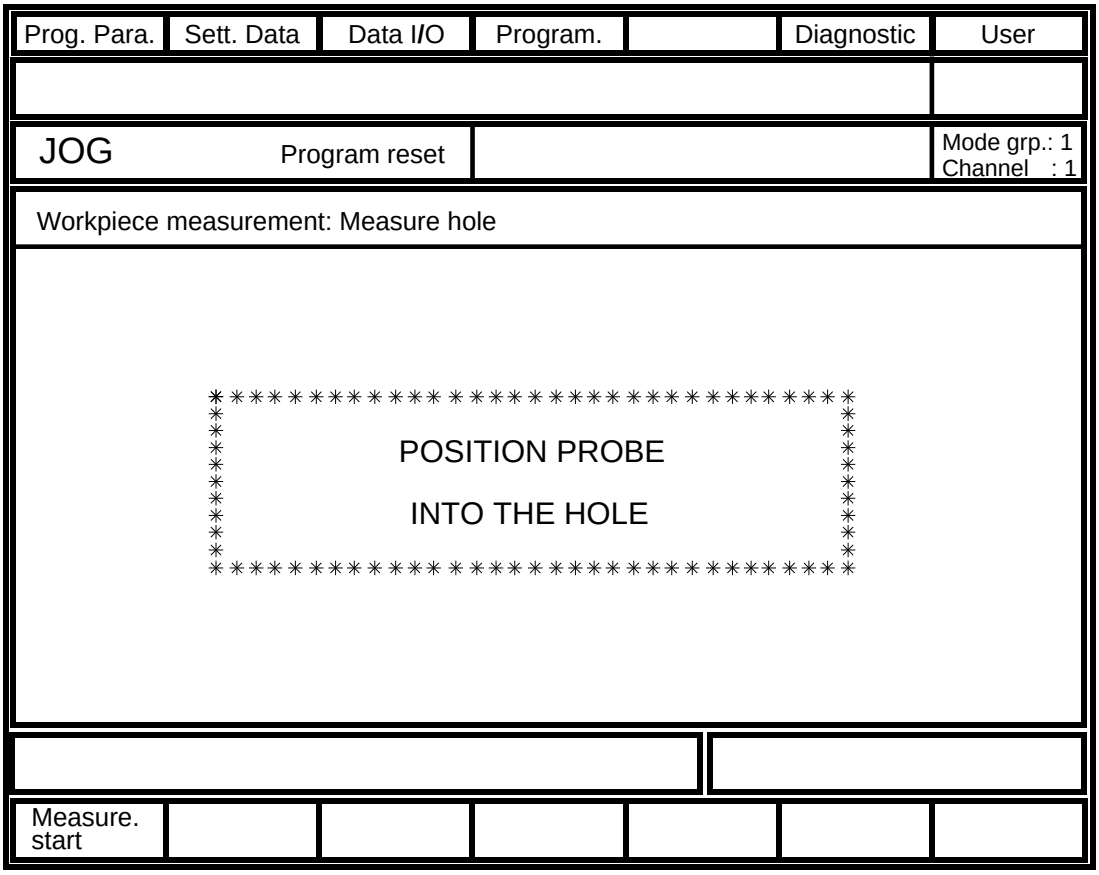
© Siemens AG 1990 All Rights Reserved

8.4.1 Measuring a hole



The only parameter that has to be set is the setpoint diameter of the hole. When you press the softkey "MEASURE", a selection display appears requesting you to position the probe at the centre of the hole.

© Siemens AG 1990 All Rights Reserved



When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected which runs in the automatic channel defined by the machine tool manufacturer. The measuring cycle is processed on NC START.

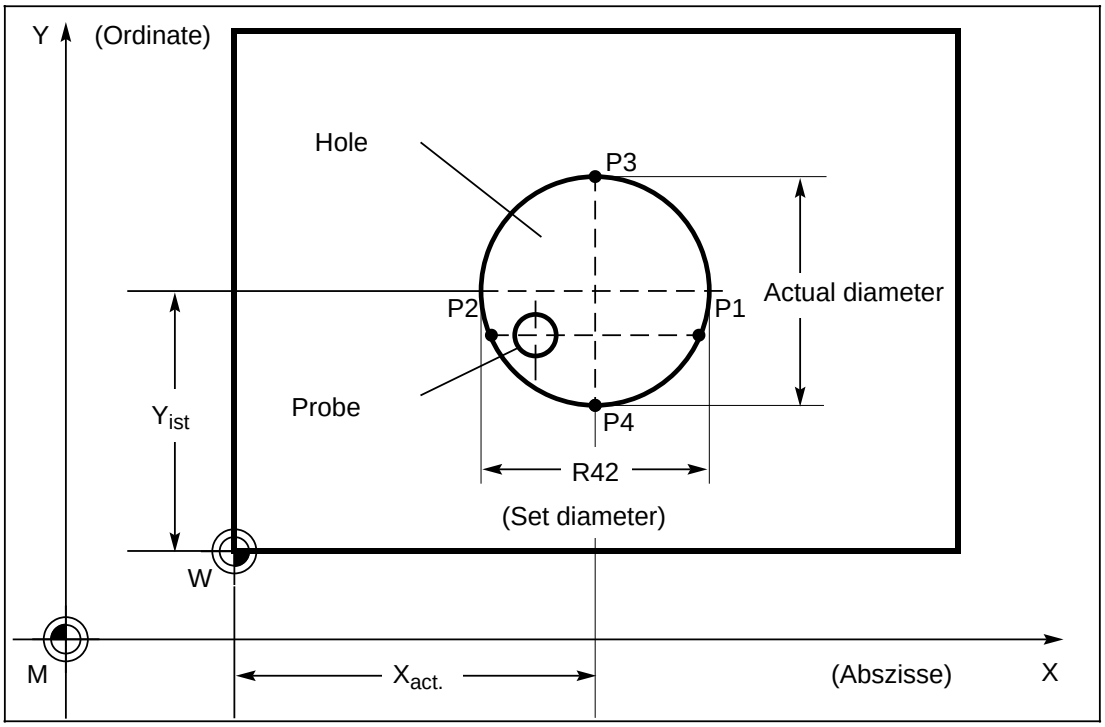
The measuring cycle measures points P1, P2, P3 and P4 in the hole in the abscissa and ordinate. The actual value of the hole diameter and the position of the centre point of the hole in the abscissa and ordinate are calculated from these four measured values with reference to the workpiece zero.

The centre point of the abscissa is calculated from points P1 and P2. Then the probe is positioned to the calculated centre point and points P3 and P4 are measured. The hole centre point along the ordinate and the hole diameter are calculated from these two points.

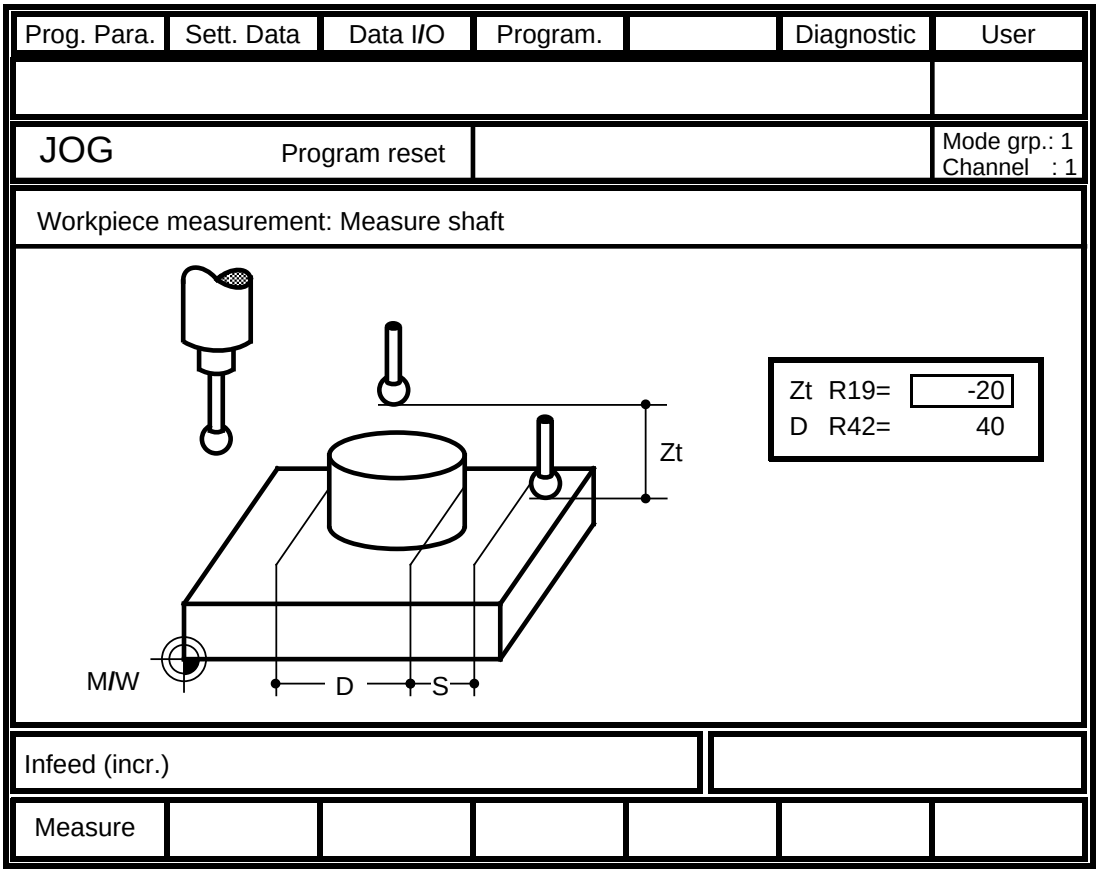
The probe is at the determined hole centre on completion of measurement. The control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the following results:

- Diameter of hole (actual dimension)
- Deviation from set diameter (difference)
- Centre point of hole in abscissa and ordinate in machine or workpiece coordinates

© Siemens AG 1990 All Rights Reserved



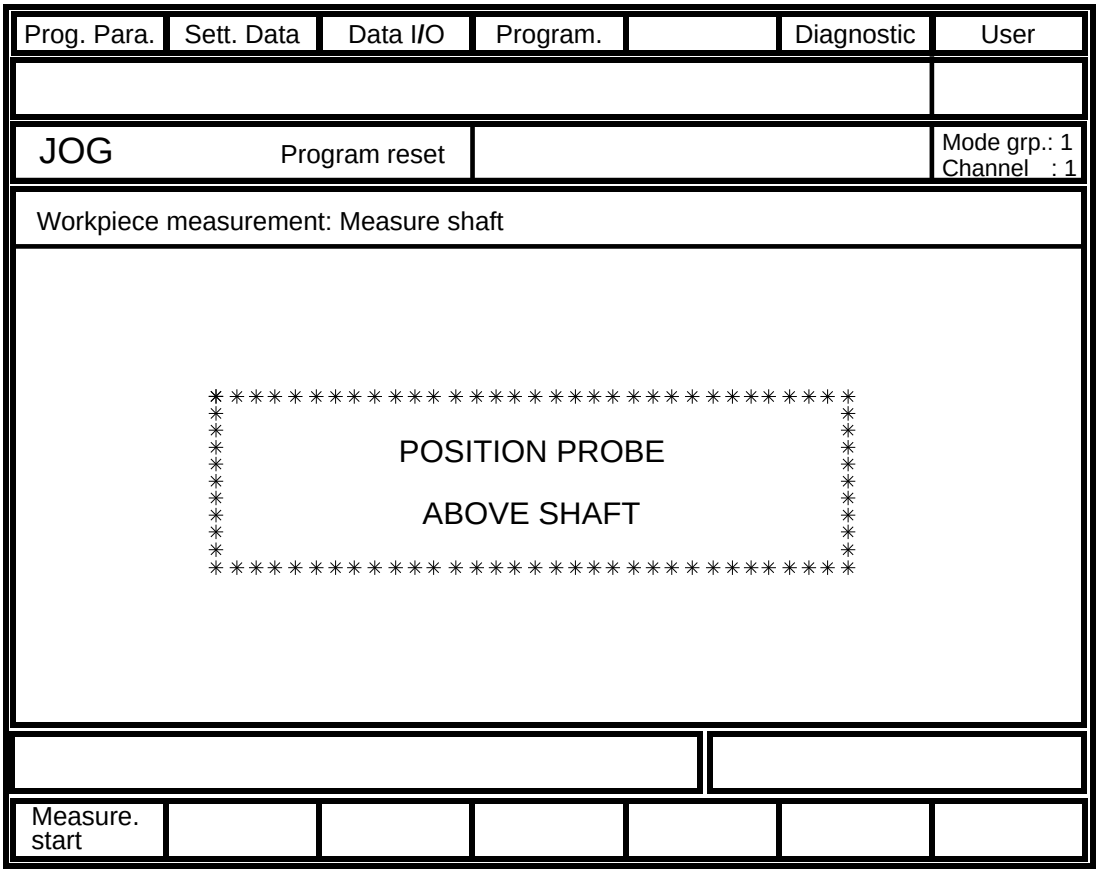
8.4.2 Measuring a shaft



The following parameters have to be set:

- The infeed along the applicate axis with sign
- Set diameter of the shaft.

When you press the softkey "MEASURE", a selection display appears requesting you to position the probe above the centre of the shaft.



When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. The measuring cycle is processed on NC START.

The measuring cycle measures points P1, P2, P3 and P4 along the abscissa and ordinate. The actual shaft diameter as well as the position of the centre of the shaft in the abscissa and ordinate are calculated from these four measured values with reference to the workpiece zero.

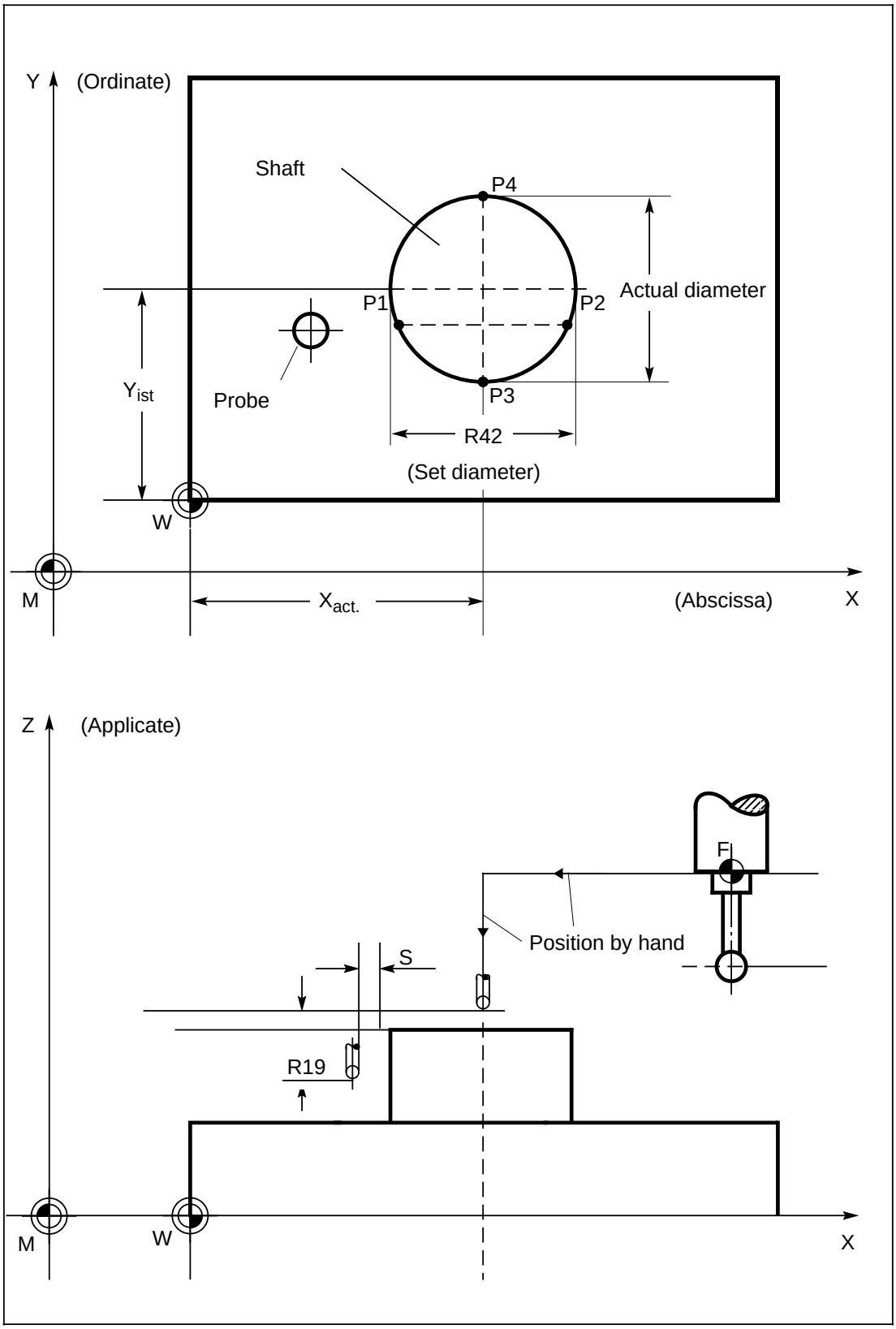
The centre point of the abscissa is calculated from points P1 and P2. Then the probe is positioned to the calculated centre point and points P3 and P4 are measured. The shaft centre point along the ordinate and the shaft diameter are calculated from these two points.

The probe is at the determined shaft centre on completion of measurement.

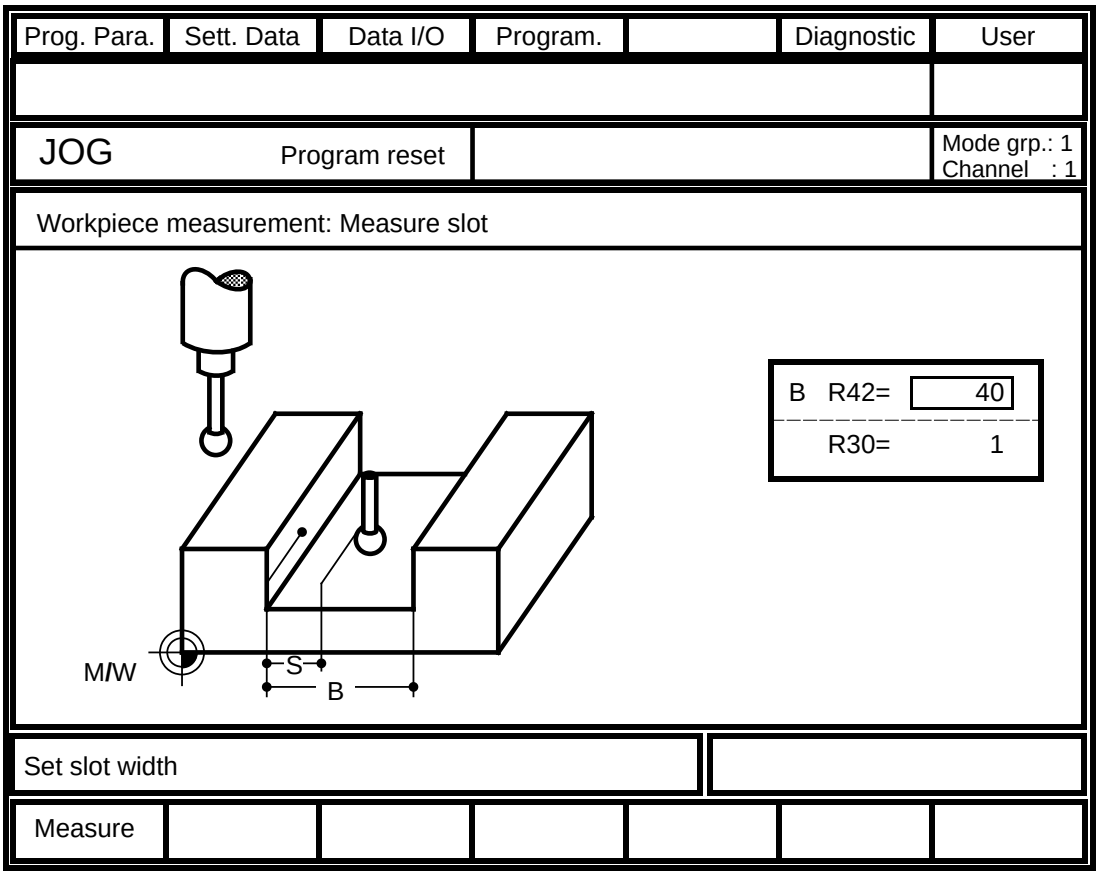
The control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the following results

- Diameter of shaft (actual dimension)
- Deviation from set diameter (difference)
- Centre point of shaft in abscissa and ordinate in machine or workpiece coordinates

© Siemens AG 1990 All Rights Reserved


Y
(Ordinate)
Shaft
P4
P1
P2
P3
Actual diameter
Probe
Y_{ist}
R42
(Set diameter)
W
M
$X_{act.}$
(Abscissa)
X
Z
(Applicate)
F
Position by hand
S
R19
M
W
X

8.4.3 Measuring a slot



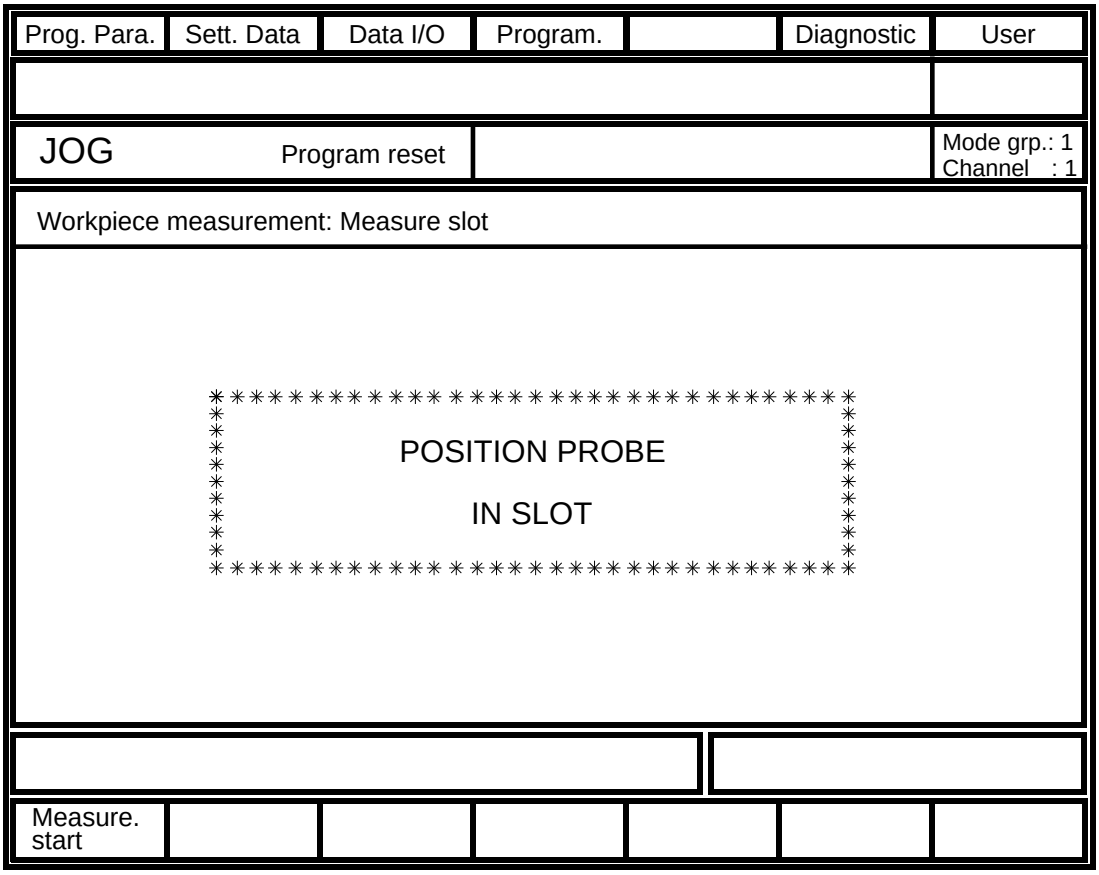
The following parameters must be set:

- Set slot width
- Number of axis along which measurement must be taken

  1 ≙ abscissa
  2 ≙ ordinate

When you press the softkey "MEASURE", a selection display appears requesting you to position the probe at the centre of the slot.

© Siemens AG 1990 All Rights Reserved



When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. The measuring cycle is processed on NC START.

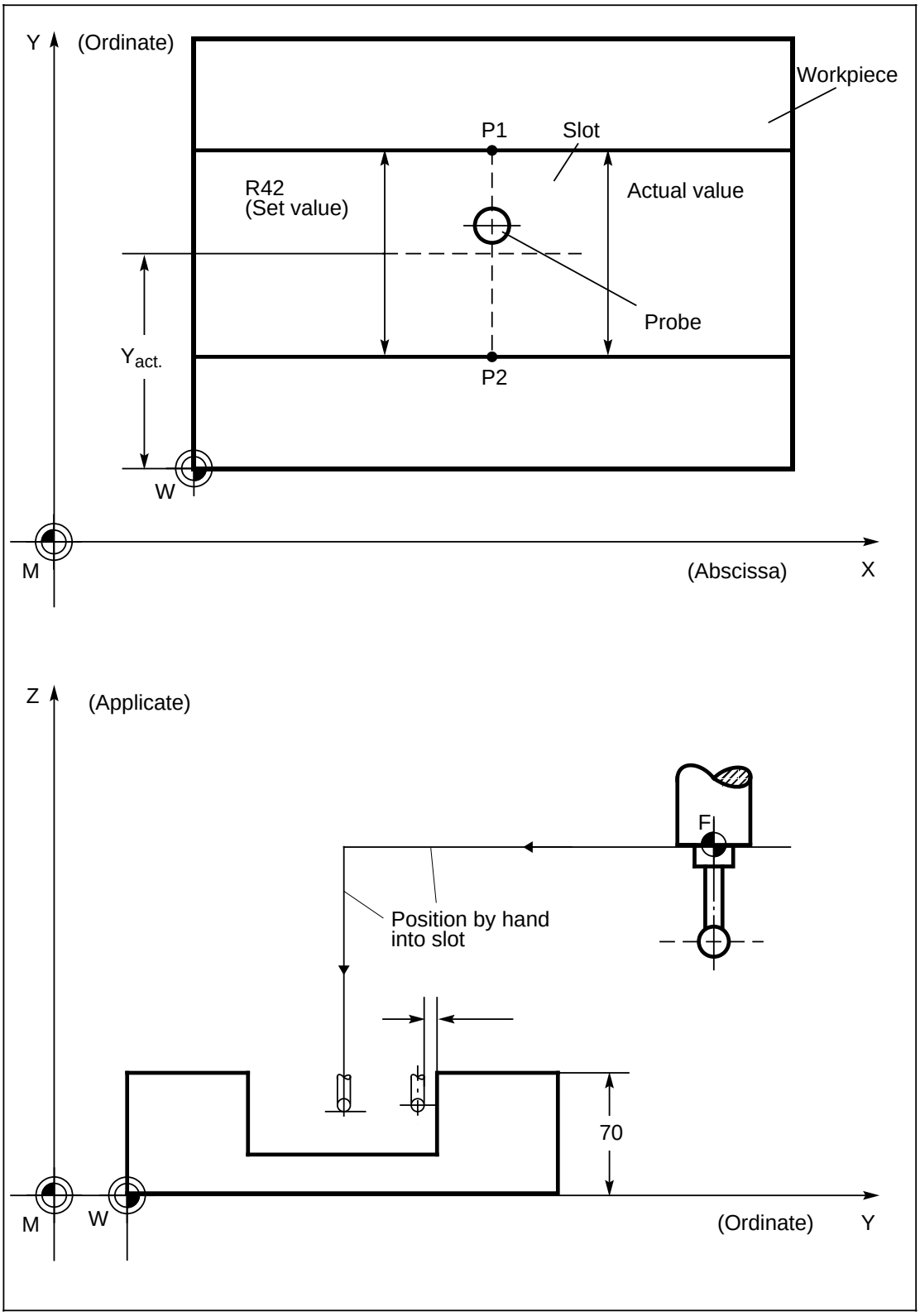
The measuring cycle measures points P1 and P2 in the slot along the measuring axis. The actual value of the slot as well as the position of the centre point of the slot along the measuring axis with reference to workpiece zero is calculated.

The probe is at the determined slot centre on completion of measurement.

The control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the following results

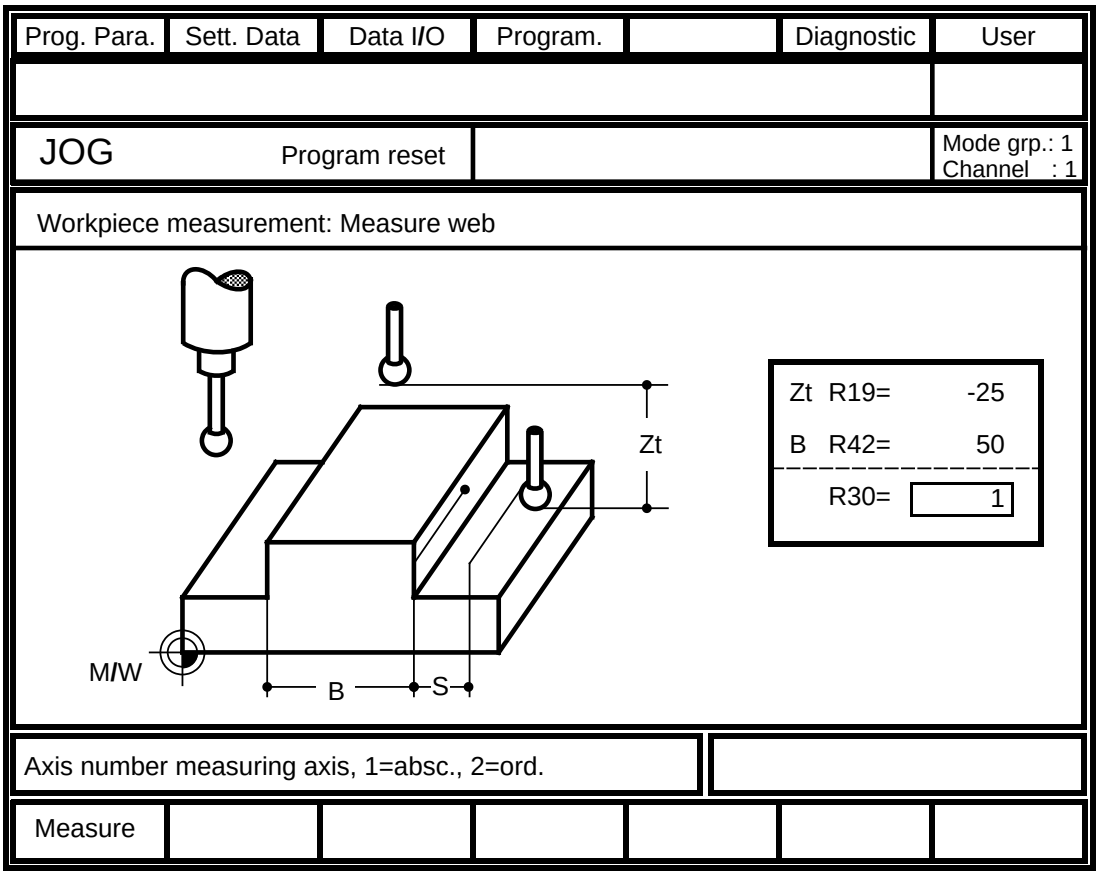
- Width of slot (actual dimension)
- Deviation from set slot width (difference)
- Slot centre in the measured axis in machine or workpiece coordinates

© Siemens AG 1990 All Rights Reserved



© Siemens AG 1990 All Rights Reserved

8.4.4 Measuring a web



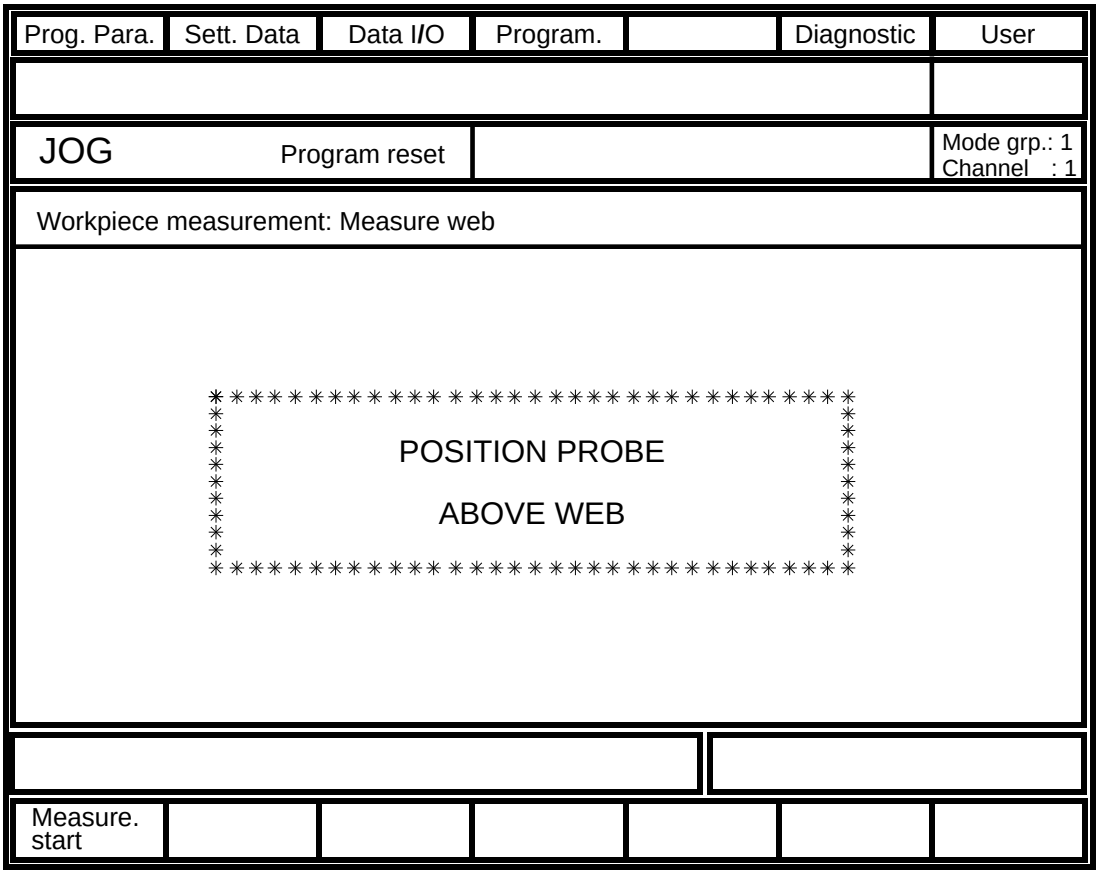
The following parameters have to be set:

- The infeed along the applicate with sign
- Set width of web
- Number of axis along which measurement is to be taken

  1 =^ abscissa
  2 =^ ordinate

When you press the softkey "MEASURE", a selection display appears requesting you to position the probe above the centre of the web.

© Siemens AG 1990 All Rights Reserved



When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. The measuring cycle is processed on NC START.

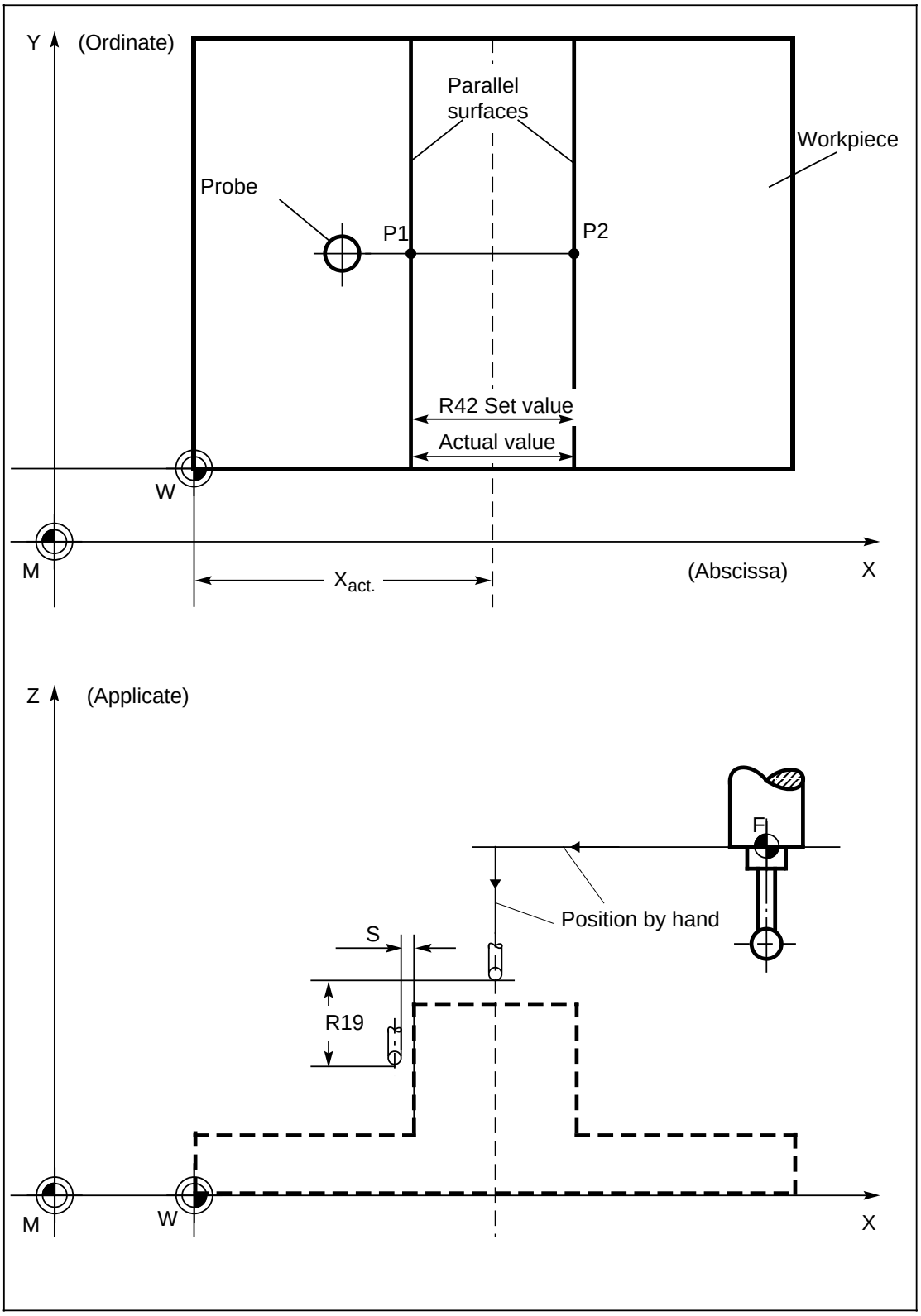
The measuring cycle measures two parallel surfaces (web) at points P1 and P2 along the measuring axis. The actual distance between the two parallel surfaces as well as the position of the centre point of the measuring axis with reference to the workpiece zero are calculated from these two values.

The probe is at the determined centre point on completion of measurement.

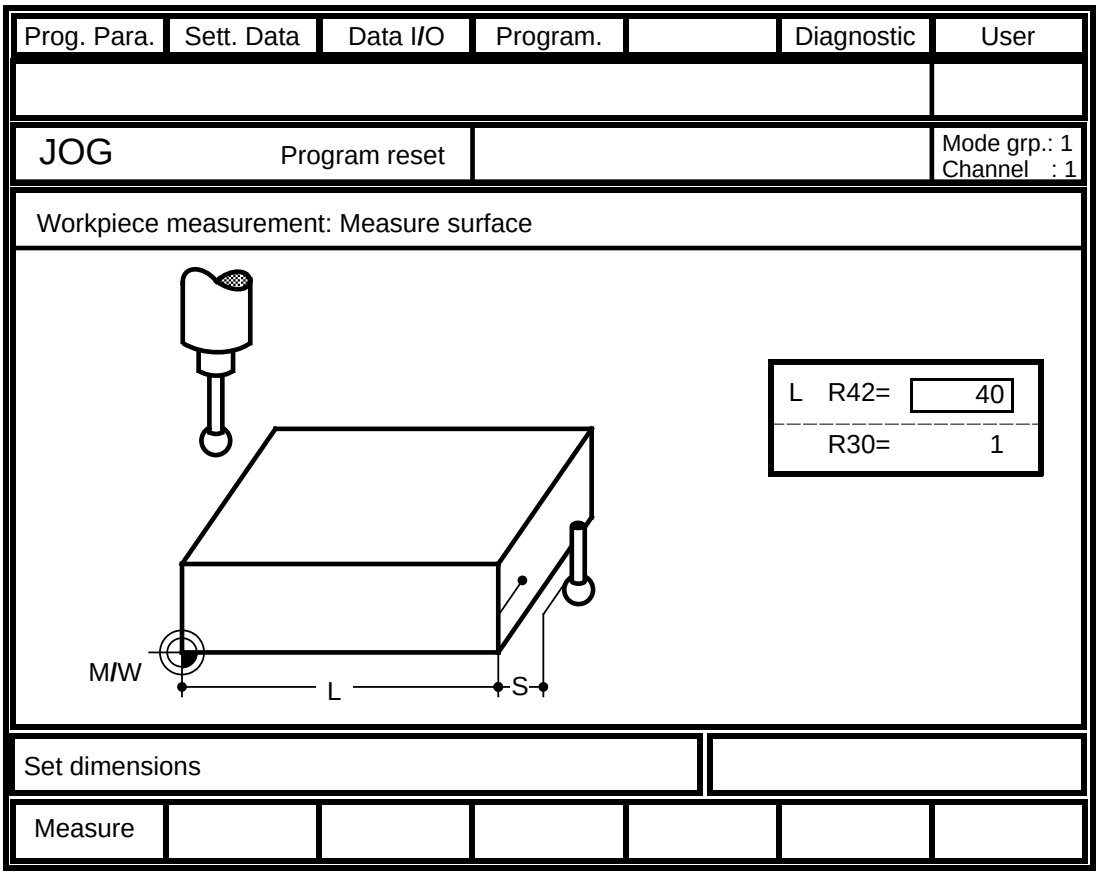
The control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the following results

- Width of web (actual dimension)
- Deviation from set web width (difference)
- Web centre in the measured axis in machine or workpiece coordinates

© Siemens AG 1990 All Rights Reserved


Y
(Ordinate)
Parallel surfaces
Workpiece
Probe
P1
P2
R42 Set value
Actual value
W
M
$X_{act.}$
(Abscissa)
X
Z
(Applicate)
F
Position by hand
S
R19
M
W
X

8.4.5 Measuring a surface



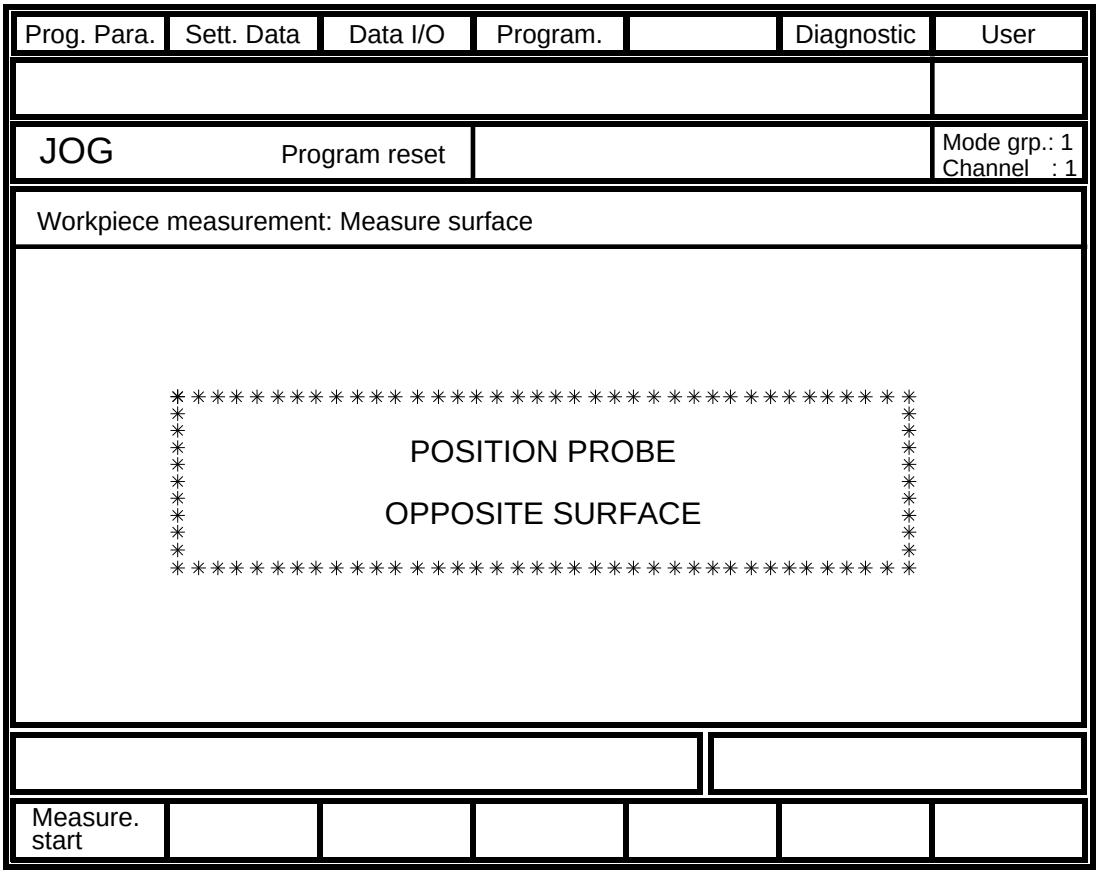
The following parameters must be set:

- Set dimension L
- Number of axis along which measurement must be taken

  1 ≙ abscissa
  2 ≙ ordinate
  3 ≙ applicate (drilling axis) (as from Version 3.5)

When you press the softkey "MEASURE", a selection display appears requesting you to position the probe opposite the surface.

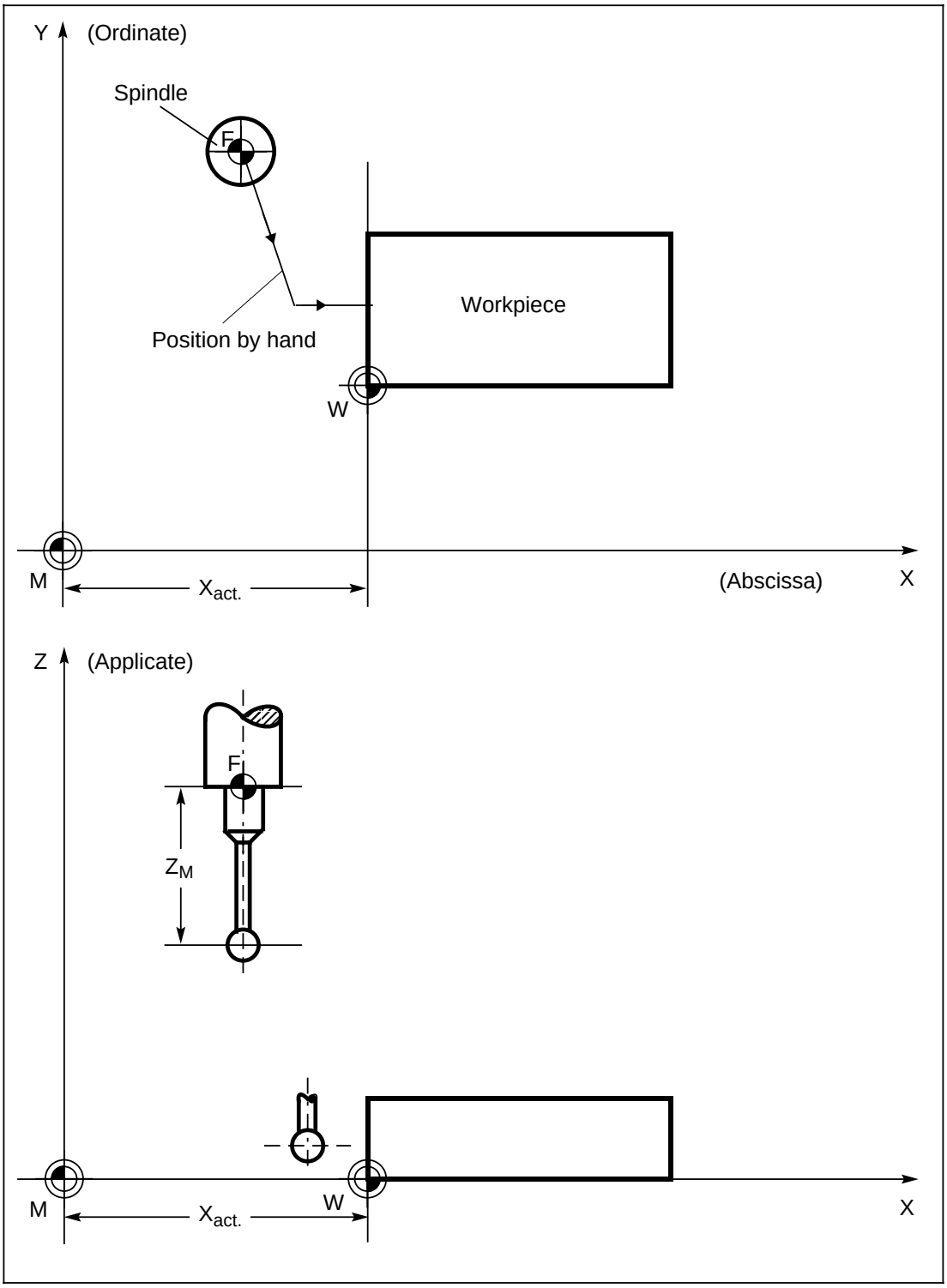
© Siemens AG 1990 All Rights Reserved



When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. The measuring cycle is processed on NC START. The measuring cycle determines the actual value of the workpiece along the selected axis and the setpoint/actual value difference.

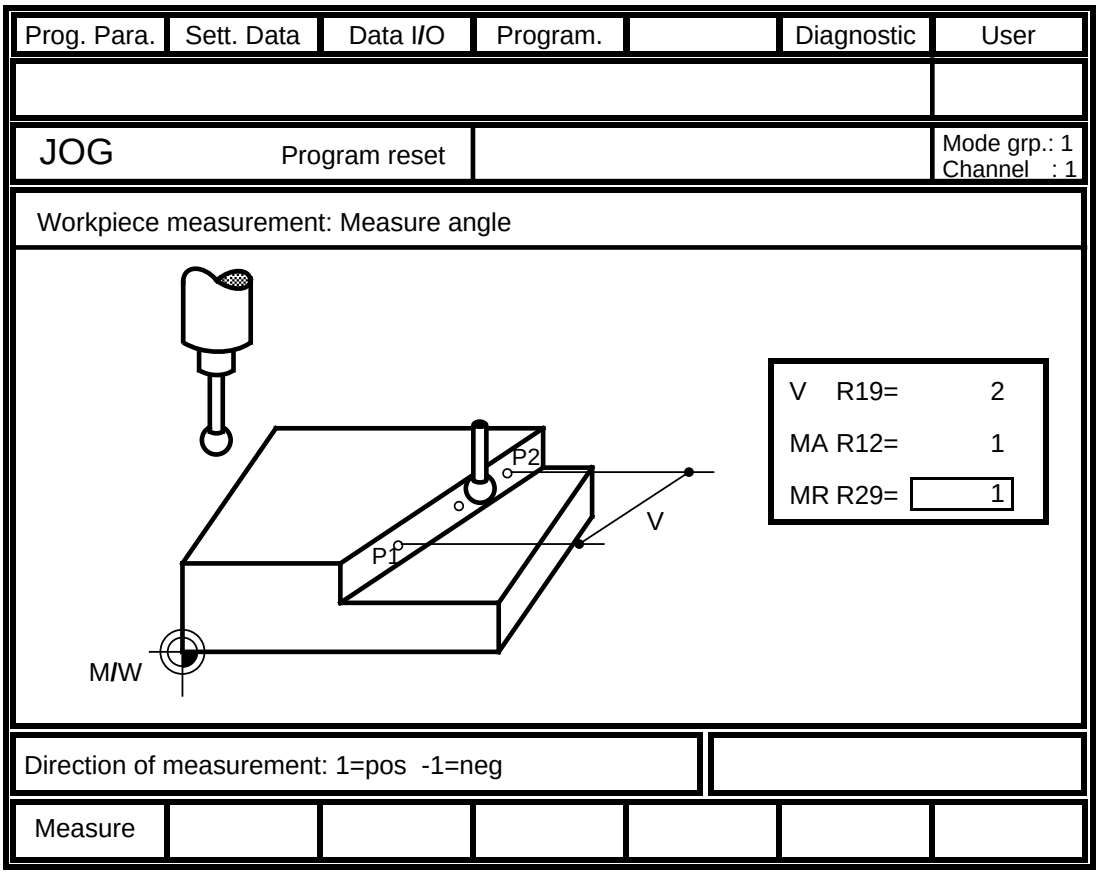
The probe is positioned the distance "a" from the measured surface on completion of measurement. The control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the measurement results.

© Siemens AG 1990 All Rights Reserved



© Siemens AG 1990 All Rights Reserved

8.4.6 Measuring an angle



The following parameters have to be set:

- The distance between the measuring points in the offset axis (positive values only)
- Number of axis along which measurement is to be taken

  1 ≙ abscissa
  2 ≙ ordinate

- Direction of measurement

  1 ≙ Measurement to be taken in positive direction
  -1 ≙ Measurement to be taken in negative direction

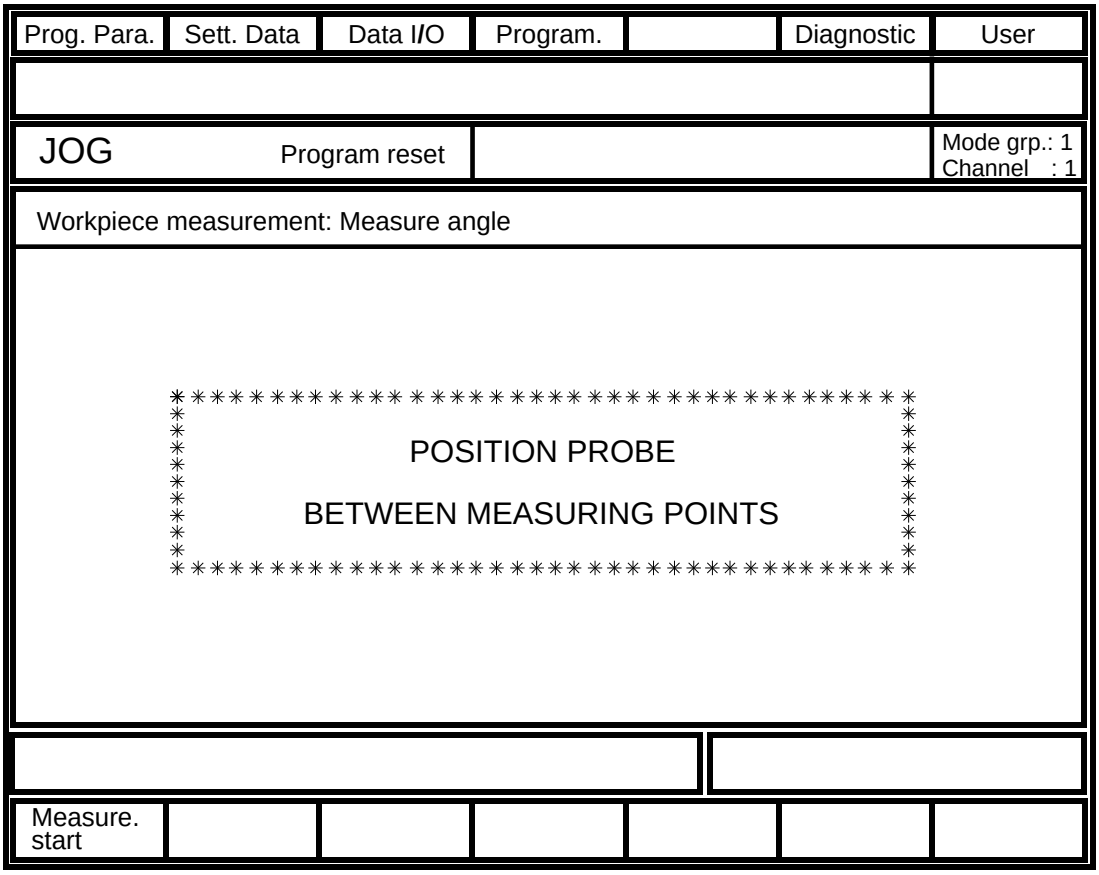
Note:

The measurement path "m" along the measuring axis must be smaller than the offset path R19. Therefore only an angle of max. 44.999° can be measured. However, the measurement can be taken from every side.

Measuring cycle L978 automatically calculates a measurement path as a function of the offset path that fulfils this condition. The measurement path S entered in the basic display is only taken into account if it is greater than the measurement path calculated by the cycle.

When you press the softkey "MEASURE", a selection display appears requesting you to position the probe between the measuring points on the workpiece.

© Siemens AG 1990 All Rights Reserved

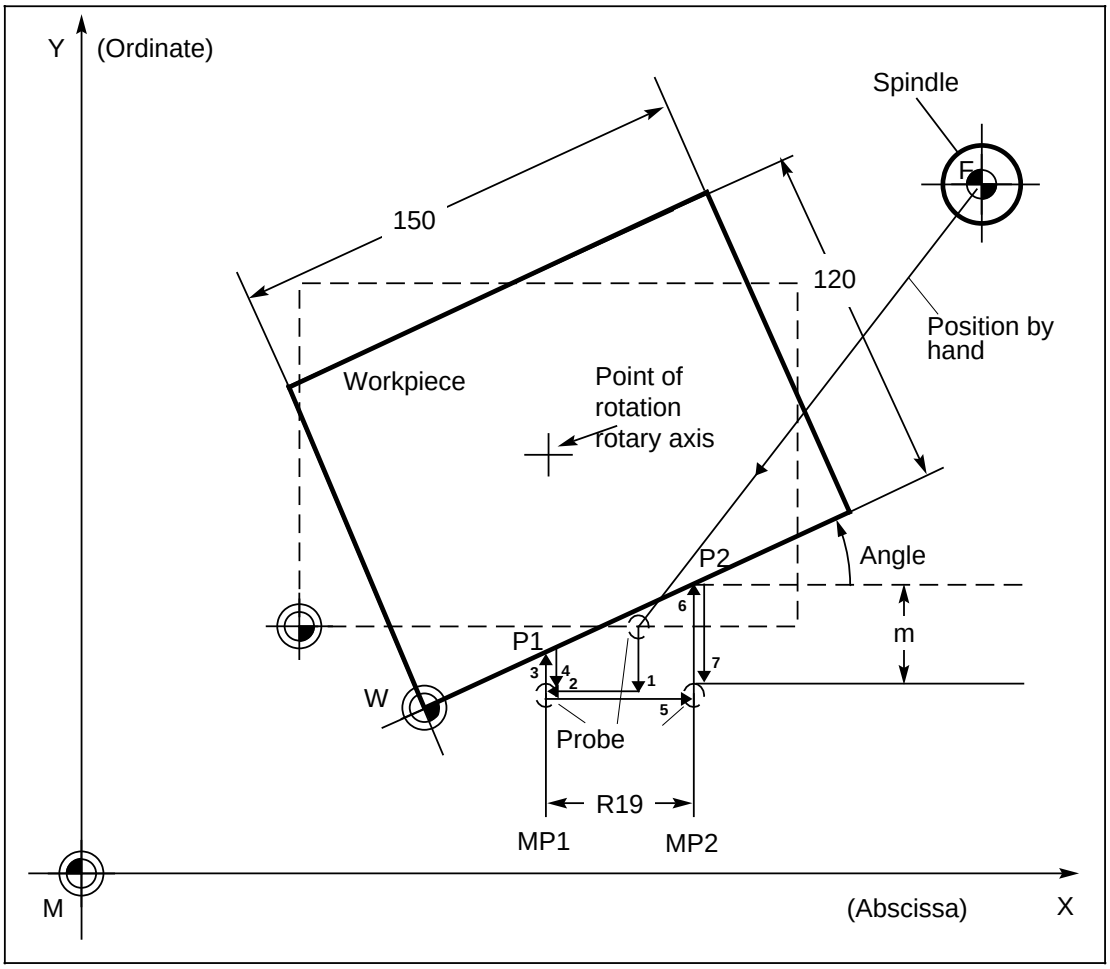


When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. The measuring cycle is executed on NC START according to the sequence 1-2-3-4-5-6-7 shown in the next display.

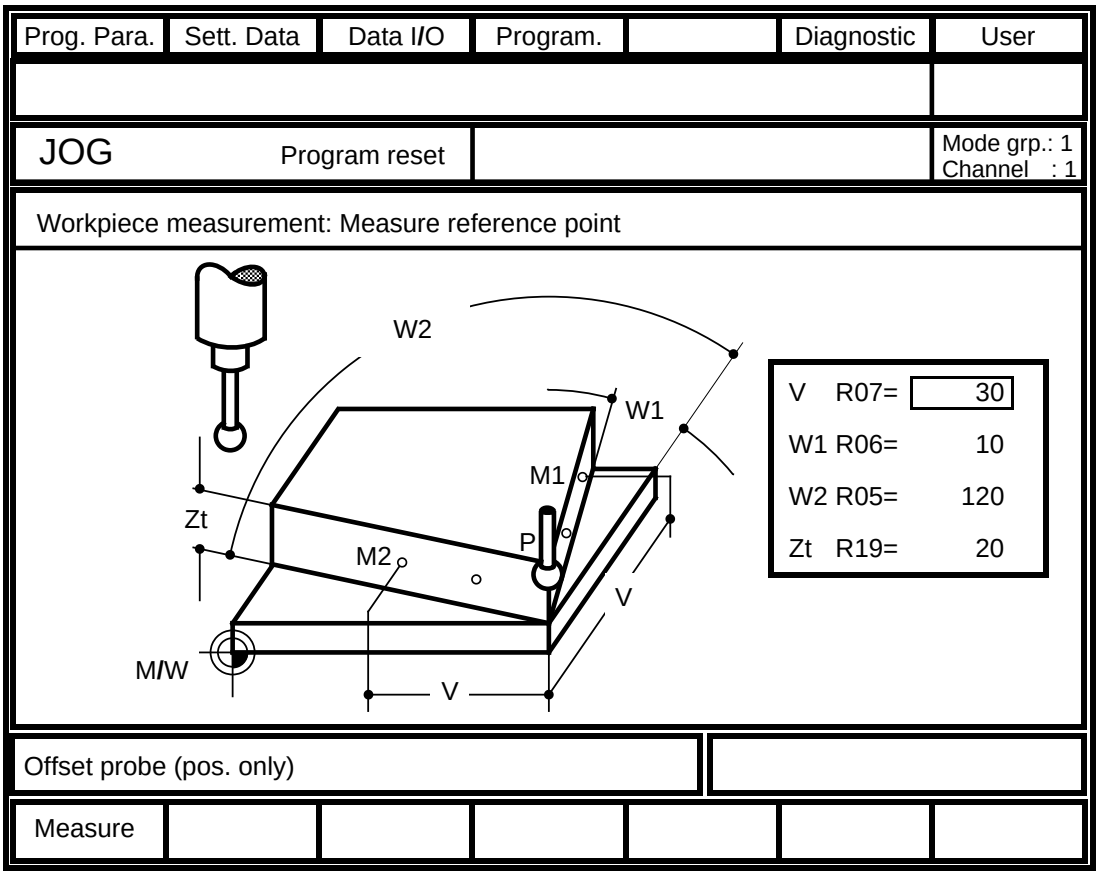
The position of the angle of a workpiece with reference to the offset axis is determined. The probe is positioned at measuring point 2 opposite the measuring surface on completion of measurement. Then the control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the following results

- Angle with reference to offset axis.

© Siemens AG 1990 All Rights Reserved


Y
(Ordinate)
Spindle
F
150
120
Position by hand
Workpiece
Point of rotation rotary axis
P2
Angle
6
m
P1
3
4
2
1
7
W
5
Probe
R19
MP1
MP2
M
(Abscissa)
X

8.4.7 Measuring a reference point on the workpiece

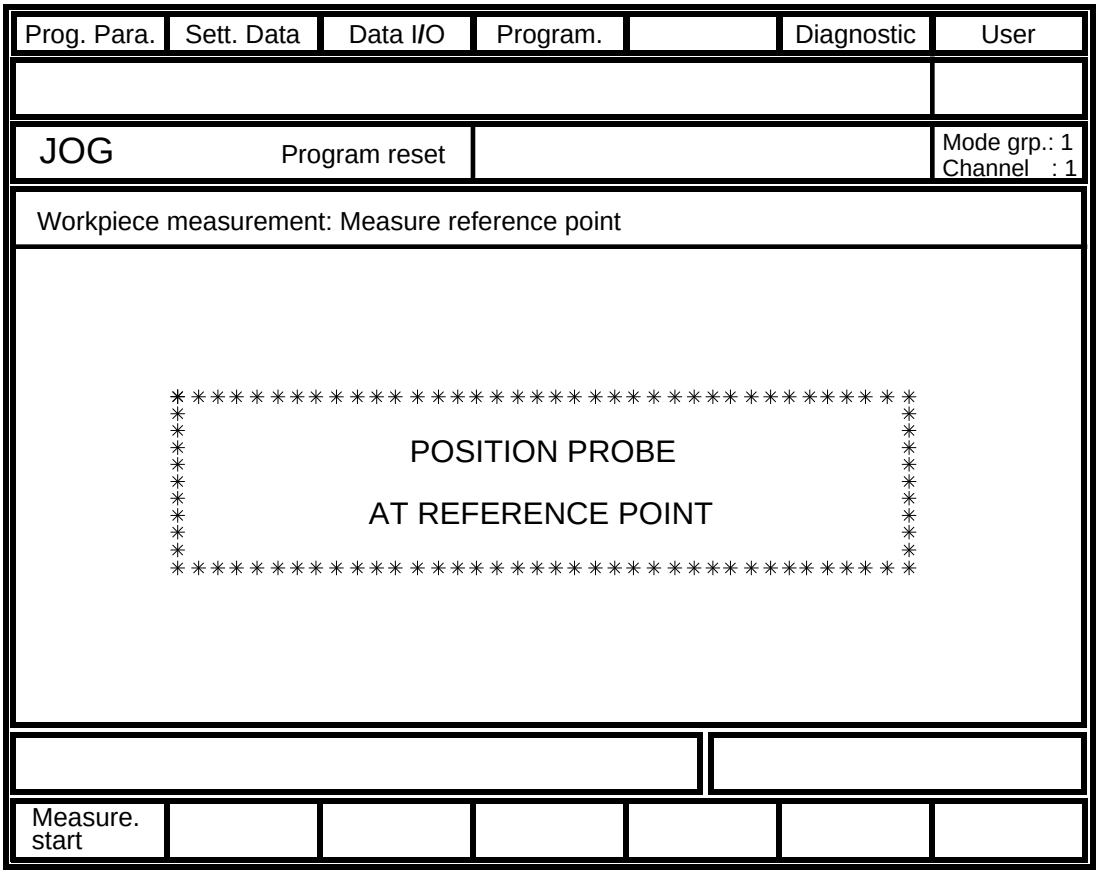


The following parameters have to be set:

- Offset of probe V (positive only)
- Angle 1 to axis 1 (abscissa)
- Angle 2 to axis 1
- Infeed in the applicate (drilling axis) incrementally to overtravelling the workpiece.

When you press the softkey "MEASURE", a selection display appears requesting you to position the probe at the reference point.

© Siemens AG 1990 All Rights Reserved

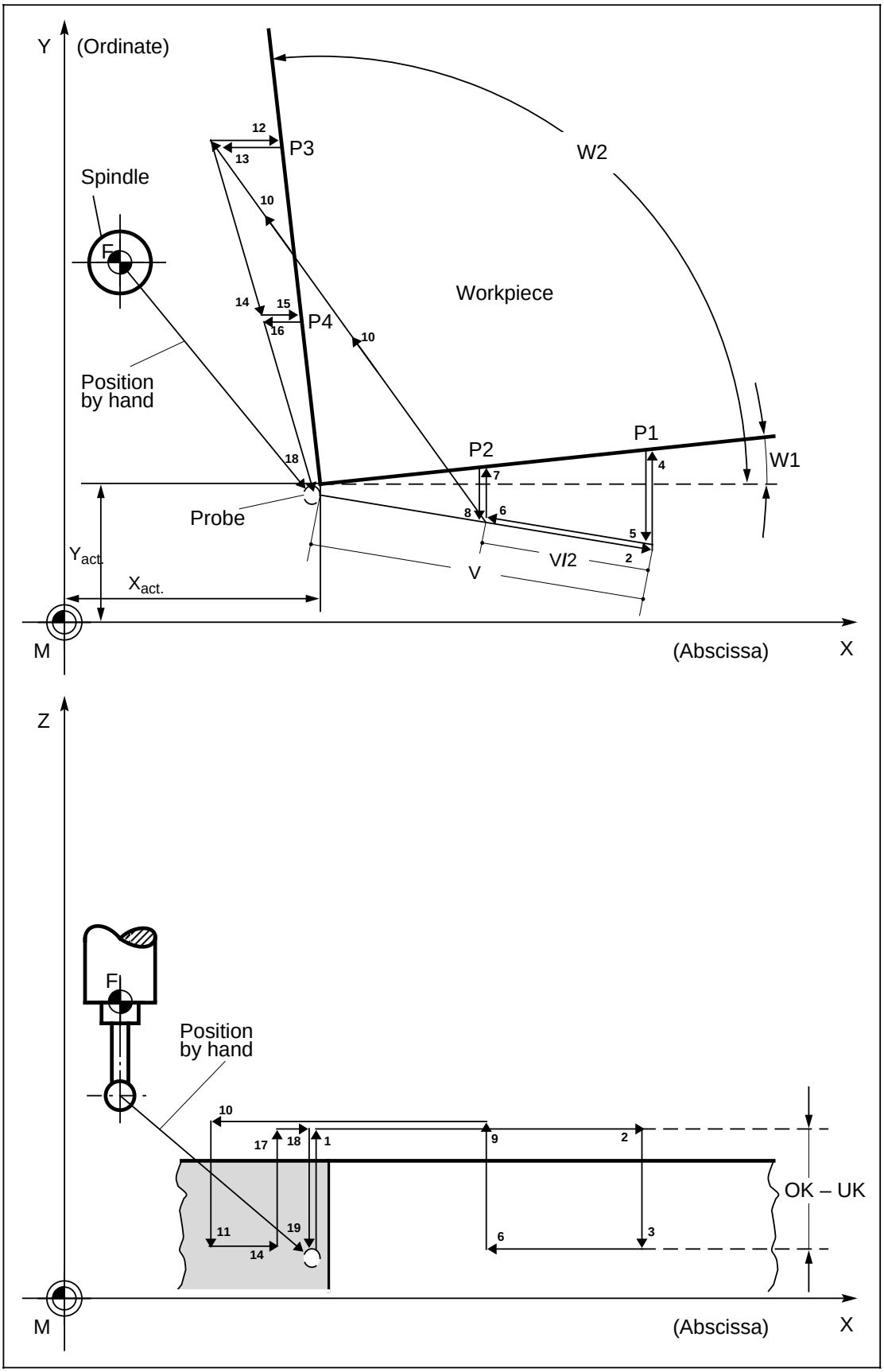


When you press the softkey "MEASUREMENT START" the control automatically switches to "AUTOMATIC" mode and measuring cycle L887 is selected. Execution of the measuring cycle is triggered on NC START. Points P1 and P2 of the first edge are measured and then P3 and P4 of the second edge.

The cycle calculates the point of intersection of the two lines and the result is the coordinates that it supplies. The probe is again positioned at the starting point on completion of measurement. Then the control automatically switches to JOG mode and the selection display "WORKPIECE MEASUREMENT" appears on the screen with the following results

- Reference point, abscissa in machine or workpiece coordinates
- Reference point, ordinate in machine or workpiece coordinates

© Siemens AG 1990 All Rights Reserved



END OF SECTION

© Siemens AG 1990 All Rights Reserved

9 Alarms

9.1 Measuring cycle alarms

Alarm Number	Alarm Group	Alarm is cancelled by ...
4000	Measuring cycles	Press acknow-ledgement key
4001 ... 4039	Measuring cycles	Press RESET key
4040 ... 4049	Measuring cycles	Press acknow-ledgement key
4050 ..4099	Mould making cycles	Press RESET key
4100 ... 4299	Standard cycles partly meas. cycles	Press RESET key
5000 ... 5299	User	

© Siemens AG 1990 All Rights Reserved

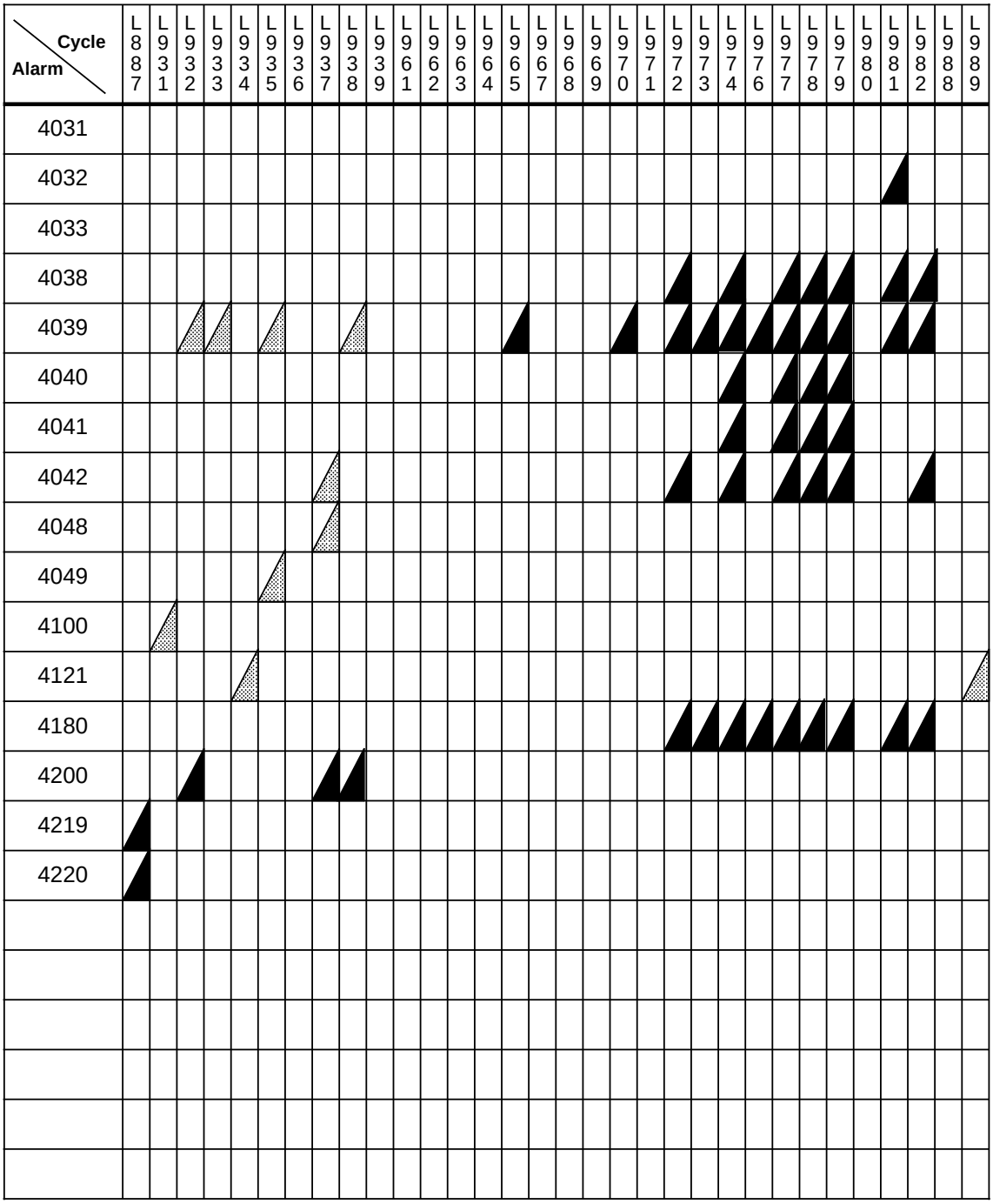
The table shows which alarms occur in the various cycles and can be output:

Cycle / Alarm	L887	L931	L932	L933	L934	L935	L936	L937	L938	L939	L961	L962	L963	L964	L965	L967	L968	L969	L970	L971	L972	L973	L974	L976	L977	L978	L979	L980	L981	L982	L988	L989
4000																																
4001			◢																													
4002			◢																													
4003			◢																													
4004																																
4005																																
4006		◢													■				■													
4007																					■	■	■	■	■	■	■		■	■		
4008			■	■															■												■	
4009		◢																					■		■	■	■					
4010					◢														■													◢
4011					◢																											◢
4012					◢									◢					■	◢	■	■	■	■	■	■	■	◢		■		◢
4013		◢																			■											
4014		◢						◢		◢																						
4015																			■													
4019													■																			
4020								◢																								
4021								◢		◢																						
4025																					■									■		
4026																					■									■		
4027																																
4030								◢		◢											■	■	■	■	■	■	■		■	■		

■ Alarm is output directly by the cycle

◢ Alarm is set and output by the calling cycle

© Siemens AG 1990 All Rights Reserved



Alarm is output directly by the cycle

Alarm is set and output by the calling cycle

© Siemens AG 1990 All Rights Reserved

9.2 Measuring cycle alarms

The following alarm texts are stored in the UMS.
They appear in the alarm line on the screen when an alarm occurs.

Alarm No.	Alarm Text	Cancel Alarm by ...
4000	Cancel alarm	
4001	Check MDC 0/1	
4002	Check MDC 2/3	
4003	Check MDC 1000 - 1019	
4004	Load MDC inhibited	
4005	MDC area exceeded	
4006	NC type or NC SW version wrong	
4007	Wrong MC programs loaded	
4008	Check MDC (Nxxxx)	
4009	Measurement not possible	
4010	Probe is defective	
4011	Probe does not switch	
4012	Probe collision	

© Siemens AG 1990 All Rights Reserved

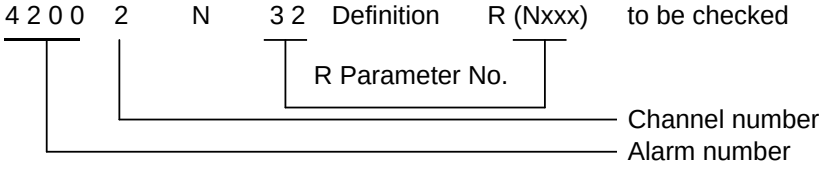
Alarm No.	Alarm Text	Cancel Alarm by ...
4013	Check type in TO memory	
4014	STACK overflow	
4015	Check ZO number	
4019	Calculation not possible	
4020	No D No. received from PLC	
4021	PLC interface not ready	
4025	Tool tip position wrong	
4026	Toolnose location 9 defined	
4027	Toolnose loc. <1 or >10 def	
4030	Safe area exceeded	
4031	Applicate outside tolerance	
4032	Ordinate outside tolerance	
4033	Appl. & ord. outside tol.	
4038	Meas. termination/RESET	

© Siemens AG 1990 All Rights Reserved

Alarm No.	Alarm Text	Cancel Alarm by ...
4039	Program RESET necessary	
4040	Oversize	
4041	Undersize	
4042	Perm. dim. diff'ce. exceeded	
4048	No active tool available	
4049	Display selection not poss.	
4100	No D number active	
4121	Spindle not in tol. range	
4180	Option not available	
4200	Check definition R (Nxxxx)	
4219	Call cycle again	
4220	Parallel lines	

Explanation of alarm 4200

Example of an alarm line



It has been detected in the cycle running in channel 2 that parameter R32 has been defined incorrectly.

© Siemens AG 1990 All Rights Reserved

9.3 Alarm description

4000	Cancel alarm
Explanation:	Alarm line must be cleared to enable further alarms to be displayed by the measuring cycles.
Remedy:	Press acknowledgement key

4001	Check MDC 0/1
Explanation:	Only on installation

4002	Check MDC 2/3
Explanation:	Only on installation

4003	Check MDC 1000 to 1019
Explanation:	Only on installation

4004	Load MDC inhibited
Explanation:	Only on installation with L898/L969 Set MD 5012 bit 2 to ”0”

4005	MDC area exceeded
Explanation:	Only on installation

4006	NC type or NC SW-Version wrong
Explanation:	NC type or NC software version wrong
Remedy:	Check MD 157

© Siemens AG 1990 All Rights Reserved

4007	Wrong MC programs loaded
Explanation:	Wrong auxiliary cycles (L93x, L96x)
Remedy:	Load auxiliary cycles from measuring cycle diskette (Version 20)

4008	Check MDC (Nxxxx)
Explanation:	Block number = MDC number (as alarm 4200)
Remedy:	Correct MDC

4009	Measurement not possible
Explanation:	Coordinate rotation or scale factor is selected
Remedy:	Deselect coordinate rotation or scale factor before measuring cycle

4010	Probe is defective
Explanation:	The probe carries 1 signal already before the measurement path ”2a” is started. • Setpoint position has been specified incorrectly (probe deflected) • Probe fault, transmits 1 signal continuously.

4011	Probe does not switch
Reason:	Probe ”2a” has been completely traversed five times without a measuring signal having been transmitted. • Set position has been incorrectly specified. • Probe does not transmit a signal despite being deflected.

4012	Probe collision
Reason:	A switching signal has been transmitted as a result of intermediate positioning operations. • Possibly collision • Set position has been incorrectly specified.

© Siemens AG 1990 All Rights Reserved

4013	**Check type in TO memory**
Remedy:	Change type in TOA memory or for type 5-8, 10-19, check the number of the measuring axis

4014	**STACK overflow**
Explanation:	Change type in TO memory or check measuring axis with types 5 - 8, 10

4015	**Check ZO number**
Explanation:	Change ZO number in the input display

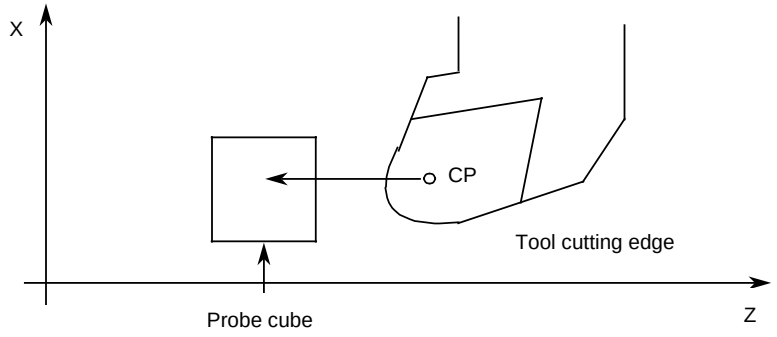
4019	**Calculation not possible (L979)**
Explanation:	The calculation results in a division by zero or a multiplication exceeding the R parameter format.
Remedy:	Change start angle (R 24) or indexing angle (R 26)

4020	**No D number received from PLC**
Explanation:	Tool not known in PLC
Remedy:	Check defining parameters R8/R9

4021	**PLC interface not ready**
Explanation:	PLC interface for tool management FB 121 DB 147 receives no feedback when a D number is requested.
Remedy:	Check PLC

© Siemens AG 1990 All Rights Reserved

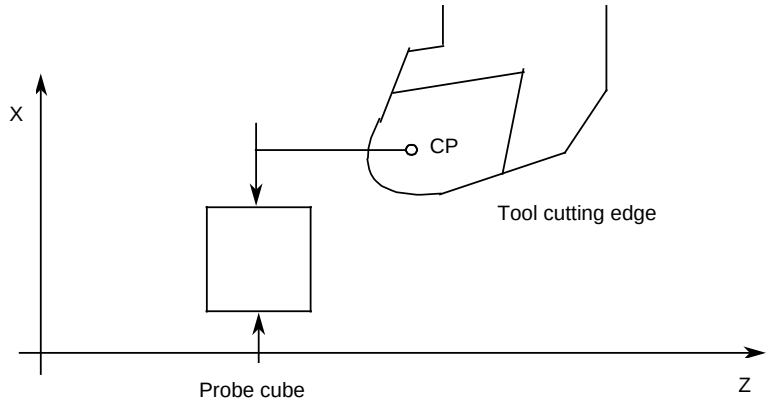
4025	**Tool tip position wrong**
Explanation:	The tool is to be measured in the X axis. The cycle wants to position the Z axis, but this would result in a collision with the measuring cube.



Tool cutting edge incorrectly positioned

Remedy:

Position tool in X axis before cycle call with greater positive value.



Tool cutting edge correctly positioned

4026	**Toolnose location 9 defined**
Explanation:	Tool point direction 9 cannot be measured with measurement variant ”Automatic measuring” (R23 > 1).
Remedy:	Use measurement variant 1.

© Siemens AG 1990 All Rights Reserved

4027	Toolnose loc. < 1 or > 10 def
Explanation:	Only the following tool point directions (tool type) are permitted: 1, 2, 3, 4, 5, 6, 7, 8, 9 and 10.

4030	Safe area exceeded
Explanation:	The set/actual difference is greater than R36. The amount of the content of the wear memory plus the compensation value is greater than the value of NC MD 208/209. The measurement result is not accepted. • Measuring error caused by probe • Measuring point contaminated No tool offset has been carried out.

4031	Applicate outside tolerance
Explanation:	

4032	Ordinate outside tolerance
Explanation:	

4033	Applic. & ord. outside tolerance
Explanation:	

4038	Meas. termination / RESET
Explanation:	Measuring error
Remedy:	Program can be continued with measurement abort signal or aborted by pressing the RESET key.

© Siemens AG 1990 All Rights Reserved

4039	Program RESET necessary
Explanation:	Measuring error
Remedy:	Program abort only possible by pressing the RESET key.

4040	Oversize
Explanation:	The set/actual difference is greater than the centre tolerance. There is too much material at the measuring point. Tool has been compensated for if R10 > 0.
Remedy:	Re-machining is possible.

4041	Undersize
Explanation:	The set/actual difference is less than the centre tolerance. There is too little material at the measuring point. Tool has been compensated for if R10 > 0.
Remedy:	Re-machining is not possible.

4042	Perm. dim. diff'ce exceeded
Explanation:	The set/actual difference is greater than R37. • Chipping of cutting edge • Too much wear in last cut, because hard spots have been encountered in the workpiece. No tool offset has been carried out.
Remedy:	Tool must be replaced.

4048	No active tool available
Explanation:	The tool to be compensated is not available in tool management. • Tool life exceeded • Twin tool not available

© Siemens AG 1990 All Rights Reserved

4049	Display selection not poss.
Explanation:	Measurement result display cannot be selected, because selection is possibly carried out by the PLC. Access to PLC data disabled by @ (PLC MD 6026.2).

4100	No D number active
Explanation:	No D number programmed in the program.
Remedy:	Enter D number in part program.

4121	Spindle not in tol. range
Explanation:	The spindle position is monitored in the cycle as soon as the function ”M19 with axis motion” is activated. The measuring cycle is not continued, until the spindle has been in the exact stop for M19 for approx. 0.2 seconds.
Remedy:	This alarm is output, if this position is not reached within 6 seconds.

4180	Option not available
Remedy:	Retrofit measurement option (Service)

4200	Check definition R(Nxxxx)
Explanation:	Defining parameter incorrectly defined. Block number N = R parameter number.
Remedy:	Correct part program.

4219	Call cycle again
Explanation:	Measurement variant selected for JOG measuring not clearly defined.
Remedy:	Select measurement variant again after RESET.

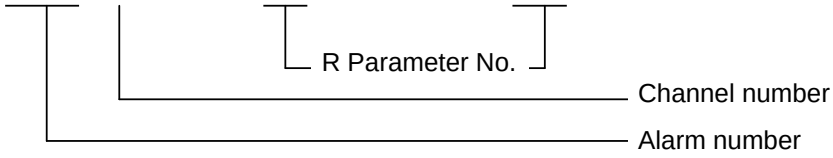
© Siemens AG 1990 All Rights Reserved

4220	**Parallel lines**
Explanation:	The cycle cannot calculate any intersection points as the lines which have been measured run parallel to each other.

Explanation of alarm 4200

Example of an alarm line

4 2 0 0 2 N 3 2 Definition R (Nxxx) to be checked



It has been detected in the cycle running in channel 2 that parameter R32 has been defined incorrectly.

END OF SECTION

© Siemens AG 1990 All Rights Reserved

10 Lists

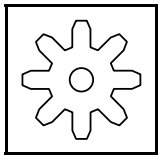
10.1 Overview of transfer parameters

Parameter definition

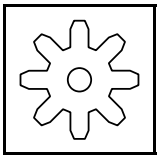
Parameter must be defined.

Parameter is not set in the cycle for the measurement variant selected.

Definition of parameter depends on the measurement variant or machine configuration.



Whether or not the parameter must be defined is determined via an MDC.
(See machine tool manufacturer's specifications)



L970	***Tool measurement / Workpiece measurement***	
	Possible axes **Abscissa / Ordinate / Applicate**	
Variant / R Param.	Tool measurement	Workpiece measurement
R23	0	1
R25	Variable measuring speed (if 0 rapid traverse/reduced rapid traverse)	
R30	Axis number of plane	
R32 ***D***	Approach position	

D = Diameter value, in the case of axis diameter programming.

© Siemens AG 1990 All Rights Reserved

L972 *Tool measurement*

	Possible axes Abscissa / Ordinate		
Variant / R Param.	Calibrate tool probe	Measure tool	Automatic tool measurement
R0			
R8			
R9			
R10			
R11		Empirical value memory number	
R12			
R13			
R18			Var. change position Ordinate
R19			Var. change position Abscissa
R20			
R21			
R22	**Tool probe No. 1..[MDC 1001]**		
R23	0	1	2 (12/22)
R24		Starting angle with tool types 26	
R25	Variable measuring speed (if 0 150/300 mm/min)		
R26			
R27	Number of measurements at same location		
R28	Multiplication factor for measurement path 2a		
R29			
R30	Axis number measuring axis		
R31			
R32			
R33	Zero offset range (lower limit)		
R34			

© Siemens AG 1990 All Rights Reserved

L972	*Tool measurement*		
	Possible axes Abscissa / Ordinate		
Variant / R Param.	Calibrate tool probe	Measure tool	Automatic tool measurement
R36	Safe Area		
R37		Dimensional difference check	
R40			
R41			
R42			

Explanation of R23:

R23= 2 1

0 Calibrate
1 Measure
2 Autom. measurement; depending on tool type Types 1 to 8

	Tool length in TO memory	
	before measurement	after measurement
0	PF	PF
1	PF	SF
2	SF	SF

Explanation of R30: **NC MD 157 = x2xx**

R30=1 Abscissa (X)
R30=2 Ordinate (Y)
R30=3 Applicate (Z)

NC MD 157 = x1xx and MDC 7002 bit 1 =0: Axes exchanged
Axis definition via R30
R30=1 Ordinate (X)
R30=2 Abscissa (Z)

NC-MD157 = x1xx and MDZ 7002 bit 1 = 1
Axes not exchanged
Axis definition via R30 after G18 plane Z X
R30=1 Abscissa (Z)
R30=2 Ordinate (X)

© Siemens AG 1990 All Rights Reserved

L973	***Workpiece measurement***
	Possible axes **Abscissa / Ordinate**
	Calibrate workpiece probe with reference data | with arbitrary data
Variant / R Param.	Hole | Cube (Surface)
R0	
R8	
R10	
R11	
R12	
R13	
R18	
R19	
R20	
R21	
R22	Probe number 1.. [MDC 1003]
R23	**Plane** xxx02 | xxx22
R24	
R25	**Variable measuring speed (if 0 150 / 300 mm/min)**
R26	
R27	Number of measurements at same location
R28	Multiplication factor for measurement path 2a
R29	
R30	Measuring axis | (Definition depending on R23)
R31	**Axial direction** | (Definition depending on R23) 0 = positive / 1 = negative
R32 ***D***	Calibration setpoint
R33 ***D***	Zero offset range (lower limit)
R34	

D = Diameter value, if measuring axis in diameter programming.

© Siemens AG 1990 All Rights Reserved

L973	*Workpiece measurement*	
	Possible axes Abscissa / Ordinate	
	Calibrate workpiece probe with arbitrary data	
Variant / R Param.	Hole	Cube (Surface)
R36 ***D***	Safe area	
R37		
R40		
R41		
R42		

D=Diameter value, if measuring axis in diameter programming

Explanation of R23:

R23= 5 4 3 2 1

2	With arbitrary data in the plane	
0	Hole	not with 1-point measurement
2	Surface	
0	No calculation of probe ball	
1	Calculation of probe ball (not with 1-point measurement)	
0/1	1 axis direction with NC type (T)	Specify measuring axis and axis direction
2	2 axis directions with NC type (T)	Specify measuring axis
0	No position determination	
1	Position determination (not with 1-point measurement)	

Explanation of R30: (see L972)

© Siemens AG 1990 All Rights Reserved

L974 *Workpiece measurement*

	Possible axes Abscissa / Ordinate						
	ZO determination	Measuring					
Variant / R Param.	1-point	1-point	1-point with reversal (M19 workpiece)	2-point at diameter	Multi-point on circumference (M19 workpiece)	Multi-point on cylinder	
R0							
R8		**Extended T address** (only if MDC 7000.0 = 1)					
R9		**Tool number (T address)** (only if MDC 7000.3 = 0)					
R10	ZO memory number	TO memory number (D number) (only if MDZ 7000.3=1)					
		Number of cutting edge (only if MDZ 7000.3=0)					
R11 **[*D*]**		**Empirical value memory number** (Average value)					
R12							
R13							
R18				Protection zone Ordinate PA (Radius)			
R19				Protection zone Abscissa LA		Offset Abscissa (increm.) LA	
R20							
R21							
R22	Probe number 1..[MDC 1003]						
R23	0	21	22	23 (24)	25	26	
R24					No. of measuring points		
R25	Variable measuring speed (if 0 150 / 300 mm/min)						

LA = Longitudinal axis
PA = Transverse axis
[*D*]= Memory content is a diameter value

© Siemens AG 1990 All Rights Reserved

L974	***Workpiece measurement***						
	Possible axes Abscissa / Ordinate						
	ZO determination	Measuring					
Variant / R Param.	1-point	1-point	1-point with reversal (M19 workpiece)	2-point at diameter	Multi-point on circumfer. (M19 workpiece)	Multi-point on cylinder	
R26			Start angle		Start angle =indexing angle		
R27	Number of measurements at same location						
R28	Multiplication factor for measurement path 2a						
R29		Weighting factor k for averaging					
R30	Measuring axis						
R31 ***D***							
R32 ***D***	Setpoint						
R33 ***D***		Zero offset range (lower limit)					
R34 ***D***		Compensation area with averaging					
R36 ***D***	Safe area						
R37 ***D***		Dimensional difference check					
R40 ***D***		Upper tolerance limit (according to drawing)					
R41 ***D***		Lower tolerance limit (according to drawing)					
R42 ***D***		Setpoint (according to drawing)					

D = Diameter value, if measuring axis in diameter programming

Explanation of R23:

R23=23	External measurement	**with** R18/R19
	Internal measurement	**without** R18/R19
R23=24	Internal measurement	**with** R18/R19

Explanation of R30: (see L972)

© Siemens AG 1990 All Rights Reserved

L976	***Workpiece measurement***				
	Possible axes **Abscissa / Ordinate / Applicate**				
	Calibrate workpiece probe with reference data			with arbitrary data	
Variant / R Param.	Hole	Ball	Cube (Surface)	Hole	Cube (Surface)
R0					
R8					
R10					
R11					
R12	Reference data 1.. [MDC 1005]				
R13	**Compensation angle for monodirectional probe (if MDC 7004/0 = 1)**				
R18					
R19					
R20					
R21					
R22	Probe type/probe no. Multidirectional probe 1.. [MDC 1003] Monodirectional probe 101..(100+ [MDC 1003])				
R23	**Plane** xxx00	xxx10	xxx20	xxx02	**Plane** xxx22
R23	**Applicate** xxx01			**Applicate** xxx23	
R24					
R25	**Variable measuring speed (if 0 150 / 300 mm/min)**				
R26					
R27	Number of measurements at same location				
R28	Multiplication factor for measurement path 2a				
R29					
R30	Measuring axis (Definition depending on R23)				
R31	Axial direction (Definition depending on R23) 0 = positive / 1 = negative				
R32 ***D***				Calibration setpoint	
R33 ***D***	Zero offset range (lower limit)				

© Siemens AG 1990 All Rights Reserved

L976	*Workpiece measurement*				
	Possible axes Abscissa / Ordinate / Applicate				
	Calibrate workpiece probe				
	with reference data			with arbitrary data	
Variant / R Param.	Hole	Ball	Cube (Surface)	Hole	Cube (Surface)
R34					
R36 ***D***	Safe area				
R37					
R40					
R41					
R42					

D=Diameter value, if measuring axis in diameter programming.

Explanation of R23

R23= 5 4 3 2 1

1: 0 With reference data in the plane
1 With reference data in the applicate
2 With arbitrary data in the plane
3 With arbitrary data in the applicate

2: 0 Hole
1 Ball
2 Surface
3 Slot
} not with 1-point measurement

3: 0 No calculation of probe ball
1 Calculation of probe ball (not with 1-point measurement)

4: 0 4 axis directions with NC type (M):
1 1 axis direction with NC type (M) Specify meas. axis and axis direction
2 2 axis directions with NC type (M) Specify measuring axis

5: 0 No position determination
1 Position determination (not with 1-point measurement)

Explanation of R30: (see L972)

© Siemens AG 1990 All Rights Reserved

L977 *Workpiece measurement*

	Possible axes Abscissa / Ordinate							
	Measuring				ZO determination			
Variant / R Param.	Hole	Shaft	Slot	Web	Hole	Shaft	Slot	Web
R0								
R8	**Extended T address** (only if MDC 7000.0 = 1)							
R9	**Tool No. (T address)** (only if MDC 7000.3 = 0)							
R10	TO memory No. (D number) (if MDC 7000.3=1) Number of cutting edge (if MDC 7000.3=0)				ZO memory number			
R11	Empirical value memory number (Average value)							
R12								
R13	Compensation angle for monodirectional slot (if MDC 7004.0=1)							
R18								
R19		Infeed Applicate		Infeed Applicate		Infeed Applicate		Infeed Applicate
R20								
R21								
R22	**Probe type/probe No.** Multidirectional probe 1..[MDC 1003] Monodirectional probe 101..(100+[MDC 1003])							
R23	1	2	11	12	21	22	31	32
R24								
R25	**Variable measuring speed (if 0 150 / 300 mm/min)**							

© Siemens AG 1990 All Rights Reserved

L977	*Workpiece measurement*							
	Possible axes Abscissa / Ordinate							
	Measuring				**ZO determination**			
Variant / R Param.	Hole	Shaft	Slot	Web	Hole	Shaft	Slot	Web
R26								
R27	Number of measurements at same location							
R28	Multiplication factor for measurement path 2a							
R29	Weighting factor k for averaging							
R30			Measuring axis				Measuring axis	
R31								
R32					Setpoint			
R33	Zero offset range (lower limit)							
R34	Compensation area with averaging							
R36	Safe area							
R37	Dimensional difference check							
R40	Upper tolerance limit (acc. to drawing)							
R41	Lower tolerance limit (acc. to drawing)							
R42	Setpoint (acc. to drawing)							

Explanation of R30: (see L972)

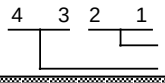
© Siemens AG 1990 All Rights Reserved

L979 *Workpiece measurement*

	Possible axes Abscissa / Ordinate / Applicate			
	ZO determination	Measuring		ZO determination
Variant / R Param.	1-point	1-point	Multi-point	Angle
R0				
R8		**Extended T address** (only if MDC 7000.0 = 1)		
R9		**Tool No. (T address)** (only if MDC 7000.3 = 0)		
R10	ZO memory no.	TO memory No. D number (s.L974/R10) Number of cutting edge (s.L974/R10)		ZO memory no.
R11	Empirical value memory number (Average value)			
R12				
R13	Compensation angle for monodir. probe (if MDC 7004.0=1) (s. L977/R13)			
R18				
R19			Infeed offset axis incremental positive only	
R20				
R21				
R22	**Probe type/probe No.** Multidirectional probe 1..[MDC 1003] Monodirectional probe 101..(100+[MDC 1003])			
R23	0 10 *	1 11 *	2 12 *	3 13 *
R24			Number of measuring points	Setpoint angle
R25	**Variable measuring speed (if 0 150 / 300 mm/min)**			

* Differential measuring

© Siemens AG 1990 All Rights Reserved

L978	*Workpiece measurement*			
	Possible axes **Abscissa / Ordinate / Applicate**			
	ZO determination	Measuring		ZO determination
Variant / R Param.	1-point	1-point	Multi-point	Angle
R26	**Start angle** (only for differential measurement)			
R27	Number of measurements at same location			
R28	Multiplication factor for measurement path 2a			
R29		Weighting factor k for averaging		
R30	Measuring axis			
R31				Axis number Rotary axis
R32 ***D***	Setpoint			Setpoint Approach position
R33 ***D***		Zero offset range (lower limit)		
R34 ***D***		Compensation area with averaging		
R36 ***D***	Safe area			
R37 ***D***		Dimensional difference check		
R40 ***D***		Upper tolerance limit (according to drawing)		
R41 ***D***		Lower tolerance limit (according to drawing)		
R42 ***D***		Setpoint (according to drawing)		

[*D*]= Memory content is a diameter value
D = Diameter value
} if measuring axis in diameter programming (not for angular measurement)

Explanation of R30: (see L972)

© Siemens AG 1990 All Rights Reserved

L979 *Workpiece measurement*

	Possible axes Abscissa / Ordinate							
	Measuring				ZO determination			
Variant / R Param.	Hole	Shaft	Slot	Web	Hole	Shaft	Slot	Web
R0	internal							
R8	**Extended T address** (only if MDC 7000.0 = 1)							
R9	**Tool No (T address)** (only if MDC 7000.3 = 0)							
R10	TO memory No. (D number) (if MDC 7000/3=1) Number of cutting edge (if MDC 7000/3=0)				ZO memory number			
R11	Empirical value number (average value)							
R12								
R13	Compensation angle for monodirectional probe (if MDC 7004.0=1)							
R18								
R19				Infeed Applicate				Infeed Applicate
R20 ***D***	**Centre point abscissa**							
R21 ***D***	**Centre point ordinate**							
R22	**Number of measuring points/probe type/probe No.** R22 = 4 3 2 1 Probe No. 0=Multidirectional probe /1= Monodirectional probe 0=3-point /1= 4-point							
R23	1	2	11	12	21	22	31	32
R24	Starting angle (from the abscissa (horizontal axis))							
R25	**Variable measuring speed (if 0 150 / 300 mm/min)**							

© Siemens AG 1990 All Rights Reserved

L979 *Workpiece measurement*

	Possible axes Abscissa / Ordinate							
	Measuring				**ZO determination**			
Variant / R Param.	Hole	Shaft	Slot	Web	Hole	Shaft	Slot	Web
R26	Indexing angle				Indexing angle			
R27	Number of measurements at same location							
R28	Multiplication factor for measurements path 2a							
R29	Weighting factor k for averaging							
R30								
R31	Velocity for circular interpolation				Velocity for circular interpolation			
R32					Set value			
R33	Zero offset range (lower limit)							
R34	Compensation area with averaging							
R36	Safe area							
R37	Dimensional difference check							
R40	Upper tolerance limit (according to drawing)							
R41	Lower tolerance limit (according to drawing)							
R42	Setpoint (according to drawing)							

D=Diameter value, if axis in diameter programming

© Siemens AG 1990 All Rights Reserved

L981	***Workpiece measurement***
	Possible axes Abscissa / Ordinate
Variant / R Param.	Searching for a hole / slot
	on face | on cylinder
R1	Set diameter
R2 ***D***	Reference plane
R10	ZO memory number
R11	
R18 ***D***	Centre point linear axis
R19	
R22	Probe No. 1.. [MDC 1003]
R23	1 | 2
R25	**Variable measuring speed (if 0 150 / 300 mm/min)**
R26	Search increment in degrees (positive only)
R27	Number of measurements at same location
R29	
R30	
R31	
R32 ***D***	Approach setpoint
R33	
R34	
R36	Safe area in degrees

D = Diameter value, if axis in diameter programming

© Siemens AG 1990 All Rights Reserved

L982 *Tool measurement*

Possible axes
Abscissa / Ordinate / Applicate

Variant / R-Param.	Calibrate tool probe	Measure tool	Automatic tool measurement
R0			
R8			
R9			
R10			
R11		Empirical value memory number	
R12			
R13			Angle offset with R23 102
R18			Variable change position Ordinate
R19			Variable change position Abscissa
R20			
R21			
R22	**Probe No. 1.. [MDC 1001]**		
R23	0	1	2 (12/22)
R24			
R25	Variable measuring speed (if 0 150/300 mm/min)		
R26		Starting angle with tool types 26	
R27	Number of measurements at same location		
R28	Multiplication factor for measurement path 2a		
R29			
R30	Axis number measuring axis		
R31			
R32			
R33	Zero offset range (lower limit)		
R34			

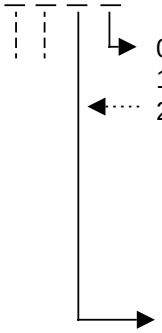
© Siemens AG 1990 All Rights Reserved

L972	*Tool measurement*		
	Possible axes Abscissa / Ordinate		
Variant R-Param.	Calibrate tool probe	Measure tool	Automatic tool measurement
R36	Safe area		
R37		Dimensional difference check	
R40			
R41			
R42			

© Siemens AG 1990 All Rights Reserved

Explanation of R23:

R23= 4 3 2 1



0 Calibrate
1 Measure
2 Automatic measurement, depending on tool type, type 1-8

	Tool length in TO memory	
	before measurement	after measurement
0	PF	PF
1	PF	SF
2	SF	SF

R23= 4 3 2 1

0 Measuring (also automatically) of L1 and/or L2 with tool types 1-10, 26, 28, 31-38 with one tool offset number

1 Measuring (automatically only) of L1 and/or L2 and R with tool types 28, 37, 38 with one tool offset number

2 Measuring (automatically only) of L1 and/or L2 and R with tool types 28, 37, 38 with two tool offset numbers

0 Compensate length and radius
1 Compensate length only
2 Compensate radius only

} If 3rd decimal place of R23=1 or 22F

Explanation of R30:

NC-MD 157 = x2xx
R30=1 Abscissa (X)
R30=2 Ordinate (Y)
R30=3 Applicate (Z)

NC-MD 157 = x1xx and MDZ 7002 bit 1 =0:
Axes exchanged
Axis definition via R30
R30=1 Ordinate (X)
R30=2 Abscissa (Z)

NC-MD 157 = x1xx and MDZ 7002 bit 1 =1:
Axes not exchanged
Axis definition via R30 after G18 plane Z X
R30=1 Abscissa (Z)
R30=2 Ordinate (X)

© Siemens AG 1990 All Rights Reserved

10.1.1 Compatibility list, Version 10 to Version 20

L971/L972

		Version 10	Version 20 onwards
R20	Omitted	Control parameter	———
R22	New	———	Tool probe No.
R23		Tool length definition	Control parameter
R25	New	———	Variable measuring speed

Function of L971 now by calling L972. L971 is only an auxiliary program.

L973 only for calibrating the workpiece probe.
No blank measurement!

		Version 10	Version 20 onwards
R11	Omitted	EV memory number	———
R21	Omitted	Control parameter	———
R22		Probe type	Probe no.
R23		Control parameter	Control parameter
R25	New	———	Var. measuring speed
R34	Omitted	Compensation area with averaging	———

L974 now also for blank measurement!

		Version 10	Version 20 onwards
R22	New	———	Probe no.
R23		Number of measurements	Control parameter
R24		Control parameter	Number of measurements
R25		Control parameter	Var. measuring speed
R26		Offset	Now R19
R35	Omitted	Tolerance	Now R40/R41

© Siemens AG 1990 All Rights Reserved

10.2 Result displays

The function of the result displays depends on the machine configuration.

(see machine tool manufacturer's specifications)

Result displays are available for displaying the measurement results. These displays are automatically shown on the screen on completion of the cycle.

The numbers in the fields of the displays shown indicate the R parameter in which the relevant data must be entered.

AUTOMATIC - CH1

Tool measurement: Measurement result Cycle 215

Input data		Output data			
Measurement variant	23	Measure (calibrate)		Actual value	Difference
D memory No.	212	L1 1.5	(Abscissa - R.)	200	201
Tool type	216	L2 1.5	(Abscissa +R.)	202	203
		D/R 1.5	(Ordinate - R.)	204	205
EV memory No.	213	L1 2.5	(Ordinate +R.)	206	207
Empirical value	214	L2 2.5	(Applicate - R.)	208	209
		D/R 2.5	(Applicate +R.)	210	211
Safe area	36				
Perm. dim. diff'ce	37				
Probe No.	218				
Data	217	Alarm No.		219	

Measurement result display for tool measurement

© Siemens AG 1990 All Rights Reserved

AUTOMATIC - CH1

Workpiece measurement: Measurement result Cycle 215

Input data		Output data		
			Actual value	Difference
Measurement variant	23			
Safe area	36	Probe ball	200	201
Lower limit	33			
		Abscissa - direction	202	203
Probe No.	218	Abscissa +direction	204	205
Data	217	Ordinate - direction	206	207
		Ordinate +direction	208	209
		Applicate - direction	210	211
		Applicate +direction	212	213
		Position abscissa	214	
		Position ordinate	216	
		Alarm No.	219	

Measurement result display for workpiece probe calibration

AUTOMATIC - CH1

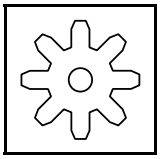
Workpiece measurement: Measurement result Cycle 215

Input data		Output data			
		Diameter /width		1st axis	2nd axis
Measurement variant	23				
Safe area	36	Set value	206	207	208
Perm. dim. diff'ce	37	Actual value	209	210	211
		Tolerance upper limit	200	201	202
D/ZO memory No.	216	Tolerance lower limit	203	204	205
		Difference	212	213	214
Probe No.	218				
Data	217	Compensation value	50		
		Alarm No.	219		

Measurement result display for workpiece measurement

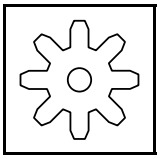
© Siemens AG 1990 All Rights Reserved

10.3 Result parameters



Display of measurement results in R parameters R40-R49 can be determined by MDC.

(See machine tool manufacturer's specifications)



Result parameters:

L972 ***Tool measurement***

P No.	Calibration		Measurement	
R200	Trigger point minus direction	Actual value Abscissa	L1 (1st cutting edge)	Actual value
R201	Trigger point minus direction	Difference Abscissa	L1 (1st cutting edge)	Difference
R202	Trigger point plus direction	Actual value Abscissa	L1 (1st cutting edge)	Actual value
R203	Trigger point plus direction	Actual value Abscissa	L1 (1st cutting edge)	Difference
R204	Trigger point minus direction	Actual value Ordinate	D/R (1st cutting edge)	Actual value
R205	Trigger point minus direction	Difference Ordinate	D/R (1st cutting edge)	Difference
R206	Trigger point plus direction	Actual value Ordinate	L1 (2nd cutting edge)	Acutal value
R207	Trigger point plus direction	Difference Ordinate	L1 (2nd cutting edge)	Difference
R208	Trigger point minus direction	Actual value Applicate	L2 (2nd cutting edge)	Actual value
R209	Trigger point minus direction	Actual value Applicate	L2 (2nd cutting edge)	Difference
R210	Trigger point plus direction	Actual value Applicate	D/R (2nd cutting edge)	Actual value
R211	Trigger point plus direction	Difference Applicate	D/R (2nd cutting edge)	Difference
R212			D memory No.	
R213			EV memory No.	
R214			Empirical value	
R215	Cycle No.			
R216			Tool type	
R217	MDC area probe data			
R218	Probe number			
R219	Alarm number			

© Siemens AG 1990 All Rights Reserved

Result parameters:

L973	*Workpiece measurement*			
P No.	**Calibration**			
R200	Probe ball diameter			Actual value
R201	Probe ball diameter			Difference
R202	Trigger point	Actual value	minus direction	Abscissa
R203	Trigger point	Difference	minus direction	Abscissa
R204	Trigger point	Actual value	plus direction	Abscissa
R205	Trigger point	Difference	plus direction	Abscissa
R206	Trigger point	Actual value	minus direction	Ordinate
R207	Trigger point	Difference	minus direction	Ordinate
R208	Trigger point	Actual value	plus direction	Ordinate
R209	Trigger point	Difference	plus direction	Ordinate
R210	Trigger point	Actual value	minus direction	Applicate
R211	Trigger point	Difference	minus direction	Applicate
R212	Trigger point	Actual value	plus direction	Applicate
R213	Trigger point	Difference	plus direction	Applicate
R214	Positional deviation abscissa			
R215	Cycle number			
R216	Positional deviation ordinate			
R217	MDC area probe data			
R218	Probe number			
R219	Alarm number			

© Siemens AG 1990 All Rights Reserved

Result parameters:

L974	*Workpiece measurement*	
P No.	**Measuring / ZO determination**	
R200 R40	Upper tolerance limit	Measuring axis
R201		
R202		
R203 R41	Lower tolerance limit	Measuring axis
R204		
R205		
R206 R42	Setpoint	Measuring axis
R207	Setpoint	Abscissa / Applicate
R208	Setpoint	Ordinate
R209 R43	Actual value	Measuring axis
R210		
R211		
R212 R44	Difference	Measuring axis
R213		
R214		
R215	Cycle number	
R216 R46	D memory number / ZO memory number	
R217	MDC area Probe data	
R218	Probe number	
R219	Alarm number	
R50	Compensation value	

© Siemens AG 1990 All Rights Reserved

Result parameters:

L976	*Workpiece measurement*			
P No.	**Calibration**			
R200	Probe ball diameter			Actual value
R201	Probe ball diameter			Difference
R202	Trigger point	Actual value	minus direction	Abscissa
R203	Trigger point	Difference	minus direction	Abscissa
R204	Trigger point	Actual value	plus direction	Abscissa
R205	Trigger point	Difference	plus direction	Abscissa
R206	Trigger point	Actual value	minus direction	Ordinate
R207	Trigger point	Difference	minus direction	Ordinate
R208	Trigger point	Actual value	plus direction	Ordinate
R209	Trigger point	Difference	plus direction	Ordinate
R210	Trigger point	Actual value	minus direction	Applicate
R211	Trigger point	Difference	minus direction	Applicate
R212	Trigger point	Actual value	plus direction	Applicate
R213	Trigger point	Difference	plus direction	Applicate
R214	Positional deviation abscissa			
R215	Cycle number			
R216	Positional deviation ordinate			
R217	MDC area probe data			
R218	Probe number			
R219	Alarm number			

© Siemens AG 1990 All Rights Reserved

Result parameters:

L977 *Workpiece measurement*

	Measurement		ZO determination	
	Hole Shaft	Slot/ Web	Hole Shaft	Slot/ Web
R40	Upper tolerance limit	Upper tolerance limit		
R41	Lower tolerance limit	Lower tolerance limit		
R42	Setpoint diameter	Setpoint width	Setpoint CP Abscissa	Set value slot/ web centre
R43	Actual value diameter	Actual value width	Setpoint CP Ordinate	
R44	Difference diameter	Difference width	Actual value CP abscissa	Actual val. slot/ web midth
R45	Actual value Abscissa	Slot/ web centre Meas. axis	Actual val. CP Ordinate	
R46	Actual value Ordinate		Difference CP Abscissa	
R47	D memory number	D memory number	Difference CP Ordinate	
R48			ZO memory number	ZO memory number
R49				
R50	Compensation value			

P No.	Measurem. /ZO determin.
R200	Upper tolerance limit Hole/shaft/slot/web
R201	Upper tolerance limit Centre point Abscissa
R202	Upper tolerance limit Centre point Ordinate
R203	Lower tolerance limit Hole/shaft/slot/web
R204	Lower tolerance limit Centre point Abscissa
R205	Lower tolerance limit Centre point Ordinate
R206	Setpoint Hole/shaft/slot/web
R207	Set value Centre point Abscissa
R208	Setpoint Centre point Ordinate
R209	Actual value Diameter Hole/shaft/slot/web
R210	Actual value Centre point Abscissa
R211	Actual value Centre point Ordinate
R212	Difference Diameter Hole/shaft/slot/web
R213	Difference Centre point Abscissa
R214	Difference Centre point Ordinate
R215	Cycle number
R216	D memory number / ZO memory number
R217	MDC area Probe data
R218	Probe number
R219	Alarm number
R50	Compensation value

© Siemens AG 1990 All Rights Reserved

Result parameters:

L978	***Workpiece measurement***	
P No.	**Measurement/ZO determination**	
R200 R40	Upper tolerance limit	Measuring axis
R201		
R202		
R203 R41	Lower tolerance limit	Measuring axis
R204		
R205		
R206 R42	Setpoint	Measuring axis
R207	Setpoint	Abscissa / Applicate
R208	Setpoint	Ordinate
R209 R43	Actual value	Measuring axis
R210		
R211		
R212 R44	Difference	Measuring axis
R213		
R214		
R215	Cycle number	
R216 R47	D memory number / ZO memory number	
R217	MDC area Probe data	
R218	Probe number	
R219	Alarm number	
R50	Compensation value	

© Siemens AG 1990 All Rights Reserved

Result parameters:

L979 *Workpiece measurement*

	Measurement		ZO determination	
	Hole Shaft	Slot/ Web	Hole Shaft	Slot/ Web
R40	Upper tolerance limit	Upper tolerance limit		
R41	Lower tolerance limit	Lower tolerance limit		
R42	Setpoint Diameter	Setpoint Width	Setpoint CP Abscissa	Setpoint Slot/ Web centre
R43	Actual value Diameter	Actual value Width	Setpoint CP Ordinate	
R44	Difference Diameter	Difference Width	Actual val. CP Abscissa	Actual val. Slot/ Web centre
R45	Actual value Abscissa	Slot/ Web centre Meas. axis	Actual val. CP Ordinate	Actual val. CP Ordinate
R46	Actual val. Ordinate		Difference CP Abscissa	Difference CP Abscissa
R47	D memory number	D memory number	Difference CP Ordinate	Difference CP Ordinate
R48			ZO memory number	ZO memory number
R49				
R50	Compensation value			

P No.	Measurem./ZO determin.
R200	Upper tolerance limit Hole/shaft/slot/web
R201	Upper tolerance limit Centre point Abscissa
R202	Upper tolerance limit Centre point Ordinate
R203	Lower tolerance limit Hole/shaft/slot/web
R204	Lower tolerance limit Centre point Abscissa
R205	Lower tolerance limit Centre point Ordinate
R206	Setpoint Hole/shaft/slot/web
R207	Setpoint Centre point Abscissa
R208	Setpoint Centre point Ordinate
R209	Actual value Diameter Hole/shaft/slot/web
R210	Actual value Centre point Abscissa
R211	Actual value Centre point Ordinate
R212	Difference Diameter Hole/shaft/slot/web
R213	Difference Centre point Abscissa
R214	Difference Centre point Ordinate
R215	Cycle number
R216	D memory number / ZO memory number
R217	MDC area Probe data
R218	Probe number
R219	Alarm number
R50	Compensation value

© Siemens AG 1990 All Rights Reserved

Result parameters:

L981	*Workpiece measurement*
P No.	**ZO determination**
R200	
R201	
R202	
R203	
R204	
R205	
R206 R40	Diameter Hole Slot
R207 R43	Setpoint Position Hole/Slot
R208	
R209	
R210 R45	Actual value Position Hole/Slot
R211	
R212	
R213 R47	Actual value Position Hole/Slot
R214	
R215	Cycle number
R216 R46	ZO memory number
R217	MDC area Probe data
R218	Probe number
R219	Alarm number
R50	Compensation value

© Siemens AG 1990 All Rights Reserved

Result parameters:

L982	*Tool measurement*			
P No.	**Calibration**		**Measurement**	
R200	Trigger point minus direction	Actual value Abscissa	L1 (1st cutting edge)	Actual value
R201	Trigger point minus direction	Difference Abscissa	L1 (1st cutting edge)	Difference
R202	Trigger point plus direction	Actual value Abscissa	L1 (1st cutting edge)	Actual value
R203	Trigger point plus direction	Actual value Abscissa	L1 (1st cutting edge)	Difference
R204	Trigger point minus direction	Actual value Ordinate	D/R (1st cutting edge)	Actual value
R205	Trigger point minus direction	Difference Ordinate	D/R (1st cutting edge)	Difference
R206	Trigger point plus direction	Actual value Ordinate	L1 (2nd cutting edge)	Actual value
R207	Trigger point plus direction	Difference Ordinate	L1 (2nd cutting edge)	Difference
R208	Trigger point minus direction	Actual value Applicate	L2 (2nd cutting edge)	Actual value
R209	Trigger point minus direction	Actual value Applicate	L2 (2nd cutting edge)	Difference
R210	Trigger point plus direction	Actual value Applicate	D/R (2nd cutting edge)	Actual value
R211	Trigger point plus direction	Difference Applicate	D/R (2nd cutting edge)	Difference
R212			D memory No.	
R213			EV memory No.	
R214			Empirical value	
R215	Cycle number			
R216			Tool type	
R217	MDC area probe data			
R218	Probe number			
R219	Alarm number			

© Siemens AG 1990 All Rights Reserved

10.4 Setting data for measuring cycles

10.4.1 Cycle setting data memory (SDC)

Empirical values and average values of the measuring cycles are stored in the cycle setting data memory (SDC).

The SDC memory is devided in the following areas:

10.4.1.1 Channel-oriented values

SDC No.	Designation	Standard value	Maximum input value	Reference system	Input unit
0 . . . 99	Siemens (Measuring cycles)	–	–	–	–
400 . . . 499	User	–	–	–	–

10.4.1.2 Channel-oriented bits

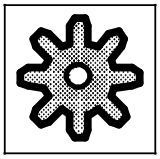
SDC No.	**Bit No.**							
	7	**6**	**5**	**4**	**3**	**2**	**1**	**0**
800 . . . 819	Siemens (Measuring cycles)							
820 . . . 849	Siemens							
900 . . . 949	User							

© Siemens AG 1990 All Rights Reserved

10.4.2 SDC Channel-oriented values

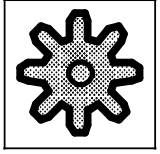
SDC No.	Designation	Standard value	Maximum input value	Reference system	Input unit
0	Internal data	–	–	–	–

SDC No.	Designation	Standard value	Maximum input value	Reference system	Input unit
1 . . . 99	Empirical values/ average values	0	0 to +/-99999999	–	–



The number of EV/AV available is determined via MDC.

(See machine tool manufacturer's specifications)



10.4.3 Channel-oriented bits

SDC No.	Bit No. 7	6	5	4	3	2	1	0
800						Display in workpiece coordinates when measuring in JOG mode 840/840C	Check Cycle MD	Check transfer parameters

Bit 3 Bit 3=1: Display in workpiece coordinates.
Bit 1 Bit 1=1: Cycle MD are checked.
Bit 0 Bit 0=1: Transfer parameters are checked.

SDC No.	Bit No. 7	6	5	4	3	2	1	0
801 . . . 809	Reserved							

END OF SECTION

© Siemens AG 1990 All Rights Reserved

11 Abbreviations

AV	Average value	
AVmem	Average value memory	
HSS	Hole/Shaft/Slot	
EV	Empirical value	
EVmem	Empirical value memory	
M version	Milling machine control	
MDC	Machine data for cycles	(Cycle machine data)
NC MD	NC machine data	
OGM	Operator guidance macro	
SDC	Setting data for cycles	(Cycle setting data)
SW	Software version	
T version	Turning machine control	
UMS	User memory submodule	

END OF SECTION

© Siemens AG 1990 All Rights Reserved