Programming manual
H-5755-8600-06-A

RENISHAW
apply innovation™

Inspection Plus software for machining centres with Fanuc and Meldas controllers

with SupaTouch optimisation

Disclaimer

WHILE CONSIDERABLE EFFORT WAS MADE TO VERIFY THE ACCURACY OF THIS DOCUMENT AT PUBLICATION, ALL WARRANTIES, CONDITIONS, REPRESENTATIONS AND LIABILITY, HOWSOEVER ARISING, ARE EXCLUDED TO THE EXTENT PERMITTED BY LAW.

RENISHAW RESERVES THE RIGHT TO MAKE CHANGES TO THIS DOCUMENT AND TO THE EQUIPMENT, AND/OR SOFTWARE AND THE SPECIFICATION DESCRIBED HEREIN WITHOUT OBLIGATION TO PROVIDE NOTICE OF SUCH CHANGES.

Trade marks

RENISHAW® and the probe symbol are registered trade marks of Renishaw plc. Renishaw product names, designations and the mark ‘apply innovation’ are trade marks of Renishaw plc or its subsidiaries.

Apple and the Apple logo are trademarks of Apple Inc., registered in the U.S. and other countries. App Store is a service mark of Apple Inc., registered in the U.S. and other countries.

Google Play and the Google Play logo are trademarks of Google LLC.

Other brand, product or company names are trade marks of their respective owners.

Renishaw plc. Registered in England and Wales.

Company no: 1106260.

Registered office: New Mills, Wotton-under-Edge, Gloucestershire, GL12 8JR, UK.

Renishaw part no: H-5755-8600-06-A

Issued: 03.2022

EQUIPMENT REGISTRATION RECORD

Please complete this form (and Form 2 overleaf, if applicable) after the Renishaw equipment has been installed on your machine. Keep one copy yourself and return a copy to your local Renishaw office (for contact details, see **www.renishaw.com/contact**). The Renishaw installation engineer should normally complete these forms.

MACHINE DETAILS

Machine description ....

Machine type ....

Controller ....

Special control options ....

....

....

RENISHAW HARDWARE	**RENISHAW SOFTWARE**
Inspection probe type	Inspection software media ....
Interface type	....
	....
Tool setting probe type	Tool setting software media ....
Interface type	....
	....

SPECIAL SWITCHING M-CODES (OR OTHER) WHERE APPLICABLE

	Dual systems only
Switch (Spin) probe on	Switch inspection probe on..........
Switch (Spin) probe off	Switch tool setting probe on ....
Start/Error signal	Other ....
	....

ADDITIONAL INFORMATION

☐ Tick box if Form 2 overleaf has been filled in.

Customer's name	
Customer's address	Date installed ....
....	
....	Installation engineer ....
....	
Customer's telephone no.	Date of training ....
Customer's contact name	

Form 2

SOFTWARE DEVIATION RECORD

Standard Renishaw kit no.	Software media nos.
Reason for deviation	
Software no. and cycle no.	Comments and corrections

The software product for which these changes are authorised is subject to copyright.

A copy of this deviation sheet will be retained by Renishaw plc.

A copy of the software amendments must be retained by the customer – they cannot be retained by Renishaw plc.

Caution – software safety

The software you have purchased is used to control the movements of a machine tool. It has been designed to cause the machine to operate in a specified manner under operator control, and has been configured for a particular combination of machine tool hardware and controller.

Renishaw has no control over the exact program configuration of the controller with which the software is to be used, nor of the mechanical layout of the machine. Therefore, it is the responsibility of the person putting the software into operation to:

- ensure that all machine safety guards are in position and are correctly working before commencement of operation;
- ensure that any manual overrides are disabled before commencement of operation;
- verify that the program steps invoked by this software are compatible with the controller for which they are intended;
- ensure that any moves which the machine will be instructed to make under program control would not cause the machine to inflict damage upon itself or upon any person in the vicinity;
- be thoroughly familiar with the machine tool and its controller, understand the operation of work co-ordinate systems, tool offsets, program communication (uploading and downloading) and the location of all emergency stop switches.

IMPORTANT: This software makes use of controller variables in its operation. During its execution, adjustment of these variables, including those listed within this manual, or of tool offsets and work offsets, may lead to malfunction. Ensure that all variable and program numbers required and/or used by the Renishaw system are not used by any other function or software package already installed on the CNC machine tool.

Caution – programming manual compatibility

This software includes a feature called SupaTouch optimisation which reduces measurement cycle time by optimising measurement and positioning feedrates. Where optimisation is not required (cycle O9800 is not used), the software will use standard Inspection Plus feedrates and two-touch measurement method.

Any text that is specific to SupaTouch optimisation is marked with the superscript ST.

Example code format

For clarity, code examples contained within this document are shown with spaces separating each input of the program call. In practice, it is not a requirement that these spaces be included.

For example, the following code:

G65 P9814 D50.005 Z100. E21. F0.8 H0.2 M0.2 Q10. R10. S1. T20. U0.5 V0.5 W2.

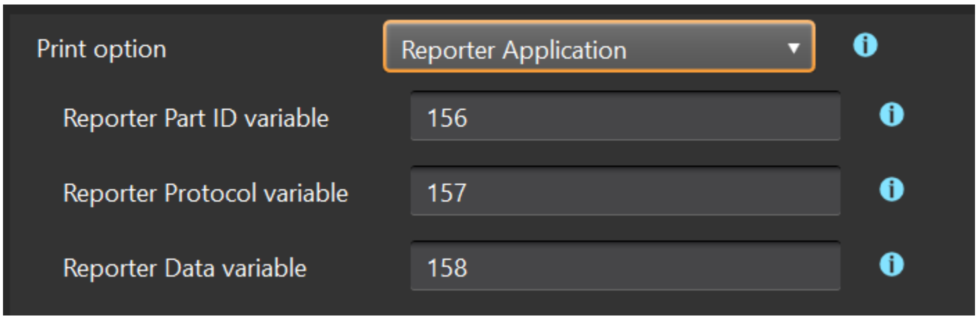
may be entered as:

G65P9814D50.005Z100.E21.F0.8H0.2M0.2Q10.R10.S1.T20.U0.5V0.5W2.

NOTE: All code examples are shown with input data followed by a decimal point. Some controllers may operate correctly with these decimal points omitted, however, care should be taken to determine that this is the case before running any programs.

Reporter

There is a Reporter option in the installation wizard which can be used to display trends of component measurement. (Reporter app v3.3 or later is required.)



This option requires the Reporter app (A-5999-4200) to be installed and connected to the machine tool to receive measured data. If the option is selected and the Reporter app is not connected, the measuring program will continue to run (See “Reporter Print” in Chapter 11, “General information”, for further information).

Machine tool apps

This software kit is supported by smartphone and on-machine apps.

Smartphone apps provide information at a user's fingertips in a simple, convenient format. Available globally in a wide range of languages, our free-of-charge apps are perfect for new and less experienced users.

On-machine apps can be seamlessly integrated with a wide range of CNC controls. Apps are installed onto a Microsoft® Windows®-based CNC control or a Windows tablet connected to the control via Ethernet.

With touch interaction and intuitive design, smartphone and on-machine apps provide significant benefits to machine tool probe users.

For more information,visit **www.renishaw.com/machinetoolapps**.

Contents

Chapter 10 Configuration

Chapter 11 General information

Appendix A Features, cycles and limitations of the Inspection Plus software

Appendix B Alternative calibration cycles

Appendix C Inspection Plus and 5-axis commands

This page is intentionally left blank.

Intended use

Renishaw Fanuc/Meldas Inspection Plus software must only be used as intended.

The software is only intended for use with Renishaw touch-trigger probes. Use of the software with non-Renishaw probes is not supported. This version of the software is for use on Fanuc/Meldas and compatible controllers only.

About the software

Renishaw Inspection Plus software described in this manual is for use on machining centres that are fitted with Fanuc/Meldas and compatible controllers.

For a comprehensive description of the features provided by the software, as well as the limitations of the software, see Appendix A, “Features, cycles and limitations of the Inspection Plus software”.

About this manual

This programming manual contains detailed information about how to use the Inspection Plus software for programming, operating and controlling your machine tool.

Comprising 11 self-contained chapters and three appendices, the manual is structured to provide the information that you require to use the Inspection Plus software effectively.

- Chapter 1, “Installing the software”, describes how to install the Inspection Plus software on your machine.
- Chapter 2, “Optional inputs”, describes the optional inputs that are available with some of the cycles.
- Chapter 3, “Cycle outputs”, provides a complete list of the outputs that are produced by some of the cycles.
- Chapter 4, “Probe start/stop and protected positioning”, describes how to use probe start (O9832), how to use probe stop (O9833) and how to use the protected positioning cycle (O9810). When used correctly, protected positioning prevents damage to the probe stylus if the probe collides with the workpiece.
- Chapter 5, “Probe calibration and SupaTouch optimisation”, explains why a probe stylus must be calibrated and how to optimise the measuring feedrates, reducing probing cycle time.
- Chapter 6, “Standard measuring cycles”, describes how to use the standard measuring cycles.
- Chapter 7, “Vector measuring cycles”, describes how to use the vector measuring cycles.

- Chapter 8, “Additional cycles”, describes how to use the cycles that are not described in previous chapters.
- Chapter 9, “Alarms and messages”, describes the cycle alarm numbers and messages that may be displayed on the screen of the machine tool controller when an error occurs. An explanation of the meaning and possible cause of each alarm message is provided, together with typical actions you must take to correct the fault causing the message.
- Chapter 10, “Configuration”, describes setting information and details about the variables used in the Inspection Plus software.
- Chapter 11, “General information”, contains general information and reference material that is relevant to the Inspection Plus software package.

Measurement values used in this manual

Throughout this manual, metric units of measurement (for example, millimetres) are used in the examples. Where appropriate, the equivalent imperial values (for example, inches) are shown in brackets.

Associated publication

When you are using the Inspection Plus software, you may find it useful to refer to the following Renishaw publication if it has been provided with the software package.

- Installation manual *Probe systems for machine tools* (Renishaw part no. H-2000-6040).

Software kit part no. A-4012-0516

The kit comprises the following item:

- Software media assembly: part no. A-4012-0518.

The software media contains the following files and folders:

\Readme.txt	This is an information file.
\Macros\<files>	This folder contains various source files, including alternative language files for messages.
\Archive\<files>	This folder contains older versions of Inspection Plus software that do not include SupaTouch optimisation or the GoProbe measurement cycles.
\Documentation\<files>	This folder contains software documentation.

Memory requirements

Establish how much free program memory is available on the machine. This must be considered when deciding which cycles to load.

Using the installation wizard to issue the software

After you have selected the cycles you wish to use, press “Run”. A message will appear showing the amount of memory that will be required in kilobytes (KB). If this is too big, press “Cancel” and modify your cycle selections, or increase the memory available on the machine.

Machine parameter settings[ST]

Be aware that this software requires certain machine parameters to be set. See “Machine parameter settings” in Chapter 10, “Configuration”.

Read-ahead control

Fast machining or smoothing control options can cause problems with block read-ahead when running a cycle. Refer to “Editing the active offset and read-ahead program O9723” in Chapter 10, “Configuration”.

Use of inch/mm units

All inputs (unless otherwise stated) described in this manual are to be inserted using the current machine units.

CAUTION: It is a feature of this software that all unit-dependent probe data is stored in metric (mm) units, regardless of the current machine units. When this data is read, it is converted as required to suit the active machine units. This differs from previous versions of inspection software.

Order of I, J and K cycle inputs

It is important that the order of these inputs is maintained when programming relevant cycles. Failure to do so may lead to incorrect moves or results.

Examples

G65 P9xxx I30. J10. K9. ✓

G65 P9xxx K9. J10. I30. ✗

Renishaw customer services

Calling Renishaw

If you have a question about the software, first consult the documentation and other information included with your product.

If you cannot find a solution, you can receive information on how to obtain customer support by contacting the Renishaw company that serves your country (for worldwide contact details, see **www.renishaw.com/contact**).

When you call, it will help the Renishaw support staff if you have the appropriate product documentation at hand. Please be prepared to give the following information (as applicable):

- The software version you are using (see the EQUIPMENT REGISTRATION RECORD form).

  NOTE: The software part number and version number are commented at the top of the settings program (O9724).

- The type of hardware that you are using (see the EQUIPMENT REGISTRATION RECORD form).
- The error number and wording of any message that appears on your screen.
- A description of what happened and what you were doing when the problem occurred.
- A description of how you tried to solve the problem.

Chapter 1

Installing the software

This chapter describes how to load the Inspection Plus software.

Contained in this chapter

Installing the software

It is important that this software is installed correctly. This means selecting the appropriate cycles and configuring them to run properly on the machine. An installation wizard is provided to prepare the software for installing, but further on-machine customisation may be required afterwards. To complete the task, it will then be necessary to run the calibration cycles to set the calibration data for the probe on the machine.

1. First, refer to Appendix A, “Features, cycles and limitations of the Inspection Plus software”, to determine whether the software is suitable for your needs. Also familiarise yourself with Chapter 10, “Configuration”.

2. Use the installation wizard to prepare the cycles for loading into the controller. Options within the wizard automatically prepare the required cycles ready for loading into the controller. If required, the wizard can also separate the cycles into individual files for loading separately. Either load the whole suite of cycles or choose a suitable subset of cycles:

Category	Cycles	Notes
Basic programs and cycles	O9700, O9701, O9721, O9722, O9723, O9724, O9725, O9726, O9727, O9729, O9731, O9732, O9800, O9801, O9832, O9833	These must always be loaded.
Standard cycles and programs	O9810, O9811, O9812, O9814, O9815, O9816, O9817	These can be added as required.
Vector cycles	O9821, O9822, O9823	These can be added as required.
Additional cycles and programs	O9730, O9735, O9818, O9819, O9820, O9834, O9835, O9843	These can be added as required.
Rotated WCS setting	O9744	This can be added as required.
Multiple probe support programs	O9712, O9713, O9714	These are for multiple probe support only.

Manual configuration of the software

As the software is issued using an installation wizard, no further adjustments should be required. However, useful information can be found in Chapter 10, “Configuration”, including details of general software settings, customising the software and variable details.

Chapter 2

Optional inputs

Many of the cycles make use of standard optional inputs. Instead of describing them each time they are required, they are described once in this chapter. You will be referred to this chapter from other chapters whenever a standard optional input is available.

Details of each non-standard optional input that is available with a cycle is provided in the relevant cycle description.

Contained in this chapter

Optional inputs

The examples described below assume that the controller has been configured for metric values (millimetres). The equivalent inch measurement values are shown in brackets.

Bb b = Angle tolerance of the surface, for example, 30° ±1° inputs A30. B1.

Example: B5. to set a tolerance of 5°.

Ee e = Experience value.

Specify the number of a spare tool offset where an adjustment value to the measured size is stored (see Chapter 11, “General information”).

Example: E21. causes the experience value stored in tool offset 21 to be applied to the measured size.

Ff f = This can be either one of the following:

1. The percentage feedback that is used when updating a tool offset (see Chapter 11, “General information”).

   Enter a value between 0 and 1 (0% and 100%).

   Default value: 1 (100%).

2. The feedrate that is used in the protected positioning cycle (O9810) (see Chapter 4, “Protected positioning cycle”).

   Example: F15. sets a feedrate of 15 mm/min.
   (F0.6 sets a feedrate of 0.6 in/min.)

Hh h = The tolerance value of a feature dimension being measured (see Chapter 11, “General information”).

Example: For a dimension of 50 mm +0.4 mm −0 mm, the nominal tolerance is 50.2 mm with H0.2.
(For a dimension of 1.968 in +0.016 in −0 in, the nominal tolerance is 1.976 in with H0.008.)

Mm m = The true position tolerance of a feature. This is a cylindrical zone about the theoretical position (see Chapter 11, “General information”).

Example: M0.1 sets a true position tolerance of 0.1 mm.
(M0.004 sets a true position tolerance of 0.004 in.)

Qq q = The probe overtravel distance for use when the default values are unsuitable. The probe will then travel beyond the expected position when it searches for a surface.

Default values: 4 mm (0.16 in) in the Z axis and 10 mm (0.394 in) in the X and Y axes.

Example: Q8. sets an overtravel distance of 8 mm.
(Q0.3 sets an overtravel distance of 0.3 in.)

Rr	r =	This is an incremental dimension that is used in external features, such as bosses and webs, to give a radial clearance from the nominal target surface prior to a Z-axis move. **Default value:** 5 mm (0.200 in). **Example:** R10. sets a radial clearance of 10 mm. (R0.4 sets a radial clearance of 0.4 in.)
R−r	−r =	This is similar to Rr, except that the clearance is applied in the opposite direction where the boss or web is located within an internal feature. **Default value:** −5 mm (−0.200 in). **Example:** R−10. sets a radial clearance of −10 mm. (R−0.4 sets a radial clearance of −0.4 in.)
Ss	s =	The number of the work offset to be updated. S0 or S53 — The external work offset. S1 to S6 or S54 to S59 — G54 to G59. **Additional work offsets (dependent on the machine options)** S101 to S400 (G54.1 P1 to G54.1 P300) — Additional work offsets option. **Examples:** S148 (G54.1 P48) S248 (G54.1 P148) New work offset = active work offset + error. New external work offset = external work offset + error. **NOTE:** The three measurement errors in X, Y and Z must be found when setting a work co-ordinate system (WCS) with software rotations active. See Appendix C, “Updating the WCS with 5-axis commands active”.
Tt	t =	The number of the tool offset to be updated. **Example:** T20. updates tool offset number 20.
Uu	u =	Upper tolerance limit. If this value is exceeded, no tool offset is updated and the cycle stops with an alarm. Where applicable, this tolerance applies to both size and position. See Chapter 11, “General information”. **Example:** U2. to set the upper tolerance limit to 2 mm. (U0.08 to set the upper tolerance limit to 0.08 in.)
Vv	v =	Null band. This is the tolerance zone in which no tool offset adjustment occurs (see Chapter 11, “General information”). **Default value:** 0 **Example:** V0.5 for a tolerance zone of ±0.5 mm. (V0.02 for a tolerance zone of ±0.02 in.)

Ww w = Print the output data.

W1. Increment the feature number only.

W2. Increment the component number and reset the feature number.

W1.1 Reporter printing. This requires the Reporter app (A-5999-4200) to be installed on the machine. Refer to Installation and user guide *Reporter for Fanuc* (Renishaw part no. H-5999-8700) for information on this product and how to use the report feature.

Example: W1.

Chapter 3

Cycle outputs

This chapter lists the variable outputs that are produced by some of the cycles. You will be referred to this chapter from other chapters when a cycle output is produced.

Contained in this chapter

Cycle outputs (table 1)

	Single surface	Web/ pocket	Bore/ boss	Internal corner	External corner	5-point rectangle	4th axis	PCD bore/boss
	G65 P9811	G65 P9812	G65 P9814	G65 P9815	G65 P9816	G65 P9817	G65 P9818	G65 P9819
#135	X position	X position	X position	X position	X position	X position		X position
#136	Y position	Y position	Y position	Y position	Y position	Y position		Y position
#137	Z position							PCD
#138	Size	Size	Size					Size
#139				X surface angle	X surface angle	Component angle	4th angle	Angle
#140	X error	X error	X error	X error	X error	X error		X error
#141	Y error	Y error	Y error	Y error	Y error	Y error		Y error
#142	Z error			Y surface angle	Y surface angle			PCD error
#143	Size error	Size error	Size error	Y angle error	Y angle error		Height error	Size error
#144				X angle error	X angle error		Angle error	Angle error
#145 *	True position error	True position error	True position error	True position error	True position error	True position error		True position error
#146	Metal condition	Metal condition	Metal condition					Metal condition
#147	Direction indicator							Hole number
#148	Out of tolerance flag (1 to 7)							
#149	Probe error flag (0 to 2)							

* True position error expressed as radial value.

Cycle outputs (table 2)

	Stock allowance	Angled single surface (AD inputs)	Angled single surface (XYZ inputs)	Angled web/pocket	3-point bore/boss	Feature to feature	X/Y angle measure
	G65 P9820	G65 P9821	G65 P9821	G65 P9822	G65 P9823	G65 P9834	G65 P9843
#135		X position	X position	X position	X position	X incremental distance	
#136		Y position	Y position	Y position	Y position	Y incremental distance	
#137			Z position			Z incremental distance	
#138		Size from start	Size (2/3D distance to centre of active WCS)	Size	Size	Minimum distance	
#139						Angle	Angle
#140		X error	X error	X error	X error	X error	
#141		Y error	Y error	Y error	Y error	Y error	
#142			Z error			Z error	
#143		Size error	Size error (2/3D distance)	Size error	Size error	Minimum distance error	Height error
#144	Maximum value					Angle error	Angle error
#145 *	Minimum value	True position error	True position error	True position error	True position error	True position error	
#146	Variation (stock)	Metal condition	Metal condition	Metal condition	Metal condition	Metal condition	
#147							
#148	Out of tolerance flag (1 to 7)						
#149	Probe error flag (0 to 2)						

* True position error expressed as radial value.

This page is intentionally left blank.

Chapter 4

Probe start/stop and protected positioning

The probe must be switched on before use with O9832. This cycle selects which calibration data to use.

The probe can be switched off with O9833.

As a probe moves around the workpiece, it is important that the stylus is protected against a collision with the workpiece. This chapter describes how to use cycle O9810 to set up the protected positioning of the probe, so that it will stop moving in the event of a collision.

Before starting, check that this cycle is available, as the full suite of cycles may not be installed on the machine.

Contained in this chapter

Probe start (O9832)

Description

This program is used to switch the probe on and can also be used to select test mode, and to open a print port in readiness for printing results in subsequent measuring cycles.

CAUTION: It is compulsory to run this cycle before other probe cycles.

A loop in the software tries to activate the probe up to four times. An alarm results if the probe does not switch on. See Chapter 10, “Configuration”, for details on disabling this feature.

Application

The probe must be loaded into the spindle and moved to a safe start plane before running this cycle. It will activate the probe and select the operational modes for subsequent cycles to use.

Format

G65 P9832 [Dd W1.]

where [ ] denote optional inputs.

Example: G65 P9832 D1. W1.

Optional inputs

D1.ST = Test mode ON. All positioning move feedrates will be reduced by 50% and a forced cycle stop will occur before each measure move. You must press cycle start to continue.

D2.ST = Production mode ON. All positioning moves will be at the maximum feedrate and unprotected, should a collision occur. This mode should only be used after initial prove-out and in situations where further collisions are unlikely.

NOTE: If the D input is not used, the cycles will run safely at optimised feedrates.

W1. = Print flag. This must be used to open the port (POPEN) ready for printing data, but only if subsequent measuring cycles use the print results (Ww) input. This input is used in conjunction with the probe stop cycle (O9833) with the W1. input.

Example

G43 H20 Z100.	Apply a tool offset and move to a safe plane.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 X—. Y—. F—.	Move to a gauging position.

Probe stop (O9833)

Description

This cycle is used to switch the probe off. There is an optional input that can be used to close the port after printing results during previous measuring cycles.

A loop in the software tries to deactivate the probe up to four times. An alarm results if the probe does not switch off. See Chapter 10, “Configuration”, for details on disabling this feature.

Application

The probe should be retracted to a safe plane before using this cycle. It will stop the probe and optionally close the print port.

Format

G65 P9833 [W1.]

where [ ] denote optional inputs.

Example: G65 P9833 W1.

Optional input

W1.	=	Print flag. This is used to close the port (PCLOS) after printing data is completed. This input is used in conjunction with the probe start cycle (O9832) with the W1 input.

Example

In the example, with a probe tool offset active, the probe is retracted to a safe start plane before it is switched off prior to a tool change.

The probe stop cycle should be called before making a G28 reference return, otherwise the G28 position may not be effective because of the small Z-axis test move within the O9833 cycle. Note, however, that the cycle does always return to its initial position before finishing.

G65 P9810 Z100.	Retract to a safe plane with the tool offset still active.
G65 P9833	Switch off the probe.
G91	
G28 Z0.	Retract.
continue	

Protected positioning (probe trigger monitoring) (O9810)

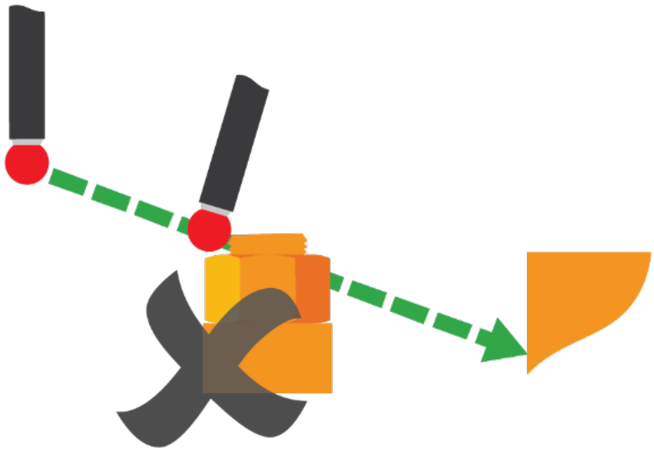


Figure 4.1 Protected positioning of the probe

Description

It is important to protect the probe stylus against damage caused by colliding with an obstacle as the probe moves around the workpiece. When this cycle is used, the machine will stop in the event of a collision.

Alternatively, the cycle can detect misloaded components (the optional M input is required).

Application

The probe is selected and moved to a safe plane. At this point the probe is made active. It can then be moved to the measuring position using this cycle.

In the event of a collision, the machine will stop. Either a “PATH OBSTRUCTED” alarm will be generated or an error flag (#148) will be set (see the Mm input).

Format

G65 P9810 Xx Yy Zz [Ff Mm C1.]

where [ ] denote optional inputs.

Example: G65 P9810 Z10. F3000. M1. C1.

Compulsory inputs

Xx	x =	
Yy	y =	The target positions for the probe positioning move.
Zz	z =	

Optional inputs

Ff f = The optimum feedrate found during optimisation is automatically used. However, this input can be used to specify a different feedrate.

CAUTION: Specifying a feedrate faster than the optimised value means that the machine will not stop in time to avoid breaking a stylus, or worse, should a collision occur. ST

M1. This will set a probe trigger flag (but without a “PATH OBSTRUCTED” alarm). The probe will not automatically return to the start point. Make a G0 or G1 move to leave the surface.

#148 = 0 No probe trigger.
#148 = 7 Probe triggered.

M2. This will set a probe trigger flag (but without a “PATH OBSTRUCTED” alarm). The probe will automatically return to the start point.

#148 = 0 No probe trigger.
#148 = 7 Probe triggered.

C1. Positioning is normally applied at the probe stylus tip position. Using this flag, it is possible to position in the spindle axis to the stylus ball centre.

Example 1: Protected positioning

G1 G54 X20. Y50.	
G43 H20 Z100.	Move to a safe plane.
G65 P9832	Switch on the probe (this includes M19 spindle orientation).
G65 P9810 Z10.	Protected positioning move (F input is optional).
G65 P9811 Z0. S1.	Single surface measurement.

Example 2: Check for a misloaded component

```
G65 P9810 Z1. F3000. M2.

IF[#148EQ0]GOTO10

#3000=100 (MISLOAD COMPONENT TOO HIGH)

N10 (CONTINUE PROGRAM)
```

This page is intentionally left blank.

Chapter 5

Probe calibration and SupaTouch optimisation

Before a probe is used, the probe and stylus must be calibrated correctly. Only when they are calibrated accurately can you achieve total quality control over your manufacturing process.

SupaTouch optimisation recommends suitable measuring and positioning feedrates based on measurement repeatability and environmental conditions. Running this cycle before calibration is optional.

This chapter explains how to use the calibration cycles and optimise your probe.

Contained in this chapter

SupaTouch optimisation cycle

Customers who wish to reduce their probing cycle time can use the SupaTouch optimisation cycle (O9800) to establish optimum measuring and positioning feedrates, based on the specific characteristics of their machine, controller and probe system.

SupaTouch optimisation will establish the following:

- Optimised measuring feedrate.
- Optimised XY and Z positioning feedrates.
- Optimised back-off distance for instances where a two-touch measuring technique is required. Measuring cycles will intelligently select either a one-touch or a two-touch measuring technique, ensuring the fastest possible cycle time.

NOTE (Fanuc only): When optimising, parameter P6201.1=1 (SEB) must be set to ensure that any servo error is compensated in the skip result. See “Machine parameter settings” in Chapter 10, “Configuration”, for further information.

If SupaTouch optimisation is not required

If SupaTouch optimisation is not required, cycle O9800 can be ignored, and calibrating the probe should be the first task. All measurement cycles will use the standard Inspection Plus two-touch measurement technique.

NOTE: If cycle O9800 has already been run, optimised measuring feedrates can be reset to default by setting #505=0 (assuming the calibration data base number #111 is set to 500). Calibration should be carried out after this change.

Why calibrate a probe?

When you fit your probe into the machine shank/holder, it is not necessary for the probe stylus to run true to the spindle centre line. A small amount of run-out can be tolerated, but it is good practice to get the stylus mechanically on-centre to reduce the effects of spindle and tool orientation errors. Without calibration of the probe, run-out will lead to inaccurate results. By calibrating the probe, the run-out is automatically accounted for.

As each Renishaw probe is unique, it is important that you calibrate it in the following circumstances:

- When your probe system is to be used for the first time.
- When a new stylus is fitted to your probe.
- When it is suspected that the stylus has become distorted or that the probe has crashed.
- At regular intervals to compensate for mechanical changes of your machine tool.
- If repeatability of relocation of the probe shank is poor. In this case, the probe may need to be recalibrated each time it is selected.

Three different operations are used to calibrate a probe. They are:

- Calibrating the stylus offset.
- Calibrating the stylus radius.
- Calibrating the probe length.

Calibrating the stylus offset

This calibration cycle determines and stores the offset of the stylus ball to the spindle centre line. The measured values are then compensated by these stored values back to the spindle centre-line position.

Calibrating the stylus radius

Calibration on a feature of known size determines the radius values of the stylus ball. These values are used by the measuring cycles to compensate the measured size.

NOTE: The stored radius values are based on the true electronic trigger points. These values are different from the physical sizes.

Calibrating the probe length

Traditionally, when calibrating a probe on a known reference surface, it determines the probe length, based on the electronic trigger point (not the stylus free length). This can lead to small positioning errors which can affect measurement, particularly if measuring non-prismatic parts. This issue is overcome, as the software determines this error and makes a suitable adjustment to set the probe stylus to its free length.

Length calibration can also be used to automatically compensate for machine and fixture height errors by calibrating on a known reference surface on the part or fixture. Absolute measured machine co-ordinates are not always the most important factor.

Calibration cycles – an overview

The following calibration cycles are described on pages 5-9 to 5-18.

Cycle O9801 K1. (or no K input) This cycle is used to set the length of the probe in its tool holder.

Cycle O9801 K4. This cycle is used to set the stylus XY offsets, the stylus ball radius values and the stylus vector radii values.

Cycle O9801 K5. The sphere cycle is recommended for complete calibration in one operation. This includes probe length calibration either on top of the sphere or at a remote Z surface position. It is important that the sphere centre position and size are accurately known.

To maintain backwards compatibility and flexibility, the following calibration cycles are also available and are described in Appendix B, “Alternative calibration cycles”.

Cycle O9801 K0. This cycle is used for centring on a reference feature, allowing the feature position to be found. Optionally use an S input to set a work offset.

Cycle O9801 K2. This cycle is used to set the stylus XY offset calibration values.

Cycle O9801 K3. This cycle is used to set the stylus XY offsets and the stylus ball radius values in the X+/− and Y+/− directions. It provides calibration data that is suitable for all measuring cycles, except for vector measuring cycles O9821, O9822 and O9823.

Use K−3. when you do not want to overwrite the stylus XY offset values that are determined when using the K2. input described above.

Notes on using the M180./M3. input

These notes refer to the calibration cycles described in this chapter and in Appendix B, “Alternative calibration cycles”.

M180. or M3. must be used when the XY position of the calibration artefact is unknown. The following restrictions should be considered before use:

- M180. requires an advanced spindle orientation option which must be provided by the machine tool builder.
- M3. can be used in all situations. The spindle will rotate at 400 r/min while measurement takes place.
- Only probes with a 360° transmission feature can use the M3. option.
- The stylus ball diameter must not be outside the following range: 1 mm to 6 mm (0.0393 in to 0.236 in).
- The stylus length must not be outside the following range: 50 mm to 200 mm (1.968 in to 7.874 in)
- The stylus run-out (stylus centre line to spindle centre line) must be less than 0.25 mm (0.0098 in).

SupaTouch optimisation cycle (O9800)

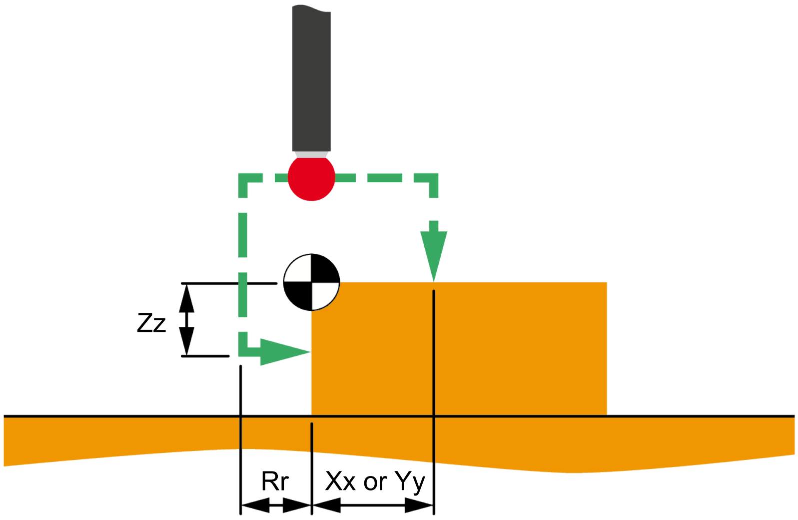


Figure 5.1 SupaTouch optimisation cycle

Description

This cycle measures an X or Y surface multiple times using fast and slow feedrates then repeats the process on a Z surface. The probe returns to the start position and waits for an M00 program stop. The calculated feedrates are displayed as follows:

#100	Maximum permissible measuring feedrate (in mm/inch).
#101	Maximum permissible Z-axis positioning feedrate (in mm/inch).
#102	Maximum permissible X-axis or Y-axis positioning feedrate (in mm/inch).

Three options are available to the user:

- Press cycle start to accept the values,
- Edit the values and press cycle start, or
- Press reset to abandon optimisation and use the standard two-touch measuring method.

When cycle start is pressed, the optimised values are automatically loaded to #500 variables (always in millimetres).

Application

Enter the approximate probe length in the relevant tool offset. Set and activate an appropriate work offset to a chosen edge, then position the probe directly above the edge and run the cycle.

Format

G65 P9800 Bb. Xx. or Yy. [Hh. Ff. Uu. Ww. Qq. Rr. Zz.]

where [ ] denote optional inputs.

Example: G65 P9800 B6. X15. Z−8. H0.002 U7. W15. Q10. R15. F9000.

Compulsory inputs

Bb	b =	The nominal diameter of the stylus ball.
Xx	x =	The absolute X-axis position for the Z touch.
or		
Yy	y =	The absolute Y-axis position for the Z touch.

Optional inputs

Hh	h =	The measurement repeatability value required from the probing system. Adjusting this value will influence the value displayed in #100. **Default value:** 0.005 mm (0.00019 in).
Ff	f =	The fast positioning feedrate limit. **Default value:** 12000 mm/min (472 in/min).
Uu	u =	The XY probe overtravel limit. Refer to the probe hardware documentation for suitable overtravel limit data. **Default value:** 11 mm (0.433 in).
Ww	w =	The Z probe overtravel limit. Refer to the probe hardware documentation for suitable overtravel limit data. **Default value:** 5 mm (0.197 in).
Zz	z =	The absolute Z-axis position for the X or Y touch. **Default value:** −10 mm (−0.393 in).

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

Assuming the calibration data base number (#111) is set to 500:

#505 Software signature – represents the optimisation status and other internal software settings.

#506 Probe system delay, including any transmission delay and probe filter settings. This is used to maintain the optimum back-off distance.

#507 Machine stopping distance for a feedrate of 1000 mm/min (39.37 in/min). This is used during measurement to ensure that skip positions are not taken while the machine is accelerating or decelerating.

#508 Measuring feedrate, transferred from #100. Measuring cycles use this value when capturing skip positions. A one-touch or two-touch measurement method is automatically selected, based on which method is the fastest.

#509 Fast positioning feedrate, transferred from #101 and #102. Measuring cycles use this value when positioning the probe prior to measurement.

Example: #509=300.089

Z feedrate = 3000 mm/min (uses the value before the decimal point only × 10).
XY feedrate = 8900 mm/min (uses the value after the decimal point only × 100000).

Example

Set the X, Y, Z values in work offset G54.

NOTE: The tool offset must be active.

O0001

G90 G80 G40 G0	Preparatory codes for the machine.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move.
G65 P9800 B6. X20. Z−6.	Optimise the probing system.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Calibration cycles

Calibrating the probe length (O9801 K1.)

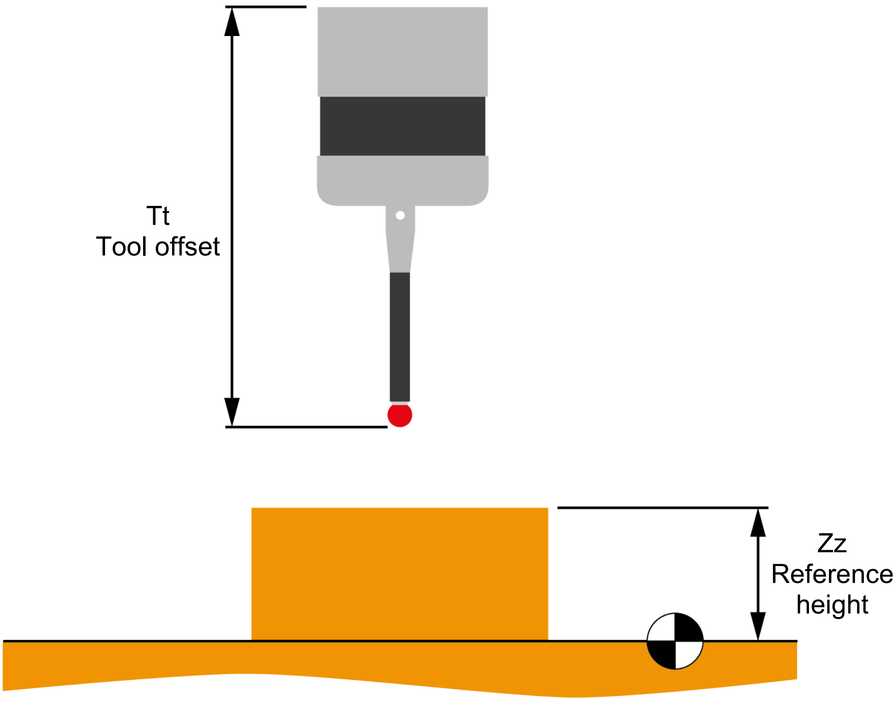


Figure 5.2 Calibrating the probe length

Description

The probe is positioned adjacent to a Z-axis reference surface. When the calibration cycle is completed, the active probe tool offset is adjusted to the reference surface.

Application

First load an approximate tool offset. Position the probe adjacent to the reference surface. When the cycle is run, the surface is measured and the tool offset is reset to a new value. The probe then returns to the start position.

Alternatively, if your machine retains the tool offset at all times, this cycle can be run directly from the MDI screen without writing a program.

Format

G65 P9801 Bb. Zz. Tt. [K1.]

where [ ] denote optional inputs.

Example: G65 P9801 B6. K1. Z0. T20.

Compulsory inputs

Bb	b =	The nominal diameter of the stylus ball.
Tt	t =	The active tool offset number.
Zz	z =	The absolute position of the reference surface.

Optional input

K1.	Tool length setting mode. This is also the cycle default if no Kk input is used.

Outputs

#518	Z calibration radius used for 3D vector measuring (ZRAD).
#519	Nominal stylus radius used for 3D vector measuring (SRAD).

The active tool offset is set.

Example

Set the X, Y, Z values in work offset G54.

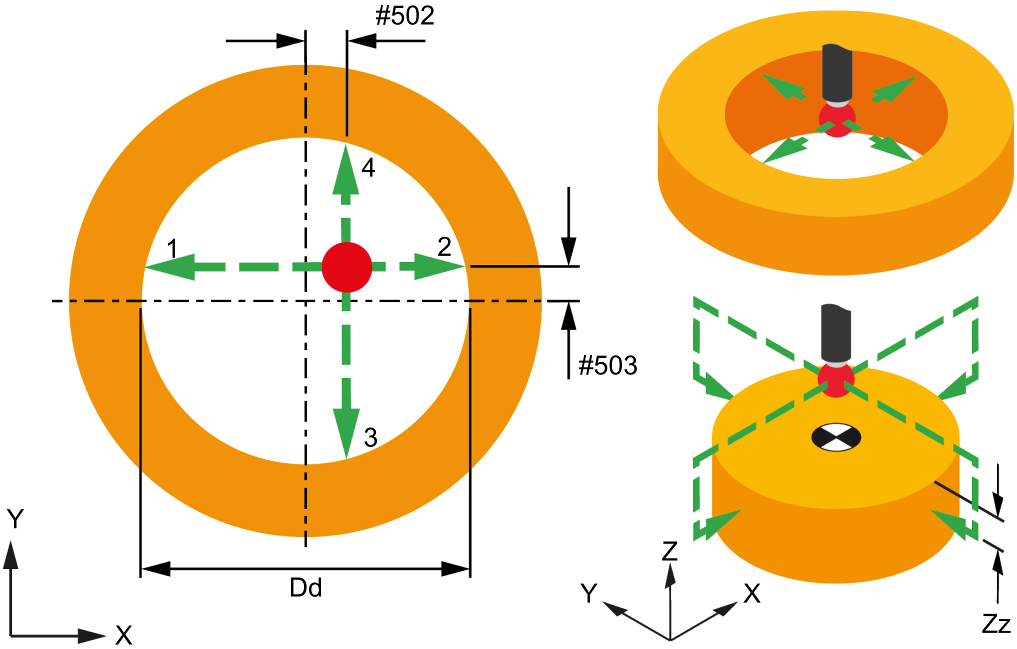
NOTE: The tool offset must be active. The active tool offset H number must be the same as the T input number (shown underlined in this example).

O0001

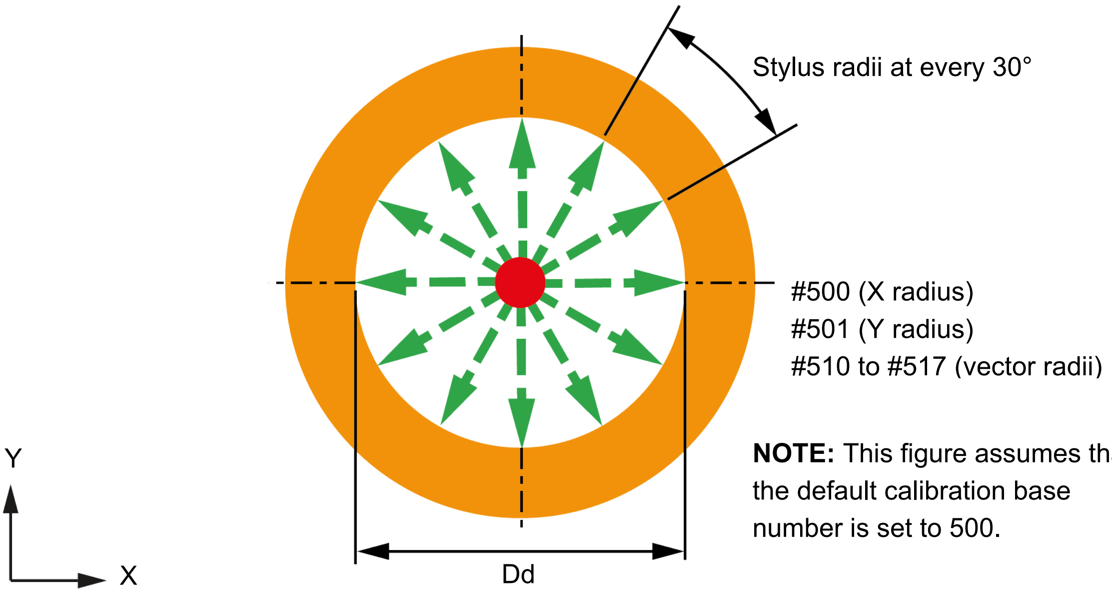
G90 G80 G40 G0	Preparatory codes for the machine.
G54 X0. Y0.	Start position.
G43 <u>H1</u> Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move.
G65 P9801 B6. K1. Z0. <u>T1.</u>	Update the probe length in the Z axis.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Calibrating the stylus ball offsets and radii (O9801 K4.)

Stylus XY offsets



Stylus ball radii



NOTE: This figure assumes that the default calibration base number is set to 500.

Figure 5.3 Calibrating the stylus ball offsets and radii

Description

The probe stylus is positioned inside a reference feature, typically a ring gauge, at a height suitable for calibration. When the cycle is completed, 12 radius values for the stylus ball are stored; one for every 30° position.

Application

The reference feature must be mounted on the machine table and its position must be accurately determined. Before running the cycle, the probe must be positioned with the spindle on the centre of the reference feature and at a suitable height, with the spindle orientation (M19) active.

Internal feature (ring gauge):	Position the stylus at a suitable height inside the feature.
External feature (cylinder):	Position the stylus at a suitable clearance position above the feature.

NOTE: If spindle 180° orientation positioning is available, use the Mm input to avoid accurate pre-positioning before running the cycle.

Format

G65 P9801 K4. Bb. Dd. [M180. Ss. Zz.]

where [ ] denote optional inputs.

Example: G65 P9801 K4. B6. D50.005 M180. Z50.

Compulsory inputs

K4.		Calibrate the offsets and radii of the stylus ball. Use K−4. when you do not want to overwrite the stylus XY offset values that are established when using the K2. input.
Bb	b =	The nominal diameter of the stylus ball.
Dd	d =	The reference size of the feature.

Optional inputs

M180.		M180. is used as a flag to orient the probe to a second spindle position (180°). This is used to automatically find the centre of the feature, and means that only approximate pre-positioning is required. Alternatively, an M3. input (rotating spindle) can be used – see “Notes on using the M180./M3. input” at the beginning of this chapter.
Zz	z =	The absolute Z-axis measuring position when calibrating on an external feature. If this is omitted, an internal reference feature cycle is assumed.

For optional input Ss, see Chapter 2, “Optional inputs”.

Outputs

The following data is stored (this assumes that the default calibration base number is set to 500):

#500	X-axis stylus ball radius (XRAD)
#501	Y-axis stylus ball radius (YRAD)
#502	X-axis stylus offset (XOFF)
#503	Y-axis stylus offset (YOFF)

#510	30° stylus ball radius (VRAD)
#511	60° stylus ball radius (VRAD)
#512	120° stylus ball radius (VRAD)
#513	150° stylus ball radius (VRAD)
#514	210° stylus ball radius (VRAD)
#515	240° stylus ball radius (VRAD)
#516	300° stylus ball radius (VRAD)
#517	330° stylus ball radius (VRAD)

#135	X position
#136	Y position

#143	X axis position (machine position)
#144	Y axis position (machine position)

Example 1: Calibrating the stylus ball offsets and radii

A tool offset must be active before running this program. If your machine does not retain the offset, then use Example 2.

Position the probe accurately on-centre in the ring gauge and at the required depth.

O0004

G90 G80 G40 G0	Preparatory codes for the machine.
G65 P9832	Switch on the probe (this includes M19).
G65 P9801 K4. B6. D50.001	Calibrate in a 50.001 mm (1.9685 in) diameter ring gauge with a 6 mm (0.236 in) diameter stylus.
G65 P9833	Switch off the probe (when applicable).
M30	End of the program.

Example 2: Calibrating the stylus ball offsets and radii (alternative method)

This example describes a complete positioning and calibration program.

Set the exact XY centre and Z top face position of the feature in a work offset (this example uses G54).

O0004

G90 G80 G40 G0	Preparatory codes for the machine.
G54 X0. Y0.	Move to the centre of the feature.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−5. F3000.	Protected positioning move into the hole.
G65 P9801 K4. B6. D50.001	Calibrate in a 50.001 mm (1.9685 in) diameter ring gauge with a 6 mm (0.236 in) diameter stylus.
G65 P9810 Z100. F3000.	Protected positioning move retract to 100 mm (3.94 in).
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Calibrating on a reference sphere (O9801 K5.)

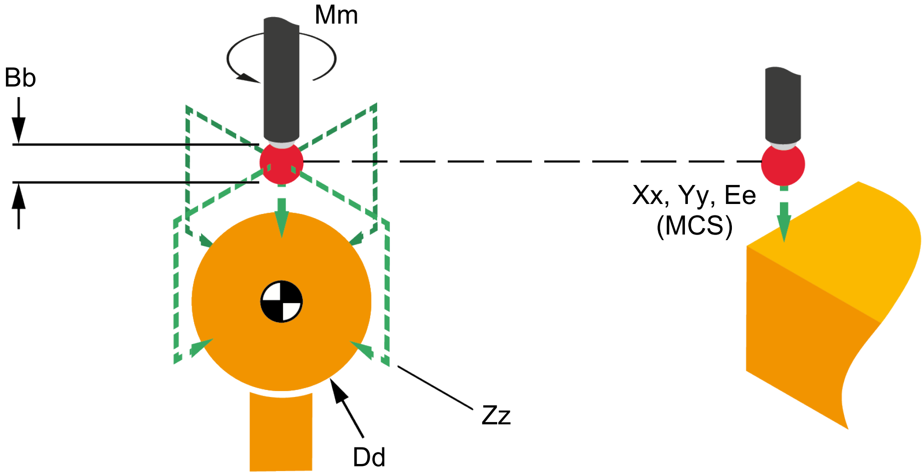


Figure 5.4 Calibrating on a reference sphere

Description

This cycle is used for calibrating the probe stylus on a reference sphere. It determines all stylus ball calibration values, including the vector radii, and also sets the probe length offset in one operation. The cycle makes all the necessary positioning and measuring moves on the sphere.

Application

The reference sphere must be rigidly mounted on the machine tool so that it can be approached from above and in all directions in the XY plane. The XYZ sphere centre must be entered into a work offset, prior to running this cycle. The XY values must be the exact centre, unless 180° spindle positioning is available. In this situation, values can be approximate (see optional inputs M180./M3).

If probe length calibration is to be performed on the sphere, enter the exact Z sphere centre position. If length calibration is performed on another surface, the approximate centre is sufficient.

Write a program that positions the probe stylus approximately 10 mm (0.394 in) above the sphere, with the probe tool offset and relevant work offset active. Then run the cycle for complete calibration.

At the end of the cycle the probe is returned to the start position.

Format

G65 P9801 K5. Zz. Dd. Bb. Tt. [M180. Ee. Xx. Yy.]

where [ ] denote optional inputs.

Example: G65 P9801 K5. Z0. D30. B6. T14. M180. E−300.157 X250. Y100.

Compulsory inputs

K5.		Calibrate on a sphere.
Bb	b =	The nominal diameter of the stylus ball.
Dd	d =	The diameter of the reference sphere.
Tt	t =	The active tool offset for length updating.
Zz	z =	The absolute Z-axis sphere centre position (typically Z0). If length calibration is performed on the sphere, the value must be exact.

Optional inputs

Ee	e =	The exact Z surface position from machine reference. This is an alternative way to set the probe length instead of on the top of the sphere. If using this method, the Z sphere height position is not critical.
M180.		M180. is used as a flag to orient the probe to a second spindle position (180°). This is used to automatically find the centre of the feature, and means that only approximate pre-positioning is required. Alternatively, an M3. input (rotating spindle) can be used – see “Notes on using the M180./M3. input” at the beginning of this chapter.
Xx	x =	The X position from machine reference. Used with the Ee input above.
Yy	y =	The Y position from machine reference. Used with the Ee input above.

Outputs

The following data is stored (this assumes that the default calibration base number is set to 500):

#500	X-axis stylus ball radius (XRAD)
#501	Y-axis stylus ball radius (YRAD)
#502	X-axis stylus offset (XOFF)
#503	Y-axis stylus offset (YOFF)

#510	30° stylus ball radius (VRAD)
#511	60° stylus ball radius (VRAD)
#512	120° stylus ball radius (VRAD)
#513	150° stylus ball radius (VRAD)
#514	210° stylus ball radius (VRAD)
#515	240° stylus ball radius (VRAD)
#516	300° stylus ball radius (VRAD)
#517	330° stylus ball radius (VRAD)
#518	Z calibration radius used for 3D vector measuring (ZRAD)
#519	Nominal stylus radius used for 3D vector measuring (SRAD)

#135	X position
#136	Y position

#143	X axis position (machine position)
#144	Y axis position (machine position)

The active tool length offset is set.

Example: Calibrating on a reference sphere using the M180./M3. input

Before running this program, the tool length offset for the probe must be active. Position the stylus approximately 10 mm (0.394 in) above, and approximately centrally over, the reference sphere.

O0002

G90 G80 G40 G0	Preparatory codes for the machine.
G54 X0. Y0.	Move to the approximate centre of the reference sphere in the XY axis.
G43 Z100. H14	Activate offset 14 and go approximately 100 mm (3.94 in) above the sphere.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z15. F3000.	Protected positioning move to approximately 15 mm (0.59 in) above the sphere.
G65 P9801 K5. Z0. D20. B6. T14. M180.	Use a 20 mm (0.7874 in) diameter reference sphere with the Z work offset set to the centre. The stylus diameter is 6 mm (0.2362 in). Set probe tool offset 14.

G65 P9810 Z100. F3000.	Protected positioning move retract to 100 mm (3.94 in).
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Chapter 6

Standard measuring cycles

This chapter describes how to use the standard measuring cycles. Before using these cycles, the radius of the stylus ball must be calibrated using cycle O9801 (see Chapter 5, “Probe calibration and SupaTouch optimisation”).

Before starting, check that the cycles are available on the machine, as the full suite of cycles may not have been installed.

Contained in this chapter

XYZ single surface measurement (O9811)

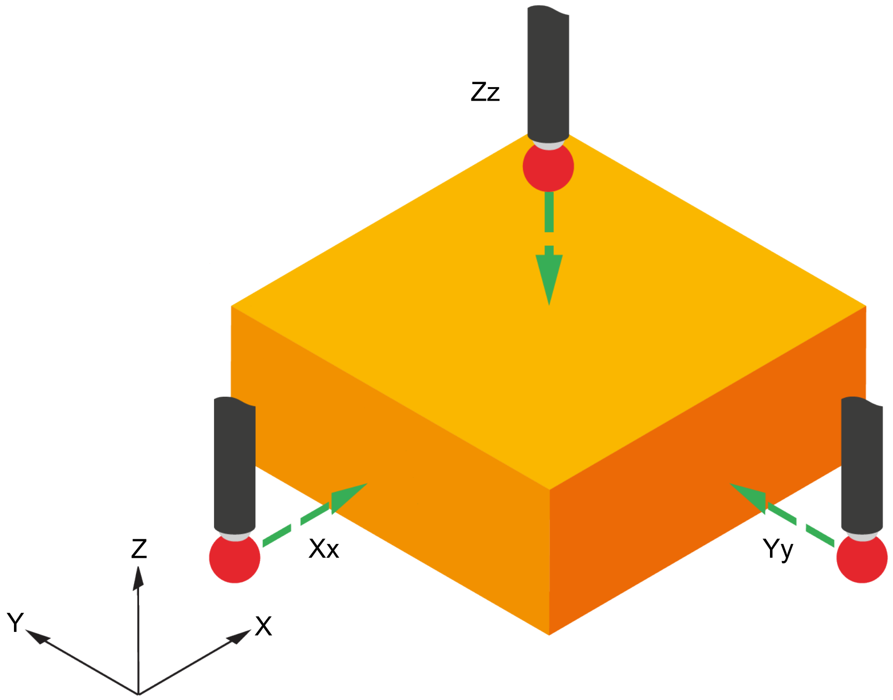


Figure 6.1 Measurement of a single surface

Description

This cycle measures a surface to establish the size or position.

Application

With its tool offset active, position the probe adjacent to the surface. The cycle measures the surface and returns to the start position.

The measured surface can be considered in one of two ways:

1. As a size, where the tool offset is updated in conjunction with the Tt and Hh inputs.

2. As a reference surface position, for the purpose of adjusting a work offset using the Ss and Mm inputs.

Format

G65 P9811 Xx. or Yy. or Zz. [Ee. Ff. Hh. Mm. Qq. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9811 X50. E21. F0.8 H0.2 M0.2 Q10. S1. T20. U0.5 V0.5 W2.

Compulsory inputs

Xx x =
or
Yy y = The surface position or size.
or
Zz z =

Optional inputs

See Chapter 2, “Optional inputs”.

Example: Measuring a single surface in X and Z

T01 M06	Select the probe.
G54 X−40. Y20.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−8. F3000.	Protected positioning move to the start position.
G65 P9811 X−50. T10.	Single surface measurement.
G65 P9810 Z10.	Protected positioning move.
G65 P9810 X−60.	Protected positioning move.
G65 P9811 Z0. T11.	Single surface measurement.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.

continue

The radius offset (10) and length offset (11) of the tool are updated by the errors of the surface positions.

Web/pocket measurement (O9812)

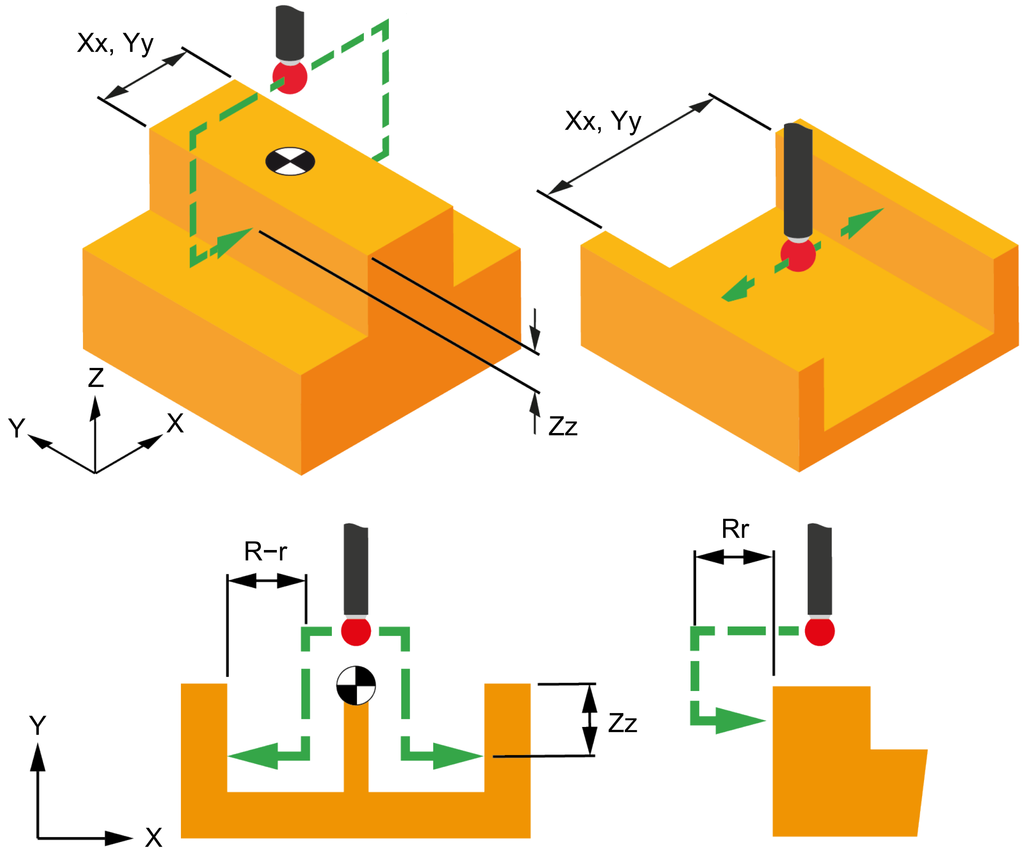


Figure 6.2 Measurement of a web or pocket feature

Description

This cycle measures a web or pocket feature using two measuring moves along the XY axis.

Application

With the probe and probe offset active, position the probe to the expected centre line of the feature and at a suitable position in the Z axis. Run the cycle with suitable inputs.

Format

G65 P9812 Xx. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

or

G65 P9812 Yy. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

or

G65 P9812 Xx. Zz. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

or

G65 P9812 Yy. Zz. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9812 X50. Z100. E21. F0.8 H0.2 M.2 Q10. R10. S1. T20. U0.5 V0.5 W2.

Compulsory inputs

Xx	x =	The nominal size of the feature when measured in the X axis.
or		
Yy	y =	The nominal size of the feature when measured in the Y axis.
Zz	z =	The absolute Z-axis position when measuring a web feature. If this is omitted, a pocket cycle is assumed.

Optional inputs

Rr	r =	This can be used, as shown in the diagrams above, to pre-position before each measurement. It can also be used for an internal pocket cycle using an R+ input (and no Zz input). The fast pre-positioning will improve cycle time on large pockets but will produce an alarm if the probe stylus is triggered during pre-positioning. **Default:** Pocket cycle with no fast pre-positioning.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

Example 1: Measuring a web

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move.
G65 P9812 X50. Z−10. S2.	Measure a 50 mm (1.968 in) wide web.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.

continue

The centre line of the feature in the X axis is stored in work offset 02 (G55).

Example 2: Measuring a pocket (referred datum)

T01 M06	Select the probe.
G54 X100. Y50.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−10. F3000.	Protected positioning move.
G65 P9812 X30. S2.	Measure a 30 mm (1.181 in) wide pocket.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.

continue

The error of the centre line is referred to the datum point X0. The revised X0 position is set in work offset 02 (G55).

Bore/boss measurement (O9814)

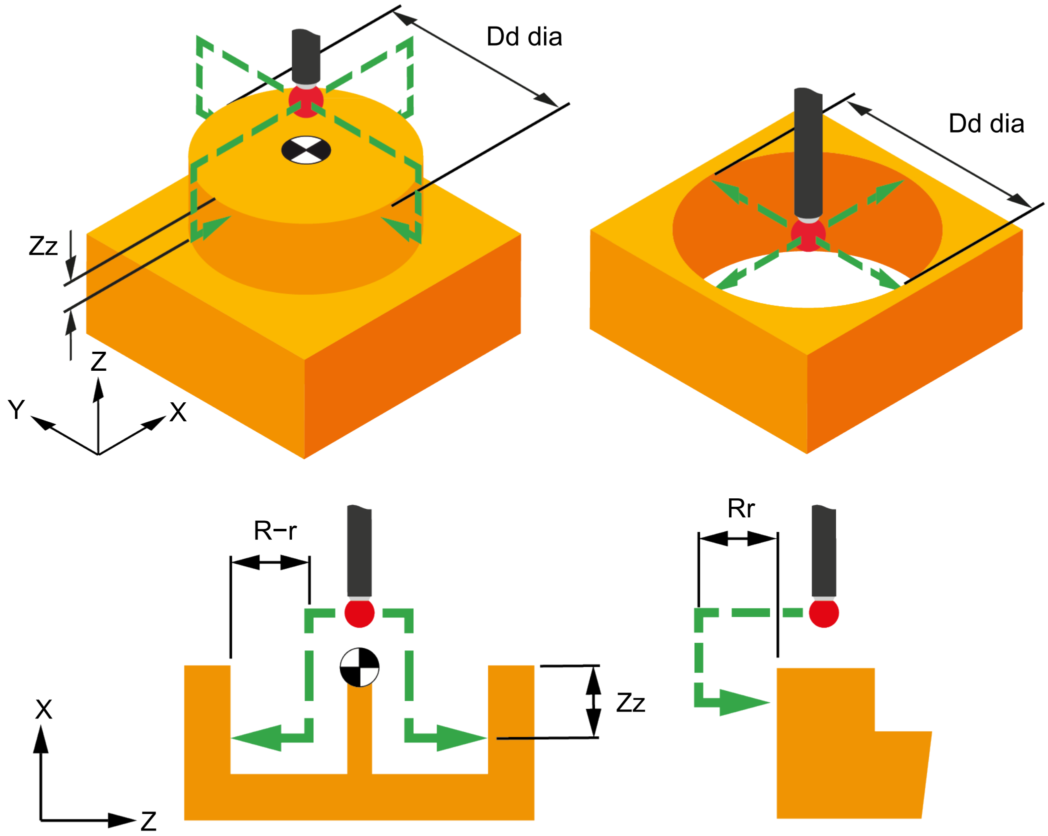


Figure 6.3 Measurement of a bore or boss feature

Description

This cycle measures a bore or boss feature using four measuring moves along the XY axis.

Application

With the probe and probe offset active, position the probe to the expected centre line of the feature and at a suitable position in the Z axis. Run the cycle with suitable inputs.

Format

G65 P9814 Dd. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]
or
G65 P9814 Dd. Zz. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9814 D50.005 Z100. E21. F0.8 H0.2 M0.2 Q10. R10. S1. T20. U0.5 V0.5 W2.

Compulsory inputs

Dd	d =	The nominal size of the feature.
Zz	z =	The absolute Z-axis position when measuring a boss feature. If this is omitted, a bore cycle is assumed.

Optional inputs

Rr	r =	This can be used, as shown in the diagrams above, to pre-position before each measurement. It can also be used for an internal bore cycle using an R+ input (and no Zz input). The fast pre-positioning will improve cycle time on large bores, but will produce an alarm if the probe stylus is triggered during pre-positioning. **Default:** Bore cycle with no fast pre-positioning.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

Example 1: Measuring a boss

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move.
G65 P9814 D50. Z−10. S2. R10.	Measure a 50 mm (1.968 in) diameter boss.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.

continue

The centre line of the feature in the X and Y axis is stored in work offset 02 (G55).

Example 2: Measuring a bore (referred datum)

T01 M06	Select the probe.
G54 X100. Y100.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−10. F3000.	Protected positioning move.
G65 P9814 D30. S2.	Measure a 30 mm (1.181 in) diameter bore.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.

continue

The error of the centre line is referred to the datum point X0, Y0. The revised X0, Y0 position is set in work offset 02 (G55).

This means the work offset is adjusted by the error between the start position and the actual centre line of the feature.

Finding an internal corner (O9815)

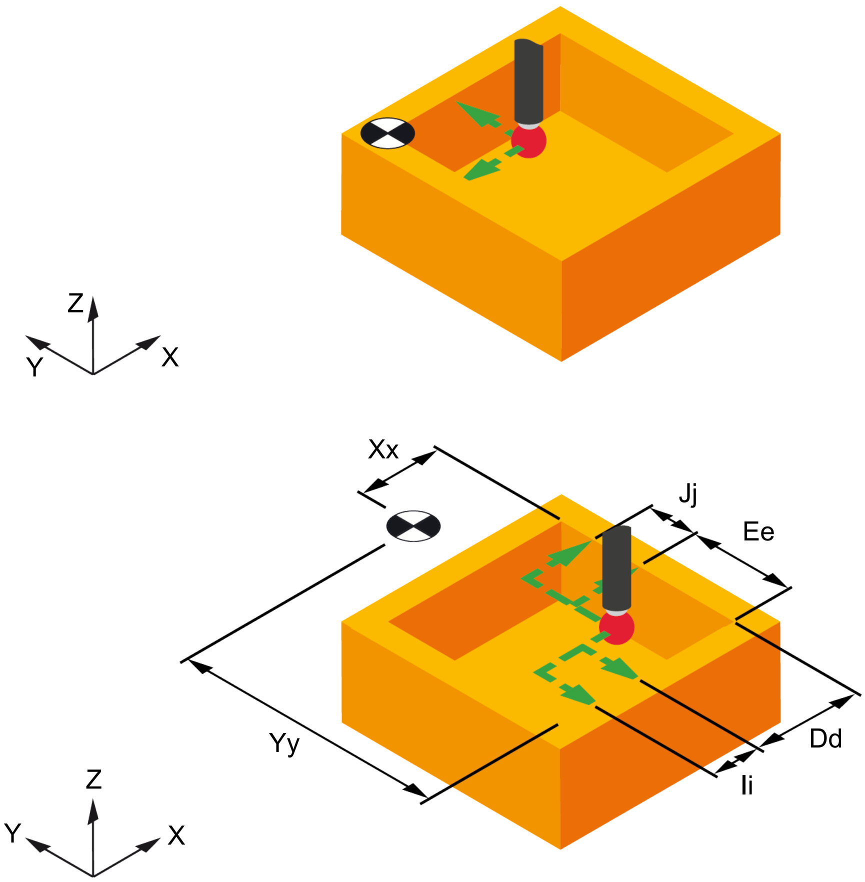


Figure 6.4 Finding an internal corner position

Description

This cycle is used to establish the corner position of a feature. A true corner intersection can be found when the corner is not 90°.

Application

With the tool offset active, position the probe at the start position. The probe measures the Y-axis surface first, then measures the X-axis surface. It then returns to the start position.

If an error occurs during the cycle, the probe returns to the start position.

Format

G65 P9815 Xx. Yy. [Bb. Dd. Ee. Ii. Jj. Mm. Qq. Ss. Uu. Ww. Zz.]

where [ ] denote optional inputs.

Example: G65 P9815 X100. Y100. B2. D10. E10. I10. J10. M.2 Q10. S1. U0.5 W2. Z−10.

NOTE: If inputs I and J are used, they **must** be stated in the order shown above.

Compulsory inputs

Xx	x =	The nominal position of the corner in the X axis.
Yy	y =	The nominal position of the corner in the Y axis.

Optional inputs

A note about inputs I and J

If the I and J inputs are both missing, only two gauging moves occur. The corner feature is assumed to be parallel to the axes.

If either I or J is missing, three gauging moves then occur and the corner feature is assumed to be 90°.

Bb	b =	Angle tolerance. This applies to both X and Y surfaces. It is equal to half the total tolerance. **Example:** ±0.25° = B0.25 tolerance.
Dd	d =	The X distance from the corner to the first measuring position. **Default value:** D0 (distance = 0) is assumed if Dd is missing.
Ee	e =	The Y distance from the corner to the first measuring position. **Default value:** E0 (distance = 0) is assumed if Ee is missing.
Ii	i =	The incremental distance to the second probing position along the X axis. This input self-calculates so that it is always a positive value. **Default value:** no move.
Jj	j =	The incremental distance to the second probing position along the Y axis. This input self-calculates so that it is always a positive value. **Default value:** no move.
Zz	z =	The Z measuring height position. It is sometimes desirable to pre-position above the feature to avoid clamps and obstacles. Using this input, the cycle will position down to the Zz height, take the measurement and retract for every measuring position. **Default value:** no move.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

The measurement values of the feature are stored in variables #135 to #149 (for details, see Chapter 3, “Cycle outputs”).

Variable #139 is the angle of the X surface and is measured from the X+ axis direction. Variable #142 is the angle of the Y surface and is also measured from the X+ axis direction.

Example: Finding an internal corner

NOTE: Co-ordinate rotation. It is possible to implement rotation using G68/G69 when the control option is available.

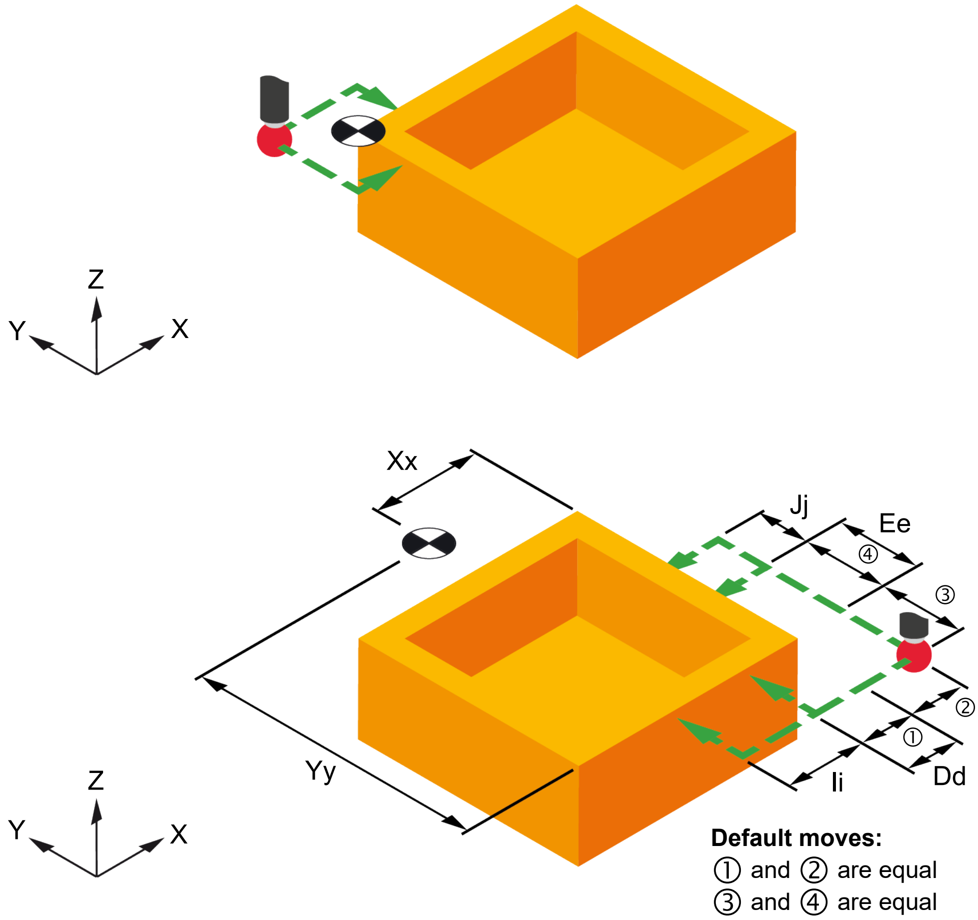
T01 M06	Select the probe.
G54 X10. Y10.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−5. F3000.	Protected positioning move.
G65 P9815 X20. Y20. I10. J10.	Corner find.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.
G17	Select the plane.
G68 X#135 Y#136 R#139	Set the rotational position and angle. (See note below.)
continue	
G69	Cancel the rotation mode.

NOTE: If the G68 data #135, #136 and #139 needs to be saved for further use, copy the data into spare #500 series variables before this G68 line of code.

Example:

```
#595=#135
#596=#136
#599=#139
G68 X#595 Y#596 R#599
```

Finding an external corner (O9816)



NOTE:
The start point establishes the distance to the first measuring position.

Figure 6.5 Finding an external corner

Description

This cycle is used to establish the corner position of a feature. A true corner intersection can be found when the corner is not 90°.

Application

With the tool offset active, position the probe at the start position. The probe measures the Y-axis surface first then measures the X-axis surface. It then returns to the start position.

If an error occurs during the cycle, the probe returns to the start position.

Format

G65 P9816 Xx. Yy. [Bb. Dd. Ee. Ii. Jj. Mm. Qq. Ss. Uu. Ww. Zz.]

where [ ] denote optional inputs.

Example: G65 P9816 X100. Y100. B2. D10. E10. I10. J10. M0.2 Q10. S1. U0.5 W2. Z10.

NOTE: If inputs I and J are used, they **must** be stated in the order shown above.

Compulsory inputs

Xx	x =	The nominal position of the corner in the X axis.
Yy	y =	The nominal position of the corner in the Y axis.

Optional inputs

A note about inputs I and J

If the I and J inputs are both missing, two gauging moves occur. The corner feature is assumed to be parallel to the axes.

If either I or J is missing, three gauging moves occur. The corner feature is assumed to be 90°.

Bb b = Angle tolerance. This applies to both X and Y surfaces. It is equal to half the total tolerance.

Example: ±0.25° = B0.25 tolerance.

Dd d = The X distance from the corner to the first measuring position.

Default value: Uses the start point and distance (① + ② method) to establish the X distance (see Figure 6.5).

Ee e = The Y distance from the corner to the first measuring position.

Default value: Uses the start point and distance (③ + ④ method) to establish the Y distance (see Figure 6.5).

Ii i = The incremental distance to the second probing position along the X axis. This input self-calculates so that it is always a positive value.

Default value: no move.

Jj j = The incremental distance to the second probing position along the Y axis. This input self-calculates so that it is always a positive value.

Default value: no move.

Zz z = The Z measuring height position. It is sometimes desirable to pre-position above the feature to avoid clamps and obstacles. Using this input, the cycle will position down to the Zz height, take the measurement and retract for every measuring position.

Default value: no move.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

The measurement values of the feature are stored in variables #135 to #149 (for details, see Chapter 3, “Cycle outputs”).

Variable #139 is the angle of the X surface and is measured from the X+ axis direction. Variable #142 is the angle of the Y surface and is also measured from the X+ axis direction.

Example: Finding an external corner

NOTE: Co-ordinate rotation. It is possible to implement rotation using G68/G69 when the control option is available.

T01 M06	Select the probe.
G54 X−10. Y−10.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−5. F3000.	Protected positioning move.
G65 P9816 X0. Y0. I10. J10.	Corner find.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.
G17	Select the plane.
G68 X#135 Y#136 R#139	Set the corner position and angle. (See note below.)
continue	
G69	Cancel the co-ordinate rotation mode.

NOTE: If the G68 data #135, #136 and #139 needs to be saved for further use, copy the data into spare #500 series variables before this G68 line of code.

Example:

```
#595=#135
#596=#136
#599=#139
G68 X#595 Y#596 R#599
```

5-point rectangle (O9817) external feature

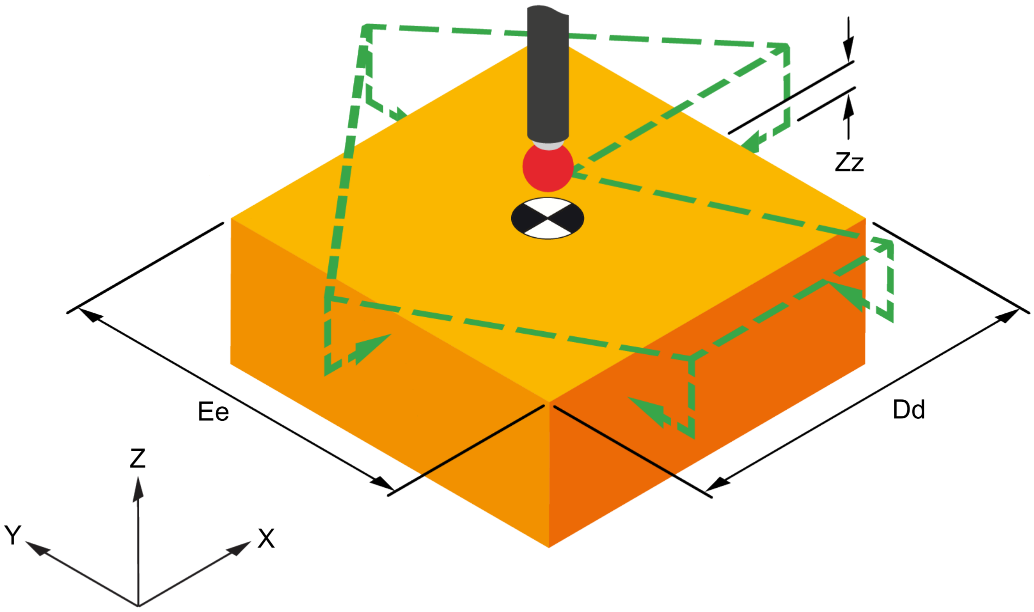


Figure 6.6 Finding the centre and angle of a rectangle (external feature)

Description

This cycle is used to establish the centre of a rectangle and its orientation. A true centre can be found even if the feature is not square to the machine axes.

Application

With the probe and probe offset active, position the probe at the nominal centre of the feature. The probe will take five measuring points before returning to the start position.

If an error occurs during the cycle, the probe returns to the start position.

Format

G65 P9817 Dd. Ee. Zz. [Aa. Bb. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9817 D100. E60. Z−10. A12. B0.5 H20. M0.1 Q10. R10. S1. T60. U2. V40. W2.

NOTE: The function of inputs Ee, Hh, Tt and Vv have been modified for this cycle. The descriptions in Chapter 2, “Optional inputs” are **not** relevant.

Compulsory inputs

Dd d = The feature nominal length in the X axis.

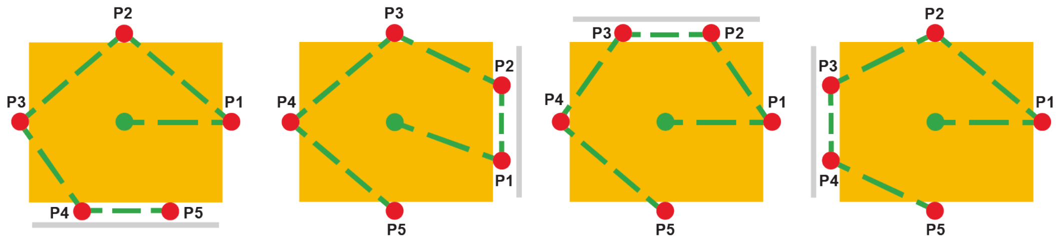
Ee e = The feature nominal length in the Y axis.

Zz z = The Z measuring height position. The cycle will position down to the Zz height, take the measurement and retract for every measuring position.

Optional inputs

Aa a = The face on which the two measurements will take place.

Default value: A14



A14 or no A input = default face **A11** = right face **A12** = top face **A13** = left face

Bb b = Angle tolerance. This applies to both X and Y surfaces. It is equal to half the total tolerance.

Example: ±0.25° = B0.25 tolerance.

Hh h = The position of points P2 and P4 in the X axis relative to the bottom left-hand corner.

Default value: P2 = 50% of Dd, P4 = 25% of Dd.

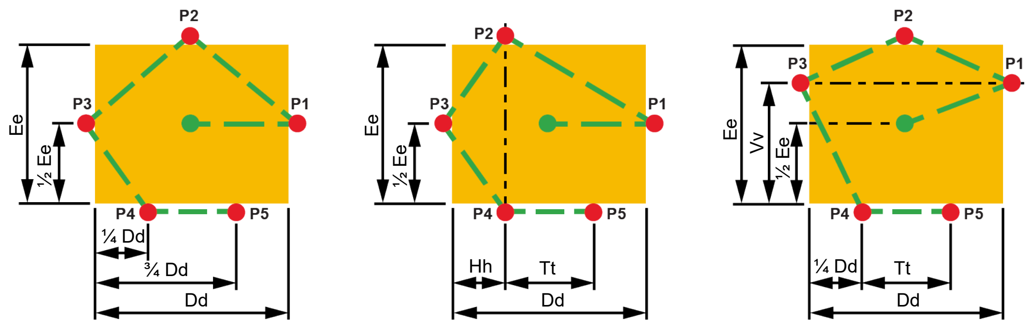
Tt t = The distance between the two measure points on the same face.

Default value: 50% of Dd

Vv v = The position of points P1 and P3 in the Y axis relative to the bottom left-hand corner.

Default value: 50% of Ee

For other optional inputs, see Chapter 2, “Optional inputs”.



Outputs

The measurement values of the feature are stored in variables #135 to #149 (for details, see Chapter 3, “Cycle outputs”).

Example

NOTE: Co-ordinate rotation. It is possible to implement rotation using G68/G69 when the control option is available.

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move.
G65 P9817 D80. E50. Z−10.	External rectangle cycle.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.
G17	Select the plane.
G68 X#135 Y#136 R#139	Set the rotational position and angle. (See note below.)
continue	
G69	Cancel the rotation mode.

NOTE: If the G68 data #135, #136 and #139 needs to be saved for further use, copy the data into spare #500 series variables before this G68 line of code.

Example:

```
#595=#135
#596=#136
#599=#139
G68 X#595 Y#596 R#599
```

5-point rectangle (O9817) internal feature

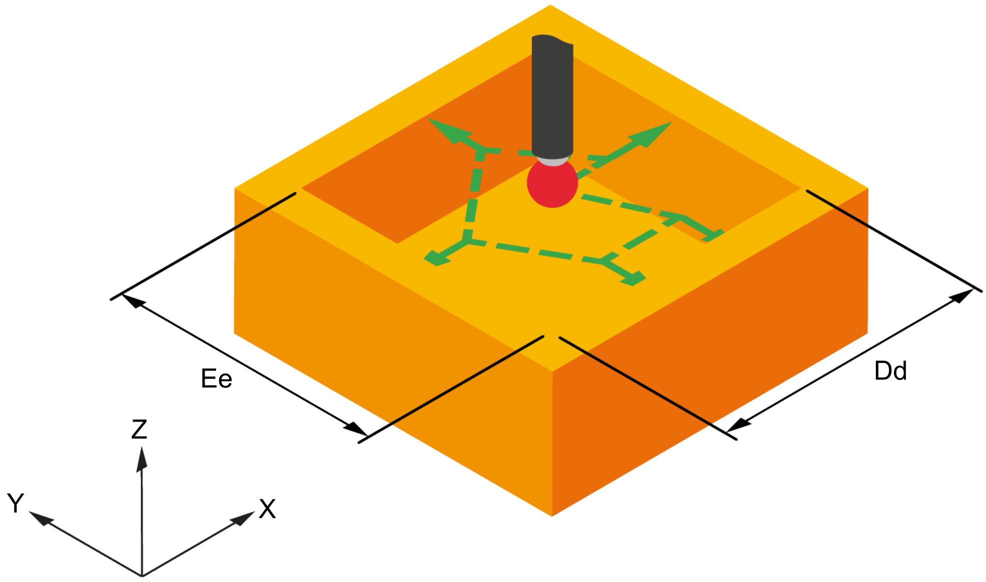


Figure 6.7 Finding the centre and angle of a rectangle (internal feature)

Description

This cycle is used to establish the centre of a rectangle and its orientation. A true centre can be found even when the feature is not square to the machine axes.

Application

With the probe and probe offset active, position the probe at the nominal centre of the feature. The probe will take five measuring points before returning to the start position.

If an error occurs during the cycle, the probe returns to the start position.

Format

G65 P9817 Dd. Ee. [Aa. Bb. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9817 D100. E60. A12. B0.5 H20. M0.1 Q10. R10. S1. T60. U2. W2.

NOTE: The function of inputs Ee, Hh, Tt and Vv have been modified for this cycle. The descriptions in Chapter 2, “Optional inputs” are **not** relevant.

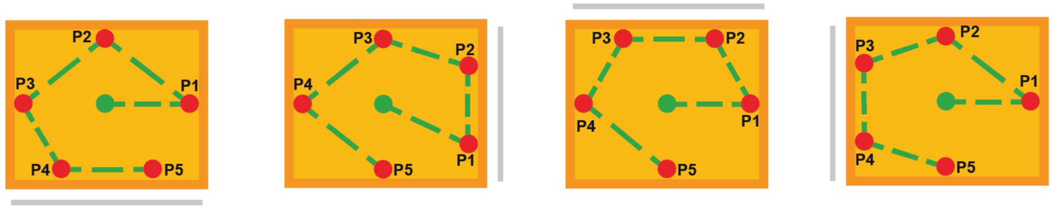
Compulsory inputs

Dd	d =	The feature nominal length in the X axis.
Ee	e =	The feature nominal length in the Y axis.

Optional inputs

Aa a = The face on which the two measurements will take place.

Default value: A14



A14 or no A input = default face **A11** = right face **A12** = top face **A13** = left face

Bb b = Angle tolerance. This applies to both X and Y surfaces. It is equal to half the total tolerance.

Example: ±0.25° = B0.25 tolerance.

Hh h = The position of points P2 and P4 in the X axis relative to the bottom left-hand corner.

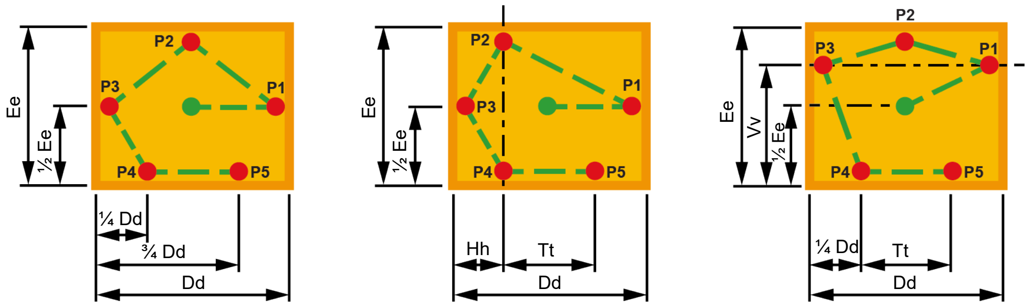
Default value: P2 = 50% of Dd, P4 = 25% of Dd.

Tt t = The distance between the two measure points on the same face.

Default value: 50% of Dd

Vv v = The position of points P1 and P3 in the Y axis relative to the bottom left-hand corner.

Default value: 50% of Ee



For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

The measurement values of the feature are stored in variables #135 to #149 (for details, see Chapter 3, “Cycle outputs”).

Example

NOTE: Co-ordinate rotation. It is possible to implement rotation using G68/G69 when the control option is available.

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−5. F3000.	Protected positioning move.
G65 P9817 D40. E30. S6.	Internal rectangle cycle.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (where applicable).
G28 Z100.	Reference return.
G17	Select the plane.
G68 X#135 Y#136 R#139	Set the rotational position and angle. (See note below.)
continue	
G69	Cancel the rotation mode.

NOTE: If the G68 data #135, #136 and #139 needs to be saved for further use, copy the data into spare #500 series variables before this G68 line of code.

Example:

```
#595=#135
#596=#136
#599=#139
G68 X#595 Y#596 R#599
```

This page is intentionally left blank.

Chapter 7

Vector measuring cycles

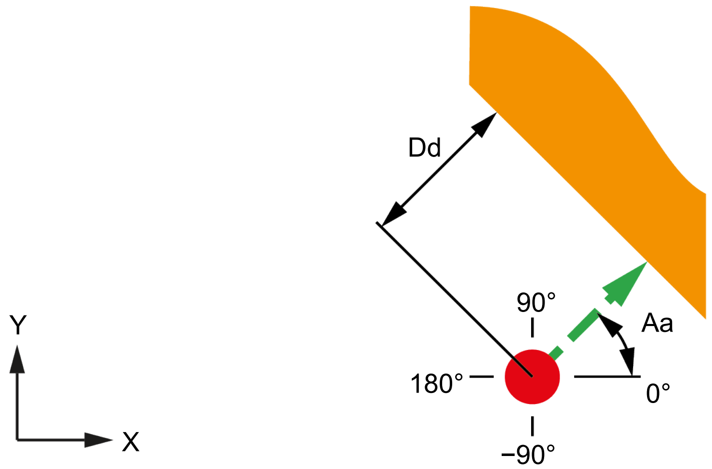
This chapter describes how to use the vector measuring cycles. Before using these cycles, the radius of the stylus ball must be calibrated using either the O9801 K4. or the O9801 K5. cycle (see Chapter 5, "Probe calibration and SupaTouch optimisation").

Before starting, check that the cycles are available on the machine, as the full suite of cycles may not have been installed.

Contained in this chapter

Angled surface measurement using the A and D inputs (O9821)

NOTE: Before using this cycle, the probe must have been recently calibrated using either the O9801 K4. or the O9801 K5. cycle to establish the vector stylus radius values (see Chapter 5, “Probe calibration and SupaTouch optimisation”).



NOTE:
Angles are in the range ±180°.
Positive (+) angle: Counterclockwise direction.
Negative (−) angle: Clockwise direction.

Figure 7.1 Measuring an angled surface

Description

This cycle measures a surface feature using one vectored measuring move along the XY axis.

Application

With the probe and probe offset active, position the probe at the expected reference point of the feature and at a suitable position in the Z axis. Run the cycle with suitable inputs.

Format

G65 P9821 Aa. Dd. [Ee. Ff. Hh. Mm. Qq. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9821 A45.005 D50.005 E21. F0.8 H0.2 M0.2 Q10. S1. T20. U0.5 V0.5 W2.

Compulsory inputs

Aa	a =	The direction of the probe measurement when measuring from the X+ axis direction.
Dd	d =	The nominal distance to the surface (radial).

Optional inputs

See Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

Example: Measuring an angled surface

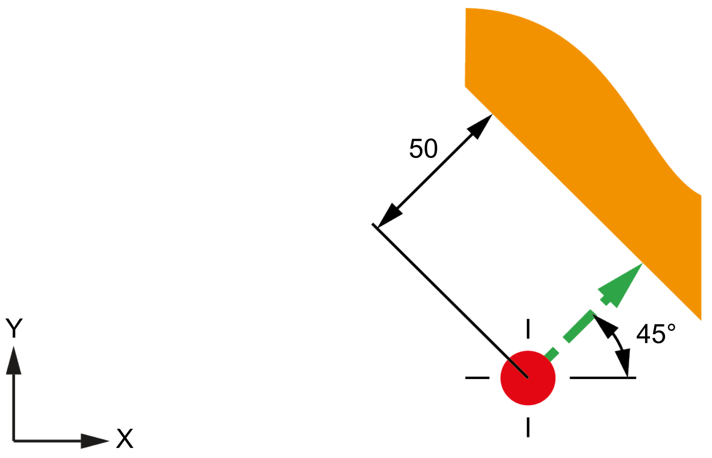


Figure 7.2 Measuring an angled surface

T01 M06	Select the probe.
G54 X−40. Y20.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−8. F3000.	Protected positioning move to the start position.
G65 P9821 A45. D50. T10.	Single surface measurement.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.

The tool radius offset (10) is updated by the error of the surface position.

Angled surface measurement using the X, Y, Z inputs (O9821)

NOTE: Before using this cycle, the probe must have been recently calibrated using either the O9801 K4. or the O9801 K5. cycle to establish the vector stylus radius values (see Chapter 5, “Probe calibration and SupaTouch optimisation”). As the stylus radius values are mapped in the XY plane only, use a RENGAGE™ probe (typically, an OMP400, RMP600 or MP700) with good 3D measuring performance.

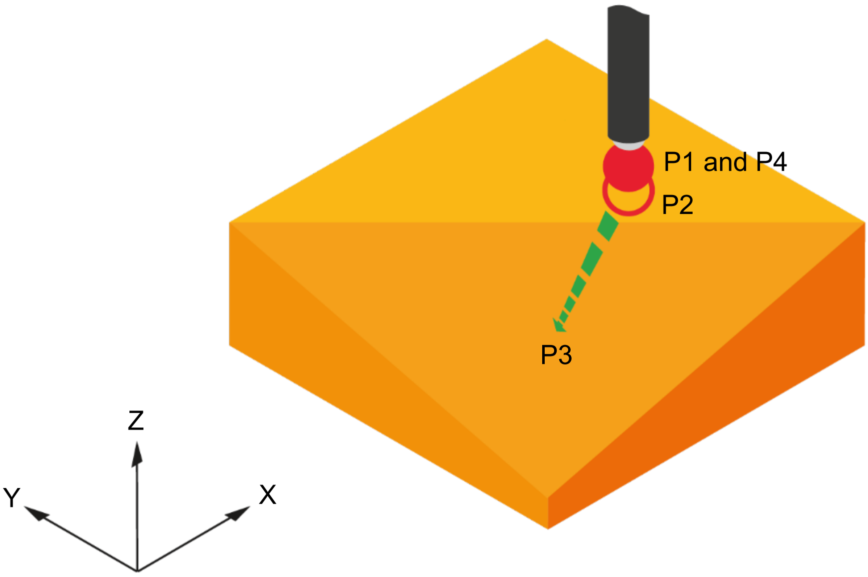


Figure 7.3 Measuring an angled surface

Description

This cycle measures a surface feature using one vectored measuring move along the XY, XZ, YZ or XYZ axis. Prior to the gauging move, the cycle will reposition the stylus ball to compensate for the XY probe offset and, if a Z-axis target position is included in the cycle call-up line, will also reposition the probe to compensate for the stylus ball radius in the Z axis.

When performing an XY plane single surface measurement, do not include a Z-axis target position in the cycle call-up line and the cycle will run at the current Z position.

NOTE: This cycle cannot be used to update the tool offset values.

Application

With the probe and probe offset active, position the probe at a suitable start point so that it will move onto the surface normal to the expected gauging point.

Format

G65 P9821 Xx. Yy. Zz. [Cc. Hh. Mm. Qq. Ww.]

where [ ] denote optional inputs.

Example: G65 P9821 X25. Y25. Z25. C1.

Compulsory inputs

At least one of these inputs is required.

Xx	x =	The X-axis target position.
Yy	y =	The Y-axis target position.
Zz	z =	The Z-axis target position.

Optional inputs

Cc — Used to adjust output values in #124, #125 and #126 (see “Outputs” below) .

NOTE: The C input does not change the cycle movements, printed results (Ww) or the values in #135, #136 and #137, these are always the surface position.

C = 2	#124 – #126 denotes the surface position, also found in #135 – #137.
C = 1	#124 – #126 denotes the stylus centre position. Values are calculated using the surface position, the approach measurement vector and the physical stylus radius (#[#111+19).
No C input	#124 – #126 denotes the spindle centreline position. No XY stylus offset or stylus radius compensation takes place.

For other optional inputs, see Chapter 2, “Optional Inputs”.

Outputs

	No C input	C = 1	C = 2
#124	No adjustment	Adjusted to centre of stylus ball in X	X position
#125	No adjustment	Adjusted to centre of stylus ball in Y	Y position
#126	No adjustment	Adjusted to centre of stylus ball in Z	Z position
#135	X position		
#136	Y position		
#137	Z position		

For other outputs, see Chapter 3, “Cycle outputs”.

Example: XZ approach to the surface at 45°

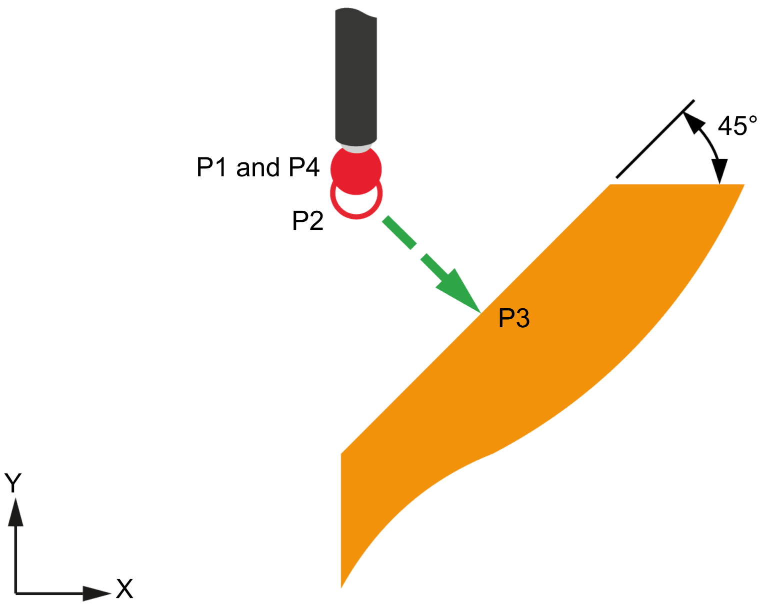


Figure 7.4 XZ surface measurement

T01 M06	Select the probe.
G54 X−5. Y20.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 X−5. Z5. F3000.	**P1**, protected positioning move to the start position (**P2**).
G65 P9821 C2. X10. Z−10.	**P3**, measure the surface and return to **P4** (see the note below).
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.

The surface position for P3 is found and the results are stored in #135, #136 and #137.

NOTE: The Z-axis movement from P1 to P2 is performed automatically to put the centre of the stylus ball on the vectored approach path to P3.

Angled web or pocket measurement (O9822)

NOTE: Before using this cycle, the probe must have been recently calibrated using either the O9801 K4. or the O9801 K5. cycle to establish the vector stylus radius values (see Chapter 5, “Probe calibration and SupaTouch optimisation”).

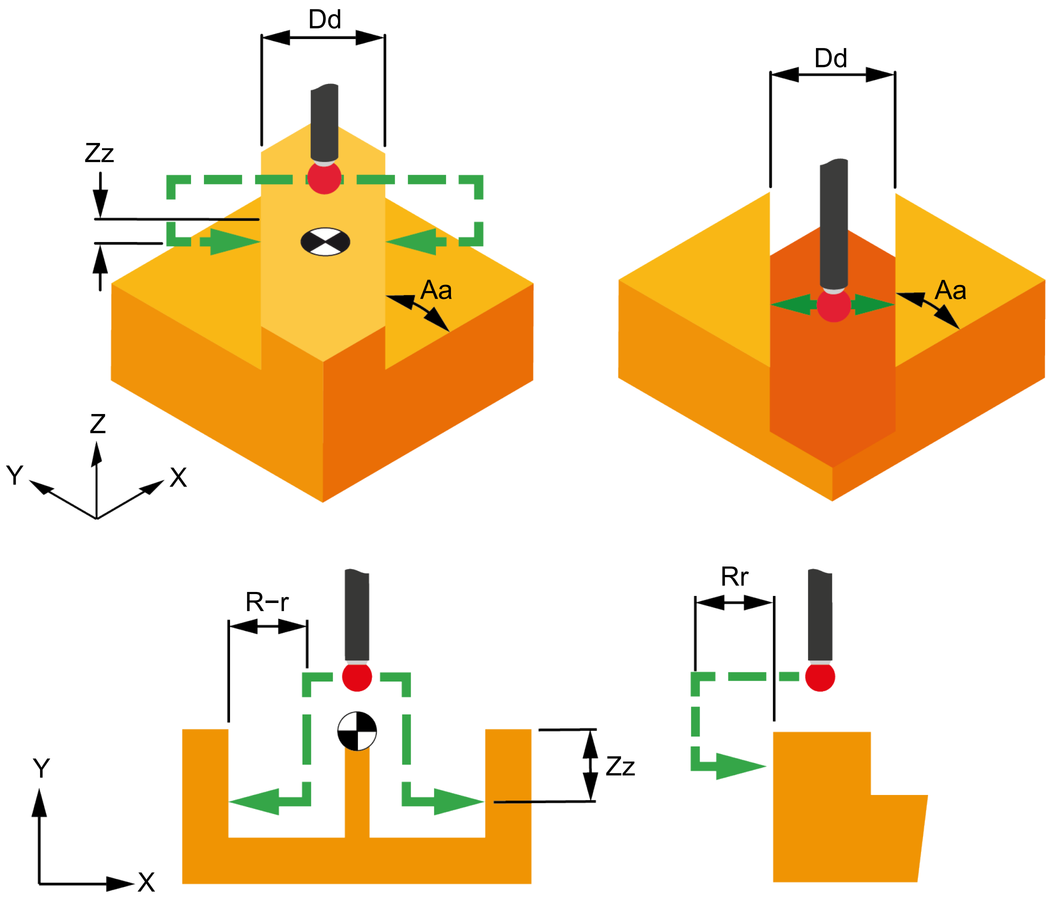


Figure 7.5 Measuring an angled web or pocket

Description

This cycle measures a web or pocket feature using two vectored measuring moves along the XY axis.

Application

With the probe and probe offset active, position the probe to the expected centre line of the feature and at a suitable position in the Z axis. Run the cycle with suitable inputs.

Format

G65 P9822 Aa. Dd. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

or

G65 P9822 Aa. Dd. Zz. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv. Ww.]

where [ ] denote optional inputs.

Example: G65 P9822 A45.005 D50.005 Z50. E21. F0.8 H0.2 M0.2 Q10. R10. S1. T20. U0.5 V0.5 W2.

Compulsory inputs

Aa	a =	The angle of the surface to be measured from the X+ axis direction.
Dd	d =	The nominal size of the feature.
Zz	z =	The absolute Z-axis position when measuring a web feature. If this is omitted, a pocket cycle is assumed.

Optional inputs

Rr	r =	This can be used as shown in the diagrams above to pre-position before each measurement. It can also be used for an internal pocket cycle using an R+ input (and no Zz input). The fast pre-positioning will improve cycle time on large pockets, but will produce an alarm if the probe stylus is triggered during pre-positioning. **Default:** Pocket cycle with no fast pre-positioning.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

Example: Measuring an angled web

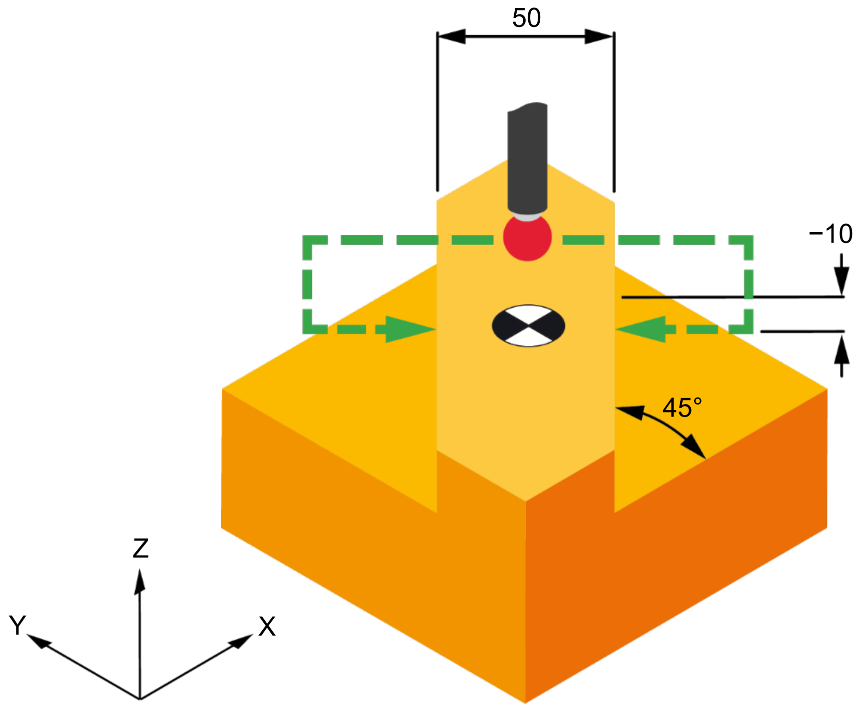


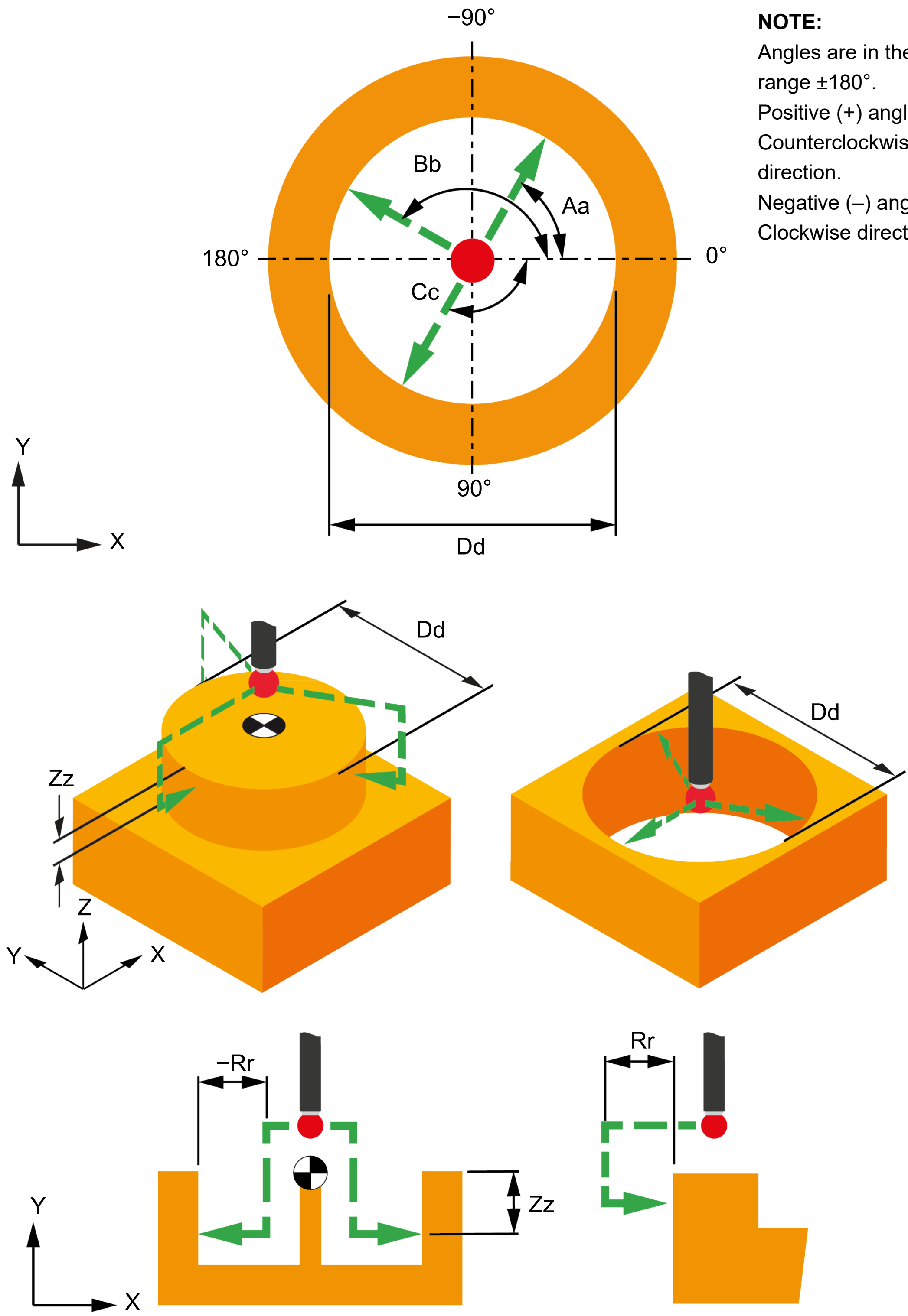
Figure 7.6 Measuring an angled web

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move.
G65 P9822 A45. D50. Z−10. S2.	Measure a 50 mm (1.9685 in) wide web at 30°.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
continue	

The centre line of the feature in the X axis is stored in work offset S02 (G55).

3-point bore or boss measurement (O9823)

NOTE: Before using this cycle, the probe must have been recently calibrated using either the O9801 K4. or the O9801 K5. cycle to establish the vector stylus radius values (see Chapter 5, “Probe calibration and SupaTouch optimisation”).



NOTE:
Angles are in the range ±180°.
Positive (+) angle: Counterclockwise direction.
Negative (–) angle: Clockwise direction.

Figure 7.7 3-point bore or boss measurement

Description

This cycle measures a bore or boss feature using three vectored measuring moves along the XY axis.

Application

With the probe and probe offset active, position the probe to the expected centre line of the feature and at a suitable position in the Z axis. Run the cycle with suitable inputs.

Format

G65 P9823 Aa. Bb. Cc. Dd. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv Ww]

or

G65 P9823 Aa. Bb. Cc. Dd. Zz. [Ee. Ff. Hh. Mm. Qq. Rr. Ss. Tt. Uu. Vv Ww]

where [ ] denote optional inputs.

Example: G65 P9823 A45.005 B150. C35.005 D50.005 Z50. E21. F0.8 H0.2 M0.2 Q10. R10. S1. T20. U0.5 V0.5 W2.

Compulsory inputs

Aa	a =	The first angle for vector measurement, measured from the X+ axis direction.
Bb	b =	The second angle for vector measurement, measured from the X+ axis direction.
Cc	c =	The third angle for vector measurement, measured from the X+ axis direction.
Dd	d =	The nominal size of the feature.
Zz	z =	The absolute Z-axis position when measuring a boss feature. If this is omitted, a bore cycle is assumed.

Optional inputs

Rr	r =	This can be used as shown in the diagrams above to pre-position before each measurement. It can also be used for an internal bore cycle using an R+ input (and no Zz input). The fast pre-positioning will improve cycle time on large bores, but will produce an alarm if the probe stylus is triggered during pre-positioning. **Default:** Bore cycle with no fast pre-positioning.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

Example: 3-point bore measurement (referred datum)

T01 M06	Select the probe.
G54 X100. Y100.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−10. F3000.	Protected positioning move.
G65 P9823 D30. A30. B150. C−90. S2.	Measure a 30 mm (1.181 in) diameter bore.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
continue	

The error of the centre line is referred to the datum point X0,Y0. The revised X0,Y0 position is set in work offset 02 (G55).

This page is intentionally left blank.

Chapter 8

Additional cycles

The Inspection Plus software contains a number of cycles that cannot be described under the headings used in previous chapters of this manual (see chapters 4 to 7 inclusive). This chapter describes how to use these cycles.

Before starting, check that the cycles are available on the machine, as the full suite of cycles may not have been installed.

Contained in this chapter

4th axis measurement (O9818)

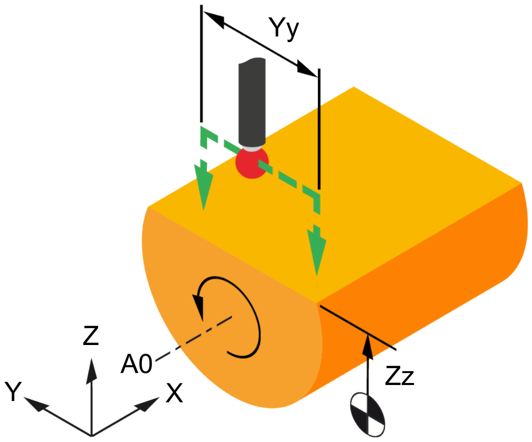


Figure 8.1 4th axis measurement (axis parallel to the Y axis) (using K1. or no Kk input)

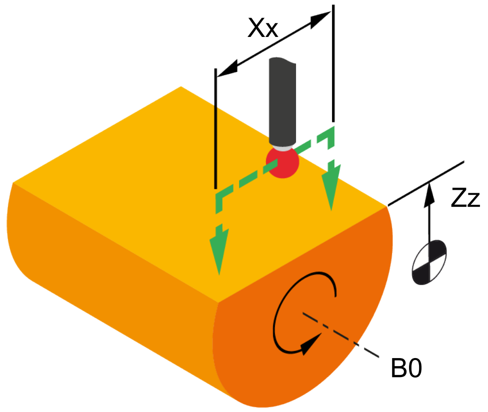


Figure 8.2 4th axis measurement (axis parallel to the X axis)

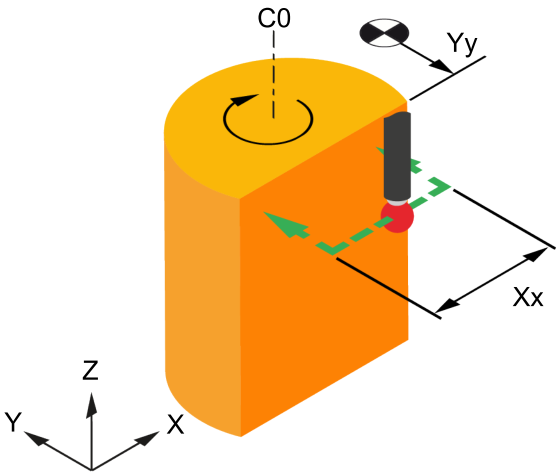


Figure 8.3 4th axis measurement (axis parallel to the X axis) (using K3. input)

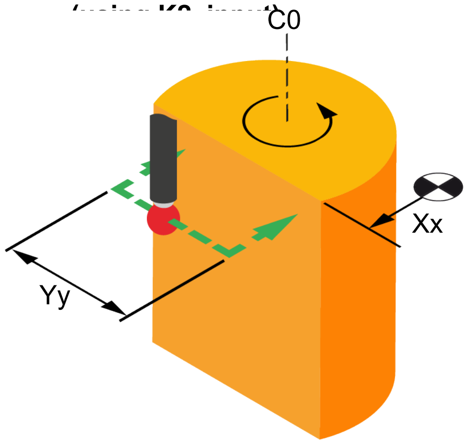


Figure 8.4 4th axis measurement (axis parallel to the Y axis) (using K4. input)

NOTE:

Angle correction to the 4th axis:

Positive (+) angle: Counterclockwise direction.

Negative (−) angle: Clockwise direction.

Description

This cycle is used to find the slope of a surface between two points; for example, Z1 and Z2. The 4th axis can then be rotated to compensate for the surface error.

It will compensate for the error with the 4th rotary axis in any of the orientations shown in Figures 8.1, 8.2, 8.3 and 8.4 above.

Application

Position the rotary axis to the expected angular position of the feature (for example, the surface normal to the Z axis). If the Ss input is used, the work offset register is adjusted by the error amount.

NOTE: To make the new work offset active on most machines, it is normally necessary to restate the work offset and move to the angular position after the cycle.

Format

K1. (A-axis setting)

G65 P9818 Yy. Zz. [Kk. Qq. Bb. Ss. Ww.]

where [ ] denote optional inputs.

Example: G65 P9818 Y100. Z50. K1. Q10. B2. S1. W2.

K2. (B-axis setting)

G65 P9818 Xx. Zz. Kk. [Qq. Bb. Ss. Ww.]

where [ ] denote optional inputs.

Example: G65 P9818 X100. Z50. K2. Q10. B2. S1. W2.

K3. (C-axis setting)

G65 P9818 Xx. Yy. Kk. [Qq. Bb. Ss. Ww.]

where [ ] denote optional inputs.

Example: G65 P9818 X100. Y50. K3. Q10. B2. S1. W2.

K4. (C-axis setting)

G65 P9818 Xx. Yy. Kk. [Qq. Bb. Ss. Ww.]

where [ ] denote optional inputs.

Example: G65 P9818 X50. Y100. K4. Q10. B2. S1. W2.

Compulsory inputs

K1. (A-axis setting)

K1.		Select the orientation of the rotary axis (in this case, the A axis).
Yy	y =	The distance between the Z1 and Z2 measurement positions in the Y axis.
Zz	z =	The expected surface position in the Z axis.

NOTE: This is also the default orientation if no Kk input is used.

K2. (B-axis setting)

K2.		Select the orientation of the rotary axis (in this case, the B axis).
Xx	x =	The distance between the Z1 and Z2 measurement positions in the X axis.
Zz	z =	The expected surface position in the Z axis.

K3. (C-axis setting)

K3.		Select the orientation of the rotary axis (in this case, the C axis).
Xx	x =	The distance between the Y1 and Y2 measurement positions in the X axis.
Yy	y =	The expected surface position in the Y axis.

K4. (C-axis setting)

K4.		Select the orientation of the rotary axis (in this case, the C axis).
Xx	x =	The expected surface position in the X axis.
Yy	y =	The distance between the X1 and X2 measurement positions in the Y axis.

Optional inputs

Bb	b =	Set a tolerance on the angular position of the feature. It is equal to half the total tolerance. **Example:** With a component dimension of 45° ±0.25° the 4th axis will be positioned to 45° and B0.25 tolerance.

For other optional inputs, see Chapter 2 “Optional inputs”.

Outputs

#139 The measured position of the 4th axis.

#143 The [Z1−Z2] value (or the [Y1−Y2] value).

#144 The angle correction value.

Customising cycle O9818

Different machines and applications may require the 4th axis system variable number and update direction to be changed. Edit cycle O9818 when it is installed to suit your machine.

Changing the axis number

#3 = 4 (_AXIS*NO)

Change the #3 value as required for each axis to be used. See the (A-AXIS), (B-AXIS) and (C-AXIS) commented sections in the cycle.

Changing the axis update direction

#4 = 1 (_1=CW*-1=CCW*UPDATE)

Change the #4 value as required for each axis to be used. See the (A-AXIS), (B-AXIS) and (C-AXIS) commented sections in the cycle.

Example 1: K2. – setting the B axis to a milled flat

T01 M06	Select the probe.
G43 H1 Z200.	Activate offset 1 and position above the surface.
G65 P9832	Switch on the probe (this includes M19).
G0 B45.	Position the B axis to 45°.
G65 P9810 X0. Y0. Z20. F3000.	Position 10 mm (0.394 in) above the surface.
G65 P9818 X50. Z10. K2. S1. B5.	Measure at 50 mm (1.9685 in) centres, update G54 and set a tolerance of 5°.
G65 P9810 Z200.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z200.	Reference return.
continue	

Example 2: K3. – setting the C axis to a milled flat

T01 M06	Select the probe.
G43 H1 Z200.	Activate offset 1 and position above the surface.
G65 P9832	Switch on the probe (this includes M19).
G0 C−90.	Position the C axis to 90°.
X−40. Y−70.	
G65 P9810 Z−10. F3000.	Position 10 mm (0.394 in) below the surface.
G65 P9818 X50. Y−50. K3. S1. B5.	Measure at 50 mm (1.9685 in) centres, update G54 and set a tolerance of 5°.
G65 P9810 Z200.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z200.	Reference return.
continue	

Bore/boss on PCD measurement (O9819)

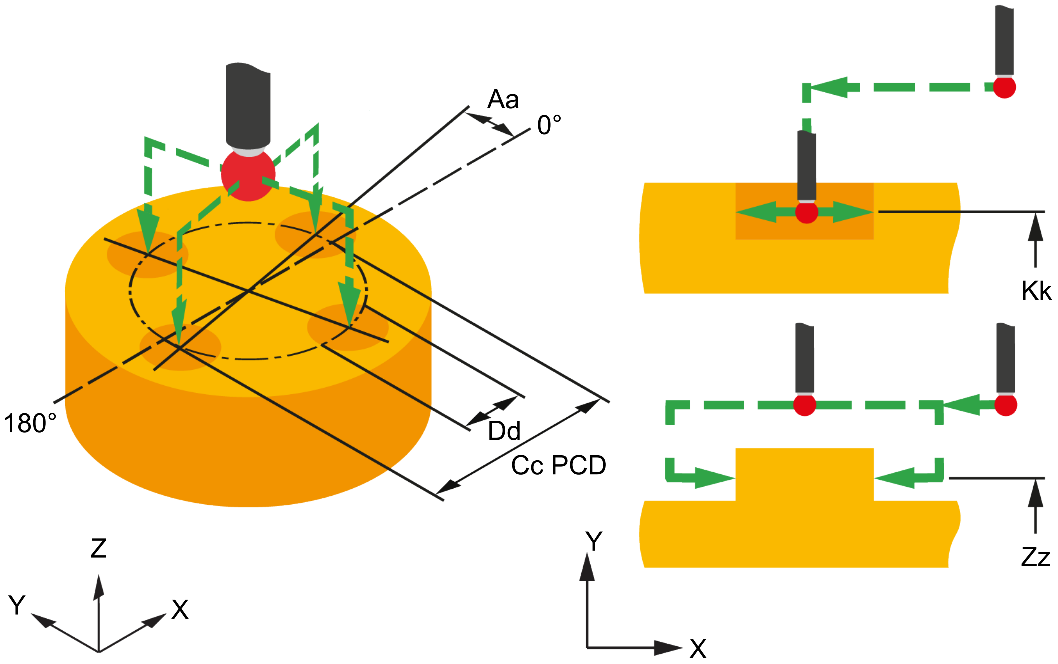


Figure 8.5 Bore/boss on PCD measurement

Description

NOTE: This cycle requires an additional cycle nesting level to other cycles included in this package. This is because it has an embedded call to the O9814 cycle.

The cycle measures a series of bores or bosses on a pitch circle diameter (PCD). All probe moves occur automatically and return to the start position at the centre of the PCD.

Application

1. Position the probe at the centre of the PCD above the component. The probe moves to each of the bore/boss features and measures each one automatically. At the end of the cycle it then returns to the PCD centre.

2. The cycle makes use of the bore/boss cycle which is nested within the moves. The cycle nesting level is four deep, which means that this cycle cannot be nested inside a customer cycle.

3. If a “probe-open” condition occurs during any of the moves between bore/boss features, a “PATH OBSTRUCTED” alarm occurs. The probe then stays in position instead of returning to the start position as is usual. This is done for safety reasons because the return path to the centre line of the PCD may be obstructed.

Format

Boss: G65 P9819 Cc. Dd. Zz. [Aa. Bb. Hh. Mm. Qq. Rr. Ww.]

or

Bore: G65 P9819 Cc. Dd. Kk. [Aa. Bb. Hh. Mm. Qq. Rr. Ww.]

where [ ] denote optional inputs.

Example: G65 P9819 C28.003 D50.005 K11. A45.005 B2. H0.2 M0.2 Q10. R10. W2.

Compulsory inputs

Cc	c =	The pitch circle diameter (PCD) of the bore/boss feature.
Dd	d =	The diameter of the bore/boss.
Kk	k =	The absolute Z-axis position at which the bore is measured.
Zz	z =	The absolute Z-axis position at which the boss is measured.

Optional inputs

Aa	a =	The angle measured from the X axis to the first bore/boss feature. **Default value:** 0.
Bb	b =	The number of bore/boss features on the PCD. **Default value:** 1.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

The feature measurements are stored in variables #135 to #149 (see Chapter 3, “Cycle outputs”).

The data listed below is output to the printer. For details of the print program output format, see Chapter 11, “General information”.

- The diameter of each bore/boss.
- The XY absolute position, angle position and pitch circle diameter of each feature.
- The feature number.
- The error of the size and position.

Example: Measuring four holes on a PCD

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z10. F3000.	Protected positioning move to 10 mm (0.394 in) above the plate.
G65 P9819 A45. B4. C100. D16. K−10.	Measure four 16 mm (0.630 in) diameter holes starting at 45°.
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.

continue

Stock allowance (O9820)

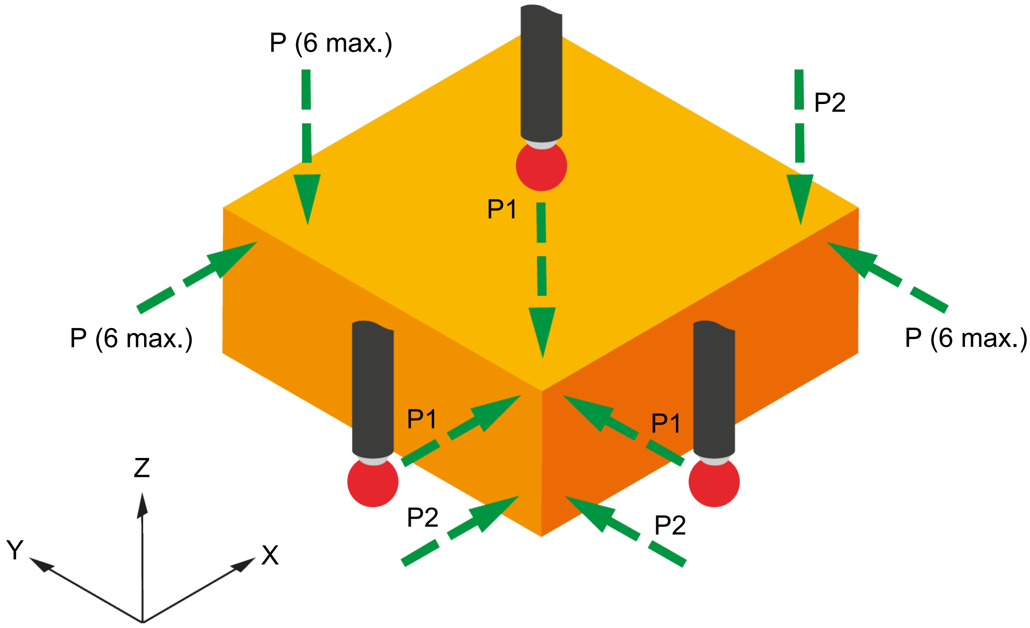


Figure 8.6 Measuring the stock allowance

Description

The cycle measures an X or Y or Z surface at defined positions to establish the maximum and minimum stock condition of the surface.

Application

The probe is positioned above the surface at the first measuring position (P1). The cycle measures the surface at this position. Additional points (P2 to P6 maximum) are measured as defined, depending on the number of sets of **I**, J, or K inputs.

NOTES:

1. When a work offset is set, the surface position is at the minimum measured position and the stock value is seen in #146.
2. When a work offset is **not** set, the nominal position is assumed and the maximum and minimum values are seen in #144 and #145 respectively.

Format

X-surface measure

G65 P9820 Xx. [Jj. Kk. Ss. Uu. Qq.]

NOTE: Successive pairs of Jj and Kk values must be in order for P2 to P6.

or

Y-surface measure

G65 P9820 Yy. [Ii. Kk. Ss. Uu. Qq.]

NOTE: Successive pairs of Ii and Kk values must be in order for P2 to P6.

or

Z-surface measure

G65 P9820 Zz. [Ii. Jj. Ss. Uu. Qq.]

NOTE: Successive pairs of Ii and Jj values must be in order for P2 to P6.

where [ ] denote optional inputs.

Example: G65 P9820 X100. J10. K11. S1. U0.5 Q5.
(Note that this shows one additional pair of [JK] values, i.e. P2.)

Compulsory inputs

Xx or Yy or Zz	x, y, z =	The nominal surface position for checking the stock allowance.

Optional inputs

I1 (P2) to I5 (P6)	i =	The X surface positions for P2 to P6 (a maximum of five additional positions).
J1 (P2) to J5 (P6)	j =	The Y surface positions for P2 to P6 (a maximum of five additional positions).
K1 (P2) to K5 (P6)	k =	The Z surface positions for P2 to P6 (a maximum of five additional positions).

Uu u = The maximum stock allowance or upper tolerance (metal condition).

1. Uu input only.
   The upper tolerance metal condition. For example, the surface is at 30 +2/−0
   G65 P9820 Z30. U2. Ii. Jj.
2. Uu and Ss inputs.
   The maximum stock allowance.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

With the Uu input only	Upper tolerance exceeded. Flag #148 is set to 3.
With the Uu and Ss inputs	Excess stock. Flag #148 is set to 6.
#144	The maximum value (metal condition).
#145	The minimum value (metal condition).
#146	The variation (stock allowance).

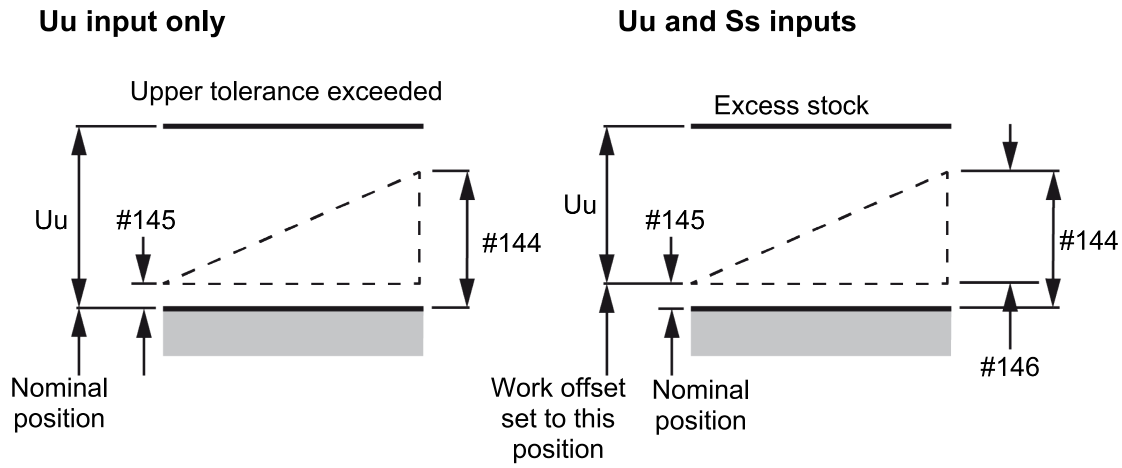


Figure 8.7 Outputs for the stock allowance cycle

Example 1: Checking a Z surface for stock variation

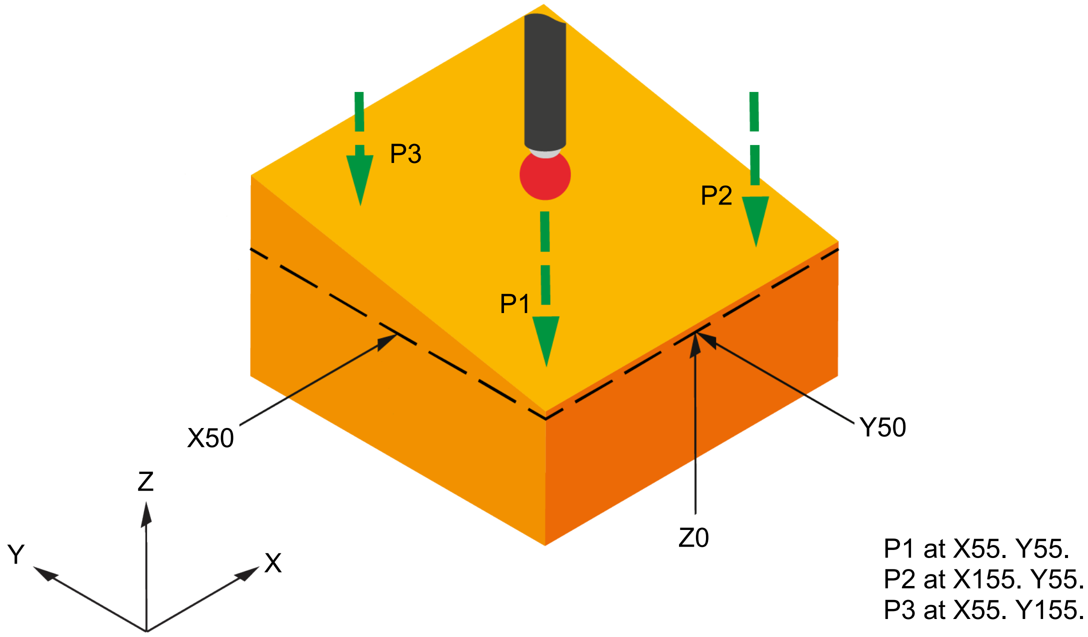


Figure 8.8 Checking a Z surface for stock variation

Select the probe.

G65 P9810 X55. Y55. Z20. F3000.	Protected positioning move to P1.
G65 P9820 Z0. I155. J55. I55. J155. U2.	Measure at P1, P2 and P3 to set a 2 mm (0.039 in) tolerance.

continue

Example 2: Checking a Y surface and updating a work offset

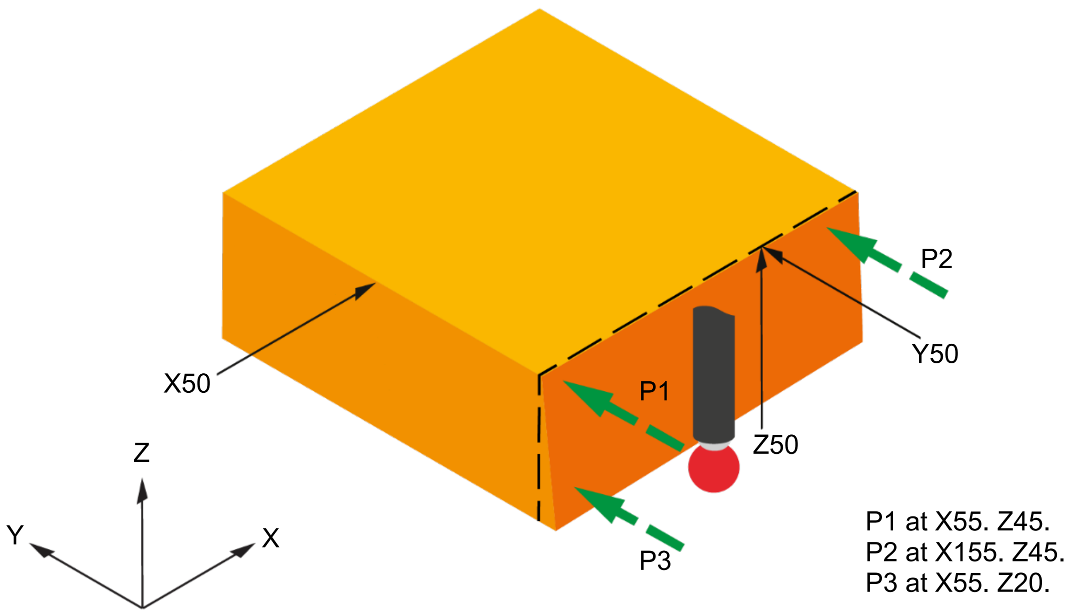


Figure 8.9 Checking a Y surface and updating a work offset

Select the probe.

G65 P9810 X55. Y40. Z45. F3000.	Protected positioning move to P1.
G65 P9820 Y50. I155. K45. I55. K20. S2	Measure at P1, P2 and P3 to set the Y-axis work offset G55 to the minimum stock position at program position Y50.
	Retract, select the tool and work offset G55 for machining the Y surface at the new Y50. surface position.

Multiple probe use

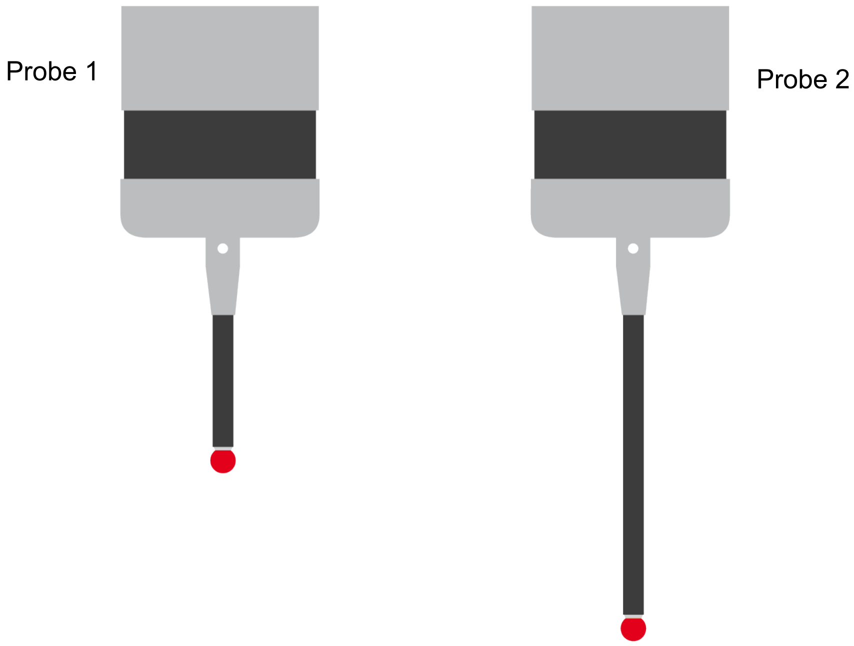


Figure 8.10 Storing multi-stylus data

Description

NOTE: This option is not enabled as standard. See Chapter 10, “Configuration”.

Up to four probes can be used with this software. This is to cater for the possibility of having similar probes but with different styli, or combinations of probes with different probe start requirements, for example an “optical on/off” and a “spin on/off” probe combination.

Application

CAUTION: It is possible for the T number to be linked to the H number, in which case just performing a tool change will enable the use of multiple probes. However, the system must be configured to do this, and it must not be assumed that this has been done.

When using multiple probes, the use of probe offset number (H) for the probe in the spindle is now restricted to those numbers which are pre-defined during the multiple probe support installation procedure.

The available probes and their associated probe offset (H) numbers must be known. The H numbers are always associated with a particular probe. It is only necessary to bring the correct probe into the spindle and activate the associated offset number (H) for that probe – the multi-probe application is then ready to use. The software will activate the correct probe data and start the probe.

In every other way the cycles will run exactly as described in this manual.

Determining feature-to-feature data in the XY plane (O9834)

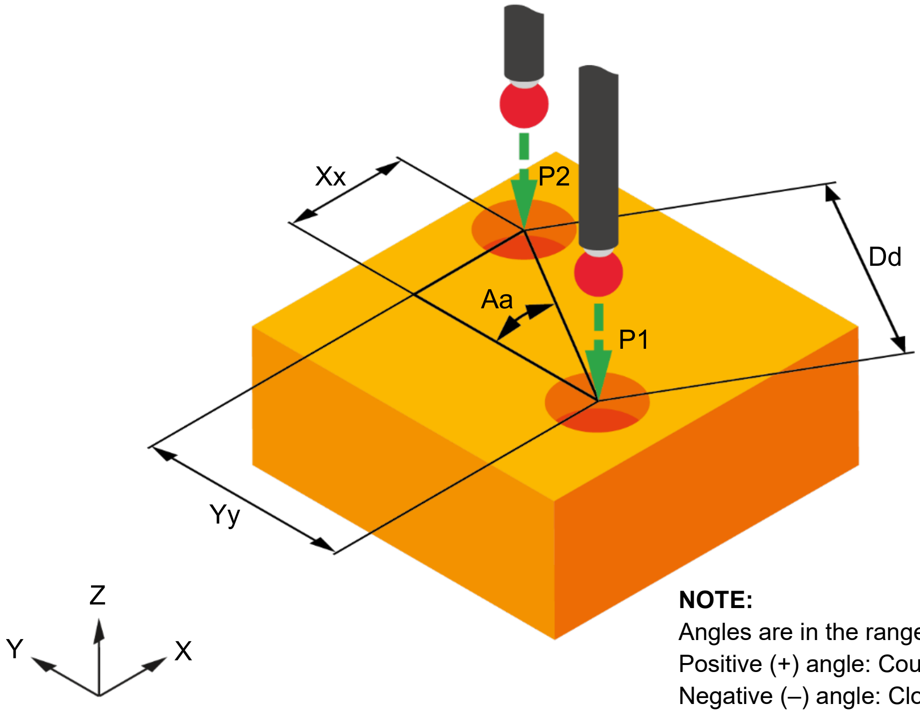


Figure 8.11 Determining feature-to-feature data in the XY plane

Description

This is a no movement cycle that is used in conjunction with two measuring cycles to determine feature-to-feature data.

Application

The first measuring cycle is run and the data is stored in variables #135 to #139 as normal.

Programming G65 P9834 without any inputs has the effect of copying the data from these variables into variables #130 to #134 for P1.

Values for P2 are obtained by running a second measuring cycle which stores new data in variables #135 to #139.

NOTE: The order of P1 and P2 is important because the data calculated is that of P2 with respect to P1.

The feature-to-feature data is established by programming G65 P9834 with suitable inputs after the second measuring cycle, to compare the data for P1 (in variables #130 to #134) with the data for P2 (in variables #135 to #139).

Format

G65 P9834 Xx. [Ee. Ff. Hh. Mm. Ss. Tt. Uu. Vv. Ww.]
or
G65 P9834 Yy. [Ee. Ff. Hh. Mm. Ss. Tt. Uu. Vv. Ww.]
or
G65 P9834 Xx. Yy. [Bb. Ee. Hh. Mm. Ss. Uu. Ww.]
or
G65 P9834 Aa. Dd. [Bb. Ee. Hh. Mm. Ss. Uu. Ww.]
or
G65 P9834 (with no inputs).

where [ ] denote optional inputs.

Examples: G65 P9834 X100. E21. F0.8 H0.2 M0.2 S1. T20. U0.5 V0.5 W2.

or

G65 P9834 Y100. E21. F0.8 H0.2 M0.2 S1. T20. U0.5 V0.5 W2.

or

G65 P9834 X100. Y100. B2. E21. H0.2 M0.2 S1. U0.5 W2.

or

G65 P9834 A45.005 D50.005 B2. E21. H0.2 M0.2 S1. U0.5 W2.

NOTES:

1. Updating a tool offset with the T input is possible only if O9811 is used for the P2 data. Otherwise a T INPUT NOT ALLOWED alarm results.
2. This cycle cannot be used in conjunction with the web/pocket cycle O9812.
3. Angles. The XY plane is with respect to the X+ axis direction. Use angles in the range ±180°.
4. When G65 P9834 (without any inputs) is used, data is copied as follows:

from	#135	to	#130
	#136		#131
	#137		#132
	#138		#133
	#139		#134

Compulsory inputs

Aa	a =	The angle of P2 with respect to P1 when measured from the X+ axis (angles are between ±180°).
Dd	d =	The minimum distance between P1 and P2.
Xx	x =	The nominal incremental distance in the X axis.
Yy	y =	The nominal incremental distance in the Y axis.
(no inputs)		This is used to store output data of the last cycle for P1 data.

Optional inputs

See Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

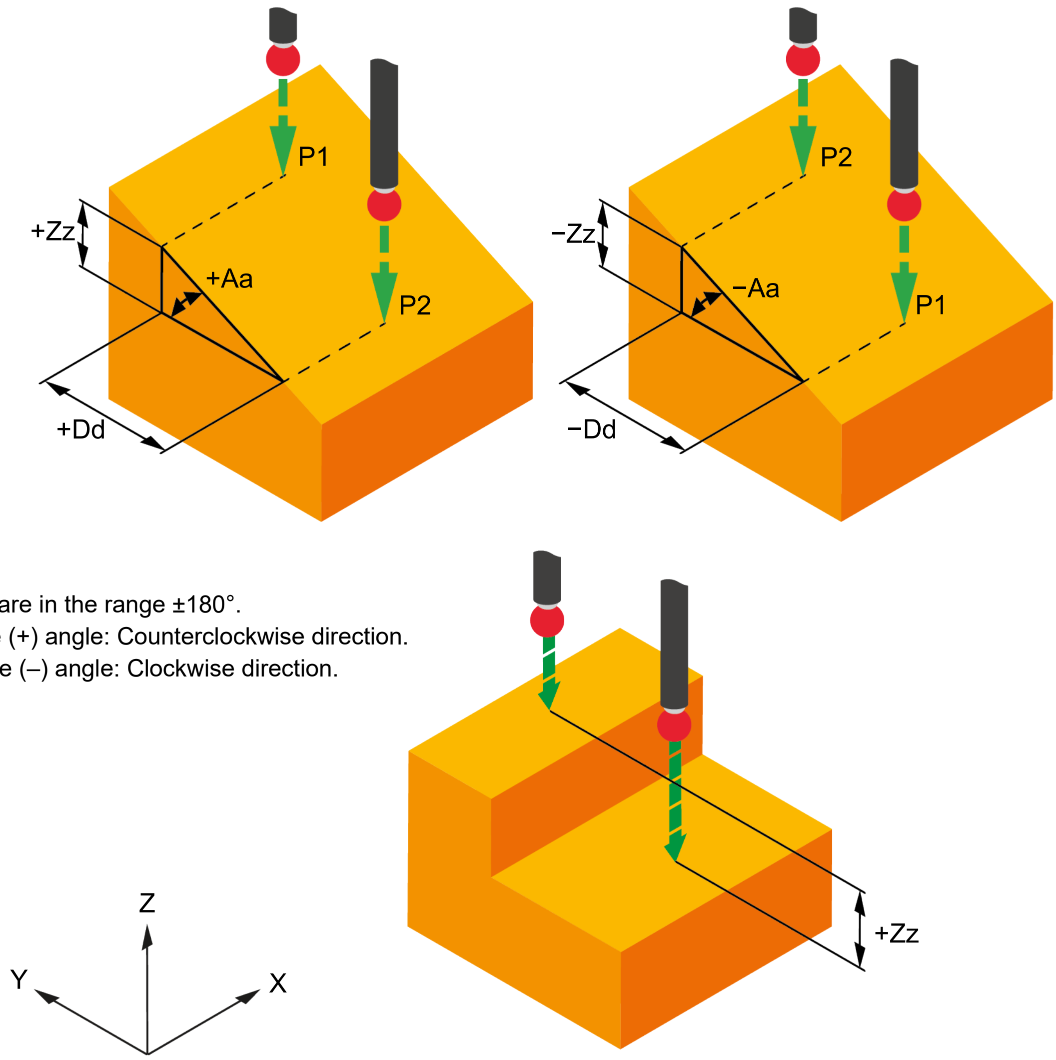
Example 1: Measuring the incremental distance between two holes

G65 P9810 X30. Y50. F3000.	Protected positioning move.
G65 P9810 Z−10.	Protected positioning move.
G65 P9814 D20.	P1 20 mm (0.787 in) bore.
G65 P9834	Store the data.
G65 P9810 Z10.	Protected positioning move.
G65 P9810 X80. Y78.867	Move to the new position.
G65 P9810 Z−10.	Protected positioning move.
G65 P9814 D30.	P2 30 mm (1.181 in) bore.
and either	
G65 P9834 X50. Y28.867 M0.1	Incremental distance measurement with 0.1 mm (0.0039 in) true position tolerance.
or	
G65 P9834 A30. D57.735 M0.1	

Example 2: Measuring a surface to bore

G65 P9810 X10. Y50. F3000.	Protected positioning move.
G65 P9810 Z−10.	Protected positioning move.
G65 P9811 X0.	P1 at X0 mm (0 in) position.
G65 P9834	Store the data.
G65 P9810 Z10.	Protected positioning move.
G65 P9810 X−50.	Move to the new position.
G65 P9810 Z−10.	Protected positioning move.
G65 P9814 D20.5	P2 20.5 mm (0.807 in) bore.
G65 P9834 X−50. H.2	Measure the distance −50 mm (−1.97 in).

Determining feature-to-feature data in the Z plane (O9834)



NOTE:
Angles are in the range ±180°.
Positive (+) angle: Counterclockwise direction.
Negative (–) angle: Clockwise direction.

Figure 8.12 Determining feature-to-feature data in the Z plane

Description

This is a no movement cycle that is used in conjunction with two measuring cycles to determine feature-to-feature data.

Application

The first measuring cycle is run and the data is stored in variables #135 to #139 as normal.

Programming G65 P9834 without any inputs has the effect of copying the data from these variables into variables #130 to #134 for P1.

Values for P2 are obtained by running a second measuring cycle which stores new data in variables #135 to #139.

NOTE: The order of P1 and P2 is important because the data calculated is that of P2 with respect to P1.

The feature-to-feature data is established by programming G65 P9834 with suitable inputs after the second measuring cycle, to compare the data for P1 (in variables #130 to #134) with the data for P2 (in variables #135 to #139).

Format

G65 P9834 Zz. [Ee. Ff. Hh. Mm. Ss. Tt. Uu. Vv. Ww.]

or

G65 P9834 Aa. Zz. [Bb. Ww.]

or

G65 P9834 Dd. Zz. [Bb. Ww.]

or

G65 P9834 (with no inputs)

where [ ] denote optional inputs.

Examples: G65 P9834 Z50. E21. F0.8 H0.2 M0.2 S1. T20. U0.5 V0.5 W2.

or

G65 P9834 A45.005 Z50. B2. W2.

or

G65 P9834 D50.005 Z50. B2. W2.

or

G65 P9834 (with no inputs)

NOTES:

1. Updating a tool offset with the Tt input is possible only if O9811 is used for the P2 data. Also the Aa and Zz and Dd and Zz inputs cannot be used when updating tool offsets as this suggests an angled surface and a T INPUT NOT ALLOWED alarm results.
2. Angles. These are with respect to the XY plane. Use angles in the range ±180°.
3. When G65 P9834 (without any inputs) is used, data is copied as follows:

from	#135	to	#130
	#136		#131
	#137		#132
	#138		#133
	#139		#134

Inputs

Aa and Zz, or Dd and Zz inputs

1. The +Dd/−Dd values should be used to indicate the direction of P2 with respect to P1.

2. Angles are between ±180°.

3. A positive Aa (+Aa) angle is in the counter-clockwise direction.

Zz input only

The +Zz/−Zz values should be used to indicate the direction of P2 with respect to P1.

Compulsory inputs

Zz	z =	The nominal incremental distance in the Z axis.
and		
Aa	a =	The angle of P2 with respect to P1 measured from the XY plane (angles are between ±180°).
Zz	z =	The nominal incremental distance in the Z axis.
and		
Dd	d =	The minimum distance between P1 and P2 measured in the XY plane.
Zz	z =	The nominal incremental distance in the Z axis.
or		
(No inputs)		This is used to store output data of the last cycle for P1 data.

Optional inputs

See Chapter 2, “Optional inputs”.

Outputs

See Chapter 3, “Cycle outputs”.

Example 1: Measuring the incremental distance between two surfaces

G65 P9810 X30. Y50. F3000.	Protected positioning move.
G65 P9810 Z30.	Protected positioning move.
G65 P9811 Z20.	P1 20 mm (0.787 in) surface.
G65 P9834	Store the data.
G65 P9810 X50.	Move to the new position.
G65 P9811 Z15.	P2 15 mm (0.591 in) surface.
G65 P9834 Z−5. H.1	The feature to feature is at −5 mm (−0.197 in).

Example 2: Measuring an angled surface

G65 P9810 X30. Y50. F3000.	Protected positioning move.
G65 P9810 Z30.	Protected positioning move.
G65 P9811 Z20.	P1 at the 20 mm (0.787 in) position.
G65 P9834	Store the data.
G65 P9810 X57.474	Move to the new position.
G65 P9811 Z10.	P2 at the 10 mm (0.394 in) position.

and either

G65 P9834 D27.474 Z−10. B.5	Measure the slope between points P2 and P1 with an angle tolerance of ±0.5°.

or

G65 P9834 A−20. Z−10. B.5	Measure the slope of −20° (in the clockwise direction) with an angle tolerance of ±0.5°.

Updating the statistical process control (SPC) tool offset (O9835)

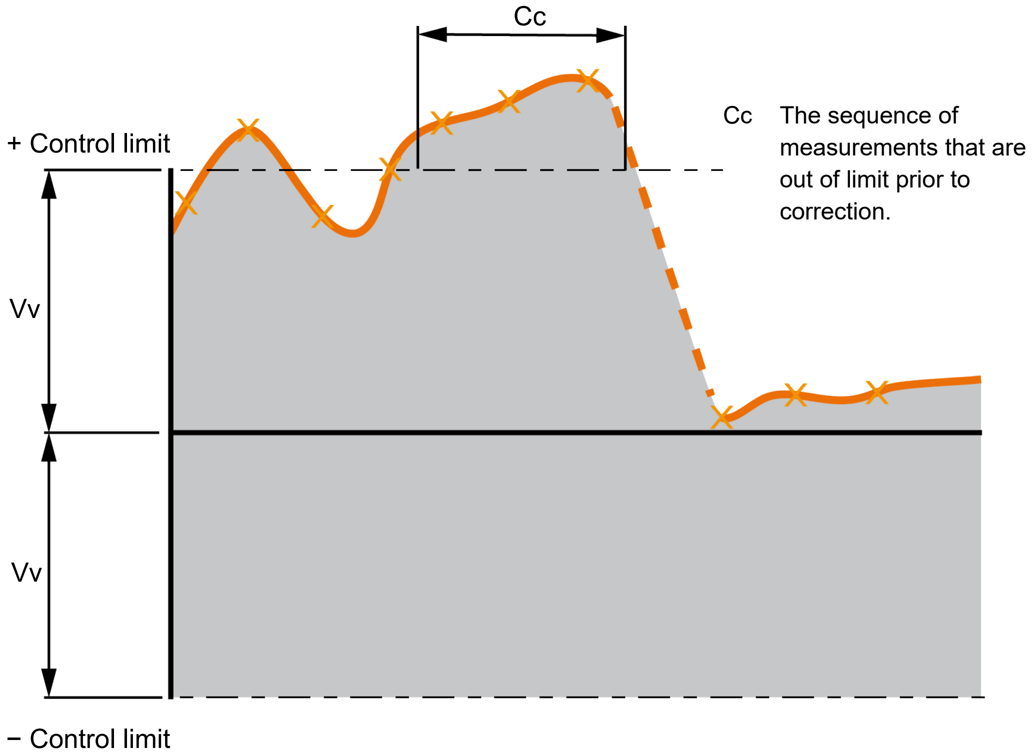


Figure 8.13 Updating the SPC tool offset

Description

This cycle can be used in conjunction with measuring cycles to control the updating of tool offsets. An update is based on the average value of a sample of measurements.

Application

A measuring cycle should be run with no tool offset update (Tt input). A component tolerance (Hh input) can be used if required.

The SPC cycle should follow. An average value is accumulated until a specified continuous sequence of values is outside the control limit. At this point the tool offset is updated, based on the average value.

IMPORTANT: Before using this cycle, set the Mm store tool offsets to 0 on the offset page.

Format

G65 P9835 Tt. Mm. [Vv. Cc. Ff. Zz.]

where [ ] denote optional inputs.

Example: G65 P9835 T20. M31. V0.25 C4. F0.8 Z1.

Compulsory inputs

Mm	m =	The spare tool offset pair that is used for storing the average value and counter. m = Accumulated average value store location. m+1 = Counter store location.
Tt	t =	The tool offset number to be updated.

Optional inputs

Cc	c =	The number of measurements that are out of tolerance before corrective action is taken. **Default value:** 3.
Zz	z =	The flag for Z length offset updating. A Z value needs to be input to force the length offset register to be updated, otherwise the radius register will be updated.

NOTE: This input is necessary only when using the Type C tool offset option.

For other optional inputs, see Chapter 2, “Optional inputs”.

Example: Updating an SPC tool offset

From previous

G65 P9814 D50. H0.5	Measure a bore to 0.5 mm (0.0197 in) tolerance.
G65 P9835 T30. M31. V0.1 C4.	T30. = The tool offset number for updating. M31. = Spare tool offsets pair (31 and 32). V0.1 = Control limit. C4. = Sequence of measurements that are out of limit.

continue

Angle measurement in the X or Y plane (O9843)

NOTE:
Angles are in the range ±180°.
Positive (+) angle: Counterclockwise direction.
Negative (−) angle: Clockwise direction.

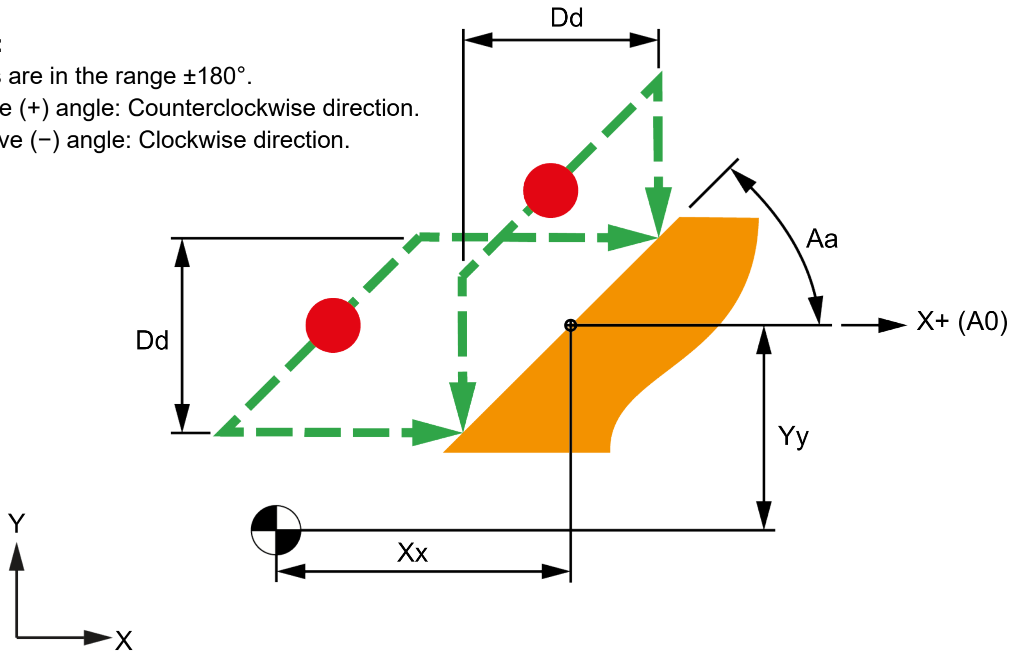


Figure 8.14 Measuring an angled surface in the X or Y plane

Description

This cycle measures an X-axis or Y-axis surface at two positions to establish the angular position of the surface.

Application

To provide a suitable start position, the stylus is positioned adjacent to the surface and at the required Z-axis position. The cycle makes two measurements, symmetrically about the start position, to establish the surface angle.

Format

G65 P9843 Xx. Dd. [Aa. Bb. Qq. Ww. Zz.]
or
G65 P9843 Yy. Dd. [Aa. Bb. Qq. Ww. Zz.]

where [ ] denote optional inputs.

Example: G65 P9843 X50. D30. A45. B0.2 Q15. W1. Z10.

Compulsory inputs

Dd	d =	The distance moved parallel to the X axis or Y axis between the two measuring positions.
Xx	x =	The mid-point position of the surface. An Xx input results in a cycle measuring in the X-axis direction.
Yy	y =	The mid-point position of the surface. A Yy input results in a cycle measuring in the Y-axis direction.

NOTE: Do not mix the Xx and Yy inputs.

Optional inputs

Aa	a =	The nominal angle of the surface. Angles are in the range ±180° and measured from the X+ axis direction. A positive angle is in a counterclockwise direction. **Default values:** X-axis measuring 90° Y-axis measuring 0°
Zz	z =	Z measuring height position. It is sometimes desirable to pre-position above the feature to avoid clamps and obstacles. Using this input, the cycle will position down to the Zz height, take the measurement and retract for every measuring position. **Default value:** no move.

For other optional inputs, see Chapter 2, “Optional inputs”.

Outputs

#139 The surface angle measured from the X+ direction.

#143 The measured height difference.

#144 The surface angle error.

Alarms

For details of the alarms, see Chapter 9, “Alarms and messages”.

Example: Measuring an angled surface

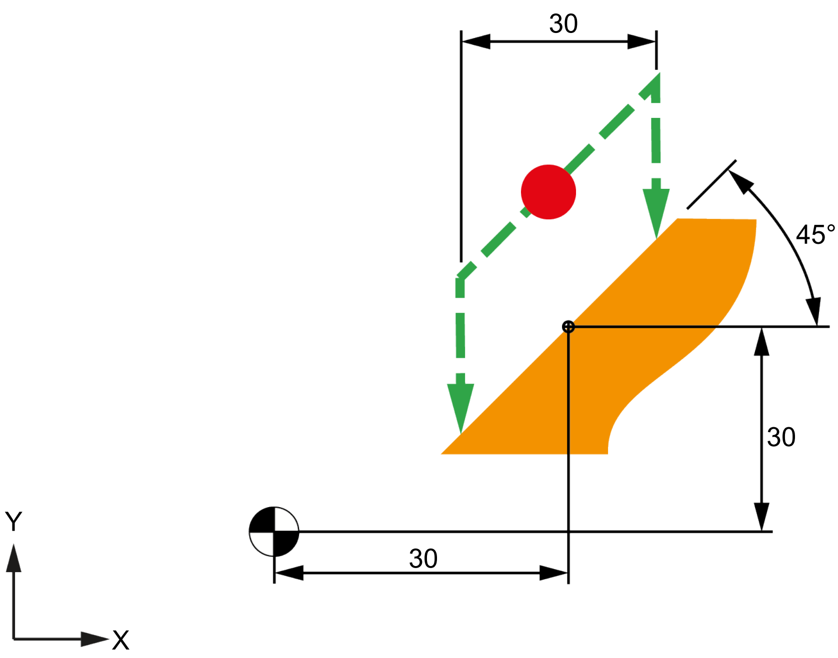


Figure 8.15 Example of an angled surface measurement

G65 P9810 X30. Y50. Z100. F3000.	Protected positioning move.
G65 P9810 Z−15.	Protected positioning move to the start position.
G65 P9843 Y30. D30. A45.	Angle measurement.
G65 P9810 Z100.	Retract to a safe position.
continue	
G17	
G68 G90 X0. Y0. R[#139]	Rotate the co-ordinate system by the angle.
continue the machining program	

NOTE: The Renishaw probe cycles cannot be used while co-ordinate rotation is in force, so use G69 to cancel co-ordinate rotation.

This page is intentionally left blank.

Chapter 9

Alarms and messages

When an error occurs during use of the Inspection Plus software, an alarm number or message is generated. This may be displayed on the screen of the controller.

This chapter describes:

- How to identify an alarm number that may be displayed on a Fanuc 0M controller.
- The meaning and likely cause of each alarm message that is displayed on the screen of the controller.

It then describes typical actions you need to take to clear the fault.

Contained in this chapter

Fanuc 0M controller alarms

Alarm messages are not displayed on the screen, only the alarm number. The alarm numbers displayed are (500 + n), where n is the alarm number.

Example: 92 (PROBE*ALREADY*TRIGGERED) is alarm 592

General alarms

Format:

		#148 flag
1 (OUT*OF*TOLERANCE)	Updates the offset if the cycle start button is pressed to continue	1
1 (OUT*OF*POSITION)		2
1 (ANGLE*OUT*OF*TOLERANCE)		4
1 (DIA*OFFSET*TOO*LARGE)		5
1 (UPPER*TOL*EXCEEDED)	No offset update if the cycle start button is pressed to continue	3
1 (EXCESS*STOCK)		6

Action: If message, press cycle start to continue.

If alarm, this is a reset condition. Restart the program from a safe position.

Format: 1 (PROBE*STOP*FAILURE)

Cause: The probe stop cycle O9833 raised this error because it failed to switch the probe off.

Action: Check if the correct switch-off sequence was added to the O9833 cycle.

If using a spin-off probe, check that the spindle speed override is not active and that sufficient time has been allowed for the spindle speed to ramp up in O9833.

Check that the probe is not faulty.

If #3006 message, press cycle start to continue.

If #3000 alarm, this is a reset condition. Restart the program from a safe position.

Format: 101 (PROBE*STARTUP*FAILURE)

Cause: The probe start cycle O9832 raised this error because it failed to switch the probe on.

Action: Check if the correct switch-on sequence was added to the O9832 cycle.

If using a spin-on probe, check that the spindle speed override is not active and that sufficient time has been allowed for the spindle speed to ramp up in O9832.

Check that the probe is not faulty.

Edit the program and start again from a safe start position.

This is a reset condition.

Format: 91 (MESSAGE)

91 (FORMAT*ERROR)

91 (A*INPUT*MISSING)
91 (B*INPUT*MISSING)
91 (C*INPUT*MISSING)
91 (D*INPUT*MISSING)
91 (E*INPUT*MISSING)
91 (F*INPUT*MISSING)
91 (I*INPUT*MISSING)
91 (J*INPUT*MISSING)
91 (K*INPUT*MISSING)
91 (M*INPUT*MISSING)
91 (S*INPUT*MISSING)
91 (T*INPUT*MISSING)
91 (U*INPUT*MISSING)
91 (V*INPUT*MISSING)
91 (W*INPUT*MISSING)
91 (X*INPUT*MISSING)
91 (Y*INPUT*MISSING)
91 (Z*INPUT*MISSING)
91 (XY*INPUT*MISSING)
91 (XYZ*INPUT*MISSING)

91 (DATA*#130*TO*#139*MISSING)

91 (H*INPUT*NOT*ALLOWED)
91 (M*INPUT*NOT*ALLOWED)
91 (S*INPUT*NOT*ALLOWED)
91 (T*INPUT*NOT*ALLOWED)
91 (A0*INPUT*NOT*ALLOWED)
91 (X0*INPUT*NOT*ALLOWED)
91 (Y0*INPUT*NOT*ALLOWED)
91 (W*INPUT*NOT*ALLOWED*FOR*REPORTER)

91 (IJK*INPUTS*5*MAX)

91 (SH*INPUT*MIXED)
91 (ST*INPUT*MIXED)
91 (TM*INPUT*MIXED)
91 (XY*INPUT*MIXED)
91 (ZK*INPUT*MIXED)
91 (XYZ*INPUT*MIXED)

91 (K*INPUT*OUT*OF*RANGE)

Action: Edit the program and start again from a safe start position.

This is a reset condition.

Format: 86 (PATH*OBSTRUCTED)

Cause: The probe has made contact with an obstruction. This occurs only during a protected positioning cycle.

Action: Edit the program. Clear the obstruction and start again from a safe position.

This is a reset condition.

Format: 87 (UNEXPECTED*PROBE*TRIGGER)

Cause: This alarm occurs when the probe has triggered several times during a monitored move without hitting a surface. The likely cause is machine vibration being transmitted to the probe stylus. If this persists, it may be necessary to contact a Renishaw representative for advice.

Action: Fix any issues and start again from a safe start position.

This is a reset condition.

Format: 88 (NO*FEEDRATE)

Cause: This occurs only during a protected positioning cycle.

Action: Edit the program. Insert the F___ code input and start again from a safe position.

This is a reset condition.

Format: 89 (NO*TOOL*LENGTH*ACTIVE)

Cause: G43 or G44 must be active before the cycle is called.

Action: Edit and start again from a safe position.

This is a reset condition.

Format: 92 (PROBE*ALREADY*TRIGGERED)

Cause: This alarm occurs if the probe is already triggered at the beginning of a measurement move.

The stylus may be in contact with a surface or the probe has failed to reseat.

This could be due to swarf trapped around the probe eyelid.

Action: Clear the fault and start again from a safe start position.

This is a reset condition.

Format: 93 (PROBE*DID*NOT*TRIGGER)

Cause: This alarm occurs if the probe did not trigger during the move.

Either the surface was not found or the probe has failed.

Action: Edit the program and start again from a safe start position.

This is a reset condition.

Format: 102 (INCREASE*PROBE*STAND-OFF)

Cause: During probe calibration a trigger has taken place while the machine was accelerating or decelerating, rendering the skip value invalid.

Action: Increase the Rr or Qq inputs or the probe stand-off distance prior to measurement.

This is a reset condition.

Format: 120 (M101*CYCLE*NOT*SUITABLE)

Cause: The machine characteristics, calculated measuring feedrate and restricted space around the GoProbe training part mean this calibration cannot be used.

Action: Please calibrate on a ring gauge or calibration sphere where the clearances are greater.

This is a reset condition.

Format: 123 (XYZ*ERROR*MISSING*FOR*WCS*UPDATE*IN*TWP/IPM)

Cause: The three measurement errors in X, Y and Z must be found when setting a rotated WCS.

Action: Ensure X, Y and Z errors are measured before a WCS update is applied.

This is a reset condition.

Messages

Message: 1 (SET*P6201.1=1*CYC*START*TO*CONTINUE)

Cause: When optimising on a Fanuc controlled machine this parameter must be set. If you are unable to change the parameter, abandon optimisation and go straight to probe calibration.

Message: 1 (MF#100*ZPF#101*XYPF#102*CYC*START*TO*SAVE)

Cause: Optimisation results are displayed in #100 to #102:

#100	Maximum permissible measuring feedrate.
#101	Maximum permissible Z-axis positioning feedrate.
#102	Maximum permissible X/Y-axis positioning feedrate.

Message: 1(PROBE*CALIBRATION*REQUIRED*AFTER*OPTIMISATION)

Cause: Optimisation has been carried out, so the probe needs to be calibrated using the new optimisation values.

Chapter 10

Configuration

This chapter contains setting information and details about the program variables used in the Inspection Plus software.

Sections in this chapter that apply to SupaTouch optimisation only are marked with the superscript ST.

Contained in this chapter

General

In general, this software is self-configuring and, apart from special applications, will run “out of the box”, provided the installation wizard was used to prepare the software correctly. Once the optional optimisation cycle and calibration cycle are completed for the first time, the cycles are ready to use. However, further manual customisation of the settings is possible. The following configuration information will be of use in this regard.

Installation wizard

This software is supplied with an installation wizard that can be launched from any PC running Windows® XP (or later versions) and will configure the software for the machine, creating machine-specific and personal customisation prior to installing the software.

Multiple probe support

The default setting is to use one probe, but up to three additional probes can be supported using the settings outlined below. The “Extended macro variables” controller option will be required to support multiple probes.

NOTE: Configuring the software for more than one probe is not recommended unless necessary, owing to the additional code overhead and the requirement for additional variables for probe data storage. In principle, it is possible to support more than four probes. If this is necessary, consult a Renishaw representative for advice on how to do this.

When modifying the software to support multiple probes:

- Edit program O9724 to enable multi-probe applications (see “Editing the settings program O9724”).
- Edit program O9732. This sets the variable base number #111 for each probe – each requires its own data storage variables. See “Multiple probe settings in O9732”.
- Edit cycle O9832. Individual probe start sequences can be set up using programs O9712, O9713 and O9714. See “Editing the probe start cycle O9832”.
- Edit cycle O9833. Individual probe stop sequences can be set up using programs O9712, O9713 and O9714. See “Editing the probe stop cycle O9833”.

- Install and edit the multi-probe programs as shown below:

Number of probes	Program
Two (one additional)	O9712
Three (two additional)	O9713
Four (three additional)	O9714

Table 10.1 Multi-probe programs

The programs provide alternative probe start and stop methods and independent overtravel limit settings for each probe (used during optimisation).

Editing the multi-probe programs O9712, O9713 and O9714

NOTE: If all probes use the same start and stop method, no individual code is required, so the standard cycles O9832 and O9833 can be used for all probes.

Cycles O9832 and O9833 must be modified to call up programs O9712, O9713 and O9714 (see “Editing the probe start cycle O9832” and “Editing the probe stop cycle O9833”).

Details of the start/stop code changes required for these cycles are similar to those outlined for O9832 and O9833.

Editing the error messages program O9700

#30=3006(3006=MESSAGE*3000=ALARM) edit =3006/=3000 as required

The Fanuc *i* Series controller supports both #3006 messages and #3000 alarms.

The #30 setting near the top of program O9700 controls the alarm/message type. When 3006 is selected, a mix of #3000 and #3006 messages and alarms is output, depending on which method is best for each alarm or message. Using the setting #30=3006 is the preferred choice – only change this if #3006 is not supported or for specific application preference.

- #3000 alarms will need a program reset to start the program again.
- #3006 messages will allow the cycle to continue if the cycle start button is pressed.

Editing the active offset and read-ahead program O9723

The primary role of this program is to read the active tool offset amount, but it also has a role in controlling read ahead.

Fast machining or smoothing control options can cause block read-ahead problems when running a cycle. Read-ahead control, in the form of G53, is resident in program O9723. This program is called at strategic positions within the cycles. If further code is required to suppress read ahead, add it to this program.

Adjustment is not normally required, but customisation is possible. Please consult a Renishaw representative for advice if changes are being considered.

Examples of changes:

- Use G31 instead of G53. The read-ahead control is performed with the G53 command, but G31 can also be used (if using G31, it may also require a feedrate, for example G31 F1).
- Add a dwell, for example, G4 X.1 (or G4 P100). Sometimes adding a small pause can help resolve spurious behaviour, but excessive dwells will impact on overall cycle time.
- It is also possible to add M98 P9723 program calls within the software at strategic positions.

Editing the settings program O9724

This program is called at the beginning of each cycle to establish the necessary modal information. The following setting adjustments can be made:

NOTE: Items marked (*) are pre-set by the installation wizard.

Base number setting (*)

CAUTION: Before editing, see the full list of variables in the section “Use of variables” later in this chapter, check the availability of free variables and consider other installed software variable requirements.

#111=500(BASE*NO)	The variable base number defines the first variable number in the range of variables used for storing setting data and probe calibration data.

Setting #120 (*)

#120=5(SELECT*OPTIONS*3=FS15/FS9*11=FS16/FS6)

NOTE: When selecting the machine controller types on Fanuc 0, 16, 18-21, 0*i*, 3x (*i* Series), check the machine parameter setting 6000.3 (V15). This could be set to either of the following:

6000.3 = 0 Use FS6/FS16 type setting.
6000.3 = 1 Use FS9/FS15 type setting.

Tool offset type	Tolerance alarm condition	Fanuc tool offset type	
		FS9, FS15 type 10/11/12/15/15*i*	FS6, FS16 type 0/6/16/18-21/30-32*i*
Type A	Flag and alarms	#120 = 1	#120 = 9
Type B	Flag and alarms	#120 = 2	#120 = 10
Type C	Flag and alarms	#120 = 3	#120 = 11
Type A	Flag	#120 = 5	#120 = 13
Type B	Flag	#120 = 6	#120 = 14
Type C	Flag	#120 = 7	#120 = 15

Table 10.2 Fanuc software setting options #120

Tool offset type	Tolerance alarm condition	Meldas tool offset type
Type 1	Flag and alarms	#120 = 1
Type 2	Flag and alarms	#120 = 3
Type 1	Flag	#120 = 5
Type 2	Flag	#120 = 7

Table 10.3 Meldas software setting options #120

Using the “flag only” method

It is expected that the settings to enable “flag only” alarms will suit FMS machining cells where the requirement is to run unmanned. The process error flag #148 will be set and it should be monitored after the relevant probe cycles for corrective action.

Example

G65 P9812 X30. H0.2	Set the tolerance on the measured size.
IF[#148EQ1]GOTO999	Test for an out-of-tolerance condition.
continue the part program	
GOTO1000	
N999 G65 P5001	Pallet change. This changes the pallet to select the next component for machining (details are machine-dependent).
GOTO1	Go to the start of the program.
N1000	
M30	

Additional multiple probes (*)

GOTO1(<DELETE*TO*ENABLE*MULTI*PROBES)

Multiple probes are not enabled as standard, but can be activated by deleting the GOTO1 line or commenting it out by enclosing it in brackets.

This will enable the use of multiple probes, but further settings will be required elsewhere (see “Multiple probe support” in this chapter).

Prove-out mode (% rate is not controlled by the wizard)

For prove-out work, the positioning feedrate is reduced by 50% (a factor of .5) as standard, but the amount in Z (#113=#113*.5) and XY (#119=#119*.5) can be modified by changing the .5 factor.

#113=#113*.5
#119=#119*.5

Setting the in-position checking tolerance (#123)

NOTE: It is not usual to modify this setting.

This is an in-position checking tolerance used within the software to validate a protected positioning or measuring move within the software. Typically, a “PROBE*ALREADY* TRIGGERED”, “PROBE*DID*NOT*TRIGGER” or “UNEXPECTED*PROBE*TRIGGER” alarm may result from this test.

Edit program O9724 as follows:

#123 = .05 (POSITION ZONE MM)
#123 = .002 (POSITION ZONE INCH)

Edit the metric (.05) and inch (.002) values as a pair to the required new tolerance.

Adjusting the back-off factor (#111+6)

NOTE: The following only applies when the optimisation cycle (O9800) has not been used and measuring cycles use the standard two-touch measurement method.

This is used to control the back-off distance in the basic move before the final gauging move. It should be fine-tuned on installation to suit the machine.

A default value of 0.25 is installed by the software. The actual factor should normally be between 0 and 1. Reduce the value to reduce the back-off distance.

Adjusting the fast positioning feedrate (#111+9)

NOTE: The following only applies when the optimisation cycle (O9800) has not been used and measuring cycles use the standard two-touch measurement method.

A default value of 5000 is installed by the software. The actual feedrate should be set between 1000 and 10000. This can be adjusted accordingly.

Editing the basic measure program O9726

```
#29=0(1-TOUCH*RETRIES)                    See note 1
#32=.1(DWELL*SG*PROBE-SECONDS)            See note 2
#33=1.1(OPTIMISED BOF)                    See note 3
```

Note 1 High measuring feedrates can cause the probe to unintentionally trigger as it moves towards the target surface. Robustness against false trigger events can be improved by increasing the #29 value.

Note 2 Dwell or wait between the fast first touch and second probing touch. For kinematic probes (such as OMP40 or OMP60) the value should be 0.1, and for RENGAGE probes (such as OMP400 or OMP600) the value should be 0.3.

Note 3 Optimisation calculates the back-off distance and further adjustment should not be necessary. However, #33 provides a further opportunity to increase or decrease the back-off distance after the first probing touch.

Editing the probe calibration cycle O9801

NOTE: Items marked (*) are pre-set by the installation wizard.

180° spindle orientation section (*)

If the machine supports this feature, enter the appropriate code(s) where shown:

```
(POSITION*SPDL*AT*0*OR*180)
..
M00(180*SPDL*POS)                     See note 1.
```

Note 1 Modify this line of code – for example to M119(180*SPDL*POS) – or replace it with several lines of code if necessary.

NOTE: Keep the comment (180*SPDL*POS) as a marker to quickly locate this section again if required.

Editing the probe start cycle O9832

NOTE: Items marked (*) are pre-set by the installation wizard.

Customising the “USER M/C START CODE” section (*)

Look for this section near the top of cycle O9832. This is where you can add code that must always be active when running the cycles.

```
(-->USER*M/C*START*CODE)
(<*ADD*M/C*START*CODES*HERE)                 See note 1
(<--USER*M/C*START*CODE)
```

Note 1 This is where machine-specific code can be placed. Multiple lines can be added if necessary.

Probe status checking (*)

```
#31=1(PROBE*STATUS*ON*CHECKING*1=ON/0=OFF)
```

Change the setting on this line in the program (mid-way down) to the setting required.

Set #31=1 to enable status checking or #31=0 to disable status checking.

If the probe start code is reliable, it may be possible to improve cycle time by omitting the built-in retries loop. However, when the same M-code is used to switch the probe ON and OFF, the status checking is required to ensure the probe is on before continuing. This also applies to spin ON/OFF probes.

Entering the probe ON code (*)

The actual code required will vary from machine to machine and depend on the probe system used. Refer to the documentation for the machine and relevant Renishaw hardware.

```
(-->*PROBE*ON)
M00(PROBE*ON)                                See note 1
G4 X2.0(PROBE*DWELL)
(<--*PROBE*ON)
```

Note 1 Add the relevant probe ON code. You can use multiple lines comprising M-codes and dwells (G4 Xxx) as required.

Enabling multi-probe start support

NOTE: If all probes use the same start and stop method, no individual code is required, so do not delete (see note 1).

```
(*)
GOTO5(<DELETE*TO*ENABLE*MULTI*PROBES)          See note 1
()
```

Note 1 To activate multiple probes, either delete the GOTO5 line or comment it out by enclosing it in brackets.

Multiple probe settings in O9732 (*)

If using multiple probes, the section at the bottom of O9732 must be edited to define the multiple probe tool offset number (H) and associated base number variable for each additional probe. Programs O9712, O9713 and O9714 may also need to be installed and edited.

```
()
#30=#0(H*OFFSET*NO*PROBE*2)          See note 1
#111=530(PROBE*2*BASE*NO)            See note 2
#30=#0(H*OFFSET*NO*PROBE*3)          See note 1
#111=760(PROBE*3*BASE*NO)            See note 2
#30=#0(H*OFFSET*NO*PROBE*4)          See note 1
#111=572(PROBE*4*BASE*NO)            See note 2
```

Note 1 For each additional probe, define a probe offset (H) number. Any left set at #30=#0 will not be used.

Note 2 For each additional probe, define a variable base number #[#111] (allow 20 variables per probe).

CAUTION: Do not overlap the multi-probe variable ranges or variables used for other purposes – the software does not check this.

Editing the probe stop cycle O9833

NOTE: Items marked (*) are pre-set by the installation wizard.

Probe status checking (*)

#31=<u>1</u>(PROBE*STATUS*OFF*CHECKING*1=ON/0=OFF)

Change the setting on this line in the program (near the top) to the setting required.

Set #31=1 to enable status checking, or #31=0 to disable status checking.

If the probe stop code is reliable, it may be possible to improve cycle time by omitting the built-in retries loop. However, when the same M-code is used to switch the probe ON and OFF, the status checking is required to ensure the probe is on before continuing. This also applies to spin ON/OFF probes.

Entering the probe OFF code (*)

The actual code required will vary from machine to machine and depend on the probe system used. Refer to the documentation for the machine and relevant Renishaw hardware.

```
(-->*PROBE*OFF)
M00(PROBE*OFF)                                  See note 1
(<--*PROBE*OFF)
```

Note 1 Add the relevant probe OFF code. You can use multiple lines comprising M-codes and dwells (G4 Xxx) as required.

Enabling multi-probe stop support

NOTE: If all probes use the same start and stop method, no individual code is required, so do not delete (see note 1).

```
()
GOTO4 (<DELETE*TO*ENABLE*MULTI*PROBES)          See note 1
()
```

Note 1 To activate multiple probes, either delete the GOTO4 line or comment it out by enclosing it in brackets.

Customising the “USER M/C STOP CODE” section (*)

Look for this section near the bottom of cycle O9833. This is where you can add code that must always be read when probing is finished.

```
(-->USER*M/C*STOP*CODE)
(<*ADD*M/C*STOP*CODES*HERE)                    See note 1
(<--USER*M/C*STOP*CODE)
```

Note 1 This is where machine-specific code can be placed. Multiple lines can be added if necessary.

CAUTION: Always check with the machine tool builder about what active modal data should remain active after running the probing cycles.

Machine parameter settings ST

The following parameter must be set:

P6201.1=1 (SEB) The skip value is automatically compensated for machine servo errors to record the actual position.

The following parameter setting is recommended:

P6200.7=1 (SKF) Dry run, override, acceleration/deceleration included. This helps to prevent machine bumping during fast operation.

The following parameter setting is known to give issues with previous software versions:

CAUTION: Seek machine tool builder advice if in doubt about this parameter. It may not be applicable for all machines and may rely on other parameter settings and machine options. Failure to re-enable this parameter setting after probing could lead to machining issues.

P2005.1=0 (FEED) If this is set to =1, severe bumping can occur during probing moves. This parameter bit must be set for each axis – X, Y and Z. If the high-speed machining option is not used, it may be possible to turn this parameter off, otherwise it will be necessary to turn it off just for probing and then back on afterwards, under program control.

Example

How to use M-codes to apply and cancel the parameter for feed forward (parameter P2005.1) for the X, Y and Z axes.

```
PARAMETER 6078=91
PARAMETER 6079=92

O9023 (M91 PRE-PROBING MACRO)
(REMOVES FEED FORWARD)
G10L50
N2005P1R01000000
N2005P2R01000000
N2005P3R01000000
G11
M99

O9024 (M92 POST-PROBING MACRO)
(REINSTATES FEED FORWARD)
G10L50
N2005P1R01000010
N2005P2R01000010
N2005P3R01000010
G11
M99
```

Use of variables

Local variables

#1 to #32	These are used within each program as required for such things as calculation.

Common variables

#100 to #110	Stored calibration data for the 5-point rectangle cycle O9817.
#111	Calibration base number variable used for storing probe data.
#112	Active vector radius used in cycles O9821, O9822 and O9823.
#113	Z fast positioning feedrate (in the units of the machine). This is read in from the #[#111+9] value (mm/min) and the units converted.
#114 ST	Used internally in O9729.
#115 ST	Used internally in O9729.
#116	Active tool offset amount.
#117	Reserved.
#118	RADIUS TOO LARGE flag in cycles O9812, O9814, O9822 and O9823 (also used for a temporary ATAN store in program O9731).
#119	XY fast positioning feedrate (in the units of the machine). This is read in from the #[#111+9] value (mm/min) and the units converted.
#120	Setting variable used in program O9724.
#121	Print option. The component number is incremented by 1 with each program heading. To reset, state #121 = 0.
#122	Print option. The feature number is incremented by 1 with each print program call. To reset, state #122 = 0.
#123	Start and end of block position zone. The normal setting is 0.05 mm (0.002 in). If the skip position is within this zone, the cycle aborts, with either a “PROBE ALREADY TRIGGERED” or “PROBE DID NOT TRIGGER” alarm.
#124	Stored X skip position at the end of the basic move cycle (O9726).
#125	Stored Y skip position at the end of the basic move cycle (O9726).
#126	Stored Z skip position at the end of the basic move cycle (O9726).
#127	X average skip position at the end of the X diameter move cycle (O9721).

#128	Y average skip position at the end of the Y diameter move cycle (O9722).
#129	Inch/metric multiplier (1/25.4 [0.03937]/1.0 factor).
#130 to #134	Saved output data for the first feature when using the feature-to-feature measurement cycle (O9834). The second feature output data is stored in common variables #135 to #139.
#135 to #149	See Chapter 3, “Cycle outputs”, for information.
#150 onwards	These are not used by the software.

Common retained variables

CAUTION: It is a feature of this software that all unit-dependent probe data is stored in millimetres, regardless of the current machine units. When this data is read, it is converted as required to suit the active machine units. This differs from previous versions of Renishaw inspection software.

#[#111+0]	(XRAD) X calibration radius.
#[#111+1]	(YRAD) Y calibration radius.
#[#111+2]	(XOFF) X-axis stylus offset.
#[#111+3]	(YOFF) Y-axis stylus offset.
#[#111+4] ST	(R-ADJ) Adjustment used for recovery measuring feedrate.
#[#111+5]	(FLAG) Software status flag used for internal setting and monitoring of the cycles.
#[#111+6]	Calculated probe delay (trigger filter + transmission delay) ST. Back-off factor, if SupaTouch optimisation has not taken place.
#[#111+7] ST	(SDF) Stopping distance factor.
#[#111+8]	(MFR) Measuring feedrate.
#[#111+9]	(FPF) Fast positioning feedrate.

#[#111+10] (30°) #[#111+11] (60°) #[#111+12] (120°) #[#111+13] (150°) #[#111+14] (210°) #[#111+15] (240°) #[#111+16] (300°) #[#111+17] (330°)	(VRAD) Vector calibration data storage.
#[#111+18]	(ZRAD) Z calibration radius used for 3D vector measuring.
#[#111+19]	(SRAD) Nominal stylus radius used for 3D vector measuring.
#[#111+n] to #[#111+(n+20)]	Multiple probe calibration data. Additional sets of probe data (#[#111+0] to #[#111+19]) must be defined for each probe (three sets maximum). The actual variables available are the limiting factor and this depends on the controller options fitted. See "Multiple probe support".

Compliance with Fanuc parameter P5006.6, P6019.4 and P6006.4 settings

This software is now compliant with these parameter settings – no software adjustment will be required.

This page is intentionally left blank.

Chapter 11

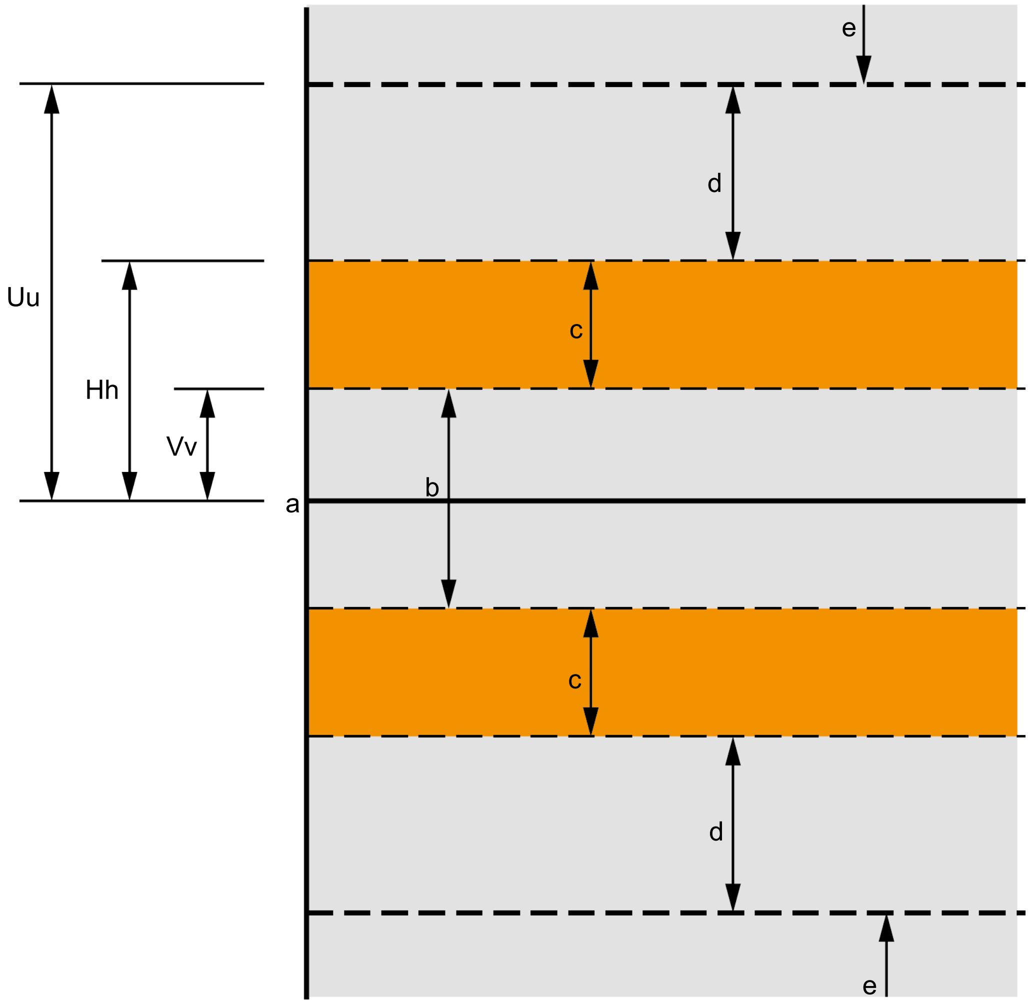
General information

This chapter contains general information and reference material that is relevant to the Inspection Plus software package.

Contained in this chapter

Tolerances

Inputs Uu, Hh and Vv apply to the size and tool offset updates only.



a = Nominal size.

b = Null band. This is the tolerance zone in which no tool offset adjustment occurs.

c = Area where the Ff input is effective in percentage feedback. F (0 to 1) gives 0% to 100% feedback to the tool offset.

Example: F0.5 will feed back 50% as the error.

d = "OUT OF TOLERANCE" alarm occurs. The tolerance value that applies to the size of the feature is defined by input Hh.

e = Uu upper tolerance. If this value is exceeded, no tool offset or work offset is updated and the cycle stops with an alarm. This tolerance applies to both size and position where applicable.

Figure 11.1 Size and tool offset update tolerances

Also see the SPC cycle (O9835) in Chapter 8, "Additional cycles", which can be used as a modified method for the feedback of tool offset corrections. Use this instead of input Ff.

True position tolerances

For a true position tolerance (Mm input), see Figure 11.2 below.

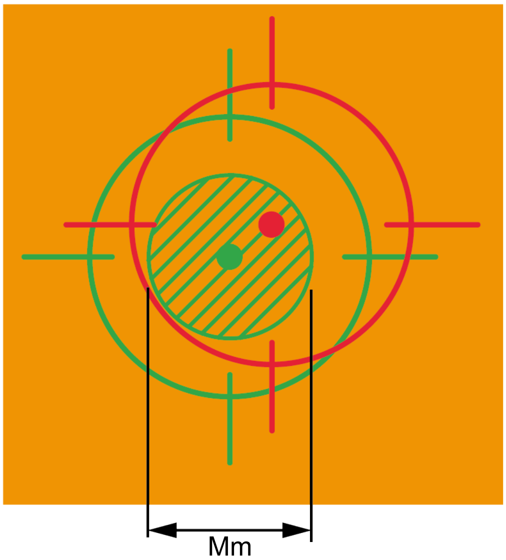


Figure 11.2 Cylinders centred on true positions

Experience values Ee

The measured size can be adjusted by an amount stored in a spare tool offset.

Example

Measure a 40 mm diameter and update tool offset 20.

G65 P9814 D40. T20. <u>E21.</u>	The experience value stored in tool length offset 21 will be added to the measured size.

Reason for using this option

Component clamping forces in some applications can influence the measured size. Therefore, an adjustment value to relate measurement to a traceable standard, such as a co-ordinate measuring machine, is desirable. Thermal effects can also be compensated for using this method.

Additional spare tool offsets

The range of spare tool offsets can be extended when either a Type B or Type C tool offset option is installed on the machine.

Experience no.	Type A	Type B	Type C
E1 to En	✓	✓	✓
E201 to E20n		✓	✓
E601 to E60n			✓

where ‘n’ is the tool offset number.

From the table, you can see that you should add either 200 or 600 to the tool offset number.

These additional tool offset registers can safely be used for both Ee experience values and also with the SPC cycle O9835 Mm input provided. The tool offset number is not used as a normal tool offset location.

Reporter Print

Reporter Print requires the Reporter app (A-5999-4200) to be installed and connected to the machine tool to receive measured data.

This section explains the variables defined during software installation and gives a programming example.

Reporter Part ID variable

The Reporter Part ID variable is a machine variable that is used to set the Part ID number. The default variable is #156, but if this is already being used by other programs, another appropriate variable can be selected during software installation.

Reporter requires the inclusion of a Part ID so that it can identify which component the measurement data is associated with.

Typically, the program number is used as the Part ID, however, setting a different ID for each start and end sequence is possible, assuming each number is unique.

The Part ID can later be renamed in the Reporter app, but the number chosen still needs to be unique.

The G-code line to set the Part ID (for example, #156=2000) must be inserted in the program before the Data Send start macro (O9735).

Reporter Protocol variable and Reporter Data variable

Protocol variable

This variable is set during software installation and is used to specify the type of data being received. The default value is 157.

If you change the default value, you will also need to change the related variable in the Reporter app settings menu. For further information, refer to the Installation and user guide *Reporter for Fanuc* (Renishaw part no. H-5999-8700).

Data variable

The data variable is set in the Reporter app configuration settings and is used to specify the base number for a range of 28 sequential machine variables required to hold data.

For example, enter the value 158 to use machine variable range #158 to #185 (#158 + 27 variables).

If you change the default value, you will also need to change the related variable in the Reporter app settings menu. For further information, refer to Installation and user guide *Reporter for Fanuc* (Renishaw part no. H-5999-8700).

NOTE: If these values are changed from their default value, ensure that no other applications or G-code programs use these variables.

On machine programming

NOTE: If Set and Inspect is connected to the machine tool, manual programming of component inspection and reporting will not be required.

Data Send start and end

Reporting is enabled and disabled using the Data Send start and end macro which is embedded within the Probe Start and Probe Stop macros. The command line is written as shown below and is enabled with a W1.1 on the end of the Probe Start and Probe Stop macros. The example assumes #156 is used for the Part ID.

G65 P9832 W1.1

Embedded Data Send start macro:

G65 P9735 A1. B1. C0. I#156

After the measuring of feature is complete, the Data Send macro must be run again.

G65 P9833 W1.1

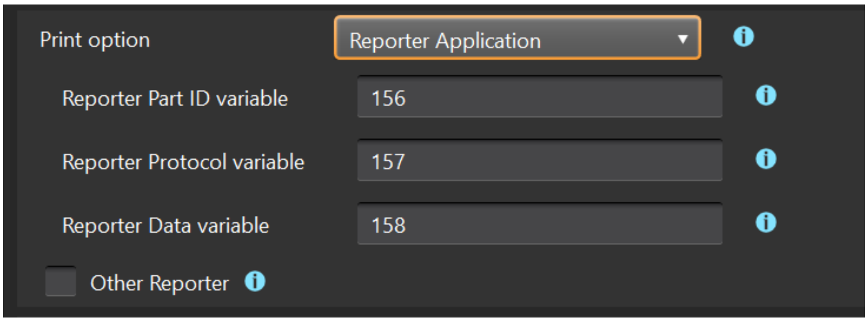
Embedded Data Send stop macro:

G65 P9735 A1. B2. C0. I#156

Description of Data Send inputs

Macro	O9735 call line	Description
Data Send – Start	`G65 P9735 A1.` **`B1.`** `C0.` **`I#156`**	A1. = informs Data Collector to expect one additional input containing data after the C input. **B1.** = informs Data Collector this is a Part Start command. C0. = not applicable (future requirement). **I#156** = informs Data Collector the variable containing the Part ID (for example #156).
Data Send – End	`G65 P9735 A1.` **`B2.`** `C0.` **`I#156`**	A1. = informs Data Collector to expect one additional input containing data after the C input. **B2.** = informs Data Collector this is a Part Stop command. C0. = not applicable (future requirement). **I#156** = informs Data Collector the variable containing the Part ID (for example #156).

NOTE: If other Reporter packages are present on the controller, the library will have the Data Send start and end macro already loaded. A tick box option is available within the wizard that pevents the Data Send (O9735) start and end macro being generated when the wizard is run.



W input for measured feature macros

The W input acts as the feature ID number. The feature number that is being measured must be added to the feature measurement macro line.

NOTE: A suffix of “.1” must be added to the W inputs to output measurement results to Reporter.

Example: XYZ single surface measurement with reporting

O2000	
G40 G80	
G91 G28 Z0.	
G90	
M6 T1	Select the probe.
G54	Start position.
G01 G43 H1 Z15. F3000.	Set the probe to the retract position away from the surface.
#156=2000	Part ID set to 2000.
G65 P9832 W1.1	Switch on the probe (this includes M19). Also starts the print option for Reporter app.
G65 P9810 Z10. F3000.	Protected positioning move to the start position.
G65 P9811 Z0.0 W1.1	Measure the surface position in Z and set the feature as 1 in Reporter app.
G65 P9810 Z15.	Protected positioning move to the retract position.
G65 P9833 W1.1	Switch off the probe (this includes M19). Also stops the print option for Reporter app.
G91 G28 Z0.	
G90	
M30	

Text Print

If the Reporter App is not installed, a formatted text print can be generated as the cycle runs. The component number can be incremented by cycle control (see input Ww in Chapter 2, “Optional inputs”). However, it must be reset external to the cycles when necessary (set #121 = 1).

Example of printing a cycle output

```
------------------------------------------------------------
DATE/   21/ 10/ 2020/  TIME/   12/ 28/  0
COMPONENT NO/     11
UNITS/MM
------------------------------------------------------------
------------------------------------------------------------
FEATURE NO/    1/  FEATURE NAME/  X POINT
------------------------------------------------------------
SIZEX/     .0000/  ACTUAL/     .3306/  TOL/  1.0000/  DEV/    .3306
------------------------------------------------------------
FEATURE NO/    2/  FEATURE NAME/  Y POINT
------------------------------------------------------------
SIZEY/− 130.0000/  ACTUAL/− 129.0937/  TOL/  1.0000/  DEV/    .9063
        +++++OUT OF POS/+++++ ERROR TP/     .4063/ RADIAL
------------------------------------------------------------
FEATURE NO/    3/  FEATURE NAME/  BORE
------------------------------------------------------------
SIZED/   23.0000/  ACTUAL/   22.9574/  TOL/  1.0000/ DEV/−   .0426
POSNX/   25.0000/  ACTUAL/   25.1471/  TOL TP/  1.0000/  DEV/    .1471
POSNY/−  44.0000/  ACTUAL/−  43.3556/  TOL TP/  1.0000/  DEV/    .6444
        +++++OUT OF POS/+++++ ERROR TP/     .1609/ RADIAL
```

NOTE: Additional “/” characters are included for use as a delimiter, allowing easy loading of the results into a spreadsheet if required.

Variables #121 and #122

When Renishaw non-contact tool setting (NCTS) software is used in conjunction with this Inspection Plus software, you must be aware that variables #121 and #122 will be overwritten by the NCTS software. These variables are used as counters in the Inspection Plus software print program. Use alternative free variables instead of #121 and #122 and edit O9730 accordingly.

Alternatively, they can be set manually in part programs:

Example: #121 = 1
#122 = 1

Considerations when using vector cycles O9821, O9822 and O9823

Vector cycles involve the mathematical operation of squared values. This can lead to precision errors if large values are used. The following factors must be considered:

Use of 3-point bore/boss cycle (O9823)

The cycle may be used to establish the centre and diameter of a bore or external feature. There is, however, a practical limitation to the use of the cycle. It is advisable to use the largest distance between contacts that is practical. The minimum conditions to give reliable data are as follows:

- 168° total span.
- 48° between any two points.

The software does not check the minimum condition inputs.

The accuracy of the result deteriorates if the minimum conditions are not followed.

Effect of vector calibration data on results

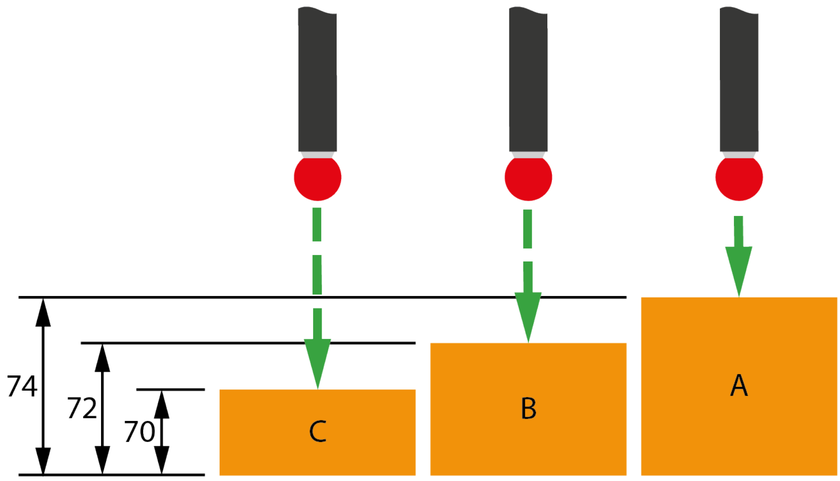
The vector calibration cycle establishes true calibration data at each 30° increment. A small error due to the probe trigger characteristics may occur at intermediate angles between the 30° calibration points. This error is small for standard machine tool probes and standard styli but, to minimise any errors, linear interpolation is applied between the calibration radii.

NOTE: In order to acheive the highest possible accuracy, always use the standard bore/boss measuring cycle (O9814) where practical.

General probing applications

Example 1: Part identification

If a group of components can be identified by a single feature, a probe can be used to inspect that feature and decide which component is present. This is done by using data from the output chart following a measuring program.



Each part surface is known to be within ±0.5

Figure 11.3 Part identification

G65 P9810 Z84. F3000.	Protected positioning move to the start position.
G65 P9811 Z70.	Single surface measurement (target C surface).
IF[#137GT73.]GOTO100	If the result is greater than 73, go to N100.
IF[#137GT71.]GOTO200	If the result is greater than 71, go to N200.
IF[#137GT69.]GOTO300	If the result is greater than 69, go to N300.

```
GOTO400
N100(PROGRAM TO MACHINE A)
continue “A” component
GOTO400
N200(PROGRAM TO MACHINE B)
continue “B” component
GOTO400
N300(PROGRAM TO MACHINE C)
continue “C” component
N400
M30
%
```

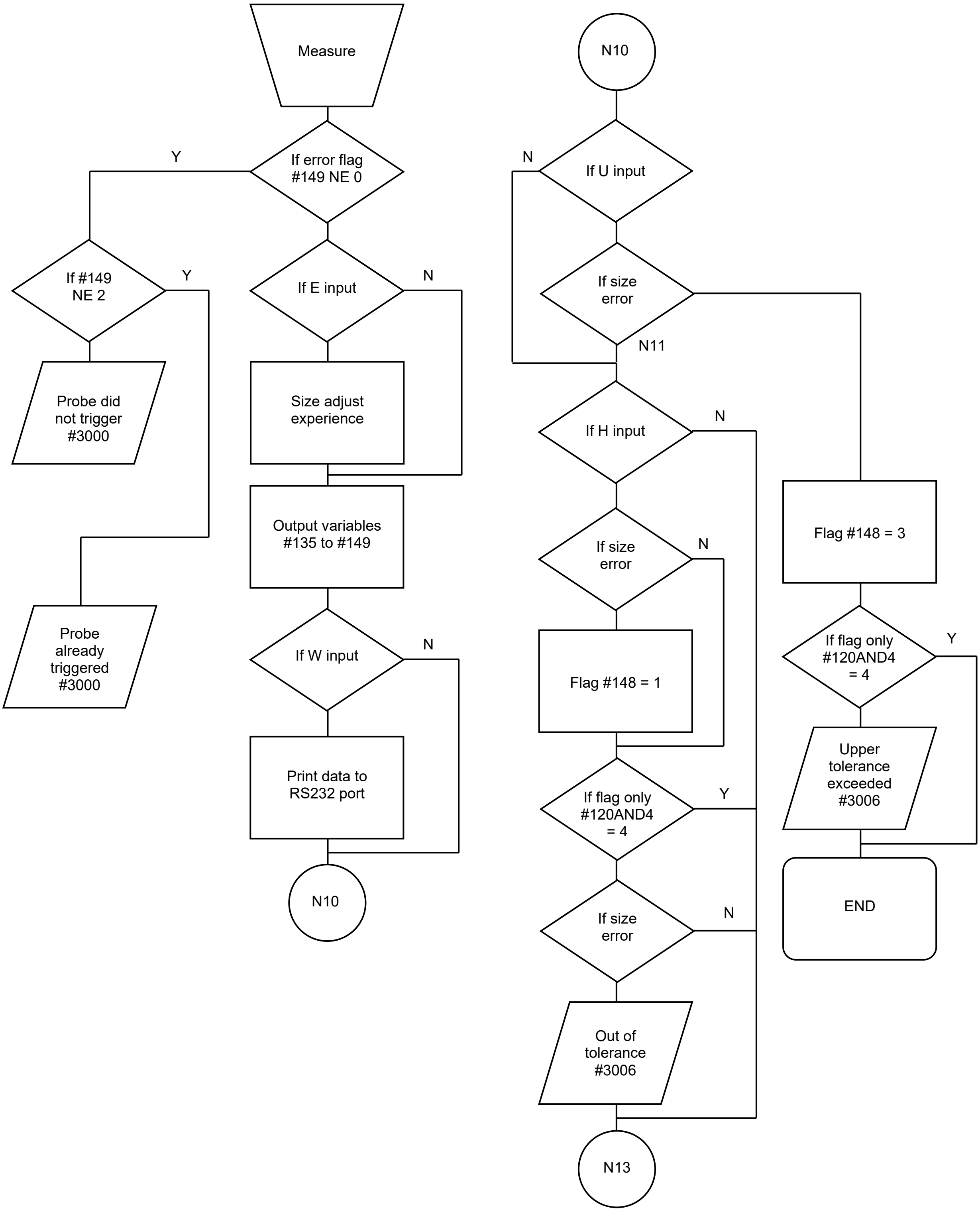
Example 2: Probe measure every nth component

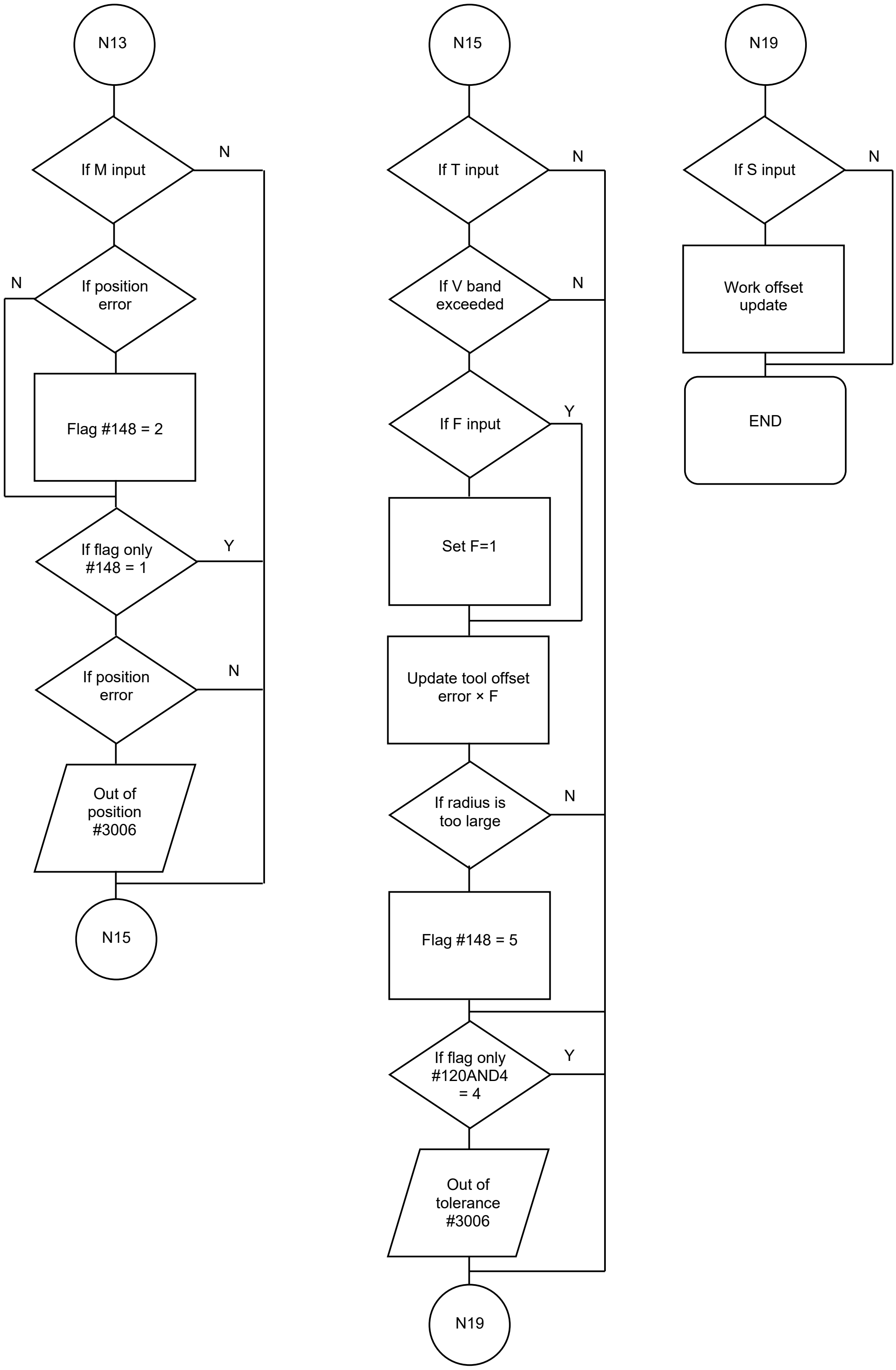
It is often a requirement to probe every nth component in the interests of reducing overall cycle time.

The following programming method can be employed:

O5000(PART PROGRAM)	
#100=0	Reset the counter.
#101=5	Count limit.
N1	
(START OF MACHINING)	
conventional part programming	
N32	
(START OF PROBE ROUTINES)	
IF[#100LT#101]GOTO33	If the counter is less than 5, go to N33.
T01 M06 (PART INSPECTION)	Select the inspection probe.
probing routines	
#100=0	Reset the counter to zero.
N33	
(CONTINUE MACHINING OR END)	
#100=#100+1	Increment the counter.
rest of the machining program	
M99 P1	Return to N1.
M30	
%	

Output flow (bore/boss and web/pocket cycles)




N13
If M input
N
If position error
N
Flag #148 = 2
If flag only #148 = 1
Y
If position error
N
Out of position #3006
N15
N15
If T input
N
If V band exceeded
N
If F input
Y
Set F=1
Update tool offset error × F
If radius is too large
N
Flag #148 = 5
If flag only #120AND4 = 4
Y
Out of tolerance #3006
N19
N19
If S input
N
Work offset update
END

Appendix A

Features, cycles and limitations of the Inspection Plus software

Contained in this appendix

Features of the Inspection Plus software

- Protected positioning.
- Measurement of internal and external features to determine both size and position. This includes:
  - Obtaining a printout of feature data.
  - Applying tolerances to both size and position.
- Additional features for feedback of errors include:
  - Experience values can be applied to the measured size.
  - Percentage feedback of the error can be applied.
  - Null band zone for no tool offset update.
  - SPC (statistical process control) feedback based on average value.
- Calculation of feature-to-feature data.
- Measurement of external and internal corners for corner surfaces which may not be parallel to an axis.
- Calibration of multiple probes.
- 4th axis datum setting and tolerancing.
- Angular measurement of features.
- Software option to turn off the tolerance alarms and provide a flag-only alarm. Suitable for flexible manufacturing systems (FMS) and unmanned applications.
- Built-in stylus collision and false trigger protection for all cycles.
- Diagnostic and format error-checking routines for all cycles.
- Self-optimisation for optimum performance.

Cycles

- Protected positioning.
- Calibration cycles (including calibrating on a sphere).
- Measurement:
  - XYZ single surface.
  - Web/pocket.
  - Bore/boss (four measuring points).
  - Internal and external corner find.
  - 5-point rectangle.
- Vectored measurement:
  - 3-point bore/boss.
  - Angled web/pocket.
  - Angled single surface.
  - XYZ angled surface.
- Additional cycles:
  - 4th axis measurement.
  - Bore/boss on a PCD.
  - Stock allowance.
  - Multiple probe use.
  - XY plane angle measurement.
  - Feature-to-feature data.
  - Updating SPC tool offset.

Limitations

General

- The probe cycles will not run if “mirror image” is active.
- Consider variable availability.
- Requires G31 skip function and custom macro.

Fanuc 10, 11, 12 and 15M controllers

Limitations

- #500 to #549 standard variable option.
- Cannot support multiple probes unless a controller option for more variables is installed.

Fanuc 6M controller

This controller is no longer supported.

Fanuc 0M controller

Limitations

- #500 to #531 standard variables (no option available).
- Cannot support multiple probes.

Fanuc 16M, 18M and 21M controllers

Limitations

- #500 to #549 standard variable option.
- Cannot support multiple probes unless a controller option for more variables is installed.

Fanuc *i* Series: 0*i*M, 16*i*M, 18*i*M, 21*i*M and 3x*i* controllers

Limitations

- #500 to #549 standard variable option.
- Cannot support multiple probes unless a controller option for more variables is installed.

Appendix B

Alternative calibration cycles

These alternative calibration cycles are available to maintain backwards compatibility and flexibility.

Contained in this appendix

Centring on a calibration feature (O9801 K0.)

NOTE: This cycle should be used only if the machine has spindle orientation for the 180° positioning.

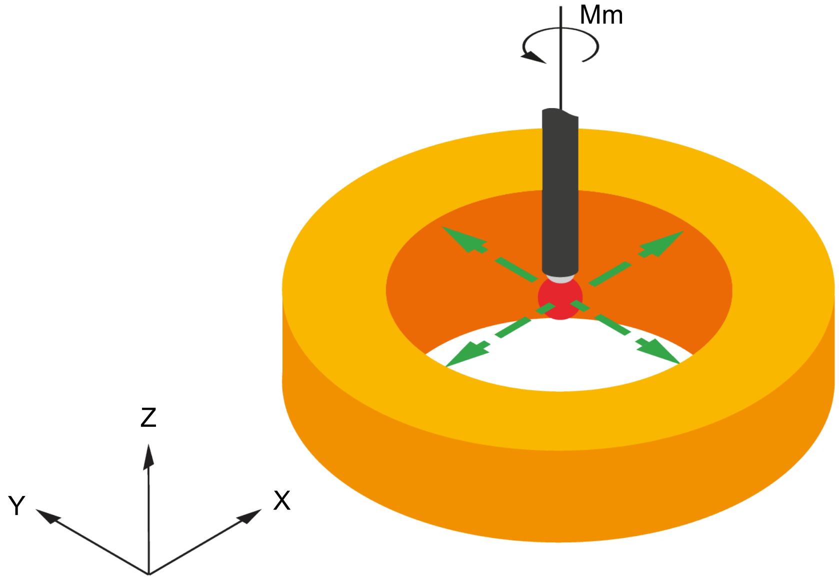


Figure B.1 Centring on a calibration feature

Description

This cycle is used to position the spindle centre on the centre line of the calibration feature.

Application

Prepare a program to position the probe stylus in the feature approximately on the centre line and at the required depth. Run the cycle to complete the measuring sequence with spindle orientation included. The cycle finishes with the spindle on the centre line.

Alternatively, if your machine retains the tool offset at all times, the cycle can be run directly from the MDI screen without writing a program.

Format

G65 P9801 K0. Dd. Bb. M180. [S1. Zz.]

Example: G65 P9801 K0. D30. Z50. B6. M180. S1.

Compulsory inputs

K0.		The mode for centring only.
Bb	b =	The nominal diameter of the stylus ball.
Dd	d =	The diameter of the feature.
M180.		M180. is used as a flag to orient the probe to a second spindle position (180°). This is used to automatically find the centre of the feature, and means that only approximate pre-positioning is required. Alternatively, an M3. input (rotating spindle) can be used – see “Notes on using the M180./M3. input” at the beginning of Chapter 5, “Probe calibration and SupaTouch optimisation”.

Optional inputs

Zz	z =	The absolute Z-axis measuring position when calibrating on an external feature. If this is omitted, a bore cycle is assumed.

For optional input Ss, see Chapter 2, “Optional inputs”.

Outputs

The spindle is centred on the reference feature.

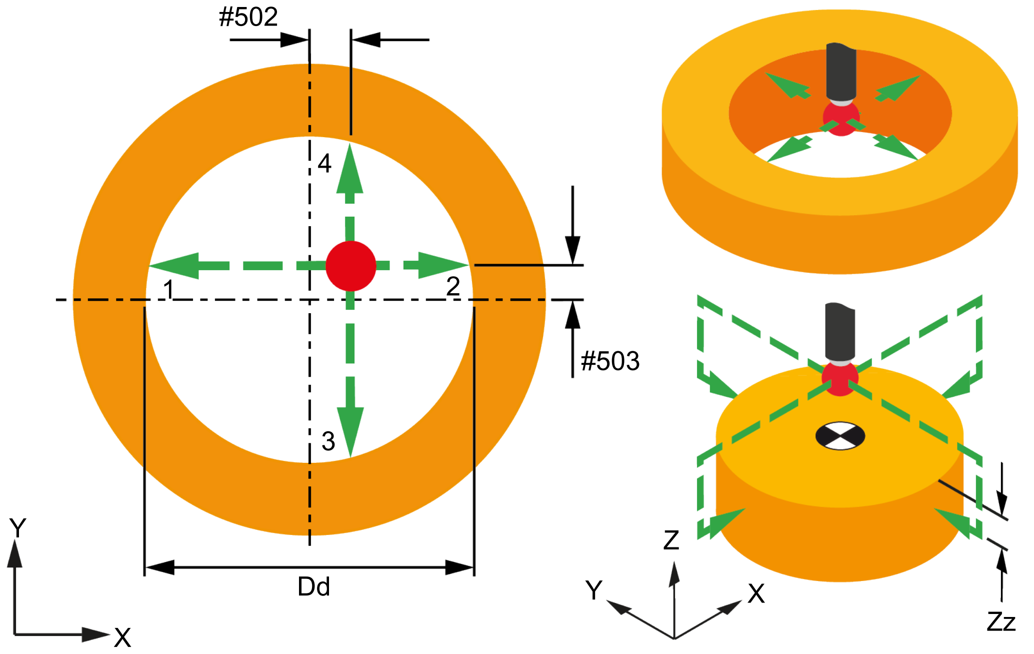
Example

Centre on a ring gauge.

O0001

G90 G80 G40 G0	Preparatory codes for the machine.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−10. F3000.	Protected positioning move.
G65 P9801 K0. D30. M180.	Measure moves to find the centre (includes 180° positioning).
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Calibrating the stylus X and Y offsets (O9801 K2.)



NOTE: This figure assumes that the default calibration base number is set to 500.

Figure B.2 Calibrating the stylus X and Y offsets

Description

The probe stylus is positioned inside a pre-machined hole at a height suitable for calibration. When this cycle is completed the stylus offset amounts in the X and Y axes are stored.

Application

Machine a hole with a suitable boring bar so that the exact centre of the hole is known. With the spindle orientation active, position the stylus to be calibrated inside the hole and the spindle on the known centre position.

NOTE: If spindle 180° orientation positioning is available, use the Mm input to avoid accurate pre-positioning before running the cycle or boring a hole.

When the cycle is run, measuring moves are made to determine the X offset and Y offset of the stylus. The probe is then returned to the start position.

Format

G65 P9801 K2. Bb. Dd. [M180. Zz.]

where [ ] denote optional inputs.

Example: G65 P9801 K2. B6. D50.005 Z50. M180.

Compulsory inputs

K2.		The flag to set the stylus offsets.
Bb	b =	The nominal diameter of the stylus ball.
Dd	d =	The nominal size of the feature.

Optional inputs

M180.		M180. is used as a flag to orient the probe to a second spindle position (180°). This is used to automatically find the centre of the feature, and means that only approximate pre-positioning is required. Alternatively, an M3. input (rotating spindle) can be used – see “Notes on using the M180./M3. input” at the beginning of Chapter 5, “Probe calibration and SupaTouch optimisation”.
Zz	z =	The absolute Z-axis measuring position when calibrating on an external feature. If this is omitted, a bore cycle is assumed.

Outputs

The following data is stored (this assumes that the default calibration base number is set to 500):

#502	X-axis stylus offset (XOFF)
#503	Y-axis stylus offset (YOFF)

Example: Calibrating the stylus X, Y offset

This example describes a complete positioning and calibration program.

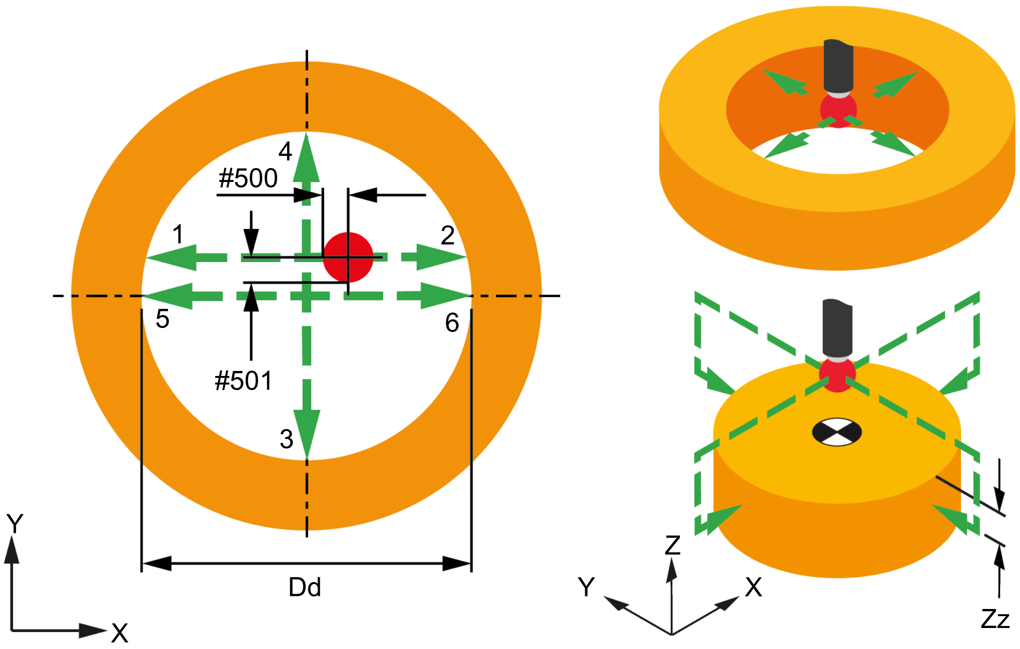
Set the exact X, Y and Z feature positions in a work offset (this example uses G54).

O0002

G90 G80 G40 G0	Preparatory codes for the machine.
G54 X0. Y0.	Move to the centre of the feature.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−5. F3000.	Protected positioning move into the hole.
G65 P9801 K2. B6. D50.	Calibrate in a 50 mm (1.97 in) diameter bored hole with a 6 mm (0.236 in) diameter stylus.
G65 P9810 Z100. F3000.	Protected positioning move retract to 100 mm (3.94 in).
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Calibrating the stylus ball radius (O9801 K−3.)

NOTE: It is possible to use K−3. (or K−4., not illustrated) to prevent the stylus offsets from being set, but otherwise they perform the same operations as either K3. or K4. shown in Chapter 5, “Probe calibration and SupaTouch optimisation”. If you intend using the vector measuring cycles (O9821, O9822 or O9823) later, choose the K4., K−4. option to include vector radii calibration.



NOTE: This figure assumes that the default calibration base number is set to 500.

Figure B.3 Calibrating the stylus ball radius

Description

This cycle is used to calibrate the stylus radius values only, whereas using K3. includes setting the stylus offsets. Otherwise, the use and operation of both cycles is similar.

Application

Clamp a calibrated ring gauge on the machine table at an approximately known position. With spindle orientation active, position the stylus to be calibrated inside the ring gauge on the approximate centre position.

When the cycle is run, six moves are made to determine the radius values of the stylus ball. The probe is then returned to the start position.

Alternatively, if your machine retains the tool offset at all times, the cycle can be run directly from the MDI screen without writing a program.

Format

G65 P9801 K−3. Bb. Dd. [Zz.]

where [ ] denote optional inputs.

Example: G65 P9801 K−3. B6. D50.005 Z50.

Compulsory inputs

K−3.		Calibrate the radius of the stylus ball.
Bb	b =	The nominal diameter of the stylus ball.
Dd	d =	Diameter of the reference ring gauge.

Optional input

ZZ	z =	The absolute Z-axis measuring position when calibrating on an external feature. If this is omitted, a ring gauge cycle is assumed.

Outputs

The following data is stored (this assumes that the default calibration base number is set to 500).

#500	X+, X−, stylus ball radius (XRAD)
#501	Y+, Y−, stylus ball radius (YRAD)

Example: Calibrating the radius of a stylus ball

This example describes a complete positioning and calibration program.

Set the approximate X, Y, Z feature positions in a work offset (this example uses G54).

O0003

G90 G80 G40 G00	Preparatory codes for the machine.
G54 X0. Y0.	Move to the centre of the feature.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G65 P9832	Switch on the probe (this includes M19).
G65 P9810 Z−5. F3000.	Protected positioning move into the hole.
G65 P9801 K−3. B6. D50.	Calibrate in a 50 mm (1.97 in) diameter bored hole with a 6 mm (0.236 in) diameter stylus.
G65 P9810 Z100. F3000.	Protected positioning move retract to 100 mm (3.94 in).
G65 P9833	Switch off the probe (when applicable).
G28 Z100.	Reference return.
H00	Cancel the offset (when applicable).
M30	End of the program.

Appendix C

Inspection Plus and 5-axis commands

Contained in this appendix

Machine configurations

Multi-axis machines are considered to be those with XYZ linear axes and one or more rotary axes. Generally, a multi-axis machine will fall into one of the following configuration types:

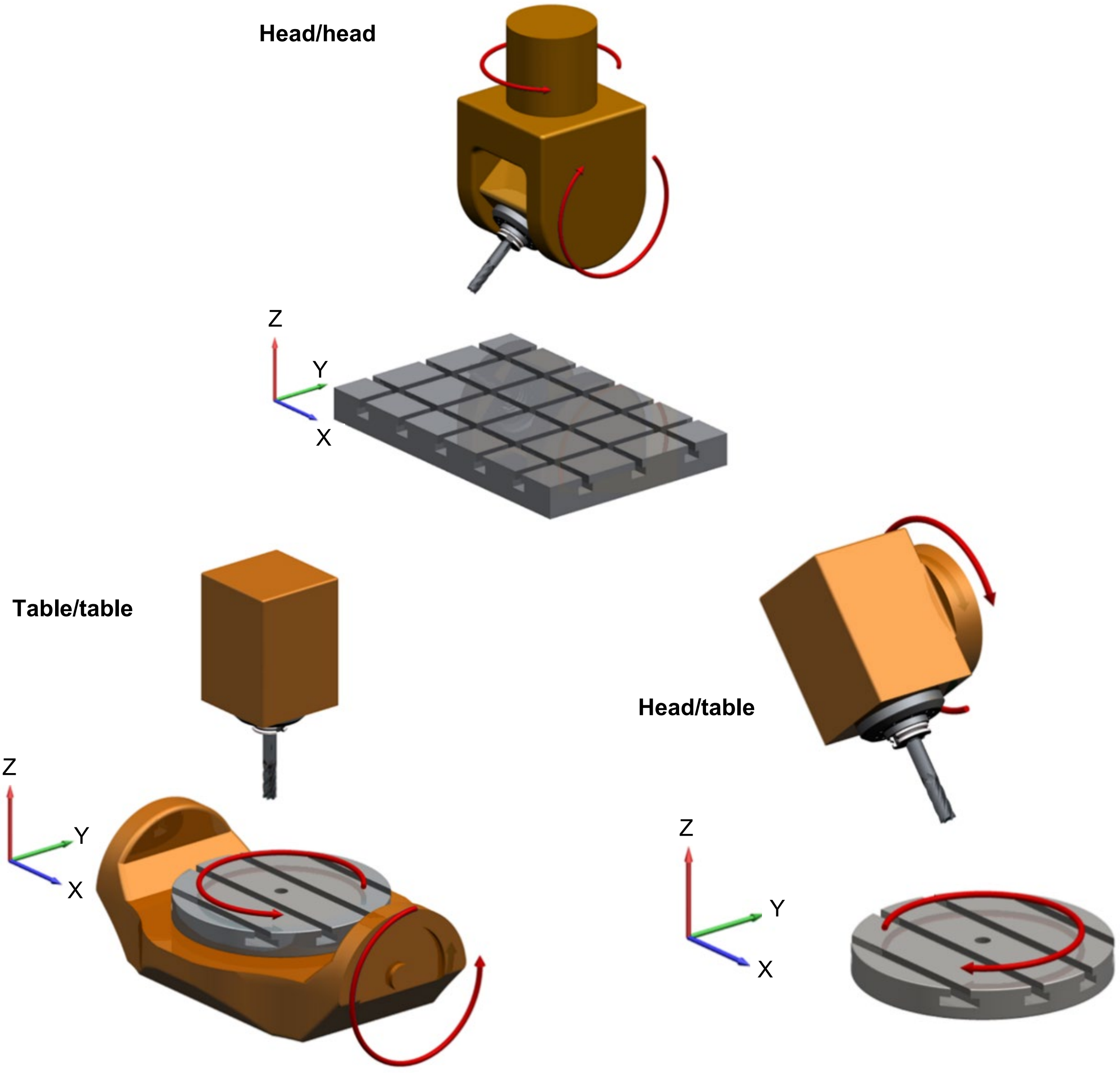


Figure C.1 Machine configurations

If your machine configuration does not follow one of these, then this software may not function correctly. Please contact your local Renishaw representative for further support.

Multi-axis commands

There are many 3+2 or 5-axis commands available on Fanuc/Meldas controllers, the most common of which are detailed on the following pages.

Renishaw Inspection Plus software can work with some of these functions active. The functionality of the Renishaw cycles is dependent on the feature co-ordinate system (FCS) in relation to the tool posture.

Alignment of the tool posture to the active FCS can be achieved using functions such as G53.1 or G53.6 or by using manual positioning methods.

Below is a diagram explaining the meaning of some of the terms which will be referred to in the following pages.

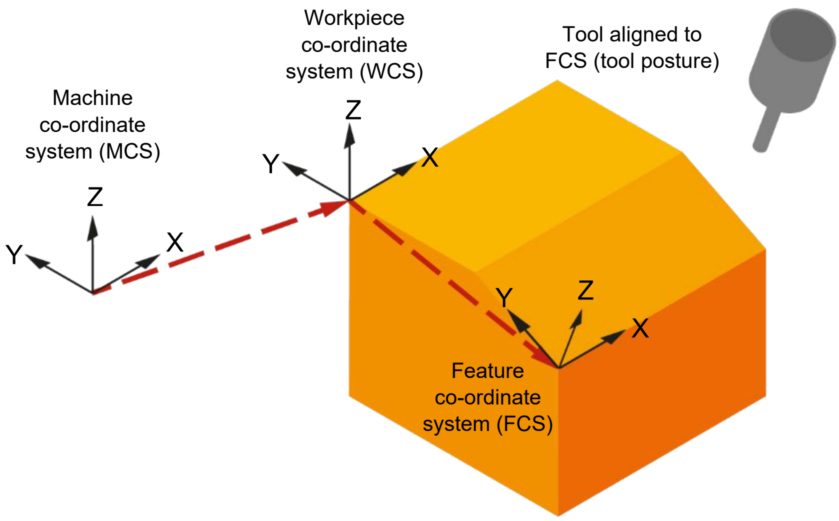


Figure C.2 Co-ordinate systems

NOTE: The following pages give an overview of typical 3+2 or 5-axis functions. However, a detailed explanation of these functions is not given, so prior knowledge is required if using them to fully implement probing routines.

FCS rotations - G68, G68.2, G68.3 and G68.4

FCS rotations are typically called Tilted Working Plane (TWP) on Fanuc controls and Inclined Plane Machining (IPM) for Meldas controls.

This function allows rotation and translation of the active WCS to a designated position called the feature co-ordinate system (FCS). Any positional data programmed with this function active will now be in relation to this new position.

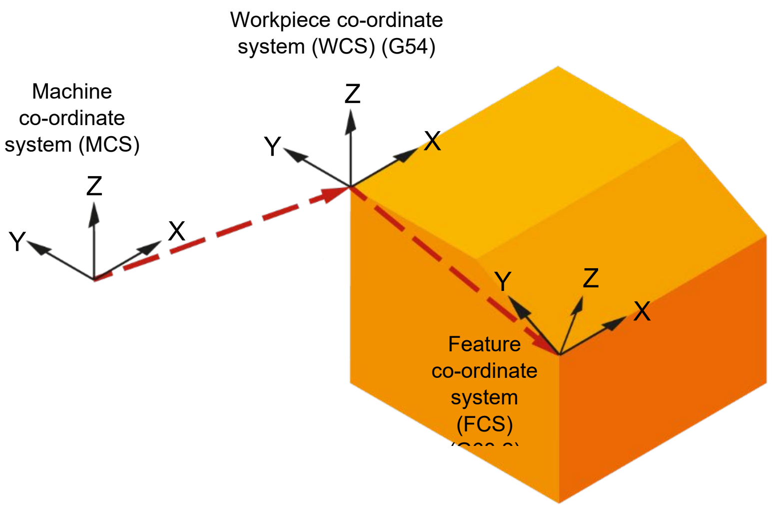


Figure C.3 FCS rotations

FCS rotation cancellation – G69

The FCS rotation functions are cancelled using a G69 command. This reactivates the original WCS.

Use with probing

Renishaw cycles will give correct measurement data with these cycles active, if the tool posture is aligned to the Z of the FCS. Setting of a rotated WCS is also available.

Example

G90 G80 G40 G0	
G69	Cancel TWP/IPM functions.
T1 M6	
G53 Z0.	
G53 A0. F1000.	
G56	Set active WCS.
G90 G1 G43 Z400. H1 F3000.	Activate probe offset.
G68.2 X0. Y0. Z0. I0. J90. K0.	Activate TWP/IPM with 90° rotation.
G53.6 H1 R1.	Align tool posture to active Z of FCS.
G65 P9810 Z100. F3000.	Protective positioning in Z.
G65 P9810 X0. Y0. F3000.	Protective positioning in X and Y.
G65 P9810 Z−6. F3000.	Protective move into bore.
G65 P9814 D50.	Measure bore.
G65 P9810 Z50. F3000.	Protective move out of bore.
G69	Cancel TWP/IPM function.
M30	

Dynamic fixture offset (DFO) – G54.2

This function allows for any mispositioning or rotation of a component from the previously programmed WCS position. This allows a program which has been created with a WCS in a specific MCS position to be run with the workpiece in a new WCS location.

This function is usually used when a component has been programmed using a CAM system. The program is created with the component in a known position in the machine. If the component is not located in the machine in the exact location where the CAM system programmed the part, then the output will be incorrect.

The G54.2 function allows a part to be mispositioned, with the error from the original to the new position input into G54.2. The original program then can be still run.

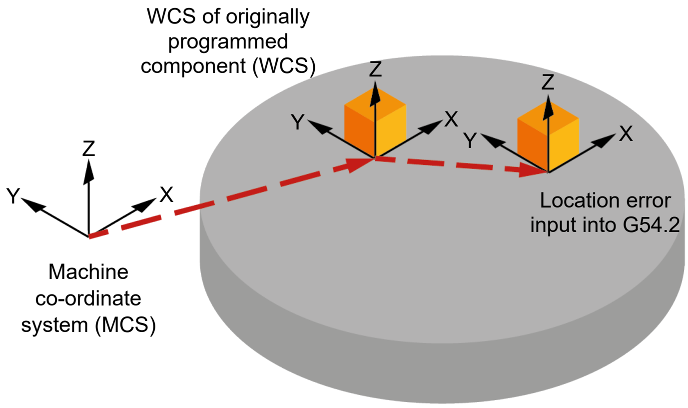


Figure C.4 Dynamic fixture offset (DFO)

DFO cancellation – G54.2 P0

The DFO functions are cancelled using a G54.2 P0 command. This reactivates the original WCS.

Use with probing

The original WCS and the G54.4 error values must be correctly set up prior to running any probing cycles.

Renishaw cycles will give correct measurement data with these cycles active, if the tool posture is aligned to the Z of the FCS.

Example

G90 G80 G40 G0	
T1 M6	
G53 Z0.	
G53 A0 F1000.	
G56	Set active WCS.
G54.2 P1	Activate the DFO.
G90 G1 G43 Z400. H1 F3000.	Activate probe offset.
G65 P9810 Z10. F3000.	Protective positioning in Z.
G65 P9810 X0. Y0. F3000.	Protective positioning in X and Y.
G65 P9810 Z−6. F3000.	Protective move into bore.
G65 P9814 D50. S5.	Measure bore and set G58 work offset.
G65 P9810 Z50. F3000.	Protective move out of bore.
G54.2 P0	Cancel DFO function.
M30	

Dynamic fixture offset (DFO) II – G54.4

This function is similar to the DFO G54.2 function, with the added ability to allow for multiple rotational errors of the component from the previously programmed WCS position. This function is set up and programmed in the same way as DFO G54.2.

Use with probing

The original WCS and the G54.2 error values must be correctly set up prior to running any probing cycles.

Renishaw cycles will give correct measurement data with these cycles active, if the tool posture is aligned to the Z of the FCS.

NOTE: Setting of a G54.4 is not currently available with this Renishaw Inspection Plus software.

Tool control point (TCP) – G43.4/G43.5

The TCP (G43.4/G43.5) function provides simultaneous axis control for moving the tool on a 5-axis control machine so that the tool tip point follows a path, at such a rate of feed as programmed in terms of the relative position of the tool to the workpiece.

This type of programming is created by using a CAM system, therefore there can be many different approaches, formats and set-ups for obtaining the correct output.

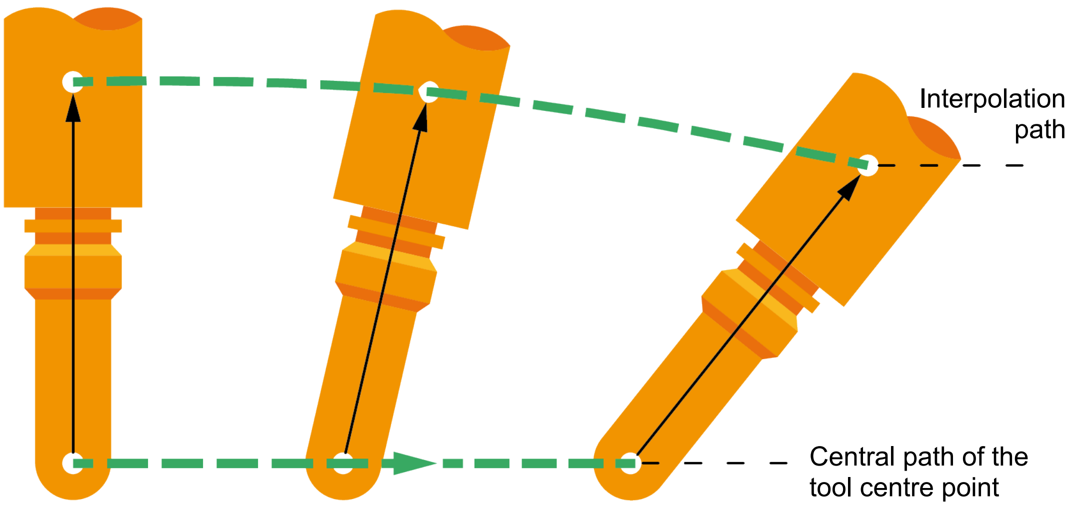


Figure C.5 Tool control point (TCP)

Use with probing

Probing can be possible when using TCP functions, however, this can only be achieved if the Z of the active co-ordinate system is aligned to the tool posture. This is often not the case, as CAM systems program using the initial WCS and translate and rotate around this.

An in-depth knowledge of these TCP full 5-axis functions and machine set-up is required to be able to implement probing routines.

NOTE: This Renishaw Inspection Plus software is not a generic package to suit any type of probing in TCP mode. For further details and support, please contact your local Renishaw representative.

Parameters

The following parameters need to be set when using this Inspection Plus software.

CAUTION: Any parameter changes may affect the functionality of other systems on the machine. Confirm any parameter changes prior to use. Existing Renishaw installations (such as tool setters and lasers) may be affected by parameter changes owing to how tool lengths are applied and calculated in the software.

The following parameters are only available on Fanuc Series 30*i* controllers (P5400.5) and above & Meldas (8713).

Parameter P5400.5 = 1 (Fanuc) \ 8713=1 (Meldas)

Set P5400 bit 5 \ 8713 to 1. This allows the co-ordinates of the skip position (#5061 to #5063) and current position (#5041 to #5043) to be read in the active feature co-ordinate system (FCS).

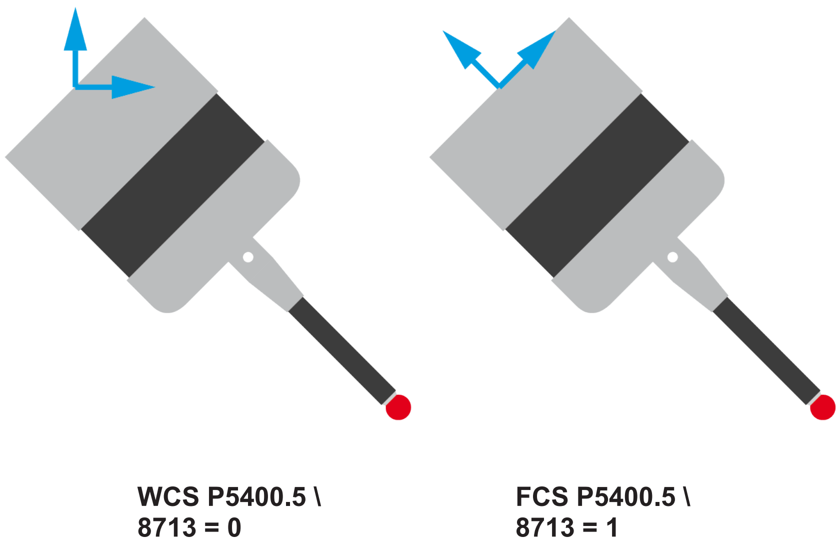


Figure C.6 Parameter P5400.5 \ 8713

With parameter P5400.5 \ 8713 set to 1, the Z of the FCS will always be aligned to the probe centre line.

Parameter P6019.4 = 1

With P6019 bit 4 set to 1, the current tool length is added to the co-ordinates of the skip position (#5061 to #5063) and current position (#5041 to #5043). This allows the end of the probe to be the controlled and measuring point.

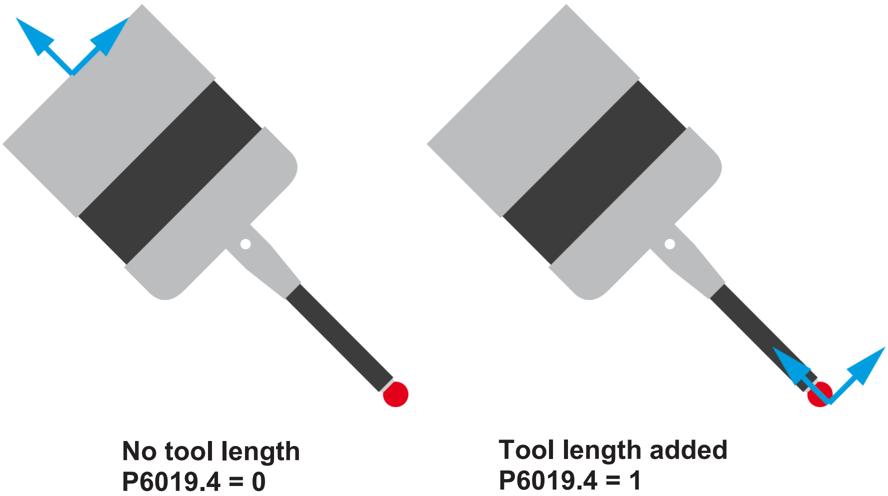


Figure C.7 Parameter P6019.4

NOTE: If parameter P6019.4 is set to 1, then #116 in O9723 needs to be set to 0. #116=0 will mean no tool length is added to any Z measurements.

Updating the WCS with 5-axis commands active

When a feature co-ordinate system (FCS) is not aligned to the machine co-ordinate system (MCS), then the Fanuc system does not allow the correct update of the work co-ordinate system (WCS).

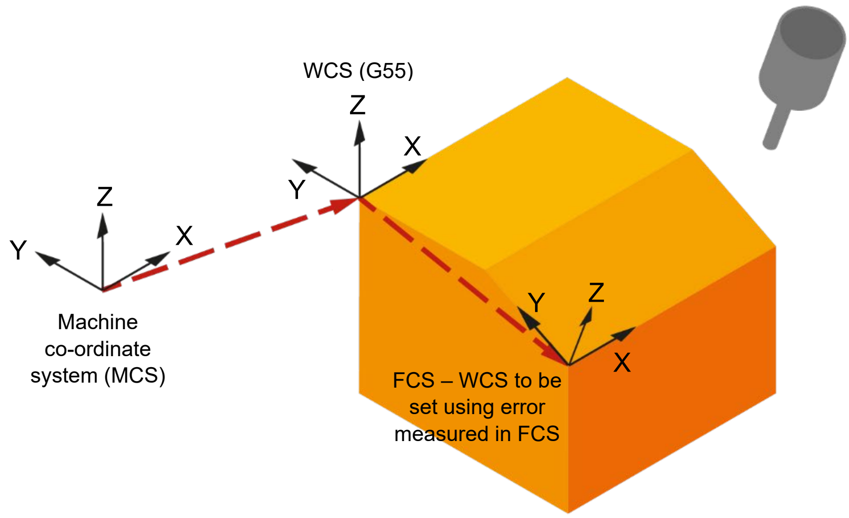


Figure C.8 Rotated WCS

Renishaw provides a solution to enable the update of a WCS by identifying the active FCS rotation and “back calculating” the errors.

The Renishaw solution consists of two parts, the first one of which is the “Identity move”. The data from the Identity move obtained here is used for further calculations in the WCS updating.

The Identity move is not required on the following Fanuc controllers, as the Identity data is supplied using system variables:

Fanuc Series 30*i*-0B
Fanuc Series 31*i*-0B

NOTE: Rotated WCS updates are directly related to machine tool performance. Inaccuracies in machine tool alignments and axis rotation centres will directly affect the tracking and updating of WCS positions.

CAUTION: Updating the extended WCS while the rotated WCS setting is active is not supported on Meldas controllers.

Calculations

To calculate the FCS errors into the MCS for setting the WCS, three errors in X, Y and Z are required.

The X, Y, Z errors are automatically stored within the macros when running measuring cycles. These errors are then used for rotated WCS setting when required. The stored error values are cleared when a rotated WCS is set.

The calculation method requires the three errors (X, Y, Z) to be present before the identity move, and so WCS, can be set.

NOTE: The three measurement errors in X, Y and Z must be found when setting a rotated WCS. Failure to achieve this will result in an alarm.

Cycle description (O9744)

This program is automatically called by the Inspection Plus software when running with any 3+2 or 5-axis functions active. This program is not used if the machine is in standard mode.

This program identifies the active FCS rotation by making a small identity move. From the resulting data, the measured errors are calculated back into the WCS for correct WCS setting or updating.

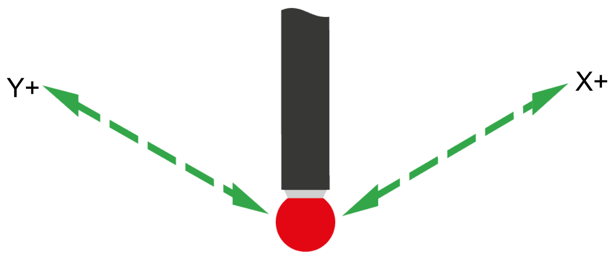


Figure C.9 XY move

O9744 (REN*FCS*TO*WCS)

#24=1.(IDENTITY*MOVE*DISTANCE*IN*MM) *** **edit 1**

#25=1000.(IDENTITY*MOVE*FEEDRATE*MM/MIN) *** **edit 2**

*** **edit 1** Adjust this value if the move causes the probe stylus to collide during the identity move.

NOTE: Reduction of this value may decrease the accuracy of the WCS calculation.

*** **edit 2** Adjust this value to suit the machine.

NOTE: Reduction of this value may increase the cycle time of the identity move.

Program example

T01 M06	Select the probe.
G54 X0. Y0.	Start position.
G43 H1 Z100.	Activate offset 1 and go to 100 mm (3.94 in) above.
G68.2 X0. Y0. Z0. I0. J45. K0.	Plane rotated by 45°.
G53.1	Rotary axes positioned normal to surface (FCS Z = tool centre line).
G65 P9832	Switch the probe on (this includes M19).
G65 P9810 X0. Y0. Z10. F3000.	Protected positioning move over the centre of the boss clear of the Z surface.
G65 P9811 Z0.	Z surface measure. (Z error stored. No update selected yet.)
G65 P9814 D50. Z−10. S2	Measure a 50 mm (1.968 in) diameter boss. (X, Y error stored. G55 set using X, Y, Z errors in rotated calculations.)
G65 P9810 Z100.	Protected positioning move.
G65 P9833	Switch the probe off (when applicable).
G28 Z100.	Reference return.
G69	
continue	

The centre line of the feature in the X and Y axis and Z surface is stored in work offset 02 (G55).

Renishaw plc

New Mills, Wotton-under-Edge
Gloucestershire, GL12 8JR
United Kingdom

T +44 (0)1453 524524
F +44 (0)1453 524901
E uk@renishaw.com

www.renishaw.com

apply innovation™

For worldwide contact details, visit
www.renishaw.com/contact

H-5755-8600-06