Programming manual
H-2000-6525-0B-A

RENISHAW
apply innovation™

Five-face contact tool setting cycles (for Fanuc and Meldas controllers)

Disclaimer

RENISHAW HAS MADE CONSIDERABLE EFFORTS TO ENSURE THE CONTENT OF THIS DOCUMENT IS CORRECT AT THE DATE OF PUBLICATION BUT MAKES NO WARRANTIES OR REPRESENTATIONS REGARDING THE CONTENT. RENISHAW EXCLUDES LIABILITY, HOWSOEVER ARISING, FOR ANY INACCURACIES IN THIS DOCUMENT.

Trade marks

Renishaw part no: H-2000-6525-0B-A

Issued: 07.2014

IMPORTANT – PLEASE READ CAREFULLY

RENISHAW PRODUCT LICENCE

Licensee: you, the person, firm or company accepting the terms of this Licence

Renishaw: Renishaw plc, New Mills, Wotton-under-Edge, Gloucestershire, GL12 8JR, United Kingdom

Product: the software, which is designed to operate on machine tool numeric controllers, supplied by Renishaw for use with Renishaw's machine tool probing systems

Licence to use: a non-exclusive licence to use the **Product** on a single machine tool only

By installing and/or using the **Product** you indicate your acceptance of the terms of this Licence.

Renishaw grants the **Licensee** a **Licence to use** the **Product** on condition the **Licensee** accepts the following terms and conditions:

1. All rights in and title to the **Product** are and shall remain vested in **Renishaw** and its licensors.

2. **Renishaw** shall replace or repair the **Product** if it does not materially perform to specification under proper use within 90 days of delivery. This warranty does not apply where the **Product** has been modified in any manner that is not specifically described in the **Product** or in the installation or programming manuals supplied with the **Product**, or where the **Product** is used with probing systems that have not been produced by **Renishaw**. Except as stated in this paragraph, all warranties, conditions and terms implied by law are excluded. In particular, no warranty is given that the **Product** is bug or error-free.

3. **NOTE - LIMITATION OF LIABILITY IN CONNECTION WITH USE OF THE PRODUCT**

   Renishaw does not exclude liability for personal injury or death caused by **Renishaw's** negligence.

   Renishaw's liability is limited to (a) the warranty contained in paragraph 2 and (b) direct losses up to a maximum of £50,000.

   Renishaw has no liability to the Licensee for any indirect, consequential or economic loss (including, without limitation, loss of data, profits or goodwill).

   The **Product** has been designed for use with **Renishaw**'s machine tool probing systems. **Renishaw** has no liability for the results of using the **Product** with another manufacturer's machine tool probing systems.

   By accepting the terms of this Licence the **Licensee** agrees that this limitation of liability is reasonable.

4. The **Licensee** may not make any copies of the **Product** except as provided in this Licence or as permitted by applicable law. The **Licensee** is authorised to make a backup copy of the **Product** for security purposes. The **Licensee** must not remove any licence and copyright notices, labels or marks contained in the original and shall ensure all copies contain such notices without modification.

5. If the **Product** contains electronic manuals the **Licensee** may print out the manuals in part or in full, provided that the print outs or copies are not supplied to any third party that is not an employee or contractor for the **Licensee** without **Renishaw's** written permission

6. The **Licensee** shall not reverse engineer, decompile, or modify the **Product** or re-use any components separately from the **Product** unless permitted by a specific instruction contained in the **Product** or the programming or installation manuals supplied with the **Product** or by applicable law provided that in the latter case, **Licensee** has first contacted **Renishaw** to request any information required to interface with **Licensee's** other software.

7. The **Licensee** shall not make the **Product** available to any third party in any manner whatsoever nor may this Licence and the **Product** be transferred to a third party without **Renishaw's** prior written agreement. Any agreement by **Renishaw** is conditional on the permitted transferee agreeing to all terms of this Licence and the **Licensee** not retaining any copies of the **Product**. Where the **Licensee** is a reseller of **Renishaw's** machine tool probing systems, **Licensee** may transfer the **Product** for ultimate use by an end user with **Renishaw's** machine tool probing systems.

8. **Renishaw** shall have the right to terminate this Licence immediately if the **Licensee** fails to comply with any of these terms and conditions. The **Licensee** agrees upon receipt of notice of termination from **Renishaw** to immediately return or destroy all copies of the **Product** in its possession or control.

9. This Licence is governed by English law and the parties submit to the exclusive jurisdiction of the English courts.

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 Customer Support office (see www.renishaw.com/contact for the address and telephone number). 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 disk(s) .....
Interface type	.....
	.....
Tool setting probe type	Tool setting disk(s) .....
Interface type	.....
	..... .

SPECIAL SWITCHING M-CODES (OR OTHER) WHERE APPLICABLE

	Dual systems only
Switch (Spin) probe on	Switch on inspection probe .....
Switch (Spin) probe off	Switch on tool setting probe .....
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 tel. no.	Date of training .....
Customer's contact name	

SOFTWARE DEVIATION RECORD

Standard Renishaw kit no.	Software disk nos.
Reason for deviation	
Software no. and macro no.	Comments and corrections

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 over 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 working correctly 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 that the machine will be instructed to make under program control will 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.

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 P9857 D50.01 Z6.0 K.01 H2.0

may be entered as:

G65P9857D50.01Z6.0K.01H2.0

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.

This page is intentionally left blank.

Table of contents

Chapter 1 Getting started

Chapter 2 Software installation

Chapter 3 Calibrating the stylus

Chapter 4 Manual length or length and radius measurement

Chapter 5 Automatic length and radius measurement

Chapter 6 Broken tool detection

Chapter 7 Thermal compensation cycle

Chapter 8 Advanced options

Chapter 9 Alarms

Chapter 1

Getting started

Before you start to use the tool setting software, take time to read this chapter. It will provide you with a basic understanding of the importance of accurately calibrating the probe you intend to use for tool setting. Only when the probe is accurately calibrated can you achieve total quality control over your manufacturing process. This chapter also provides you with some guidance regarding the most suitable operating conditions for your probe.

Contained in this chapter

Why calibrate your probe?

In Chapter 3 of this manual you will find details of how to calibrate your Renishaw tool setting probe. But why is it so important that your probe is calibrated?

After your probe has been assembled and mounted on the machine table, it is necessary to align the faces of the stylus with the machine's axes to avoid probing errors when setting tools. Take care with this operation – you should try to get the faces aligned to within 0.010 mm (0.0004 in) for normal use. This is achieved by manually adjusting the stylus with the adjusting screws provided, and using a suitable instrument such as a DTI clock mounted in the machine spindle.

After the probe has been correctly set up on the machine, the probe must be calibrated. Calibration cycles are provided for this task. The purpose is to establish the trigger point values for the measuring face of the probe's stylus under normal measuring conditions. The calibration values are stored in macro variables for computation of the tool size during tool setting cycles.

Values obtained are axis trigger positions (in machine co-ordinates). Any errors due to machine and probe triggering characteristics are automatically calibrated out in this way. These values are the electronic trigger positions under dynamic operating conditions, and not necessarily the true physical stylus face positions.

NOTE: Poor repeatability of probe trigger point values indicates that either the probe/stylus assembly is loose or a machine/probe fault exists. Further investigation is required.

As each Renishaw tool setting probe system is unique, it is essential that you calibrate it in the following circumstances:

- If it is the first time your probe system is to be used.
- If a new stylus is fitted to your probe.
- If it is suspected that the stylus has become distorted or that the probe has crashed.

Notes on tool speed and feedrates

CAUTION: Setting a tool by rotating it against the stylus is suitable for most tools. However, some tools, such as those with carbide tips or delicate cutting teeth, may suffer from cutting edge deterioration as a result of contact with the stylus under these conditions.

The following parameters for operating conditions have been found by experience to suit Renishaw tool setting probes. Improvement and optimisation may be possible for specific applications.

The table-mounted probe is suitable for setting tool lengths (non-rotating). Cycles are also provided with the capability to set rotating tools for length and radius.

First touch spindle RPM

RPM for the first move onto the probe is calculated from a surface cutting speed of 60.0 m/min (197.0 ft/min). This is maintained within the range 150 rpm to 800 rpm and relates to a range of 24.0 mm to 127.0 mm (0.95 in to 5.0 in) diameter cutters. The surface cutting speed is not maintained outside this range.

First touch feedrate

The feedrate is calculated as follows:

F= 0.15 × rpm F units mm/min.

Second touch spindle RPM

800 rpm.

Second touch feedrate

4.0 mm/min feedrate (0.16 in/min) resolution 0.005 mm/rev (0.00020 in/rev).

Tool offset types supported

Positive tool offset applications

The tool setting system software is ideally suited to setting tools using positive tool offset values that represent the physical length of the tool.

Throughout this guide the descriptions refer to positive tool offset applications. The software can also be used in applications where negative tool offset values are used or where all tool offset values are entered as ± values relative to a master tool.

Negative tool offset applications

The offset value entered is the distance the tool tip must be moved from the home position to reach the zero (0) position of the part program (air-gap method), rather than the physical length of the tool.

Chapter 2

Software installation

The tool setting software is supplied with standard settings. These may be adjusted during installation to suit a specific machine. This chapter describes how to adjust the settings.

Contained in this chapter

Introduction

The software is supplied on CD. When the CD is inserted into the PC it will automatically launch a “wizard”. The screen will display the following options:

- Programming manual.
- Readme file.
- Generate macros.

Click on “Generate macros” and enter appropriate values in the fields; information will be displayed to help you. When you have completed all the fields, click on the “Run” key at the bottom of the screen. The system will now generate the macros required for your machine set-up. The macros will be stored on your PC in the directory and file shown above the Run key. These macros can now be loaded to the machine.

If for any reason the wizard does not work, this chapter will help you to edit the macros manually to suit your machine. Do this as follows:

1. From your PC, explore the CD and locate a folder called “Macros”.
2. Open this folder and locate a file called “Macro”.
3. Copy this file onto your PC.
4. Either edit this file on your PC or load it to your machine and edit it there.

Macro variables

The following variables are used by the tool setting system software:

- #500-series macro variables are used for the calibration data.
- #100-#149 series macro variables are used for the setting data.
- Macro variables #1 to #31 are reserved for locally defined data.

Variable #120 is used to define the base number of the calibration data variables. This number can be changed to avoid conflicts with other software applications.

Settings data macro O9750

Read the following parameter descriptions then edit macro O9750 as required.

NOTE: All values must be in metric.

#101 First probe touch feedrate.

Default: 200 mm/min

#102 Tool offset type.

1 = A type, one register per tool

2 = B type, two registers per tool – geometry and wear

3 = C type, four registers per tool – length geometry and wear and radius geometry and wear

Further information regarding tool offset types for other controls can be found in the Readme file.

#103 Single-sided measurement setting (see page 2-5).

#104 Probe orientation (see page 2-5).

#105 Back-off distance (see page 2-6).

Default: 0.3 mm

#106 Two tool setting probes (0 = No, 1 = Yes).

Default: 0

#109 Tool offset register type (1 = Radius, 2 = Diameter).

Default: 1

#110 Tool diameters larger than this size rotate.

Default: 10 mm

#111 Tool diameters larger than this size are measured single-sided (see page 2-5).

Default: 100 mm

#112 Reserved for future use.

#113 Initial approach clearance point above the stylus.

Default: 100 mm

#114 Secondary approach clearance point above the stylus.

Default: 10 mm

#117 Default overtravel distance.

Overtravel is the distance towards the stylus that the tool is permitted to move before a PROBE FAIL alarm is initiated.

Default: 5 mm

#118 OTS/RTS tool setting probe (Yes = 1, No = 0).

Default: 0

#119 Reserved for future use.

#120 The base number for #500 series calibration data.

Default: 520

#121=1 Machine X axis.)
#122=2 Machine Y axis. > Modify for multi-axis option only (see Chapter 8,
#123=3 Machine Z axis.) “Advanced options”).

#124 ‘Long tool/Short tool’ search feedrate.

Default: 2000 mm/min

#125 Radial clearance.

Radial clearance is the distance between the tool and the stylus when moving down the side of the stylus.

Default: 5 mm

#127 Rapid traverse feedrate.

Default: 5000 mm/min

#138 Long tool value.

Default: 0 (option not active) (see page 2-6)

#139 Short tool value.

Default: 0 (option not active) (see page 2-6)

#145 In-position zone, used for checking whether the stylus is already triggered at the beginning of the measurement move. Typically this value does not require adjustment.

Default: 0.005 mm

Probe orientation (#104) and single-sided diameter measurement (#103)

#103, #104 and #111 must be set in the settings macro (O9750).

#104 is the orientation of the probe.

#103 selects which side of the stylus is used when measuring diameters larger than the value set in #111 (single-sided measurement).

Examples:

Option	Probe orientation #104	Side selection #103
1	2	1
2	2	-1
3	1	-1
4	1	1
5	-2	-1
6	-2	1
7	-1	1
8	-1	-1

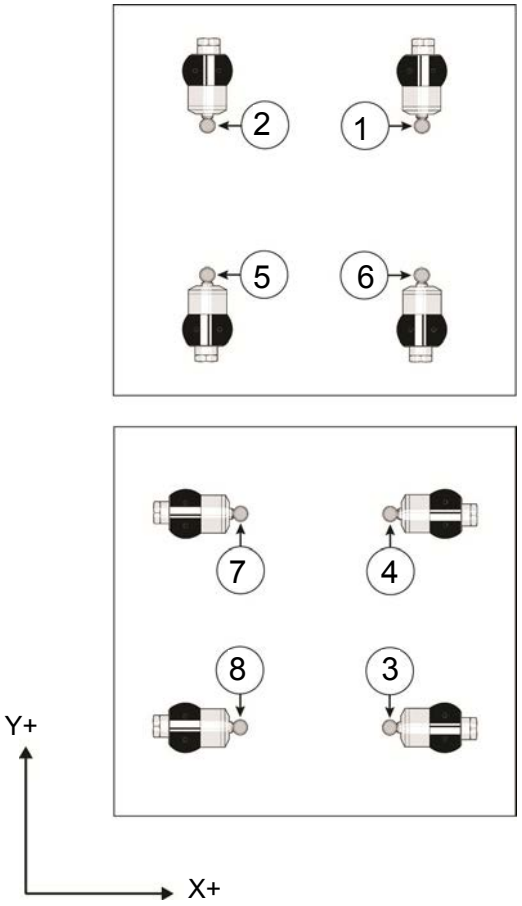


Figure 2.1 Orientation of the probe and single-sided diameter measurement settings

Adjusting the back-off distance #105

A back-off distance #105 is provided for adjusting the distance moved off the surface prior to the final measuring move.

The software loads a default value of 0.3 mm when first run. This stored value in #105 should be optimised for a minimum cycle time.

Adjust the back-off distance #105 by repeating the static length setting cycle. Reduce the #105 value each time until the tool just clears the stylus surface prior to the second touch.

NOTE: When the value is too small, a PROBE OPEN alarm occurs.

‘Long tool/Short tool’ option (#138 and #139)

This function is used only in program O9857 (automatic length setting) and can only be used when measuring tools on-centre.

The ‘Long tool/Short tool’ option is enabled by entering the maximum tool length into #138 and the minimum tool length into #139 in the settings macro O9750. The tool setting cycle will automatically search for and measure the length of a tool within the minimum and maximum lengths set. No tool offset is required in the tool offset page.

The cycle will automatically move the spindle to the home position in the tool axis. It will then position centrally over the stylus and feed at the rapid traverse rate (#127 set in O9750) to the long tool position above the stylus. It will then feed the tool towards the stylus at the feedrate set in #124, until a trigger is detected. If the tool is not detected within the set range, a PROBE FAIL alarm will be displayed.

Settings in O9750

#138 Maximum tool length.
#139 Minimum tool length.
#124 Search feedrate.

NOTE: If #138 and #139 are set to zero, the ‘Long tool/Short tool’ search will be disabled. In this case, the approximate tool length must be stored in the offset register prior to measurement, or the Y input must be programmed.

CAUTION: If ‘Long tool/Short tool’ search is enabled and a D input is programmed that is greater than the setting in #110 (tool diameters larger than this size rotate), the Y input must be used (approximate tool length).

Chapter 3

Calibrating the stylus

This chapter describes how to calibrate the probe's stylus on the machine. This must be done before using the tool setting cycles.

Contained in this chapter

Calibrating the stylus – macro O9855

Description

Macro O9855 is used for calibrating the probe's stylus.

Select the master tool in MDI mode and position it centrally over the probe's stylus using either jog or handwheel mode. The diameter and length of the master tool must be known.

The cycle moves the master tool from the start position to the stylus face(s), as specified by the probe orientation variable #104 in settings macro O9750. Calibration values are found, or calculated, for the stylus position (metric only).

Application

1. Set the probe's stylus faces parallel to the axes (or parallel to the top face, if a round stylus is used).

2. Load the master setting tool into the spindle using a program command or MDI mode.

3. Prepare a simple program to call the cycle, using the G65 P9855 command. Enter other optional inputs (see "Inputs").

4. Before running the calibration cycle, the master tool length must be input in the tool offset page.

5. **IMPORTANT:** Ensure that the calibration tool has minimal run-out and the exact stylus size is input into the program call line. Position the tool at a suitable start point, using either jog or handwheel mode, so that it is centrally over the stylus and within approximately 50 mm (2 in) of the top face.

6. Run the O9855 cycle. The tool should move down 15 mm (0.59 in) and then the alarm "PROBE FAIL" should be raised. This confirms that the setting of #112 is correct.

   If the tool does not move down 15 mm (0.59 in), but instead moves up or down by the tool offset length less 15 mm (0.59 in), adjust setting #112 in program O9750.

7. Position the tool 10 mm (0.393 in) above the stylus and run the cycle O9855.

Format

G65 P9855 Rr Tt Xx Yy [Cc Qq Uu Vv Zz]

or

G65 P9855 Dd Rr Tt [Cc Qq Uu Vv Zz]

where [ ] denotes optional inputs.

Inputs

Cc	=	The distance from the top face (Z) to the underside of the stylus. (This must be input if using measuring cycles that feed upwards.)
Dd	=	The diameter of the round stylus if X and Y inputs are not used (see Figure 3.3).
Qq	=	The overtravel distance.
Rr	=	The actual diameter of the master setting tool.
Tt	=	The tool length offset to use. (OFFSET MUST BE SET CORRECTLY.)
Uu	=	X step-over distance, used during spindle axis calibration.
Vv	=	Y step-over distance, used during spindle axis calibration.
Xx	=	The stylus width (see Figure 3.2).
Yy	=	The stylus width (see Figure 3.2).
Zz	=	The distance from the top face of the stylus to the measuring point on the side faces.

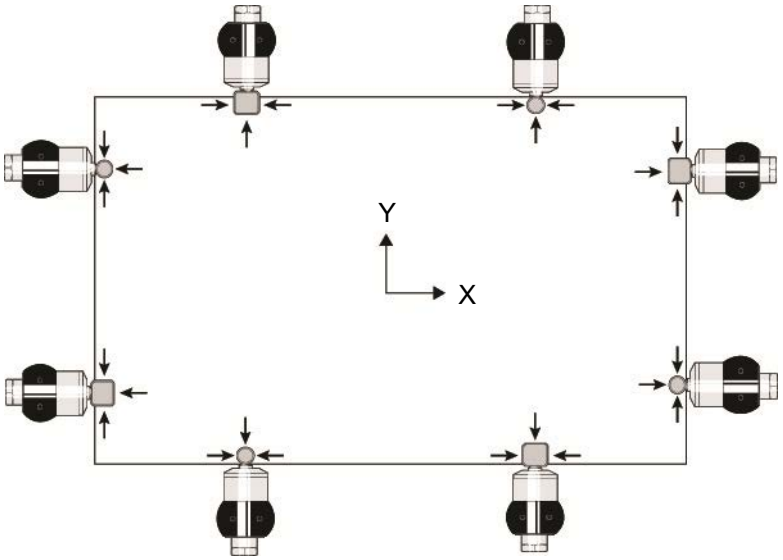


Figure 3.1 Example of machine tool movements

Calibration examples

Setting the XY stylus

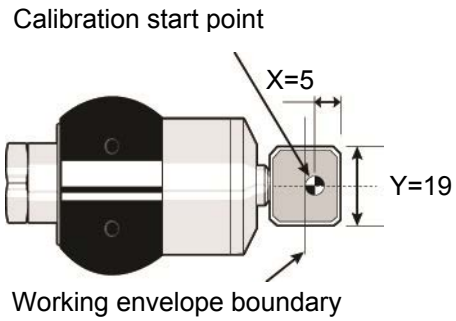


Figure 3.2 Setting the XY stylus

This will enable the stylus to be positioned just inside the working envelope of the machine.

Example:

Position the calibration tool 10 mm (0.393 in) above the top face of the stylus, as shown in Figure 3.2.

G65 P9855 R6.0 T21. X5.0 Y19.0

After calibration, tools will be measured 5 mm (0.196 in) from the edge of the stylus.

Setting a round stylus

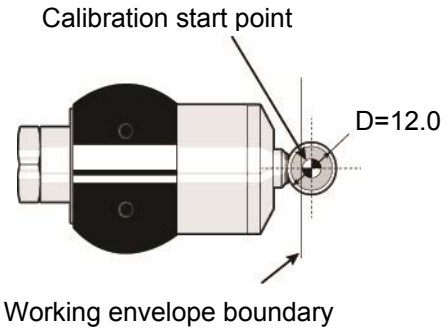


Figure 3.3 Setting a round stylus

Example:

Position the calibration tool 10 mm (0.393 in) above the top face of the stylus, as shown in Figure 3.3.

G65 P9855 D12.0 R6.0 T21.

Parameter store for calibration data

Variable #120 is used to define the base number of the calibration data variables. This number can be changed to avoid conflicts with other software applications.

The following parameters are set automatically during the calibration cycles (in metric units).

#520 (520 + 0) Z-axis position of the top face of the stylus – static tools.

#521 (520 + 1) +X-axis move position of the stylus face – rotating tools.

#522 (520 + 2) –X-axis move position of the stylus face – rotating tools.

#523 (520 + 3) +Y-axis move position of the stylus face – rotating tools.

#524 (520 + 4) –Y-axis move position of the stylus face – rotating tools.

#525 (520 + 5) Z-axis position of the bottom face of the stylus – rotating tools.

#526 (520 + 6) Difference between rotating tools and static tools.

#527 (520 + 7) Thermal compensation reset.

NOTES: Two probes will require 23 consecutive free variables.

Entering input data on the cycle call line will override any other default conditions.

This page is intentionally left blank.

Chapter 4

Manual length or length and radius measurement

This chapter describes how to use the manual tool length or length and radius setting cycle. This cycle is used to measure the tool length or length and radius by manually positioning the tool 10 mm (0.393 in) above the top of the stylus.

Contained in this chapter

Manual length or length and radius setting cycle – macro O9856

Description

This cycle is used to manually measure the length or length and radius of a tool.

Application

The tool should be manually positioned 10 mm (0.393 in) from the stylus before running the cycle. No tool offset should be active.

If there is no B input, the cycle will drive the tool towards the stylus and measure the length only. To measure the length and radius, use the B3. input.

Format

G65 P9856 [B3. Dd Tt]

where [ ] denotes optional inputs.

Example: G65 P9856

This will measure the current spindle tool length on-centre.

Example 2: G65 P9856 D80.

This will rotate the tool and measure the length.

Example 3: G65 P9856 B3. D80.

This will rotate the tool and measure the length, then measure the radius.

Inputs

B3.	=	Measure the length and radius of the tool. If there is no B input, only the length will be measured.
Dd	=	The diameter of the tool being measured. This input is used when the tool is to be rotated during the measuring cycle.
Tt	=	The tool offset to be updated. **Default value:** Current spindle tool.

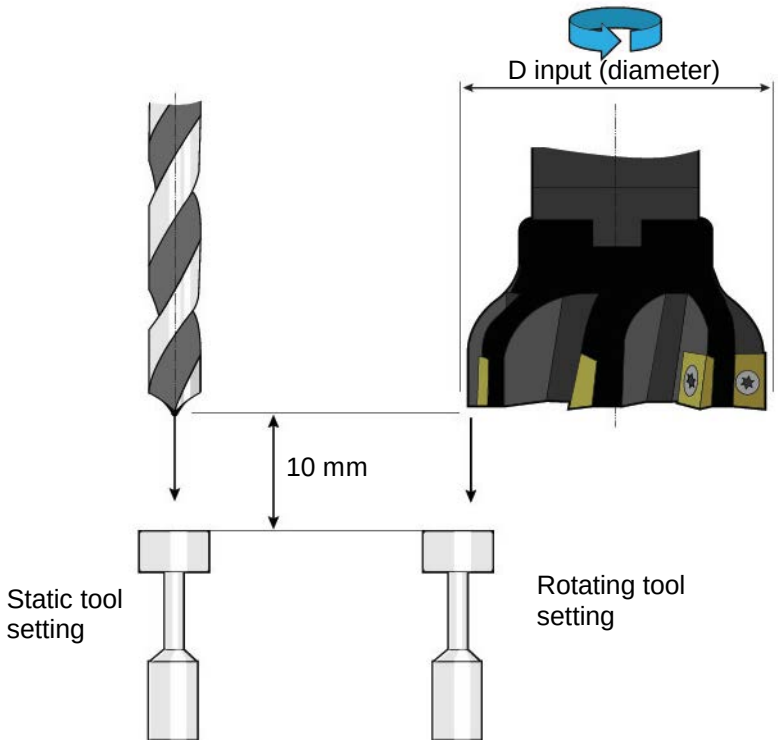


Figure 4.1 Manual positioning of tools before running cycle

This page is intentionally left blank.

Chapter 5

Automatic length and radius measurement

This chapter describes how to use the automatic length and radius measurement cycles.

Contained in this chapter

Automatic length setting – macro O9857

NOTE: Before using this cycle, the probe must have been calibrated. If #138 and #139 are set to zero, the 'Long tool/Short tool' search will be disabled. In this case, the approximate tool length must be stored in the offset register prior to measurement, or the Y input must be programmed.

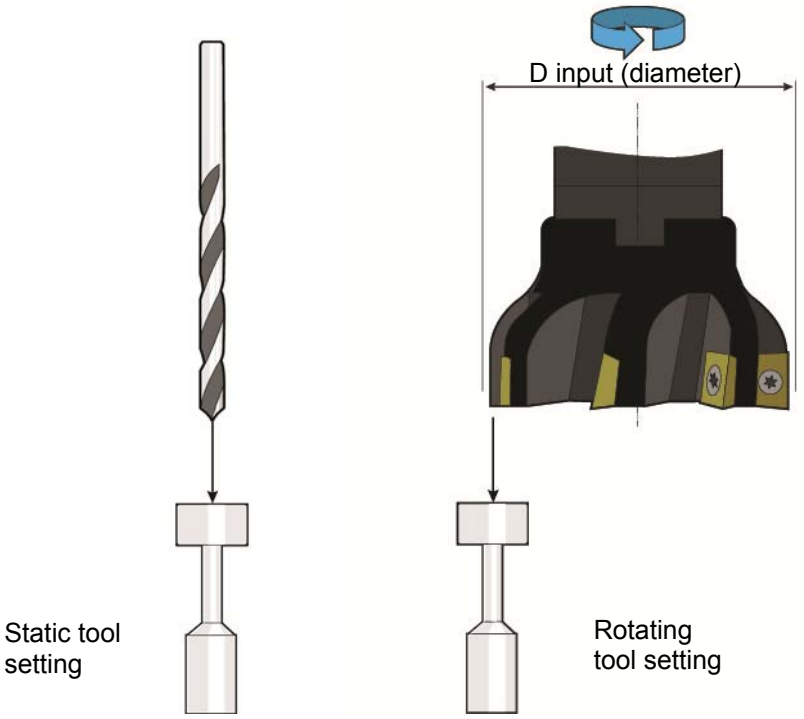


Figure 5.1 Tool length measurement

Description

This cycle is used to measure the effective cutting length of either a rotating or a non-rotating tool by taking a measurement on the tool setting stylus.

Application

The tool must be called into the spindle before the cycle is run.

The cycle automatically moves the tool to the initial clearance position (#113) above the stylus, and then to the correct position for measurement, before feeding to the secondary clearance position (#114) prior to the measurement move.

Alternatively, the 'Long tool/Short tool' search function can be used (see page 2-6).

After measurement, the tool returns to the home position in the Z axis.

Format

G65 P9857 [B1. Dd Hh Kk Mm Qq Tt Yy]

where [ ] denotes optional inputs.

Example: G65 P9857

This will measure the current spindle tool on centre.

Inputs

B1. = Set the length of the tool.

Default value: B1.

Dd = The cutter diameter (omit for non-rotating operation).

+d = right-handed cutting tool.

–d = left-handed cutting tool.

Example: D80. calls an 80 mm (3.15 in) diameter right-handed cutting tool.

Hh = The tolerance value that defines when the tool length is out of tolerance.

When this input is used, the tool offset is not updated if the tool length is found to be out of tolerance.

Default value: No tolerance check.

Kk = An experience value for length.

This value is the difference between the measured length and the effective length of the tool when under load during the cutting process.

Default value: Not used.

Mm = Tool out of tolerance flag.

Using M1. prevents a tool OUT OF TOLERANCE alarm from being raised.

Qq = The overtravel distance.

Default value: 5.0 mm

Tt = The tool offset to be updated.

Default value: Current spindle tool.

Yy = The approximate tool length value.

Default: No input (uses the values in the tool length register).

Outputs

The following outputs are set or updated when this cycle is executed:

	Set tool length.
#146	Out of tolerance flag. This is set when the measured tool length is out of tolerance, provided the H input is used. (1 = Out of tolerance, 0 = In tolerance).

Example 1: Length tool setting – non-rotating

G65 P9857 T2.	Enter set-up data. Measure length, set tool offset 2.

Example 2: Length tool setting – rotating

G65 P9857 D80.

Automatic radius/diameter setting – macro O9857

NOTE: Before using this cycle, the probe must have been calibrated. If the Y input is not being used, approximate tool offset values MUST be stored in the tool registers.

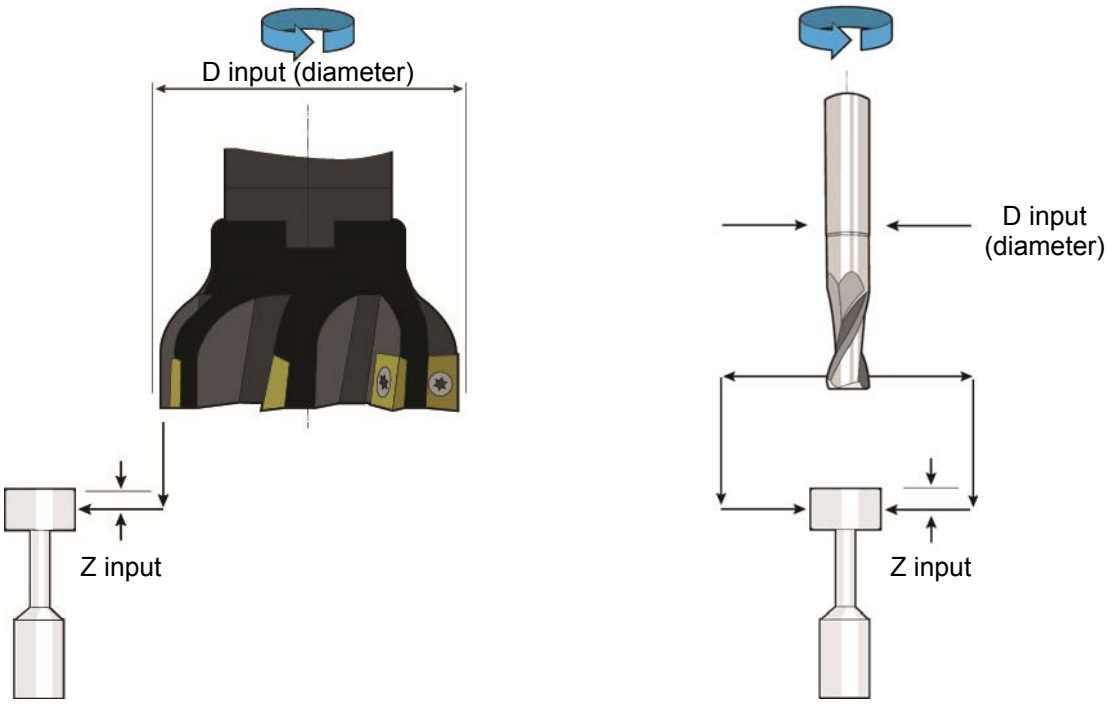


Figure 5.2 Tool cutting radius measurement

Description

This cycle is used to measure the effective cutting radius of a rotating tool by taking either one or two measurements on the tool setting stylus. The value of #111 in the settings data macro O9750 determines whether one or two measurements are taken. Tools that have a diameter greater than the value defined in #111 are measured single-sided.

Application

The tool must be called into the spindle with the correct tool length offset before the cycle is run.

The cycle first moves the tool to the centre position of the stylus and to the correct position to make either a single-touch or double-touch measuring move, as shown in the figure above. The tool then returns to the Z safe home position.

Format

G65 P9857 B2. Dd [Ee Hh Jj Mm Qq Tt Ww Yy Zz]

where [ ] denotes optional inputs.

Inputs

B2. = Measure the radius of the tool.

Dd = The cutter diameter.

+d = right-handed cutting tool.

–d = left-handed cutting tool.

Example: D80. defines an 80 mm (3.15 in) diameter right-handed cutting tool.

Ee = The tool offset to be updated if offset type A. If offset type B or C, then the current spindle tool is set as default.

Hh = The tolerance value that defines when the tool diameter is out of tolerance. When this input is used, the tool offset is not updated if the tool diameter is found to be out of tolerance.

Default value: No tolerance check.

Jj = An experience value for diameter or radius.

This value is the difference between the measured diameter/radius of the tool and the actual diameter/radius when the tool is under load during the cutting process.

Default value: Not used.

NOTE: For cutter centre-line programming applications, entering the nominal size as an experience value will result in the error being stored instead of the radius/diameter of the cutter.

Mm = Tool out of tolerance flag.

Using M1. prevents a tool OUT OF TOLERANCE alarm from being raised.

Qq = The overtravel distance.

Default value: 5.0 mm

Tt	=	The tool offset to be updated. **Default value:** Current spindle tool.
Ww	=	The extra Z clearance above the stylus when setting a diameter. **Example:** W20. will position 20 mm (0.79 in) + #114 above the stylus.
Yy	=	The approximate tool length value.
Zz	=	The measuring position on the stylus face. This is the Z-axis position from the top face of the stylus at which measurement takes place. **Default value:** 5.0 mm (0.197 in)

NOTE: A D input is compulsory if used with a B2., B3. or B4. input.

Outputs

The following outputs are set or updated when this cycle is executed:

	Set tool radius/diameter.
#146	Out of tolerance flag. This is set when the measured tool length is out of tolerance, provided the H input is used. (1 = Out of tolerance, 0 = In tolerance).

Example 3: Length tool setting – rotating with spigot

G65 P9857 D80. W30.

Automatic length and radius setting – macro O9857

NOTE: Before using this cycle, the probe must have been calibrated. If the Y input is not being used, approximate tool offset values MUST be stored in the tool registers.

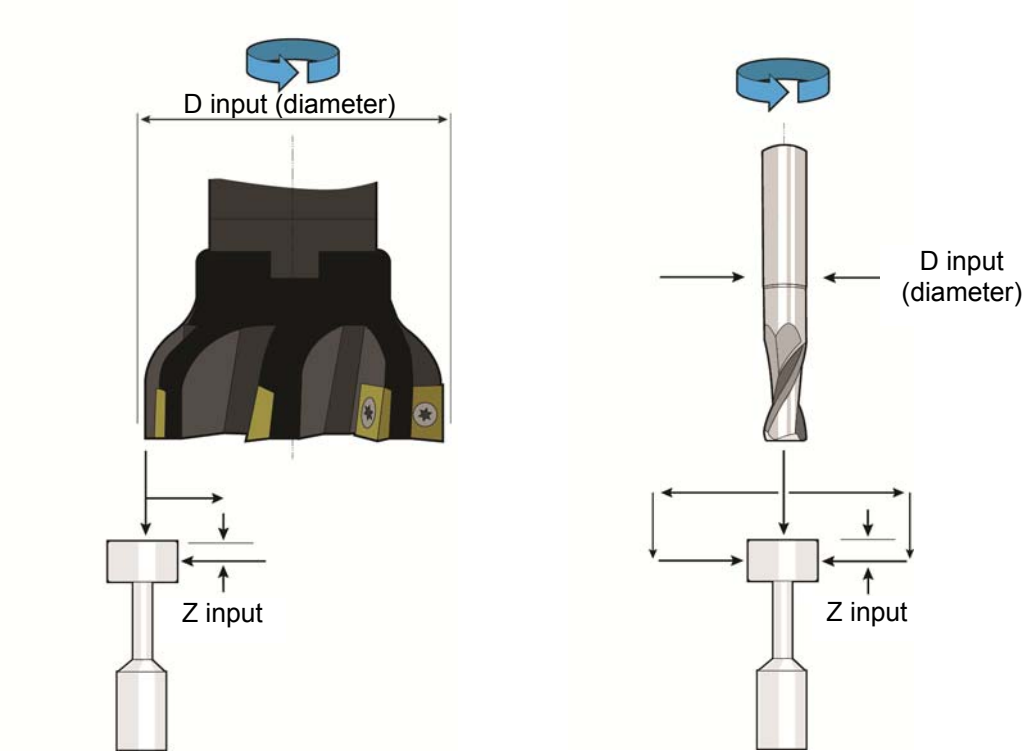


Figure 5.3 Cutting radius of rotating tool measurement

Description

The tool must be called into the spindle before the cycle is run.

This cycle combines the tool length measuring cycle (see “Automatic length setting” on page 5-2) and the tool radius/diameter measuring cycle (see “Automatic radius/diameter setting” on page 5-5).

Figure 5.3 shows the combined cycle moves. Single-sided or double-sided measurement is determined by the setting of #111 in the settings data macro O9750. Tools that have a diameter greater than the value defined in #111 are measured single-sided.

Length and radius values are written into the tool offset register. The wear registers are zeroed and the values are placed in the geometry registers.

Format

G65 P9857 B3. Dd [Ee Hh Jj Kk Mm Qq Tt Ww Yy Zz]

where [ ] denotes optional inputs.

Example:

G65 P9857 B3. D31. J.01 K.008 T1. Y125. Z10.

Inputs

B3. = Measure the length and radius of the tool.

Dd = The cutter diameter.

+d = right-handed cutting tool.

–d = left-handed cutting tool.

Example: D80. defines an 80 mm (3.15 in) diameter right-handed cutting tool.

Ee = The tool offset to be updated if offset type A. If type B or C offset, then the current spindle offset is set.

Hh = The tolerance value that defines when the tool is out of tolerance.

When this input is used, the tool offset is not updated if the tool is found to be out of tolerance.

Default value: No tolerance check.

Jj = An experience value for diameter or radius.

This value is the difference between the measured diameter/radius and the actual diameter/radius of the tool when under load during the cutting process.

Default value: Not used.

NOTE: For cutter centre-line programming applications, entering the nominal size as an experience value will result in the error being stored instead of the radius/diameter of the cutter.

Kk = An experience value for length.

This value is the difference between the measured length and the actual length of the tool when under load during the cutting process.

Default value: Not used.

Mm = Tool out of tolerance flag.

Using M1. prevents a tool OUT OF TOLERANCE alarm from being raised.

Default value: No flag set.

Qq = The overtravel distance.

Default value: 5.0 mm

Tt = The tool offset to be updated.

Default value: Current spindle tool.

Yy = The approximate tool length value.

Ww = The extra Z clearance above the stylus when setting a diameter.

Example: W20. will position 20 mm (0.79 in) + #114 above the stylus.

Zz = The measuring position on the stylus face.

This is the Z axis position from the top face of the stylus at which measurement takes place.

Default value: 5.0 mm (0.197 in)

NOTE: A D input is compulsory if used with B2., B3. or B4. input.

Outputs

The following outputs are set or updated when this cycle is executed:

Set tool length.

Set tool radius/diameter.

#146 Out of tolerance flag. This is set when the measured tool length is out of tolerance, provided the H input is used.

(1 = Out of tolerance, 0 = In tolerance).

Automatic length setting, feeding upwards – macro O9857

NOTE: Before using this cycle, the probe must have been calibrated using a C input. If the Y input is not being used, approximate tool offset values MUST be stored in the tool registers.

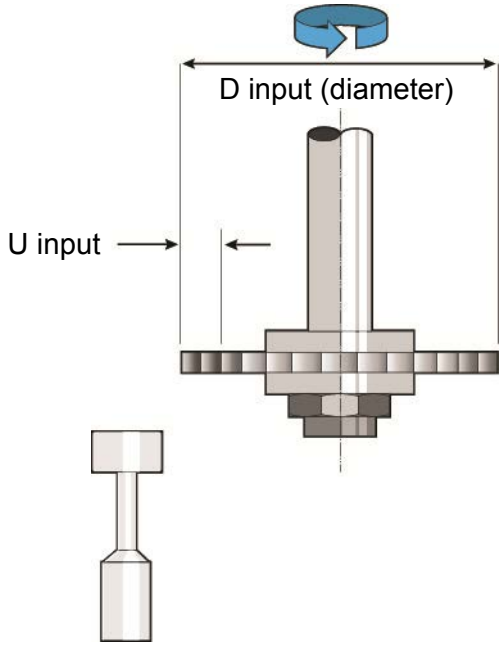


Figure 5.4 Tool length measurement

Description

This cycle is used to measure the effective length of the back edge of a rotating tool, such as a slitting saw, back boring bar or internal groove tool.

Application

The tool must be called into the spindle before the cycle is run.

The cycle automatically moves the tool to the initial clearance position (#113) above the stylus and then to the correct position for measurement, before feeding to the secondary clearance position (#114) prior to the measurement move. After measurement, the tool returns to the home position in the Z axis.

Where space is restricted from the outer radius of the tool to position under the stylus, a U input can be used to limit the distance that the tool point will position from the edge of the stylus.

Format

G65 P9857 B4. Dd [Hh Kk Mm Qq Tt Uu Yy]

where [ ] denotes optional inputs.

Example

G65 P9857 B4. D80. H6.

Inputs

B4. = Set the upper edge length of tool.

Dd = The cutter diameter.

+d = right-handed cutting tool.

–d = left-handed cutting tool.

Example: D80. defines an 80 mm (3.15 in) diameter right-handed cutting tool.

Hh = The tolerance value that defines when the tool length is out of tolerance.

When this input is used, the tool offset is not updated if the tool length is found to be out of tolerance.

Default value: No tolerance check.

Kk = An experience value for length.

This value is the difference between the measured length and the effective length of the tool when under load during the cutting process.

Default value: Not used.

Mm = Tool out of tolerance flag.

Using M1. prevents a tool OUT OF TOLERANCE alarm from being raised.

Qq = The overtravel distance.

Default value: 5.0 mm

Tt = The tool offset to be updated.

Default value: Current spindle tool.

Uu	=	The incremental radial distance for positioning under the stylus. **Default value:** 2 mm (0.078 in)
Yy	=	The approximate tool length offset.

NOTE: A D input is compulsory if used with B2., B3. or B4. input.

Outputs

The following outputs are set or updated when this cycle is executed:

	Set tool length.
#146	Out of tolerance flag. This is set when the measured tool length is out of tolerance, provided the H input is used. (1 = Out of tolerance, 0 = In tolerance).

This page is intentionally left blank.

Chapter 6

Broken tool detection

This chapter describes how to use the broken tool detection cycle for rotating tools. The cycle is used to position the edge of a tool against the stylus face to check that an edge is still present.

Contained in this chapter

Broken tool detection cycle – macro O9858

NOTE: The tool must have been previously set using the tool setting cycle O9857.

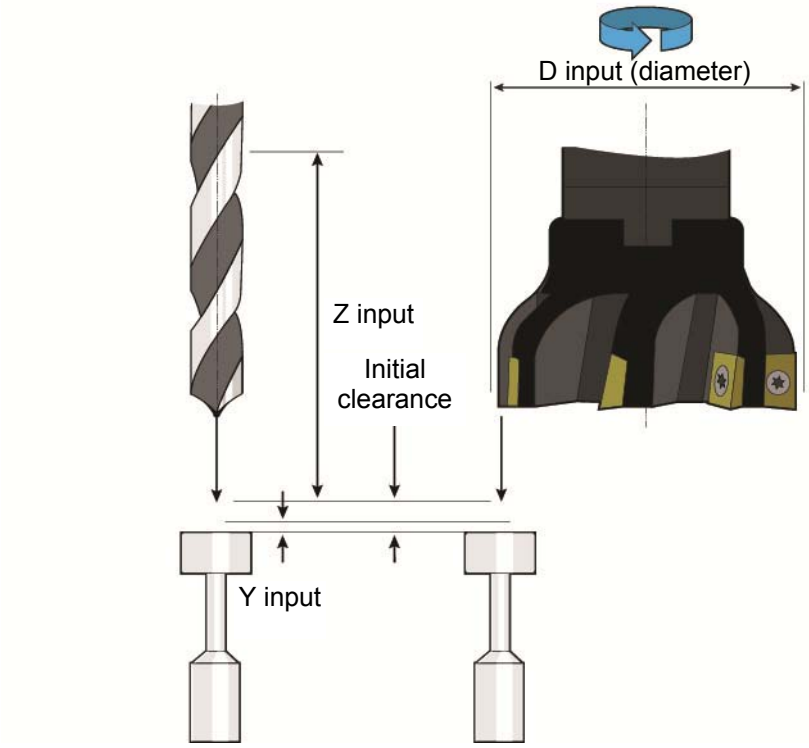


Figure 6.1 Broken tool checking of a rotating tool

Description

This cycle is used to check the length of a rotating tool for a broken tool condition. The cycle also checks for a 'long tool' condition, where the tool has possibly pulled out during machining.

The spindle returns to a safe position and then automatically moves the tool to a position above the stylus prior to checking its length.

NOTE: All rotating broken tool checks are done on the top face of the stylus.

Format

G65P9858 [Dd Hh Mm Tt Yy Zz]

where [ ] denotes optional inputs.

Inputs

Dd = The diameter of the tool.

Without a D input, the tool is checked “on-centre”.

Hh = Tolerance value that defines when the tool is broken. Both broken and long tool conditions are checked. If the default H input is used, the cycle will make a single touch on the stylus using the feedrate stored in #101 (drills, taps etc.). If the H input is less than 0.5 mm (0.02 in), the standard two touch feedrates are used.

Default value: 0.5 mm (0.02 in)

Mm = Tool out of tolerance flag.

Using M1. prevents a BROKEN TOOL or TOOL PULLOUT alarm from being raised (see example below).

Tt = The tool offset number to check. If no T input is entered, the current H offset is used.

Yy = Rapid position above the stylus. Without a Y input, the tool is positioned to #114 in the settings macro O9750.

Zz = The tool moves to this clearance position above the stylus before and after the cycle is run.

With no Z input, the tool retracts to the home position and then runs the cycle and returns to the home position when the cycle is finished. The tool offset will need to be re-applied if the tool is to be used again.

Outputs

The following output is set or updated when this cycle is executed:

#146 Out of tolerance flag.

1 = Broken tool/Tool pull-out, 0 = Good tool.

Example of using the M1. input

The M1. input will suppress the BROKEN TOOL/TOOL PULLOUT alarm, and only put a value into #146. This value can be used to call additional cycles to fix the problem.

```
G65 P9858 M1.
IF[#146EQ0] GOTO20
```

This section will contain corrective actions; for example, select a sister tool for use or select a new pallet or component.

```
N20 (CONTINUE CYCLE)
```

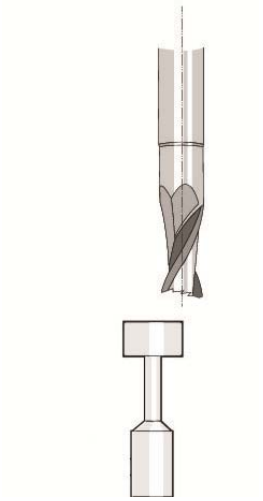
Example 1: Checking a drill for a broken tool condition

```
%O1                          Part program.
T12
G54 G00 X100. Y100.
G0 G43 Z100. H12.
S3000 M3
……..                         Machining program.

G65 P9858 H.1                Check tool length.
M30
%                            End program.
```

Figure 6.2
Checking a drill

Example 2: Checking an end mill for a broken tool condition



```
%O1                          Part program.
T11
G54 G00 X10. Y50.
G0 G43 Z100. H11.
S1500 M3
……..                         Machining program.
G65 P9858 D12. H.05          Check tool length.
M30
%                            End program.
```

Figure 6.3
Checking an end mill

Chapter 7

Thermal compensation cycle

This chapter describes how to use the thermal compensation cycle. The cycle is used to check thermal drift on the machine tool.

Contained in this chapter

Thermal compensation cycle – macro O9859

NOTE: The probe must be calibrated before using the thermal compensation cycle.

Description

This cycle is used to check thermal drift on the machine.

The spindle returns to a safe position and then automatically moves the tool to 3 mm (0.118 in) above the stylus prior to measuring. The length of the tool must be stored in the tool offset register.

Application

The cycle has two functions:

1. Set base data – measures the X, Y and Z faces of the stylus and saves the positions in macro variables. The locations are set on the input line.

2. Measure and compare – measures the X, Y and Z faces of the stylus and compares the results to the base data, thus showing thermal drift. The differences in X, Y and Z will be output into #100, #101 and #102, respectively. If they are out of tolerance (H), an alarm will be raised.

Format

G65 P9859 Cc Dd Xx Yy Zz [Hh Mm Tt Ww]

where [ ] denotes optional inputs.

Inputs

NOTE: Entering input data on the cycle call line will override any other default conditions.

Cc = Set base data or measure and compare:

C1. = measure and store base data.
C2. = measure and compare to base data.

Dd = The diameter of the tool or arbour.

Hh = The tolerance value for comparison (cannot be used with C1).

Mm	=	Tool out of tolerance flag. Using M1. prevents an OUT OF TOLERANCE alarm from being raised.
Tt	=	The tool to be used for measurement.
Ww	=	The measuring position on the stylus face. This is the Z-axis position from the top face of the stylus at which measurement takes place. **Default value:** 5.0 mm (0.197 in)
Xx	=	X-axis stylus position storage location. **Example:** X650. Stores X axis data in #650.
Yy	=	Y-axis stylus position storage location. **Example:** Y651. Stores Y axis data in #651.
Zz	=	Z-axis stylus position storage location. **Example:** Z652. Stores Z axis data in #652.

NOTE: If X, Y or Z inputs are not used, the associated axis will be omitted.

Outputs

The following output is set or updated when this cycle is executed:

#100	X-axis comparison error.
#101	Y-axis comparison error.
#102	Z-axis comparison error.
#103	Out of tolerance flag (0 = No error, 1 = Error).

Example 1: Setting base data

G65 P9859 C1. D6.95 X650. Y651. Z652.

Example 2: Measure and compare data

G65 P9859 C2. D6.95 H.05 X650. Y651. Z652.

This will measure the stylus and show the difference between the base data and new positions for all three axes. If it exceeds ±0.05 mm (0.00197 in) in any direction, an alarm will be raised.

Chapter 8

Advanced options

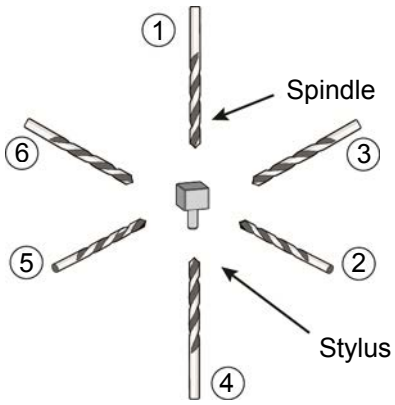
This chapter describes advanced options and functions within the software package.

Contained in this chapter

Multi-axis option

The multi-axis option should be used when the spindle axis is not the Z axis. Three settings in program O9750 must be edited and a modification to O9751 may be required.

Setting variables #121, #122 and #123 (O9750)



Spindle orientation	#121	#122	#123	#104 (Recommended)
1	1	2	3	1, -1, 2, -2
2	3	2	1	-1
3	1	3	2	-2
4	1	2	-3	1, -1, 2, -2
5	1	3	-2	-2
6	3	2	-1	-1

Adjusting safe spindle retract position (O9751)

Line N100 to line N101 may require editing to ensure that the spindle retracts to a safe position before and after running cycles.

This is only required when using multi-axis configuration, and is dependent on machine set-up, axis and probe position.

Example:

G53 Y0. changed to G53 Y600.

G53 X0. changed to G53 X-600.

NOTE: Further edits may be required, depending on the machine configuration.

Twin probe option

This option should be used when two tool setters are fitted on one machine, typically pallet machines or machines with a partition. #106=1 must be set in program O9570 and recognition code must be added in designated places in programs O9750 and O9855. Recognition code could be a machine axis position or a flag or marker supplied by the machine tool builder.

Examples of pallet recognition

O9750 / O9855

.....

.....

Replace

M0(EDIT*SECOND*PROBE*RECOGNITION*HERE)

with

IF[#1032 EQ 2] GOTO46 Flag or marker, designating pallet 2.

...

Example of dividing door

O9750 / O9855

.....

.....

Replace

M0(EDIT*SECOND*PROBE*RECOGNITION*HERE)

with

IF[#5021 GT 1000] GOTO46 X-axis machine value, designating partition position.

...

NOTE: With two probes, 23 consecutive free variables are required to store the calibration data.

Single probe, two spindle orientation option

This option should be used if tool setting is required in two different orientations, typically horizontal and vertical. Edit the software as above for twin probes, but replace pallet recognition with spindle orientation identifiers. The example below details other edits that may be necessary when switching orientations.

The software will not function correctly with G68 active (co-ordinate rotation). This must be cancelled using G69 before any tool measurement and re-applied afterwards.

Example

O9750 / O9855

...

...

IF[#106EQ0]GOTO30

IF[#5025EQ0]GOTO46 Select second probe if horizontal orientation.

#[#120]=#[#120+8](Z+FACE*STATIC)

#[#120+1]=#[#120+9](X+STATIC)

...

...

N46

(SECOND*PROBE*SIDE)

#103=2. New single-sided measure setting for horizontal orientation.

#121=3. Radial measure = Z axis.

#122=2. No measure = Y axis.

#123=1. Length measure = X axis.

#[#120]=#[#120+15](Z+FACE*STATIC)

#[#120+1]=#[#120+16](X+STATIC)

...

...

NOTE: With vertical and horizontal spindle, 22 consecutive free variables are required to store the calibration data.

Extended stylus life option

This option is designed to stop excessive wear at the centre of the stylus and is available in cycles O9857 and O9858. The position of the first (fast) touch can be adjusted by editing #12 at the top of each cycle, the second (slow) touch is made in the stylus centre.

NOTE: #12=0 is set during installation. Values must be in MM units.

```
O9857(REN*TOOL*AUTO*SET)
M5
#12=-2.(STEP*OFF*FROM*CENTRE*IN*MM)

O9858(BROKEN*TOOL*CYCLE)
#12=2.(STEP*OFF*FROM*CENTRE*IN*MM)
```

Custom G-code option (Fanuc only)

CAUTION: The machine tool builder and the relevant Fanuc documentation should be consulted before adjusting any machine parameters.

Linking a G-code to measurement cycles significantly reduces input code and simplifies the measurement process. Permanent parameter changes must be made and these may vary for each installation.

A folder is provided on the installation CD called “G_CODE programs” which contains three programs.

- O9010 Automatic length measure
- O9011 Automatic length and diameter measure
- O9012 Manual length and diameter measure

These programs may require editing to suit different tool change configurations and set-up commands. Edits should only be made by an experienced engineer.

In the example below G700 to G702 have been linked to these programs.

Fanuc parameter	G-code number	Linked program number
6050	700	O9010
6051	701	O9011
6052	702	O9012

Programming using G-codes

G700 and G701 will perform a tool change and measure the tool. If T is omitted, the current tool in the spindle is measured. The manual cycle G700 does not include a tool change; the cutting tip must be positioned 10 mm (0.4 in) above the stylus.

Commonly used inputs

T = Tool number.

D = Tool diameter.

Y = Approximate length.

E = Diameter offset number (ISO type A tool offset).

NOTE: All inputs are the same as described in Chapter 4. The tool approaches the stylus using the settings in macro O9750. If ISO A type offsets, an E input must be used to select the offset number for the radius stored size.

Examples of custom G-code

G700 T2.	Automatic length measure (static).
G700 T2. D30.	Automatic length measure (rotating).
G701 T3. D16.	Automatic length and diameter measure (rotating).
G701 T4. D50. Y125.	Automatic length and diameter measure, position above the stylus using approximate tool length.
G702 T5.	Manual length measure (static).
G702 T5. D50.	Manual length measure (rotating).
G702 B3. T5. D30.	Manual length and diameter measure (rotating).

Chapter 9

Alarms

When an error occurs during use of the software, an alarm is generated and displayed on the screen of the control.

This chapter describes the meaning and likely cause of each alarm message that may be displayed. It then describes typical actions you should take to clear the fault.

Contained in this chapter

Message **“PROBE*OPEN”**

Cause The probe is triggered at the beginning of a measuring move.

Action Edit the back-off factor (#105) in program O9750. The default value is 0.3.

Message **“PROBE*FAIL”**

Cause The probe does not register a trigger during a measuring move.

Action Correct the error and restart the program.

Message **“MISSING*INPUT”**

Cause When a compulsory input is missing, one of the following alarms will be generated.

“D*INPUT*MISSING”
“Y*INPUT*MISSING”

Action Edit the program input line to include the compulsory input.

Message **“H*INPUT*NOT*ALLOWED”**

Cause This alarm is generated if the H input is used with the C1. input.

Action Delete the H input or use the C2. input and restart.

Message **“MISSING*DATA*IN*O9750”**

Cause This alarm is generated if the settings data macro O9750 has not been edited or inputs are missing.

Action Edit the setting data macro O9750 and restart the cycle.

Message **“TOOL*PULL*OUT”**

Cause This alarm is generated if the tool is pulled out from the collet, giving a false tool length.

Action Inspect, adjust and re-measure the tool.

Message **“BROKEN*TOOL”**

Cause This alarm is generated if the tool is broken.

Action Inspect and then replace the tool and reset the tool length.

Message **“SAME*T-D*OFFSET”**

Cause The same tool offset number has been used for the length and diameter/radius.

Action Edit the macro input line then run the macro again.

Message **“FORMAT*ERROR”**

Cause R or X and Y inputs are missing from the macro call line for calibration macro O9855, or T and E inputs are incorrect (type A offset only).

Action Edit the macro input line then run the macro again.

Message **“TOOL*OUT*OF*RANGE”**

Cause This alarm is generated if the T input has a negative value.

Action Edit the macro input line then run the macro again.

Message **“OUT*OF*TOLERANCE”**

Cause The measured length or diameter of the tool is out of tolerance. A positive or negative limit has been exceeded. This may be caused by a broken tool.

Action Inspect and replace the tool if necessary and re-measure the tool length.

Message **“CHECK*PARM*5006.6*SETTING”**

Cause #112 is set incorrectly in O9750.

Action Check parameter 5006.6 and 6004.4 and set #112 in macro O9750 accordingly.

Message **“TOOL*OFFSET*ACTIVE”**

Cause This alarm is generated if a tool offset is active.

Action Ensure the correct offset type is used in settings data macro O9750.

Message **“THERMAL*COMP*TOLERANCE*EXCEEDED”**

Cause The value from the temperature compensation cycle is greater than the specified tolerance.

Action Check the value.

Message **“Y*INPUT*OUT*OF*RANGE”**

Cause The specified Y value is outside the ‘Long tool/Short tool’ range set in settings data macro O9750.

Action Ensure the correct Y value is used on the program input line. If so, adjust the ‘Long tool/Short tool’ values in settings data macro O9750.

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

RENISHAW
apply innovation™

For worldwide contact details, please visit our main website at www.renishaw.com/contact

H-2000-6525-0B