Programming manual
H-2000-6001-00-A

RENISHAW
apply innovation™

Contact tool setting cycles for Fanuc and Meldas controllers

Backward-compatible inputs

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-2000-6001-00-A

Issued: 08.2021

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 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 macro 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 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. 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 – using cycles with pre-select tool commands

When using the ‘T’ tool pre-select command after the tool change, you must use the T input on the macro call block, otherwise the pre-selected tool will be set/used.

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 B2. D80. W30.

may be entered as:

G65P9857B2.D80.W30.

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.

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.

Download on the App Store GET IT ON Google Play

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**.

Table of contents

Chapter 1 Before you begin

Chapter 2 Software installation

Chapter 3 Calibrating the stylus

Chapter 4 Manual cycles

Chapter 5 Automatic cycles

Chapter 6 Broken tool detection

Chapter 7 Thermal compensation cycle

Chapter 8 Advanced options

Chapter 9 Alarms

Chapter 1

Before you begin

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

Intended use

Renishaw contact tool setting (CTS) cycles for Fanuc and Meldas controllers must only be used as intended.

The software is only intended for use with Renishaw contact tool setting probes. Use of the software with non-Renishaw probes is not supported. This version of the software is for use on Fanuc and Meldas controllers only.

About the software

Renishaw CTS cycles for Fanuc and Meldas are designed to work with a range of Renishaw contact tool setting probes and to be compatible with a range of Renishaw software programs.

The cycles provide an easy and intuitive way for customers to measure a wide range of tooling. The software provides cycles to calibrate the contact tool setting probe, measure tools, check tools for broken or pulled out conditions and to check the thermal drift of the machine.

About this manual

This manual contains detailed information on the Renishaw CTS cycles for use on Fanuc and Meldas controllers. The aim is to guide the user through the process of calibrating and using a Renishaw contact tool setting probe. It contains separate sections for calibration, manual and automated operational modes, broken tool checking and thermal compensation.

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 – aim 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 dial test indicator (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 faces 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

The tool setting cycles use static measurement (non-rotating tool) when the tool diameter is less than the stylus diameter, and dynamic measurement (rotating tool) when it is larger.

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.

First touch spindle speed

The spindle speed for the first move onto the probe is calculated from a surface cutting speed of 60 m/min (197 ft/min). This is maintained within the range 150 r/min to 800 r/min and relates to a range of 24 mm to 127 mm (0.95 in to 5 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 × r/min   F units mm/min.

NOTE: If a C input (number of teeth) is used, the feedrate will be calculated per tooth.

Second touch spindle speed

800 r/min.

Second touch feedrate

Feedrate 4 mm/min (0.16 in/min), resolution 0.005 mm/rev (0.00020 in/rev).

Features of the CTS software

The CTS software provides the following measuring and calibration features:

Measuring macro features

Five measuring macros provide the following features:

- Macro O9856: used for measuring the length and diameter of the cutting tool with manual positioning.
- Macro O9857: used for measuring the length and diameter of the cutting tool with automatic positioning.
- Macro O9858: used for broken tool checking.
- Macro O9859: used for thermal compensation measurement.
- Macro O9921: GoProbe tool setting cycle.

Calibration macro features

One calibration macro provides the following features:

- Macro O9855: used for calibrating the positions of the stylus in the spindle axis, radial axis and stem axis.

Service macro features

The measuring and calibration macros are supported by the service macros listed below:

- Macro O9735: Data Send macro (used for the Reporter app).
- Macro O9750: used for the settings data.
- Macro O9751: used for start-up functions.
- Macro O9752: used for the measuring routine.
- Macro O9753: used for the G31 routine.
- Macro O9754: used for the G0/G1 routine.
- Macro O9755: used for retract positioning.
- Macro O9759: used for error messages.
- Macro O9773: used for the Reporter app.
- Macro O9890: used for Tool setter ON commands.
- Macro O9891: used for Tool setter OFF commands.

Software memory requirements

The CTS system software requires approximately 41 KB of part-program memory.

If your controller is short of memory, the following macros need not be loaded, or may be deleted after use.

Measuring and calibration macros

- Macro O9855 (tool setter stylus calibration routine): approximately 6 KB of memory.
- Macro O9856 (manual positioning tool setting routine): approximately 4 KB of memory.
- Macro O9857 (automatic positioning tool setting routine): approximately 13 KB of memory.
- Macro O9858 (broken tool detection): approximately 3 KB of memory.
- Macro O9859 (thermal compensation routine): approximately 4 KB of memory.
- Macro O9921 (GoProbe cycles): approximately 3 KB of memory.

Cycle input compatibility

The software allows the user to choose between current standard cycle inputs or backward-compatible cycle inputs. Backward-compatible inputs cover previous versions of the contact tool setting software up to version AG (2020). If current standard cycle inputs are selected (#143 = 0, see settings information for details), programming information must be taken from H-2000-6525. The following is a list of functions that are unavailable when using backward-compatible cycle inputs.

- Off centre long tool / short tool approach method (#141 = 2, see settings information for details).
- Accurate calibration of the underside of the stylus for higher accuracy when measuring the upper edge of a tool (O9857 B4).
- Measure / check / control tolerancing options.
- Separate length and radius tolerancing option.
- Reporter functionality.

Compatibility selection should be carefully considered when used in conjunction with Renishaw GUI products.

If using Set and Inspect up to version 4.0, cycle input compatibility must be set to backward-compatible. For versions 4.0 to 4.1, current standard must be used.

For version 4.2 or later, both packages must be set to the same compatibility but either can be used.

If using Fanuc GoProbe iHMI or GoProbe GUI (for Mitsubishi M80/M800S), backward-compatible cycle inputs must be used.

The GoProbe Smartphone app is unaffected by these changes and can be used with all versions of this kit.

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.

Example

Home position, to the zero (0) position of the part program = −1000 mm (−39.4 in).

A master calibration tool of 150 mm (5.9 in) is used (offset register value = −850 mm (−33.5 in)).

The longest tool for the machine is 200 mm (7.87 in) long.

The shortest tool for the machine is 50 mm (1.97) long.

In the setting data macro (O9750), variables #110 and #111 must be set as follows:

#110 = −800.0 Maximum length tool.

#111 = −950.0 Minimum length tool.

Relative to a master tool with zero (0) tool offset value

The master tool offset register is set to zero (0) and all other tool offset registers are set as ± values relative to the master tool.

Example

Home position, to the zero (0) position of the part program = −1000 mm (−39.4 in) (but this is not important).

A master calibration tool of 150 mm (5.9 in) is used (offset register value = 0).

The longest tool for the machine is 200 mm (7.87 in) long.

The shortest tool for the machine is 50 mm (1.97) long.

In the setting data macro (O9750), variables #110 and #111 must be set as follows:

#110 = 50.0 Maximum length tool.

#111 = −100.0 Minimum length 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 with an installation wizard to aid in customisation of the cycles to fit the specific machine tool. Load the wizard onto a PC from the software media provided, run it, and fill out the required fields to compile the software. The completed software can then be loaded to the machine tool.

Failure to use the installation wizard will result in an alarm being issued by all cycles.

Macro variables

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

- #500-series macro variables are used for calibration data.
- #100 to #149 series macro variables are used for settings 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

All settings are set via the installations wizard, although if settings are required to be changed then read the following variable descriptions and then edit macro O9750 as required.

NOTE: All values must be in metric.

#101 A tool having a diameter greater than the specified value is set on one side of the stylus only.

To set a large tool on the positive side of the stylus, enter a positive value.

To set a large tool on the negative side of the stylus, enter a negative value.

Default: 100 mm (3.937 in)

#102 First touch feedrate.

This is used after a long tool / short tool move, or when moving from the secondary clearance position when using the known tool length approach method during static measurement.

Default: 200 mm/min (7.874 in/min)

#107 The spindle axis (Sp) safe position in machine co-ordinates from which all cycles begin (excluding calibration).

Default: 0 mm

#108 Tool offset type.

1 = Type A, one register per tool.

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

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

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

#109 The setting for the tool offset register type, which may be in either radius or diameter values.

1 = Radius
2 = Diameter

Default: 1

#110 The maximum length of the tool. This defines the rapid approach height of the spindle nose above the stylus.

Default: 0 mm

#111	The minimum length of the tool. This defines the lowest measuring height of the spindle nose above the stylus. **Default:** 0 mm
#113	The accessible faces of the stem (St) axis (see “Probe access” on page 2-7).
#114	The accessible faces of the radial (Ra) axis (see “Probe access” on page 2-7).
#117	Default overtravel distance. Overtravel is the distance past a nominal target, during a measuring move, that the tool is permitted to move before an alarm is raised. **Default:** 5 mm (0.197 in)
#120	The base number for #500-series calibration data. The base number defines the address of the first variable in the set of variables that are used for storing calibration data. The default address is 520 (#520). Changing the #120 value in the settings data macro (O9750) will change the variable range. **Default:** 520
#121=1 #122=2 #123=3	Machine axis number for the stem axis) Machine axis number for the radial axis > Machine axis number for the spindle axis) Modify for multi-axis option only (see Chapter 8, “Advanced options”).
#124	Reserved for future use.
#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 (0.197 in)
#126	The accessible faces of the spindle axis (Sp) (see “Probe access” on page 2-7).
#127	The feedrate used for rapid traverse. **Default:** 5000 mm/min (197 in/min)
#128	The long tool / short tool approach feedrate. This defines the feedrate for the initial long tool / short tool approach move. **Default:** 2000 mm/min (79 in/min)
#138	Tools with diameters larger than this value will rotate during measurement. **Default:** 10 mm (0.394 in)

#139 Initial approach clearance position above the stylus. This is the target position of the tool tip during the initial rapid move when using the known tool length approach method.

Default: 100 mm (3.937 in)

#140 Secondary approach clearance position above the stylus. This defines the second approach position when using the known tool length approach method. It is also used as the clearance position above the stylus before and after radial measurement.

Default: 10 mm (0.394 in)

#141 Approach method.

0 = Long tool / short tool search: select this option if the tool length is unknown. The value in the tool offset is irrelevant. The maximum and minimum tool values (#110 and #111) define the search distance.

1 = Known tool length: select this option when the tool length is known. The value in the tool offset is used to position the tool above the stylus.

NOTES:

Tools with a diameter greater than the value set in #138 will always use the known tool length approach method.

The known tool length approach method reduces measurement cycle time, however, a collision risk exists if the tool offset value is incorrect.

2 = Only available when using current standard cycle inputs.

#142 Stylus level tolerance.

This is the maximum allowable level tolerance for the top face of the stylus. During calibration, an alarm will be issued if the stylus level exceeds this value.

Default: 0.015 mm (0.00059 in)

NOTE: This feature is only used in GoProbe tool setter check cycle M200.

#143 Cycle input compatibility.

This option can be used to allow the cycles to run using inputs compatible with previous versions of contact tool setting software (version AG and earlier). However, new functionality will be unavailable if this option is chosen. Compatibility with any GUI software must also be carefully considered (see section “Cycle input compatibility” in Chapter 1 for more information).

0 = Current standard inputs to be used.

1 = Backward-compatible inputs to be used.

NOTE: The programming instructions for using current standard inputs can be found in programming manual *Contact tool setting cycles for Fanuc and Meldas controllers* (Renishaw part no. H-2000-6525.

#145 Static 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 (0.00020 in)

#144=1	Machine stem axis identifier	1 = X	)	Modify for multi-axis option only
#146=2	Machine radial axis identifier	2 = Y	>	(see Chapter 8, “Advanced
#147=3	Machine spindle axis identifier	3 = Z	)	options”).

Probe access

#113, #114 and #126 must be set in the settings macro (O9750).

#113 controls access to the stylus in the stem (St) axis, #114 in the radial (Ra) axis and #126 in the spindle (Sp) axis.

NOTE: #113 = 2 should only be used when the stylus configuration allows for full access to both stem faces.

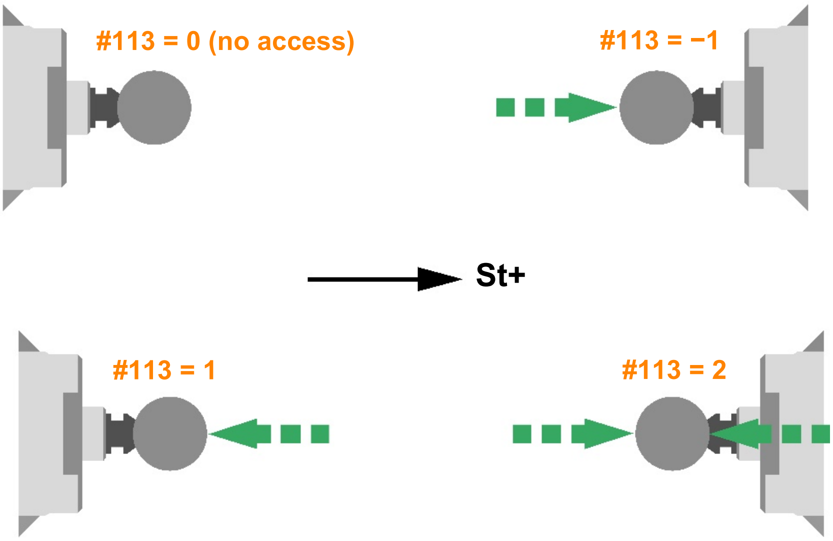

Figure 2.1 Stem (St) axis access (#113)

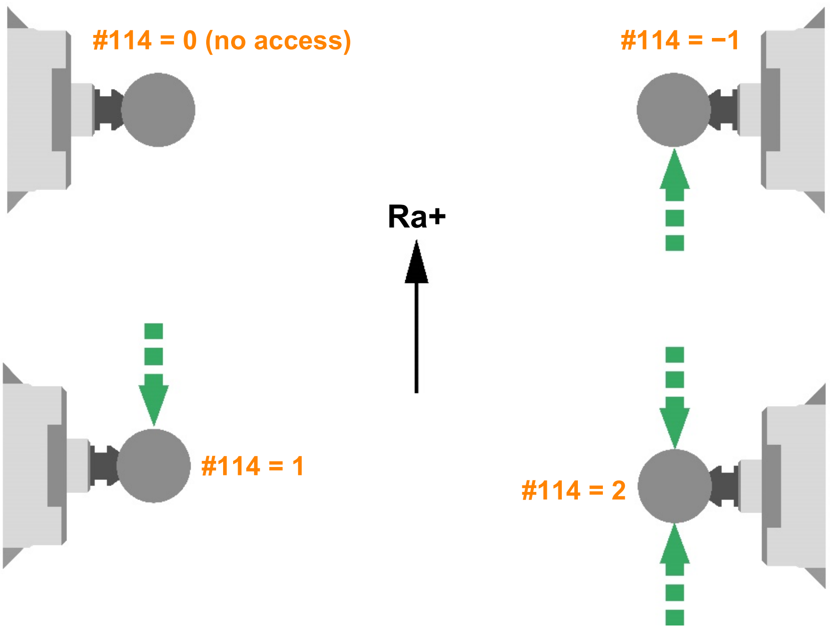

Figure 2.2 Radial (Ra) axis access (#114)

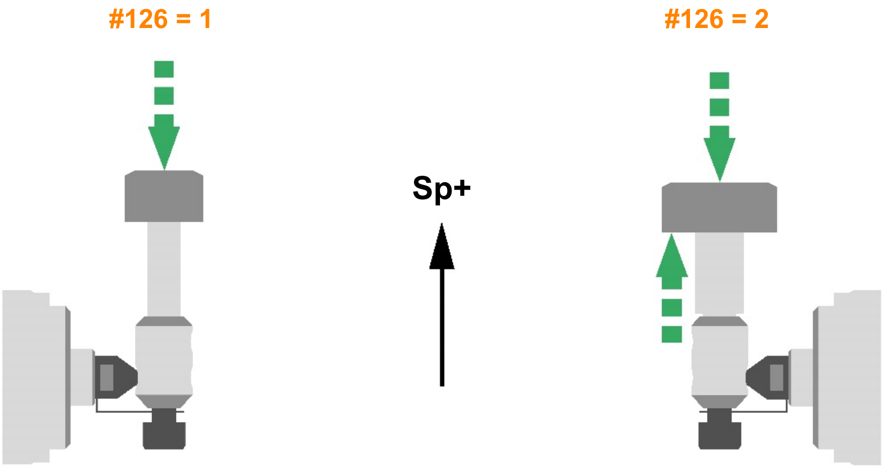

Figure 2.3 Spindle (Sp) axis access (#126)

Any combination of the above variables is possible, however, to measure a tool diameter on the ‘underside’ of the stylus (#126 = 2), at least one radial or stem face must be accessible.

Adjusting the back-off distance

A back-off distance is provided for adjusting the distance the tool moves off the surface of the stylus after the first touch prior to the final measuring move.

The software loads a default value of 0.25 mm when first run. This value is stored in base number plus 7 (#120 + 7). For example, if #120 = 500, the back-off distance is stored in #527.

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

NOTE: If the value is too small, a “PROBE*ALREADY*TRIGGERED” alarm occurs.

Long tool / short tool option

This function is used only in macro O9857 (automatic length setting).

The long tool / short tool option is enabled by entering the maximum tool length into #110 and the minimum tool length into #111 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 retract position in the spindle (Sp) axis. It will then position over the stylus and feed at the rapid traverse rate to the long tool position above the stylus. It will then feed the tool towards the stylus at the feedrate set in #128, until a trigger is detected. If the tool is not detected within the set range, a "PROBE*DID*NOT*TRIGGER" alarm will be displayed.

Settings in O9750

#107	Retract position
#127	Rapid traverse feedrate
#110	Maximum tool length
#111	Minimum tool length
#128	Search feedrate

NOTE: If #141 is set to 1, this will disable the long tool / short tool option. The tool offset must then be correct, or a Y input must be used (approximate tool length).

This page is intentionally left blank.

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.

NOTE: If programming using current standard inputs, use programming manual *Contact tool setting cycles for Fanuc and Meldas controllers* (Renishaw part no. H-2000-6525).

Contained in this chapter

Calibrating the stylus – macro O9855

Description

This cycle 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 manual 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 access variables in settings macro O9750. Calibration values are found, or calculated, for the stylus position (stored in metric units and converted when required).

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 correct stylus size is input into the program call line.
6. Position the tool at a suitable start point, using either manual or handwheel mode, so that it is centrally over the stylus and within approximately 10 mm (0.394 in) of the top face, 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 (Sp) 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 3.2).

Qq = The overtravel distance.

Default: Overtravel default set in #117 in the settings macro (O9750)

Rr = The actual diameter of the master setting tool.

Tt = The tool length offset to use.

CAUTION: The exact length of the master tool must be entered in the appropriate tool offset (Tt).

Uu = Stem (St) axis step-over distance, used during spindle axis calibration.

Vv = Radial (Ra) axis step-over distance, used during spindle axis calibration.

Xx = The distance between the start position and the accessible face of the stylus in the stem (St) axis (see Figure 3.1).

Yy = The radial (Ra) axis stylus width (see Figure 3.1).

Zz = The distance from the top face of the stylus to the measuring point on the side faces.

Default value: 5 mm (0.197 in)

Calibration examples

Setting a square stylus

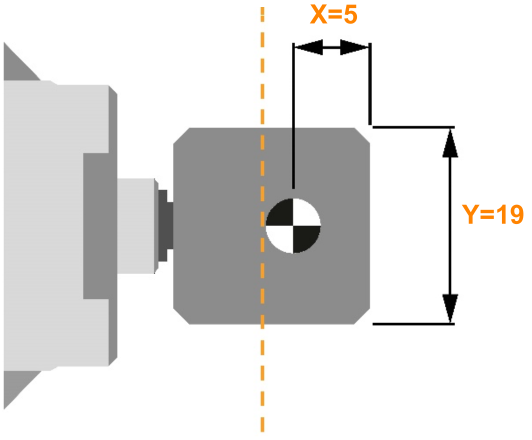

Figure 3.1 Setting a square stylus

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

Example:

Position the calibration tool 10 mm (0.394 in) above the top face of the stylus.

G65 P9855 R6. T21. X5. Y19.

After calibration, tools will be measured 5 mm (0.197 in) from the edge of the stylus.

Setting a round stylus

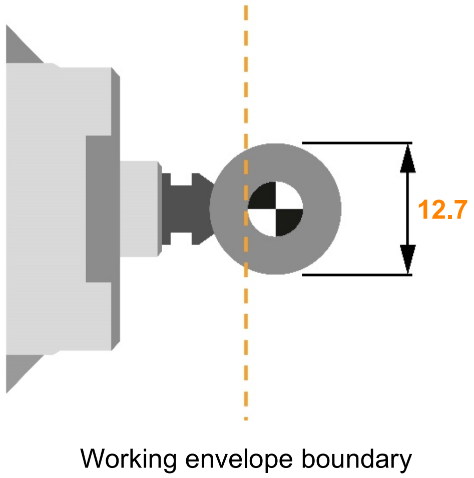

Figure 3.2 Setting a round stylus

Example:

Position the calibration tool 10 mm (0.394 in) above the top face of the stylus.

G65 P9855 D12.7 R6. T21.

Spindle axis calibration point shift

If required, the calibration tool can be offset from the start position when calibrating in the spindle (Sp) axis direction. This is especially useful when using a calibration tool with a hollow centre. See Figure 3.3 for details.

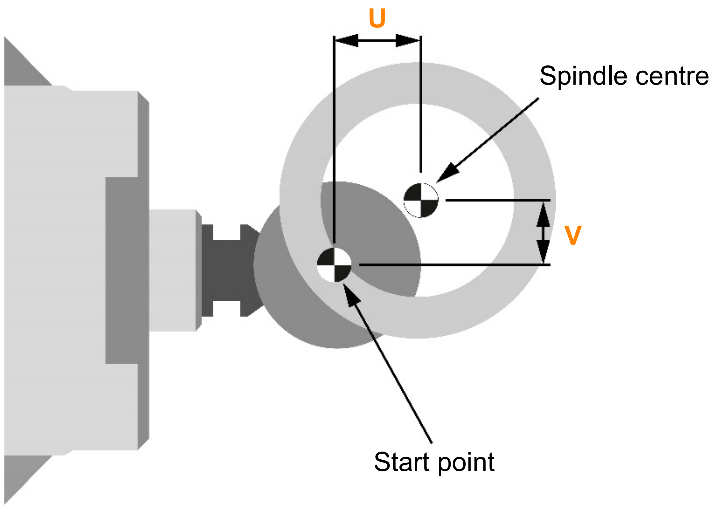

Figure 3.3 U and V inputs

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)	Sp axis position of the top face of the stylus – static tools.
#521 (520 + 1)	Sp axis position of the bottom face of the stylus – static tools.
#522 (520 + 2)	+Ra axis position of the stylus face – rotating tools.
#523 (520 + 3)	−Ra axis position of the stylus face – rotating tools.
#524 (520 + 4)	+St axis position of the stylus face – rotating tools.
#525 (520 + 5)	−St axis position of the stylus face – rotating tools.
#526 (520 + 6)	Difference between rotating tools and static tools.
#528 (520 + 7)	Reserved for back-off distance.

NOTES:

Multiple probes or multiple axis configurations will require multiple free variables for the parameters listed above. For convenience, each probe can have its own base number.

Multiple probe or axis configurations should be edited using the installation wizard.

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

This page is intentionally left blank.

Chapter 4

Manual cycles

This chapter describes how to use the manual tool length and manual tool length and radius/diameter cycles.

NOTE: If programming using current standard inputs, use programming manual *Contact tool setting cycles for Fanuc and Meldas controllers* (Renishaw part no. H-2000-6525).

Contained in this chapter

Manual length or length and radius setting cycle – O9856

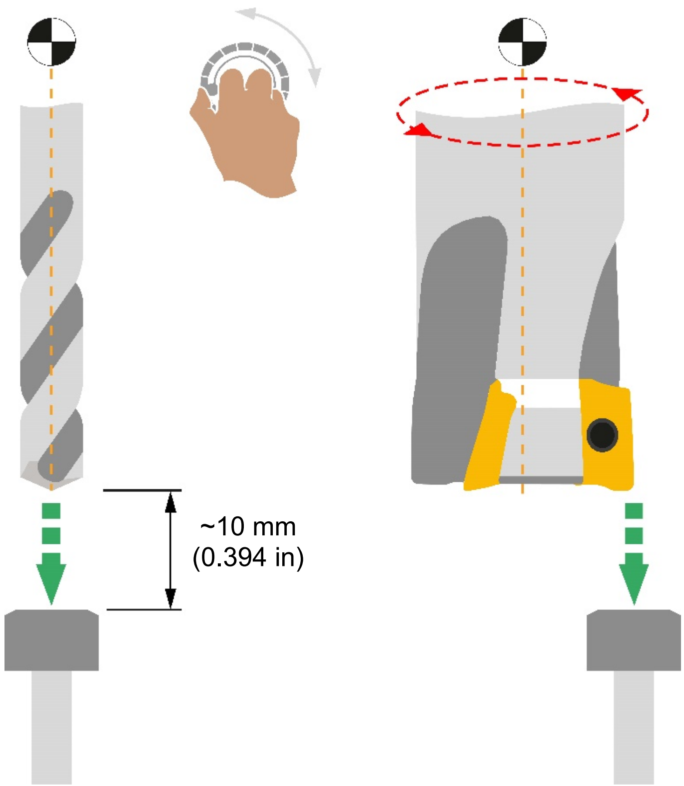

Figure 4.1 Manual length or length and radius setting

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.

NOTE: If the tool diameter is less than the value in #138 in the settings macro (O9750), length measurement will be taken while the tool is static. If the tool diameter is greater than the value in #138, length measurement will be taken while the tool is rotating. The tool is always rotating for diameter measurement.

Format

G65 P9856 [B3. Dd Tt]

where [ ] denotes optional inputs.

Example: G65 P9856

The length of the current spindle tool will be measured whilst the tool is static.

Example 2: G65 P9856 D80.

The length of the current spindle tool will be measured whilst the tool is rotating.

Example 3: G65 P9856 B3. D80.

The length and radius of the current spindle tool with an 80 mm (3.15 in) diameter will be measured, with the tool rotating.

Inputs

B3. = Measure the length and radius/diameter 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 required when using B3. It can be used when the tool is to be rotated during the measuring cycle and should be the nominal diameter of the tool.

+D = right-handed cutting tool.
–D = left-handed cutting tool.

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

Tt = Length offset number.

This is the offset location in which the measured tool length is stored when it needs to be different from the active tool number.

Default value: Current tool number.

This page is intentionally left blank.

Chapter 5

Automatic cycles

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

NOTE: If programming using current standard inputs, use programming manual *Contact tool setting cycles for Fanuc and Meldas controllers* (Renishaw part no. H-2000-6525).

Contained in this chapter

Automatic length setting – O9857

NOTE: Before using this cycle, the probe must have been calibrated. If the approach method (#141) is set to 1, the known length tool approach method will be used. In this case, if the Y input is not used, the approximate tool length MUST be stored in the offset register prior to measurement. This will also be the case if the approach method (#141) is set to 0 and the tool diameter is larger than the value in #138.

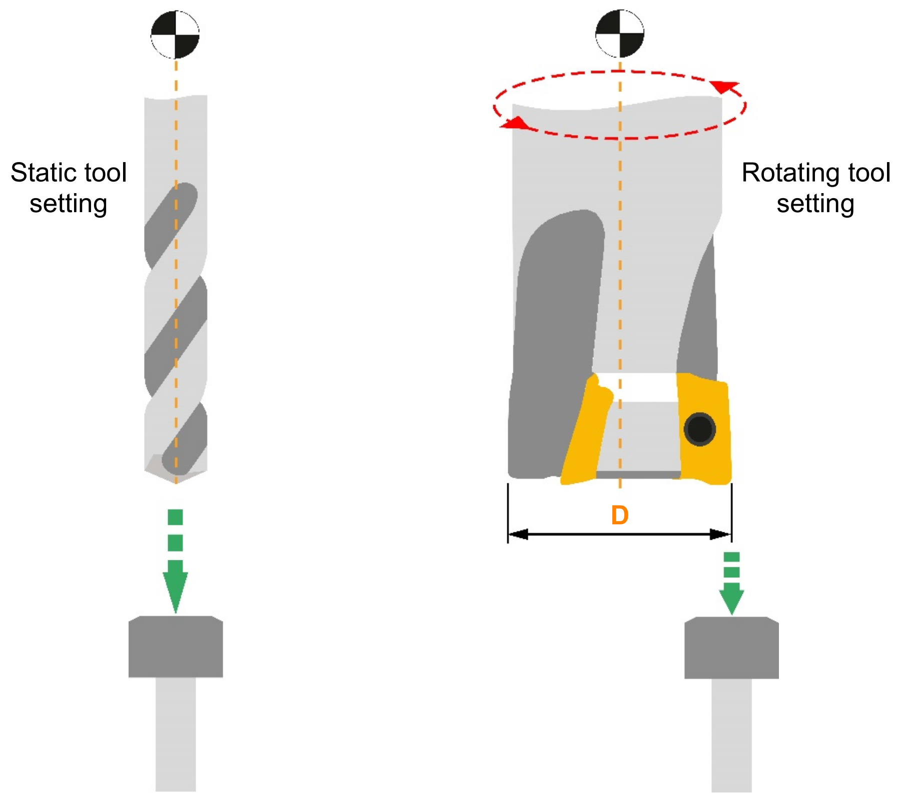

Figure 5.1 Measuring the tool length

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 retract position (#107) in the spindle (Sp) axis before moving to the correct position for measurement. It then approaches the stylus based on the approach method setting (#141).

After measurement, the tool returns to the retract position (#107) in the spindle (Sp) 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 diameter of the tool being measured.

This input is used when the tool is to be rotated during the measuring cycle and should be the nominal diameter of the tool.

+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 = Experience value for the length.

This value is the difference between the measured length of the tool and the actual length when the tool is under load during the cutting process. It is used to refine the measured length, based on previous experience of how the effective length differs from the measured length when the tool is under load.

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: Overtravel default set in the settings macro (O9750)

Tt	=	Length offset number. This is the offset location in which the measured tool length is stored when it needs to be different from the active tool number. **Default value:** Current tool number.
Yy	=	Approximate tool length value. **Default value:** Not used (value obtained from tool length register).

Outputs

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

Set tool length.

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

0 = In tolerance
1 = Out of 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.	Measure length with rotation of 80 mm diameter tool
	Set current spindle tool

Automatic radius/diameter setting – O9857

NOTE: Before using this cycle, the probe must be calibrated. If the approach method (#141) is set to 0 or 1 and the Y input is not being used, approximate tool offset values must be stored in the tool registers.

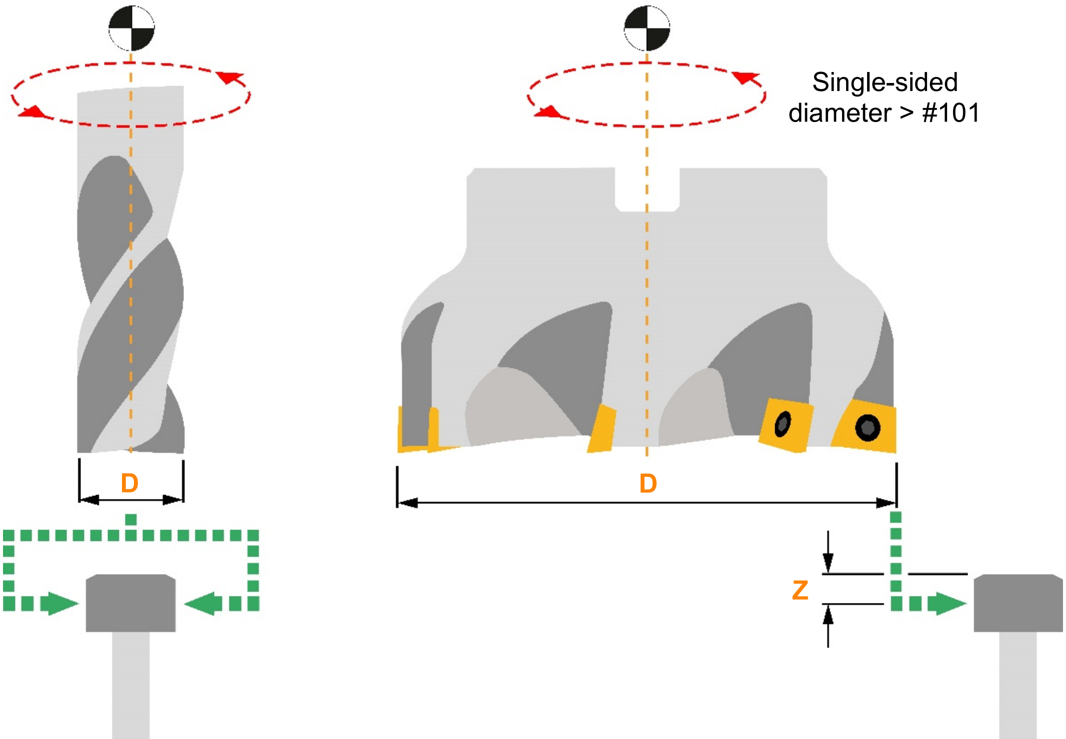

Figure 5.2 Tool cutting radius measurement

Description

This cycle is used to measure the effective cutting radius of a rotating tool by taking either single-sided or double-sided measurements on the tool setting stylus. The value of #101 in the settings data macro O9750 determines whether single-sided or double-sided measurement is used. Tools that have a diameter greater than the value defined in #101 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 moves the tool to the spindle (Sp) axis retract position (#107) and then approaches the stylus using the selected approach method (#141) to the correct position for either a single-sided or double-sided measuring move, as shown in the figure above. The tool then returns to the spindle (Sp) axis retract position (#107).

Format

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

where [ ] denotes optional inputs.

Example: G65 P9857 B2. D80.

Inputs

B2. = Measure the radius/diameter of the tool.

Dd = The diameter of the tool being measured.

This input is used when the tool is to be rotated during the measuring cycle and should be the nominal diameter of the tool.

+D = right-handed cutting tool.
–D = left-handed cutting tool.

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

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

Ee = Diameter offset number.

This is the offset location in which the measured radius/diameter of the tool is stored.

Default: When offset types have separate registers for length and radius, the active tool offset number is used.

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 = Experience value for the radius/diameter.

This value is the difference between the measured radius/diameter of the tool and the actual radius/diameter when the tool is under load during the cutting process. It is used to refine the measured radius/diameter, based on previous experience of how the effective radius/diameter differs from the measured radius/diameter when the tool is under load.

Default: 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 full 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:** Overtravel default set in #117 in the settings macro (O9750)
Tt	=	Length offset number. This is the offset location in which the measured tool length is stored when it needs to be different from the active tool number. **Default value:** Current tool number.
Ww	=	The extra spindle (Sp) axis clearance above the stylus when setting a diameter, typically used with slitting saws when a nut extends below the measured face. **Example:** W20. will position 20 mm (0.79 in) + #140 above the stylus.
Yy	=	Approximate tool length value. **Default value:** Not used (value obtained from tool length register).
Zz	=	Measuring height of the tool. This is the spindle (Sp) axis position from the end face of the tool at which measurement of the radius/diameter takes place. **Default value:** 5 mm (0.197 in)

Outputs

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

	Set tool radius/diameter.
#148	Out of tolerance flag. This is set when the measured radius/diameter of the tool is out of tolerance. 0 = In tolerance 2 = Out of tolerance

Example 1: Radius/diameter tool setting – slitting saw, rotating

G65 P9857 B2. D80. W30.	Measure the radius/diameter of an 80 mm diameter tool with an extra 30 mm clearance height when over the stylus

Automatic length and radius setting – O9857

NOTE: Before using this cycle, the probe must be calibrated. If the approach method (#141) is set to 1, the known tool length approach method will be used. In this case, if the Y input is not used, the approximate tool length must be stored in the offset register prior to measurement. This will also be the case if the approach method (#141) is set to 0 and the tool diameter is larger than the value in #138.

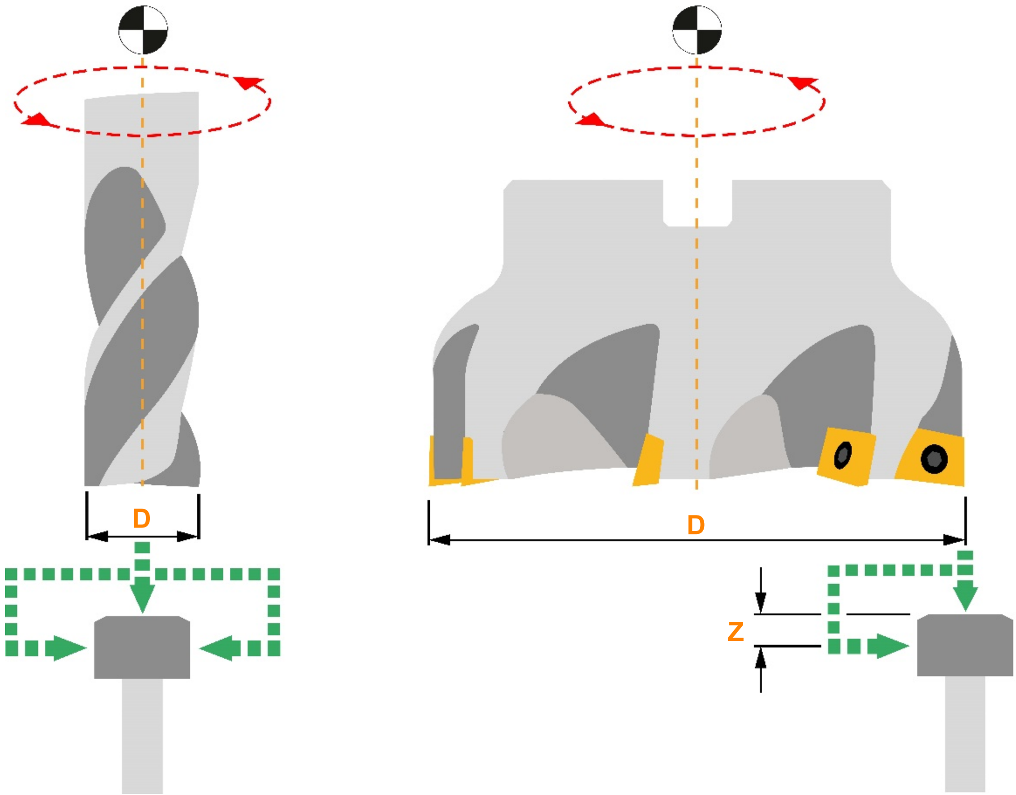

Figure 5.3 Measuring the cutting radius of a rotating tool

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 – O9857" on page 5-2) and the tool radius/diameter measuring cycle (see "Automatic radius/diameter setting – O9857" on page 5-5).

Figure 5.3 shows the combined cycle moves. Single-sided or double-sided measurement is determined by the setting of #101 in the settings data macro O9750. Tools that have a diameter greater than the value defined in #101 are measured single-sided.

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/diameter of the tool.

Dd = The diameter of the tool being measured.

This input is used when the tool is to be rotated during the measuring cycle and should be the nominal diameter of the tool.

+D = right-handed cutting tool.

–D = left-handed cutting tool.

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

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

Ee = Diameter offset number.

This is the offset location in which the measured radius/diameter of the tool is stored.

Default: When offset types have separate registers for length and radius, the active tool offset number is used.

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 = Experience value for the radius/diameter.

This value is the difference between the measured radius/diameter of the tool and the actual radius/diameter when the tool is under load during the cutting process. It is used to refine the measured radius/diameter, based on previous experience of how the effective radius/diameter differs from the measured radius/diameter when the tool is under load.

Default: 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 full radius/diameter of the cutter.

Kk = Experience value for the length.

This value is the difference between the measured length of the tool and the actual length when the tool is under load during the cutting process. It is used to refine the measured length, based on previous experience of how the effective length differs from the measured length when the tool is under load.

Default: 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: Overtravel default set in #117 in the settings macro (O9750)

Tt = Length offset number.

This is the offset location in which the measured tool length is stored when it needs to be different from the active tool number.

Default value: Current tool number.

Yy = Approximate tool length value.

Default value: Not used (value obtained from tool length register).

Ww = The extra spindle (Sp) axis clearance above the stylus when setting a diameter.

Example: W20. will position 20 mm (0.79 in) + #140 above the stylus.

Zz = Measuring height of the tool.

This is the spindle (Sp) axis position from the end face of the tool at which measurement of the radius/diameter takes place.

Default value: 5 mm (0.197 in)

Outputs

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

Set tool length and radius/diameter.

#148 Out of tolerance flag. This is set when the measured length or radius/diameter of the tool is out of tolerance.

0 = In tolerance
1 = Length out of tolerance
2 = Radius out of tolerance
3 = Length and radius out of tolerance

Example: Length and Radius/diameter tool setting – rotating tool

G65 P9857 B3. D80. E21. T1. Set the tool length offset (1) and radius offset (21).

Automatic length setting, feeding upwards – 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.

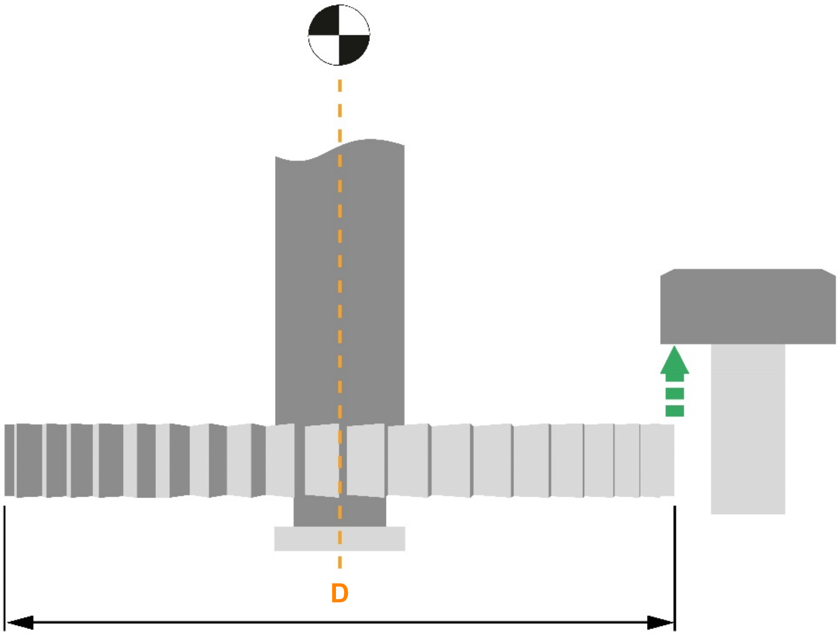

Figure 5.4 Tool length measurement

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 first moves the tool to the spindle (Sp) axis retract position (#107). The upper edge will be measured as shown in Figure 5.4 above. The tool then returns to the spindle (Sp) axis retract position (#107).

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 the tool.

Dd = The diameter of the tool being measured.

This input is used when the tool is to be rotated during the measuring cycle and should be the nominal diameter of the tool.

+D = right-handed cutting tool.

–D = left-handed cutting tool.

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

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

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 = Experience value for the length.

This value is the difference between the measured length of the tool and the actual length when the tool is under load during the cutting process. It is used to refine the measured length, based on previous experience of how the effective length differs from the measured length when the tool is under load.

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: Overtravel default set in #117 in the settings macro (O9750)

Tt = Length offset number.

This is the offset location in which the measured tool length is stored when it needs to be different from the active tool number.

Default value: Current tool number.

Uu = The incremental radial distance for positioning under the stylus (see Figure 5.5).

Default value: 2 mm (0.078 in).

Yy = Approximate tool length value.

Default value: Not used (value obtained from tool length register).

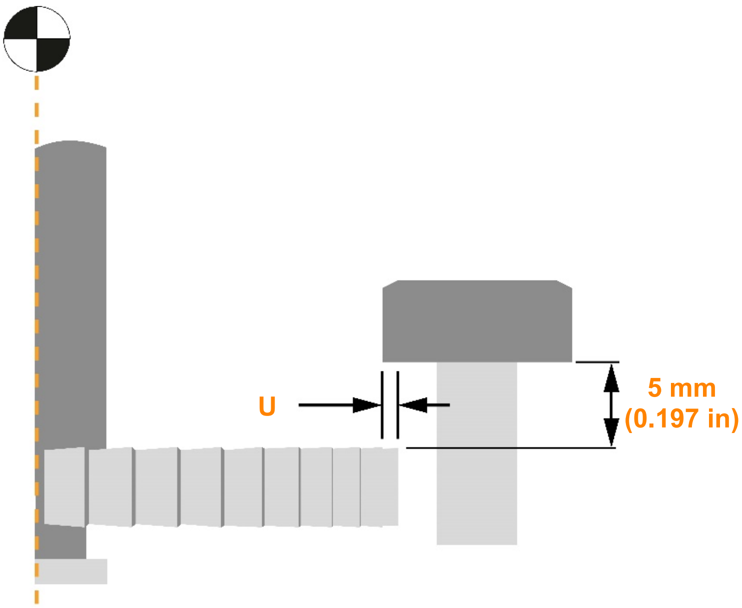

Figure 5.5 Measuring the tool length

Outputs

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

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

Example: Length tool setting feeding upwards

G65 P9857 B4. D80.	Measure the top face of an 80 mm diameter tool.

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.

NOTE: If programming using current standard inputs, use programming manual *Contact tool setting cycles for Fanuc and Meldas controllers* (Renishaw part no. H-2000-6525).

Contained in this chapter

Broken tool detection cycle – O9858

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

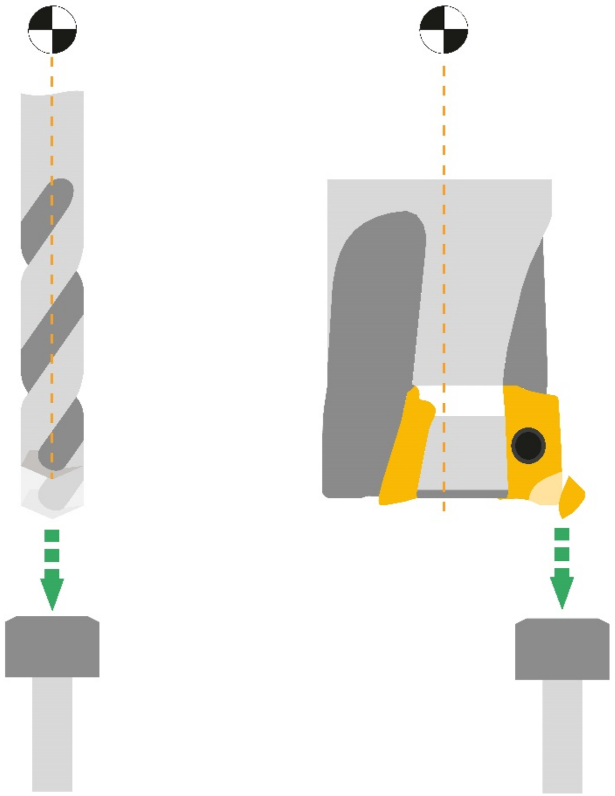

Figure 6.1 Broken tool checking of a rotating tool

Description

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

The cycle automatically moves the tool to the retract position (#107) in the spindle (Sp) axis then 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.

Example: G65 P9858

Inputs

Dd = Nominal diameter of the tool.

Hh = The tolerance value that defines when the tool is broken. If the default H input is used, the cycle will make a single touch on the stylus using the feedrate stored in #102. If the H input is less than 0.5 mm (0.02 in), the standard two-touch feedrates are used. If the D input is used, the feedrate will be calculated based on that value, and a two-touch routine will be used regardless of the H value. The cycle will check for both broken tool and long tool conditions.

Default value: 0.5 mm (0.02 in)

Mm = Tool out of tolerance flag.

Using M1. prevents a “BROKEN*TOOL” or “LONG*TOOL” alarm from being raised.

Tt = Length offset number.

This is the offset location in which the measured tool length is stored if it needs to be different from the active tool number.

Default value: Current tool number.

Yy = Rapid position above the stylus. Without a Y input, the tool is positioned to the secondary approach clearance point (#140) set 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 retract position and then runs the cycle and returns to the retract 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:

#148 Out of tolerance flag.

0 = Good tool
1 = Broken tool
2 = Long tool

Example of using the M1. input

The M1. input will suppress the “BROKEN*TOOL” or “LONG*TOOL” alarm, and just assign a value to #148. This value can be used to call additional cycles to fix the problem.

G65 P9858 M1.
IF[#148EQ0] GOTO20

These cycles will consist of corrective actions; for example, selecting a sister tool for use or selecting a new pallet or component.

N20 (CONTINUE CYCLE)

Example 1: Checking a drill for a broken tool condition

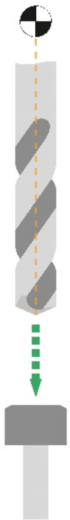

Figure 6.2
Checking a drill

%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.

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

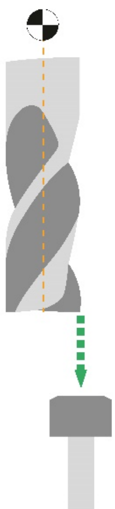

Figure 6.3
Checking an end mill

%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.

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.

NOTE: If programming using current standard inputs, use programming manual *Contact tool setting cycles for Fanuc and Meldas controllers* (Renishaw part no. H-2000-6525).

Contained in this chapter

Thermal compensation cycle – O9859

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

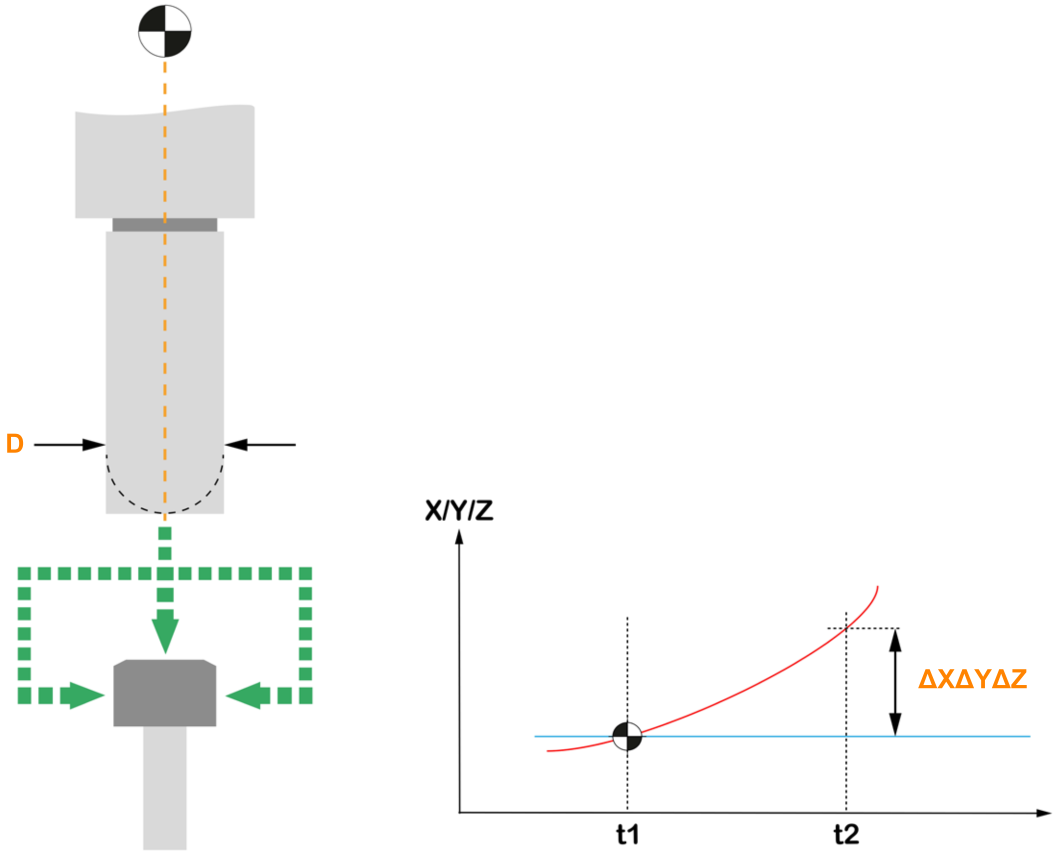

Figure 7.1 Thermal compensation cycle

Description

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

The cycle automatically moves the tool to the retract position (#107) in the spindle (Sp) axis then to a position above the stylus, before moving 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. To set base data – it 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. Only accessible faces can be measured.

2. To measure and compare – it 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 exceed the tolerance (H), an alarm will be raised.

Format

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

where [ ] denotes optional inputs.

Example: G65 P9859 C1. D16. X650. Y651 Z652

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 actual diameter of the master setting tool.

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 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. Only accessible faces can be measured.

Outputs

The following outputs are 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

Axis-swapping option

The axis-swapping option is used to define the stem (St), radial (Ra) and spindle (Sp) axis orientation of the probe. Six settings in program O9750 must be set correctly.

Setting variables

The installation wizard is used to configure the six variables required for probe orientation set-up. Variables #121, #122 and #123 should be set to the corresponding axis numbers of the machine and its orientation, while variables #144, #146 and #147 are used to identify the axis internally to the software. They are restricted to values 1 = X, 2 = Y and 3 = Z and can be arranged subject to the required probe orientation. Adjusting these values by hand is not recommended, however, the installation wizard should be used to produce the values which can then be manually entered on the machine if required.

Adjusting the spindle axis retract position (#107)

The spindle axis retract position can be used to specify a safe position for the spindle axis before a cycle and to return to after a cycle is complete. The position specified should be in machine co-ordinates.

NOTE: For the majority of installations, #121, #122 and #123 will be the same as #144, #146 and #147 respectively. However, on a non-standard machine where, for example, the axis numbers are X = 1, Z = 2 and Y = 4 and the desired probe orientation is St axis in X, Ra axis in Y and Sp axis in Z, the required set-up would be as follows:

#121=1(X)
#122=4(Y)
#123=2(Z)
#144=1(X)
#146=2(Y)
#147=3(Z)

Multiple probe or orientation option

This option can be used when multiple probes are present or to enable a single probe to be used from multiple orientations. It is also possible to combine multiple probes and multiple orientations.

CAUTION: This should be configured using the installation wizard due to the complexity involved.

Each orientation or probe will require selection. This can be done using pallet recognition or just machine position. Code will need to be inserted into the installation wizard that can be used to select the correct probe orientation and settings from the settings macro. The number of possible set-ups is currently limited to four, however this could be extended by a custom solution.

Examples of pallet recognition

IF[#1032 EQ 2]GOTO1000	Flag or marker, designating pallet 2. GOTO1000 designated for probe/orientation 1. This code will be required in programs O9750, O9890 and O9891.

Example of dividing door using position

IF[#5021 GT 1000]GOTO2000	X-axis machine value, designating partition position. GOTO2000 designated for probe/orientation 2.

Example using horizontal orientation

IF[#5025 EQ 0]GOTO3000	Select third probe/orientation if horizontal orientation. GOTO3000 designated for probe/orientation 3.

NOTE: With two or more probes, more free variables are required to store the calibration data. Each probe will use the same number of variables but can have individual base numbers. The base numbers are stored in settings program O9750.

Extended stylus life option

This option is designed to stop excessive wear at the centre of the stylus and is available with cycles O9857 and O9858. The position of the touches in the spindle (Sp) axis can be adjusted by editing #12 at the top of each cycle.

NOTE: #12=0 is set during installation. Values must be in millimetres. Negative and positive values are permissible.

```
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)
```

Chapter 9

Alarms

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

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*ALREADY*TRIGGERED”**
Cause	The probe is triggered at the beginning of a measuring move.
Action	Adjust the back-off distance (see page 2-8).

Message	**“PROBE*DID*NOT*TRIGGER”**
Cause	The probe does not register a trigger during a measuring move.
Action	Correct the error and restart the program.

Message	**“H*INPUT*NOT*ALLOWED”**
Cause	This alarm is generated by the thermal compensation cycle if the H input is used with the C1. input.
Action	Delete the H input or use the C2. input and restart.

Message	**“LONG*TOOL”**
Cause	This alarm is generated if the tool is pulled out from the collet, giving a false tool length.
Action	Inspect, adjust and remeasure 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	**“FORMAT*ERROR”**
Cause	Inputs or a combination of inputs on the call line are in error. See the relevant manual section for the required cycle.
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 **“R*INPUT*MISSING”**

Cause A compulsory R input is missing.

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

Message **“C*INPUT*MISSING”**

Cause A compulsory C input is missing.

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

Message **“W*INPUT*MISSING”**

Cause A compulsory W input is missing.

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

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 **“B4*#126*INPUTS*MIXED”**

Cause This alarm is generated by the automatic length setting cycle O9857 when attempting to use a B4. input with the spindle (Sp) axis restricted in O9750 (#126=1).

Action If access is possible, edit the settings data macro O9750 and restart the cycle (further calibration may be required). Otherwise, this cycle cannot be used.

Message **“LENGTH*OUT*OF*TOLERANCE”**

Cause The measured length 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 remeasure the tool length.

Message	**“RADIUS*OUT*OF*TOLERANCE”**
Cause	The measured radius 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 remeasure the tool radius.

Message	**“OUT*OF*TOLERANCE”**
Cause	The measured length and radius of the tool are out of tolerance. Positive or negative limits have been exceeded. This may be caused by a broken tool.
Action	Inspect and replace the tool if necessary and remeasure the tool dimensions.

Message	**“THERMAL*COMP*TOLERANCE*EXCEEDED”**
Cause	The value from the temperature compensation cycle is greater than the specified tolerance.
Action	Check the value.

Message	**“D*INPUT*MISSING”**
Cause	A compulsory D input is missing.
Action	Edit the program input line to include the compulsory input.

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, visit
www.renishaw.com/contact

H-2000-6001-00