

Tool setting for machining centres

© 1995 – 2004 Renishaw plc. All rights reserved.

Renishaw® is a registered trademark of Renishaw plc.

This document may not be copied or reproduced in whole or in part, or transferred to any other media or language, by any means, without the prior written permission of Renishaw.

The publication of material within this document does not imply freedom from the patent rights of Renishaw plc.

Disclaimer

Considerable effort has been made to ensure that the contents of this document are free from inaccuracies and omissions. However, Renishaw makes no warranties with respect to the contents of this document and specifically disclaims any implied warranties. Renishaw reserves the right to make changes to this document and to the product described herein without obligation to notify any person of such changes.

Trademarks

RENISHAW® and the probe emblem used in the RENISHAW logo are registered trademarks of Renishaw plc in the UK and other countries.

apply innovation is a trademark of Renishaw plc.

All other brand names and product names used in this document are trade names, service marks, trademarks, or registered trademarks of their respective owners.

Renishaw part no: H-2000-6082-0C-A

First issued: August 1995
Revised: February 1998
July 1998
August 2003

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 (refer to the manual 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									
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 50%; padding: 5px; vertical-align: top;"> Dual systems only Switch (Spin) probe on</td> <td style="width: 50%; padding: 5px; vertical-align: top;"> Switch on inspection probe</td> </tr> <tr> <td style="padding: 5px; vertical-align: top;"> Switch (Spin) probe off</td> <td style="padding: 5px; vertical-align: top;"> Switch on tool setting</td> </tr> <tr> <td style="padding: 5px; vertical-align: top;"> Start/Error signal</td> <td style="padding: 5px; vertical-align: top;"> Other</td> </tr> <tr> <td style="padding: 5px;">.....</td> <td style="padding: 5px;">.....</td> </tr> </table>		Dual systems only Switch (Spin) probe on	Switch on inspection probe	Switch (Spin) probe off	Switch on tool setting	Start/Error signal	Other
Dual systems only Switch (Spin) probe on	Switch on inspection probe								
Switch (Spin) probe off	Switch on tool setting								
Start/Error signal	Other								
.....								
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 70%; padding: 5px; vertical-align: top;">ADDITIONAL INFORMATION</td> <td style="width: 30%; padding: 5px; vertical-align: top;"> <input type="checkbox"/> Tick box if Form 2 overleaf has been filled in. </td> </tr> </table>		ADDITIONAL INFORMATION	<input type="checkbox"/> Tick box if Form 2 overleaf has been filled in.						
ADDITIONAL INFORMATION	<input type="checkbox"/> Tick box if Form 2 overleaf has been filled in.								
Customer's name..... Customer's address..... Customer's tel. no..... Customer's contact name.....	Date installed								
	Installation engineer								
	Date of training.....								

SOFTWARE DEVIATION RECORD

Standard Renishaw kit no.	Software disk nos.
Reason for deviation	
Software no. and macro no.	Comments and corrections
<p>The software product for which these changes are authorised is subject to copyright.</p> <p>A copy of this deviation sheet will be retained by Renishaw plc.</p> <p>A copy of the software amendments must be retained by the customer – they cannot be retained by Renishaw plc.</p>	



Caution – Software safety

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

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

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

This page is intentionally left blank

Table of contents

Before you begin

Before you begin	1
List of associated publications.....	2
Features of the tool setting software	2
Measurement values used in this manual	3
Warnings, cautions and notes	3
Software kit	4
Macro memory requirements	5
Renishaw support services	6
Calling a Renishaw subsidiary office.....	6

Chapter 1 – Getting started

Why calibrate your probe?	1-2
Notes on tool speed and feed rates	1-3
First touch spindle RPM.....	1-3
First touch feed rate	1-3
Second touch spindle RPM.....	1-3
Second touch feed rate.....	1-4
Tool offset methods.....	1-4

Chapter 2 – Software installation

Checks and adjustments.....	2-2
Active offset software adjustments.....	2-3
Software adjustment	2-4
Mazak/Meldas controls	2-4
Fanuc/Yasnac/Haas controls	2-4
G91G28 return problems	2-4
Adjusting back-off distance #506	2-5

Chapter 3 – Macro variables

Introduction	3-2
Automatically set variables.....	3-2
Manually set variables.....	3-3
Tool offsets.....	3-5
Fanuc 0M, 6M, 16M and 18M controllers.....	3-5
Fanuc 10M to 15M controllers.....	3-5
Haas controller	3-5
Meldas controller.....	3-5
Yasnac MX3, J50 controller	3-5
Yasnac I80 (M80), J300 controller	3-6
Base number settings.....	3-6
Example of macro edits O9799.....	3-8
Renishaw tool select macro	3-9

Chapter 4 – Probe calibration

Calibrating the probe	4-2
Calibration for length setting using macro O9851	4-2
Calibration of XY centre position and stylus size using macro O9852	4-3
XY calibration on a round stylus.....	4-3
XY calibration on a cube stylus	4-4
Preparing a customised installation and calibration program.....	4-5

Chapter 5 – Tool setting macro cycles

Manual length setting – macro O9851	5-2
Manual diameter setting – macro O9852	5-4
Automatic length and diameter setting – macro O9853	5-7

Chapter 6 – Tool breakage detection

Tool breakage detection – macro O9853.....	6-2
--	-----

Chapter 7 – Macro alarms

Fanuc 0M controller alarms.....	7-2
Alarms	7-2

Appendix A – Haas tool setting

Introduction	A-2
Changes and differences	A-2
Macro variables.....	A-3
Automatically set variables.....	A-3
Manually set variables	A-4
Preparing a customised installation and calibration program.....	A-6
Sample program.....	A-7

This page is intentionally left blank

Before you begin

This programming manual contains detailed information about how to use the tool setting software.

Split into seven self-contained chapters, the manual is structured to provide the information that you require to use the tool setting software effectively.

- Chapter 1 – Getting started explains why your tool setting probe must be calibrated before you start using it.
- Chapter 2 – Software installation describes how to install and customise the tool setting software on your machine.
- Chapter 3 – Macro variables describes how to use the macro variables that are required by the macro cycles.
- Chapter 4 – Probe calibration describes how to calibrate the probe to be used before using the tool setting software.
- Chapter 5 – Tool setting macro cycles describes how to use the manual length setting macro (O9851), manual diameter setting macro (O9852), and automatic length and diameter setting macro (O9853).
- Chapter 6 – Tool breakage detection describes how to use macro O9853 to detect broken tools.
- Chapter 7 – Macro alarms describes the macro alarm numbers or messages that may be displayed on the screen of the machine tool controller when an error occurs. An explanation of the meaning and possible cause of each alarm message is provided, together with typical actions you should take to correct the fault causing the alarm.

List of associated publications

When you are working with the tool setting software, you may find it useful to refer to the following Renishaw publications:

- Probe systems – installation manual for machine tools (Renishaw part no. H-2000-6040).
- Probe installation manual for HAAS VF series machines (Renishaw part no. H-2000-6066).
- Probe software for machine tools – data sheet (Renishaw part no. H-2000-2289). You will find a copy of this data sheet at the back of this manual.

Features of the tool setting software

The tool setting software provides the following features:

- Tool length setting, with automatic offset correction.
- Rotating tool diameter setting of single-point and multiple-tip tools.
- Rotating tool length setting of single-point and multiple-tip tools.
- Fully automated measurement cycle with tool change positioning and offset correction.
- Broken tool detection.
- Integral calibration cycles.

Measurement values used in this manual

Throughout this manual, metric units of measurement, e.g. millimetres, are used in the examples. The equivalent imperial measurements, e.g. inches, are shown in brackets.

Warnings, cautions and notes

Throughout this manual, warnings, cautions, and notes have the following meanings:

Warning – this is information which, if disregarded, could lead to the injury or death of an individual.

Caution – this is information which, if disregarded, could lead to damage to equipment or to software or stored data.

Note – this provides additional information to assist the reader when reading a particular paragraph.

Software kit

The tool setting software is supplied on one floppy disk for each different controller type. The Renishaw part numbers for the software kit and the floppy disk supplied as part of the kit are as follows:

Controller type	Kit no.	Floppy disk no.
Fanuc 0M (Macro B)	A-4012-0584	A-4012-0583
Fanuc 6M	A-4012-0584	A-4012-0583
Fanuc 10/11/12M	A-4012-0584	A-4012-0583
Fanuc 15/16/18M	A-4012-0584	A-4012-0583
Haas	A-4012-0635	A-4012-0634
Meldas M3, M310, M320, M330, M335 and M520	A-4013-0007	A-4013-0008
Yasnac MX3, J50	A-4014-0018	A-4014-0019
Yasnac M80 (I80), J300	A-4014-0018	A-4014-0019

Macro memory requirements

This section lists the amount of memory (in Kbytes) that is required by each macro supplied on the tool setting software floppy disk. Before you load macros, you should first work out the total amount of memory required by the macros you wish to load. Next, you should check that the machine controller has sufficient memory capacity for these macros.

If you have a version of the tool setting software supplied in Mylar paper tape format, use the following conversion data to convert from length to Kbytes or vice-versa.

Conversion: 1Kb = 2.5m (8.2ft)
 8Kb = 20m (65.6ft)

The total amount of memory required for all macros in this file is 9.4Kb. The memory requirements for each macro are as follows:

Macro number and function		Memory (Kbytes)
O9799	Variable store	1.2
O9850	Tool select	0.1
O9851	Length tool setting	2.2
O9852	Diameter tool setting	4.1
O9853	Auto length/diameter setting	1.8

Renishaw customer services

Calling a Renishaw subsidiary office

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

If you cannot find a solution, you can receive information on how to obtain customer support by contacting the Renishaw subsidiary company that serves your country.

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

- The version of the product you are using (see the *Equipment Registration Record* form).
- The type of hardware that you are using (see the *Equipment Registration Record* form).
- The exact wording of any messages that appear on your screen.
- A description of what happened and what you were doing when the problem occurred.
- A description of how you tried to solve the problem.

Chapter 1

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?	1-2
Notes on tool speed and feed rates	1-3
First touch spindle RPM	1-3
First touch feed rate	1-3
Second touch spindle RPM	1-3
Second touch feed rate	1-4
Tool offset methods	1-4

Why calibrate your probe?

In chapter 4 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?

When your probe is assembled and mounted on the machine table, it is necessary to align the stylus faces with the machine axes to avoid probing errors when setting tools. It is worth taking 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.

When the probe has been correctly set up on the machine, it is time to calibrate the probe. Calibration cycles are provided for this task. The purpose is to establish the probe stylus measuring face trigger point values 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 imperative 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 feed rates



CAUTION: Setting tools by rotating 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 metres/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 feed rate

The feed rate is calculated as follows:

$$F = .16 \times \text{rpm} \quad F \text{ units mm/min (diameter set).}$$

$$F = .12 \times \text{rpm} \quad F \text{ units mm/min (length set).}$$

Second touch spindle RPM

800 rpm.

Second touch feed rate

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

Tool offset methods

The tool setting software runs with the following tool offset methods:

1. Positive type tool offsets (gauge line to tool tip).
2. Master type tool offsets (master tool has zero length offset, all other tools are referenced to it).

NOTE: It is not possible to run this software with 'air gap' type tool offsets.

'Air gap' description:

Negative tool lengths. The spindle axis move distance necessary to reach the reference surface with the tool point.

This method requires recalibration at each job set up. The master tool 'air gap' length also changes for each job set up.

Chapter 2

Software installation

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

Contained in this chapter

Checks and adjustments	2-2
Active offset software adjustments.....	2-3
Software adjustment	2-4
Mazak/Meldas controls	2-4
Fanuc/Yasnac/Haas controls	2-4
G91G28 return problems	2-4
Adjusting back-off distance #506	2-5

Checks and adjustments

NOTE: If your machine is fitted with a Haas controller, you should also refer to appendix A, “Haas tool setting” for further information applicable to this controller.

- Check that the probe system is functional and that the stylus faces have been set parallel to the axes. You will find this described in the appropriate probe installation manual.
- Set the software variable base number in macro O9799. You will find a description of how to edit the base number setting in chapter 3, “Macro variables”.
- Set the macro variables to suit your machine. You will find a description of how to do this in chapter 3, “Macro variables”.
- Check for an active tool offset. You will find a description of how to do this in the section titled “Active offset software adjustments” later in this chapter.
- Configure the tool select macro if you intend using macro O9853. You will find a description of how to modify the tool select macro in chapter 3, “Macro variables”.
- Calibrate the probe fully using macros O9851 and O9852. You will find a description of how to calibrate your probe in chapter 4, “Probe calibration”.
- Set a tool using the manual-jog macro cycles O9851 and O9852 to establish tool geometry values. You will find this described in chapter 5, “Tool setting macro cycles”.
- Adjust the back-off distance #506 using macro cycle O9851. You will find this described later in this chapter in the section titled “Adjusting back-off distance #506”.
- Finally, test the auto-setting cycle O9853 using the same tool.

Active offset software adjustments

Carry out the following test during installation of the software to check for safe probe cycle operations.

Perform this test away from the probe and any other obstruction.

1. Enter value(s) into an active tool offset register, e.g. offset number 1.

Example: 25mm (1.0in) in the geometry offset.
5mm (0.2in) in the wear offset (if applicable).

2. Run the safe operation test as shown below.

```
%  
O0001(REN SAFE OPERATION TEST)  
G65P9851K1.          (Any small value in K is suitable – the  
                      default value is 1mm [0.04in])  
  
M30  
%
```

The Z or spindle axis should move down towards the stylus a total distance of 14mm (0.56in), i.e. by the software default amount.

3. If the distance travelled includes the tool offset amount(s), corrective action is required (see the section titled “software adjustment” later in this chapter).

Repeating the test

Repeat the test described in steps 1 to 3 above for all possible error causing conditions.

Typical test conditions include:

- Immediately after a power-up situation.
- Immediately after a previous program has finished.
- After pressing the reset button.
- After trying a program G28G91Z0 return.
- After trying a manual return sequence.
- Any other typical preferred ways of working on your machine.

The purpose of the test is to give confidence that the software is safe to use for all normal operating conditions. Any error-causing conditions found can be corrected at this stage if preferred; if not, at least you will be aware of any sequence to avoid.

Software adjustment

Mazak/Meldas controls

The software is supplied as standard to read any active tool offset amount(s) by reading the last active H word (this assumes the last H offset is still active). This is done because there is no system variable to read directly the active tool offset amount.

Fanuc/Yasnac/Haas controls

The software is supplied as standard to read the active tool offset amount using variable #5083.

This method normally works on all Fanuc controls, (not Mazak/Meldas). However, due to the machine builder parameter settings this may also give problems, particularly if the geometry or wear offset is not included in the active offset amount.

In case of difficulty, this can be changed to the Mazak/Meldas method by deleting or commenting out the following line at the end of macro O9799.

```
N110
#149=#5083          Delete or comment out this line (#149=#5083)
#31=#0
M30
```

If this change does not resolve the problem(s), it may be necessary to be aware of the error-causing condition(s), and work around them.

G91G28 return problems

If the G28G91Z0 return causes a problem, this can usually be avoided by using a G53G90Z0 return (see the section titled “Renishaw tool select macro” in chapter 3, “macro variables”).

Adjusting back-off distance #506

The static or non-rotating length setting uses the standard Renishaw two-touch method of measurement.

A back-off distance factor #506 is provided for adjusting the move distance off the surface prior to the final measuring move.

The software loads a default value of 0.5 when first run. This stored value in #506 should be optimised for minimum cycle time.

Adjust the back-off distance factor #506 by repeating the static length setting cycle O9851, reducing the #506 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 results.

This page is intentionally left blank

Chapter 3

Macro variables

This chapter describes the use of macro variables. Standard settings are installed as the software is run. The macro variables should be set prior to use.

Contained in this chapter

Introduction	3-2
Automatically set variables.....	3-2
Manually set variables	3-3
Tool offsets.....	3-5
Fanuc 0M, 6M, 16M and 18M controllers	3-5
Fanuc 10M to 15M controllers	3-5
Haas controller.....	3-5
Meldas controller.....	3-5
Yasnac MX3, J50 controller	3-5
Yasnac I80 (M80), J300 controller	3-6
Base number settings	3-6
Example of macro edits O9799	3-8
Renishaw tool select macro	3-9

Introduction

It is important that the base number is set to establish which variables are to be used by the software. See the section titled “base number settings” in this chapter if the default number shown below is not suitable.

The standard software setting variable base number for all controllers is 520, with the exception of the Haas controller.

The standard software setting variable base number for the Haas controller is 550 (see appendix A, “Haas tool setting” for details).

Automatically set variables

The following variables are set automatically during complete calibration. It is not necessary for you to preset the values.

NOTE: Variables marked * are distances in the machine co-ordinate system, and not the program co-ordinate system.

Setting variable		Internal variable
#520 (520 + 0)*	Z calibration value (non-rotating tools)	#107
#521 (520 + 1)*	Z calibration value (rotating tools)	#113
#522 (520 + 2)	Stylus size for diameter setting	#110
#523 (520 + 3)*	X axis stylus centre position	#111
#524 (520 + 4)*	Y axis stylus centre position	#112

Manually set variables

All of the following variables **MUST** be set before the cycles are used.

Setting variable		Internal variable
#525 (520 + 5)	Z approach position (used in cycle O9853 only) The first fast positioning move to the position where the tool offset is applied (height above stylus). This is shown as (B) in figure 5.3 of chapter 5, "tool setting macro cycles".	#115
#526 (520 + 6)	Z clearance position (used in cycle O9853 only) The position above the stylus for clearance moves around the stylus (height above stylus). This is shown as (C) in figure 5.3 of chapter 5, "tool setting macro cycles".	#116
#527 (520 + 7)	Tools above this diameter rotate (used in cycle O9853 only).	#117
#528 (520 + 8)	Maximum cutter diameter size.	#121
#529 (520 + 9)	Tool offset type (see the section titled tool offsets later in this chapter for the set value) e.g. #529 = 1. (A type)	#108
#530 (520 + 10)	Probe orientation. It is necessary to define the diameter measuring axis, and the radius offsetting direction for rotating length setting as follows (see Figure 3.1):	#120
	set = 1. Diameter setting:	Along the X axis.
	Rotating length setting:	Cutter radius offset in the Y– direction.
	set = –1. Diameter setting:	Along the X axis.
	Rotating length setting:	Cutter radius offset in the Y+ direction.
	set = 2. Diameter setting:	Along the Y axis.
	Rotating length setting:	Cutter radius offset in the X– direction.
	set = –2. Diameter setting:	Along the Y axis.
	Rotating length setting:	Cutter radius offset in the X+ direction.

NOTE: This figure shows the possible positions of the probe.

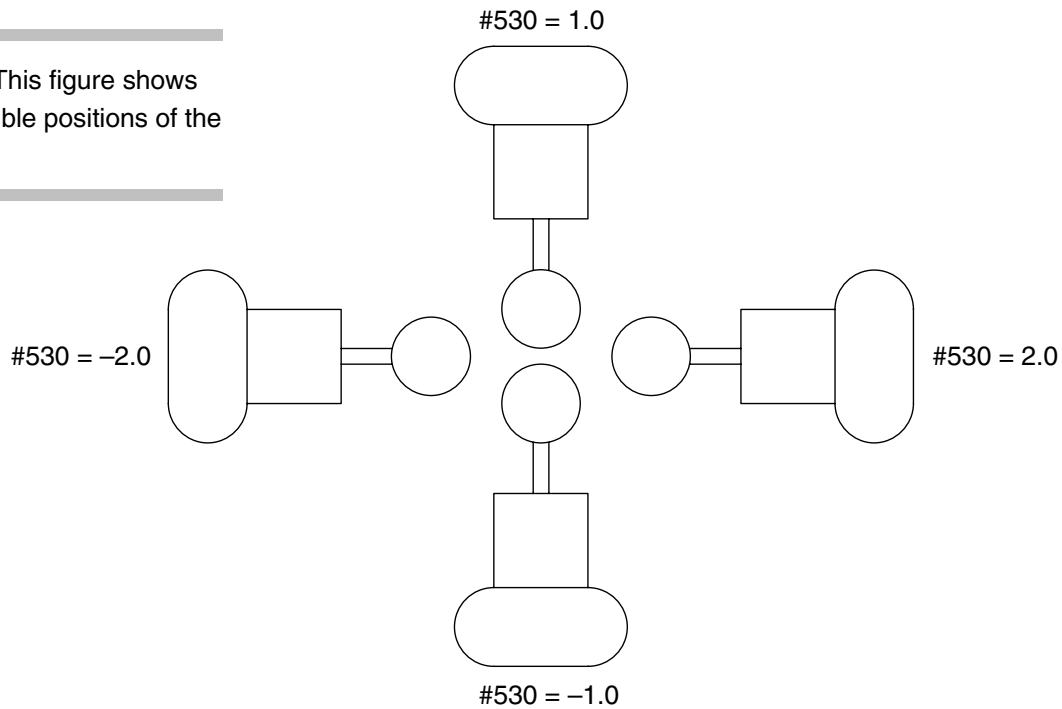


Figure 3.1 #530 (assuming a default base number of 520)

#531 (520 + 11) Flag for inch or metric setting data

set = 0. Storing metric data in variables.

set = 1. Storing inch data in variables.

This flag must be set to represent the units entered in the previous setting variables.

Tool offsets

Settings for #529 (520 + 9) as follows:-

Fanuc 0M, 6M, 16M and 18M controllers

Set = 1.	(A type)	Single offset register	(1 register)
Set = 2.	(B type)	Length, geometry and wear register	(2 registers)
Set = 3.	(C type)	Length, radius geometry and wear register	(4 registers)

Fanuc 10M to 15M controllers

Set = 11.	(A type)	Single offset register	(1 register)
Set = 12.	(B type)	Length, geometry and wear register	(2 registers)
Set = 13.	(C type)	Length, radius geometry and wear register	(4 registers)

Haas controller

Set = 13.	(-)	Length, radius, geometry and wear register	(4 registers)
-----------	-------	--	---------------

Meldas controller

Set = 11.	(Type 1)	Single offset register	(1 register)
Set = 13.	(Type 2)	Length, radius, geometry and wear register	(4 registers)

Yasnac MX3, J50 controller

Setting	H/D common type	Length	Radius
Set = 10	(basic)	H1 to H99	D1 to D99
Set = 10	(299 pair)	H1 to H299	D1 to D299

Setting	H/D separated type	Length	Radius
Set = 11	(basic)	H1 to H49	D1 to D49
Set = 12	(299 pair)	H1 to H149	D1 to D149

Yasnac I80 (M80), J300 controller

Setting	H/D common type	Length	Radius
Set = 0	(basic)	H1 to H99	D1 to D99
Set = 0	(299 pair)	H1 to H299	D1 to D299
Set = 0	(999 pair)	H1 to H999	D1 to D999
Set = 0	(1199 pair)	H1 to H1199	D1 to D1199

Setting	H/D separated type	Length	Radius
Set = 1	(basic)	H1 to H49	D1 to D49
Set = 2	(299 pair)	H1 to H149	D1 to D149
Set = 3	(999 pair)	H1 to H499	D1 to D499
Set = 4	(1199 pair)	H1 to H599	D1 to D599

Base number settings

The base number defines the first variable in the set of variables used for setting and calibration data. The default is 520, i.e. #520. It may be changed by editing the settings macro O9799. The base number edit is found near the top of this macro.

Reason for changing the base number

The default setting uses variables #520 through to #531 inclusive. This suits all listed controls except the Fanuc 6M controller.

When this variable range is already used for other purposes it will be necessary to define a different range:

- An additional retained common macro variable option may be used.
- Spare tool offsets may be used. Use a 2000 series system variable base number, e.g. 2088 to use offsets 88 through to 99.



CAUTION: If tool offset registers are used, it is not possible to switch between inch and metric units using G20/G21, because the tool offset data is converted automatically.

When installing the tool setting software without a Renishaw inspection package, use the default settings unless #520 to #530 are being used for other purposes.

When using the software in conjunction with other Renishaw software, avoid macro #500 series clashes by changing the base number.

Fanuc 6 systems

Only #500 to #509 variables are available.

It is necessary to use tool offsets. Set the base number to 2088.

Alternatively, set the base number to 500.

Hard-code the data in internal variable number #120 in macro O9799 (see the section titled “example of macro edits O9799” later in this chapter).

Vector software packages

Variables #500 to #549 are used (change the base number).

Inspection plus software

Use the standard base number (#520), unless the multi-stylus calibration feature is used. In this case #500 to #549 are used (change the base number).

Example of Macro Edits O9799



CAUTION: #107, #113, #110, #111 and #112 contain calibration data and must be updated if recalibration is performed.

Hard-coded data edits:

O9799(REN SETTING)

(40120583.0D)

#30=520(EDIT BASE NO.)

(#[#30+11]=0)(1-INCH 0-MET STORE FLAG)

G90G80G40G0

... continues ...

N104

IF[#118NE2]GOTO105

#[#30+2]=#110*#31

#[#30+3]=#111*#31

#[#30+4]=#112*#31

GOTO106

N105

#107=#[#30+0]/#31(Z POS. STATIC)

#113=#[#30+1]/#31(Z POS. ROTATING)

#110=#[#30+2]/#31(STYLUS SIZE)

#111=#[#30+3]/#31(X POS.)

#112=#[#30+4]/#31(Y POS.)

#115=#[#30+5]/#31(Z APPROACH)

#116=#[#30+6]/#31(Z CLEARANCE)

#117=#[#30+7]/#31(TOOLS ABOVE THIS ROTATE)

#121=#[#30+8]/#31(MAX. CUTTER DIA.)

#108=#[#30+9](OFFSET TYPE)

#120=#[#30+10](PROBE DIREC.)

N106

... continues ...

M99

set #30=520

set #[#30+11]=0

Remove brackets to activate.

set #120=-1.

(options 1., -1., 2., -2.)

NOTE: Any of the variables between blocks N105 and N106 may be hard-coded with actual values to prevent loss of data and to avoid using the #500 series variables.
e.g. #117 = 20.0/#31 (TOOLS ABOVE THIS ROTATE) 20.0 mm.

Renishaw tool select macro

This macro is supplied with the following default values. It should be modified to suit your specific machine requirements for tool selection (see the caution and notes below).



CAUTION: The auto cycle G65 P9853 is completed with a G28 return to home position. This cancels the active tool offset. Any program move following this call is applied without the tool offset being active. The result may cause a collision if the offset is not re-applied first.

```
O9850(REN TOOL SELECT)
G91G28Z0
G90
IF[#20EQ0]GOTO1
M06T#20
N1
M99
```

-
- NOTES:**
1. It may be possible on later controls to use G90 G53 Z____ return. This offers a safe alternative method, and all the G91 G28 Z____ returns should be replaced when possible (modify macros O9850 and O9853).
 2. Some machines may use G30 instead of G28 for tool change return position.
 3. On some machines it is not possible to select a tool already in the spindle. Insert the following line (*) to avoid this problem:

```
IF[#20EQ0]GOTO1
* IF[#4120EQ#20]GOTO1
M06T#20
N1
M99
```

This page is intentionally left blank

Chapter 4

Probe calibration

Before a probe is used, it is important that you calibrate it correctly. This chapter describes how to carry out the calibration. If you need to know more about calibrating a probe, you will find helpful information contained in chapter 1, “getting started”.

Contained in this chapter

Calibrating the probe.....	4-2
Calibration for length setting using macro O9851	4-2
Calibration of XY centre position and stylus size using macro O9852	4-3
XY calibration on a round stylus	4-3
XY calibration on a cube stylus	4-4
Preparing a customised installation and calibration program.....	4-5

Calibrating the probe

Carry out the following actions to fully calibrate the probe on the machine table. This procedure establishes the stylus face trigger positions. If you intend only to perform length setting, using macro O9851, follow the procedure described in the section titled “Calibration for length setting using macro O9851”.

The following procedures use macros O9851 and O9852. If you are unsure how to use these macros, refer to chapter 5, “Tool setting macro cycles” before following the examples below.

NOTES: The positions are found with respect to the machine reference point, i.e. the machine co-ordinates.
The spindle does not rotate during calibration cycles.

Calibration for length setting using macro O9851

Use a master tool (reference arbor) of known length in the spindle. Alternatively, it is sometimes possible to use the spindle nose (zero tool length).

FORMAT G65 P9851 Kk [Qq Zz]
 [] Denotes optional inputs

Example: G65P9851K149.536Q5.Z-15.5

INPUTS Input descriptions are the same as described in macro O9851, but the Kk input is specifically for calibration and is described here. Refer to chapter 5 - Tool setting macro cycles for other input descriptions.

Kk k = indicates a calibration cycle. Enter the exact length of the master tool (reference arbor).

EXAMPLE 1**Calibration using a reference arbor, e.g. MDI mode**

Jog to the start position, i.e. position a reference arbor 10.0mm (0.4in) above the stylus. The reference arbor moves down to qualify on top of the stylus (non-rotating), and then returns automatically.

NOTE: After the first touch, two other controlled touches occur on top of the stylus automatically.

G65P9851 K149.536 K149.536 = Calibration length of reference arbor.

The surface position of the stylus with respect to the machine reference is found and calibration values are stored.

Calibration of XY centre position and stylus size using macro O9852



CAUTION: To ensure that the master tool (reference arbor) calibrates on a diameter, it should be a solid cylinder type without flutes, however no spindle rotation will occur during calibration.

XY calibration on a round stylus

X axis and Y axis positioning are achieved with two separate operations of macro O9852.

1. Decide which axis is to be used to measure the tool diameters.
Set the probe direction variable #530 (assuming the default base number) for the opposite axis measuring direction. For example, if tool setting in the Y-axis is required, select the X-axis #530 = 1 for the first operation.
2. Position the master tool (reference arbor) 10 mm (0.4 in) above and approximately on-centre of the stylus.

- Run diameter calibration macro cycle O9852. This establishes the X-axis position (see the example below). At the end of the cycle the spindle is returned to the stylus centre ready for the next stage.



CAUTION: Do not move the spindle before completing steps 4 and 5.

4. Change variable #530 to the final operating direction, e.g. #530 = 2.
5. Run diameter calibration macro O9852 again. This establishes the Y-axis position and stylus size. At the end of the cycle the spindle is returned to the stylus centre (see the example below).

XY calibration on a cube stylus

When using a cube stylus, accurate on-centre positioning in both directions is not necessary.

1. Position the master tool (reference arbor) 10mm (0.4in) above and approximately on-centre of the stylus.
2. Set the correct axis direction value to #530 (assuming the default base number) then run the macro.
3. Run diameter calibration macro O9852. This establishes the centre position and the stylus size. At the end of the cycle the spindle is returned to the stylus centre (see the example below).

FORMAT

G65 P9852 Ss Kk [Zz]

[] Denotes optional inputs

Example: G65P9852S20.001K10.Z-15.5

INPUTS

Input descriptions are the same as described in macro O9852, but the Ss and Kk input must always be used for calibration and are described here. Refer to chapter 5, “Tool setting macro cycles” for other input descriptions.

Ss s = Master tool (reference arbor) diameter.
Enter the exact size.

Kk k = indicates a calibration cycle. Enter the nominal stylus size.

EXAMPLE 2

Calibration using a master tool (reference arbor)

Use a master tool (reference arbor) of known diameter in the spindle.

Two measuring moves along a specified axis occur – one on each side of the stylus, and at a distance of 14.0 mm (0.55 in) below the start point – using standard settings, e.g. MDI Mode.

Jog to the approximate centre of the stylus and to 10.0 mm (0.4 in) above the top cube surface.

G65P9852 S20.001 K10.0 S20.001 = 20.001 mm (0.787 in)
reference arbor size.

K10.0 = 10.0 mm (0.394 in)
nominal stylus size.

The following calibration data is stored:

- Calibration size of stylus.
- Stylus centre line position for the chosen axis.

Preparing a customised installation and calibration program

NOTE: For Haas controllers refer to appendix A, “Haas tool setting”.

It is possible to prepare a customised program for installing set-up data and calibrating the probe system. This is useful when the software is to be installed on several similar machines.

Manually position the reference tool to approximately 10mm (0.4in) above the stylus and approximately on the centre-line of the stylus.

When the custom program is run, all macro variable data is set and the probe is fully calibrated.

EXAMPLE 3**Showing a typical program to set up and calibrate on a round stylus**

- Tool setting in the Y axis direction.
- Rotating length setting by radius offsetting in the X+ direction.

The reference tool makes the following measurements:

1. Z measure on top of the stylus (three touches).
2. X diameter measure on both sides of the stylus (the direction depends on the #530 setting).
3. Y diameter measure on both sides of the stylus (the direction depends on the #530 setting).
4. Return to 10mm (0.4in) above the stylus and on-centre.

%

O8000 (CUSTOM CALIBRATION)

#506= 0.1 (SURFACE BACK-OFF DISTANCE)

#525= 100. (RAPID APPROACH POS. IN Z)

#526= 10. (CLEARANCE POS. IN Z)

#527= 10. (TOOLS ABOVE THIS ROTATE)

#528= 89. (MAX. CUTTER DIAMETER)

#529= 13. (WORK OFFSET TYPE)

#531= 0 (INCH/METRIC STORED DATA)

(LENGTH CALIB)

G65P9851K95.03

(X DIAM CAL)

#530= 1. (X-AXIS SELECT)

G65P9852S10.0K12.7Z-15.(S-TOOL DIA K-STYL SIZE)

(Y DIAM CAL)

#530= -2. (Y-AXIS SELECT)

G65P9852S10.0K12.7Z-15.(S-TOOL DIA K-STYL SIZE)

M30

%

Chapter 5

Tool setting macro cycles

This chapter describes how to use the tool setting software macro cycles. The cycles are used for manual tool length and diameter setting, automatic tool length and diameter setting, and broken tool detection.

Contained in this chapter

Manual length setting – macro O9851	5-2
Manual diameter setting – macro O9852	5-4
Automatic length and diameter setting – macro O9853	5-7

Manual length setting – macro O9851

NOTE: The probe must first be calibrated (see use of Kk input).

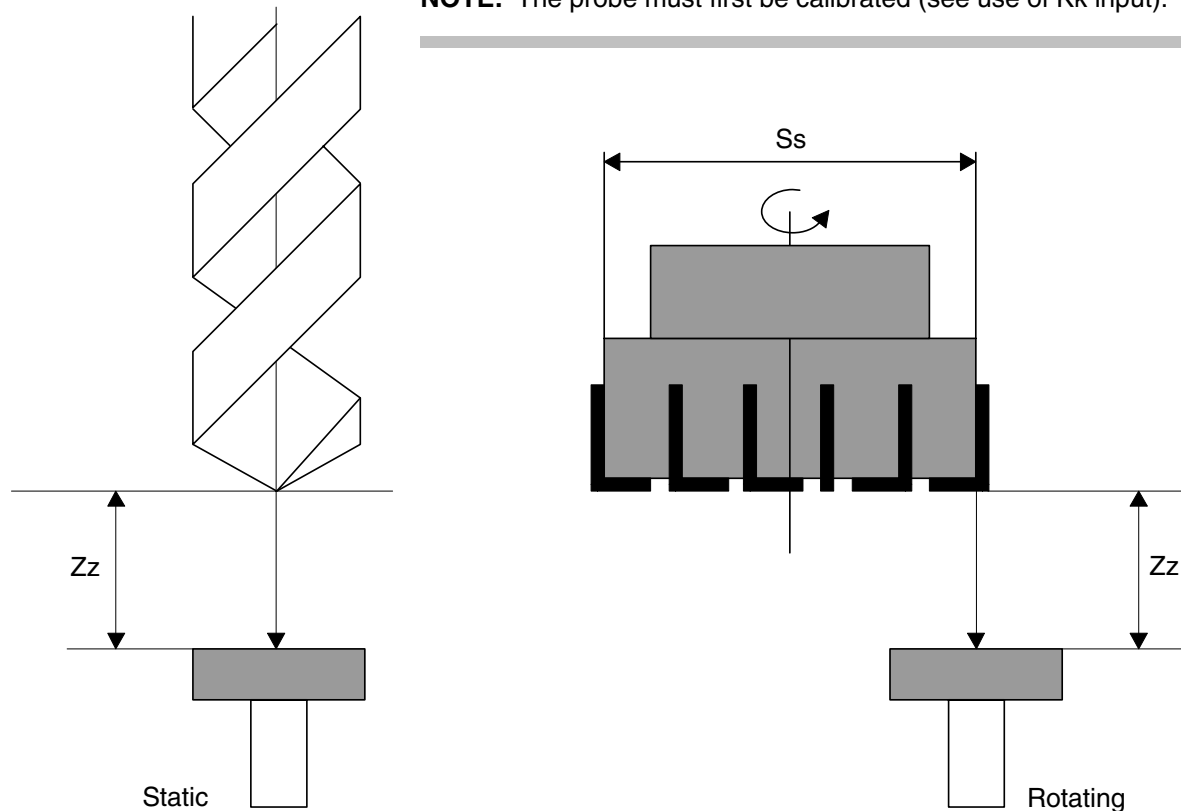


Figure 5.1 Tool length measurement

DESCRIPTION

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

APPLICATION

Jog the spindle to position a cutter tooth directly over the probe stylus within 10.0 mm (0.4 in) of the surface. The cycle can be run either by writing a small program to call the macro with suitable inputs or alternatively, on some machines, using the manual data input (MDI) method. The tool returns to the Z clearance position above the stylus.

The total Z axis movement with default Zz and Qq values is 14.0 mm (0.55 in).

FORMAT

G65 P9851Ss Kk Tt [Qq Zz Mm Hh]
[] denotes optional inputs

Example: G65P9851S80.K149.54T8.Q5.Z-15.5M30H.5

INPUTS

- Ss s = Cutter diameter or reference tool diameter (omit for non-rotating operation).
- S+s +s = Right-handed cutting tools.
- S-s -s = Left-handed cutting tools.
e.g. S80. = 80 mm (3.15 in) diameter cutter – right-handed cutting tool.
- Kk k = Calibration cycle. Refer to chapter 4, “probe calibration”.
- Tt t = Tool offset number (not required when calibrating.)
- Qq q = Probe overtravel distance
(4.0 mm [0.16 in] default).
- Zz z = Incremental depth for measurement from the start position (–10.0 mm [–0.4 in] default). The z value is normally a minus (–) value.
- Mm m = A spare tool offset number to use as a broken tool flag location. Refer to chapter 6, “tool breakage detection”.
- Hh h = Tolerance is set to the $\pm h$ value programmed. Refer to chapter 6, “tool breakage detection”.

EXAMPLE 1**Length tool setting – non-rotating, e.g. MDI mode**

Jog the cutter to the start position, i.e. position a cutter tooth 10.0mm (0.4in) above the stylus.

G65P9851T8. T8. = tool offset number 8 set
for length

EXAMPLE 2**Length tool setting – rotating, e.g. MDI mode**

Jog the cutter to the start position, i.e. position a cutter tooth 10.0 mm (0.4 in) above the stylus.

G65P9851 S80. T8. S80. = cutter diameter.

T8. = tool offset number 8
set for length.

ALARMS

Refer to chapter 7, “macro alarms” for further details of alarms and the actions you should take to correct the fault causing an alarm.

Manual diameter setting – macro O9852

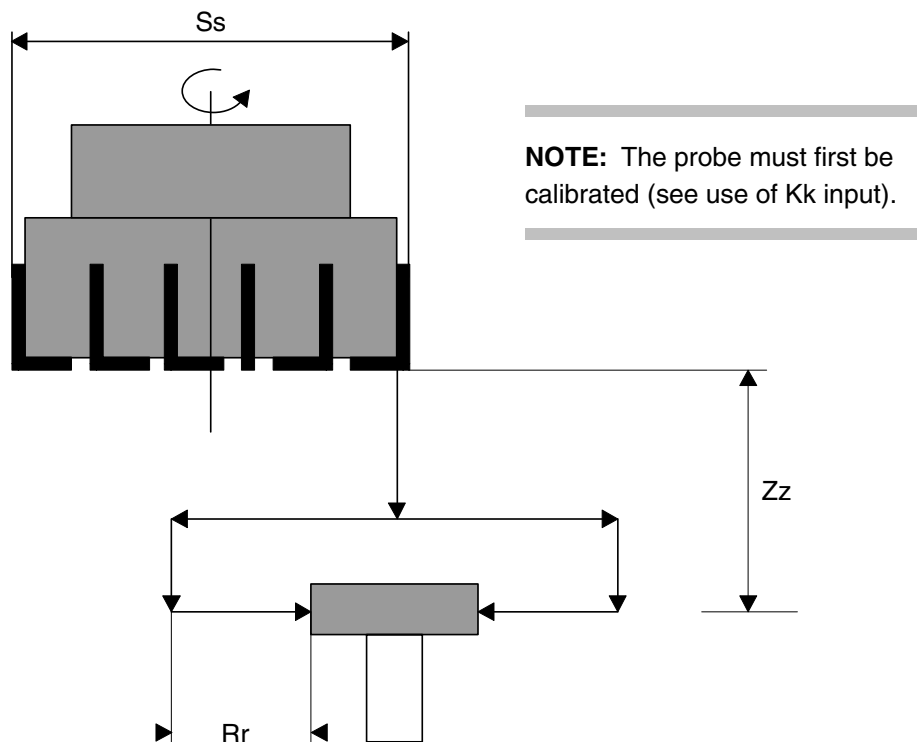


Figure 5.2 Tool cutting radius measurement

DESCRIPTION

This cycle is used to measure the effective cutting radius of a rotating tool by taking two measurements, one on either side of the tool setting stylus.

APPLICATION

Jog the spindle to position a cutter tooth directly over the probe stylus within 10.0 mm (0.4 in) of the surface. The cycle can be run either by writing a small program to call the macro with suitable inputs or alternatively, on some machines, using the manual data input (MDI) method.

The cycle first moves the cutter in X and Y over the stored centre position of the stylus before completing two measuring moves, one on either side of the stylus, with the tool rotating. The tool then returns to the Z clearance position above the stylus, and on the stylus centre-line.

FORMAT

G65 P9852 Ss Kk Dd [Zz Rr Mm Hh Ii]
[] denotes optional inputs

Example: G65P9852S80.K10.0D8.Z-20.5R3.M30H.5I.01

INPUTS

Ss s = Cutter diameter or reference tool diameter.

S+s +s = Right-handed cutting tools.

S-s -s = Left-handed cutting tools.
e.g. S80. = 80 mm (3.15 in) diameter cutter –
right-handed cutting tool.

Kk k = Calibration cycle. Enter the stylus size. For further
information, refer to chapter 4, “probe calibration”.

Dd d = Tool radius offset number to be updated (not
required when calibrating using the Kk input).

Zz z = Incremental depth for measurement from the start
position (–15.0 mm [–0.60 in] default Z axis
movement). The z value is normally a minus (–)
value.

Rr r = Overtravel distance, and radial clearance when
moving down the side of the stylus (4.0 mm
[0.16 in] default).

Mm m = A spare tool offset number to use as a broken tool
flag location. Refer to chapter 6, “tool breakage
detection”.

Hh h = The tolerance is set to the $\pm h$ value programmed.
Refer to chapter 6, “tool breakage detection”.

li i = Size adjustment to compensate for cutting
conditions. A positive value sets the tool radius
small by the stated amount, e.g. I=.01 sets the
cutter radius small by 0.01.

It may also be used to create zero nominal tool
radius values by entering the nominal radius of the
cutter.

EXAMPLE**Radius tool setting**

Two measuring moves occur, one on either side of the stylus, following an XY axis move to the centre, e.g. MDI Mode.

Jog the cutter to the start position, i.e. position a cutter tooth 10 mm (0.4 in) above the stylus.

G65P9852 S80. D8. S80. = cutter diameter
(this input is used to
calculate the clearance
moves and spindle rpm).

D8. = tool offset number 8
set for radius.

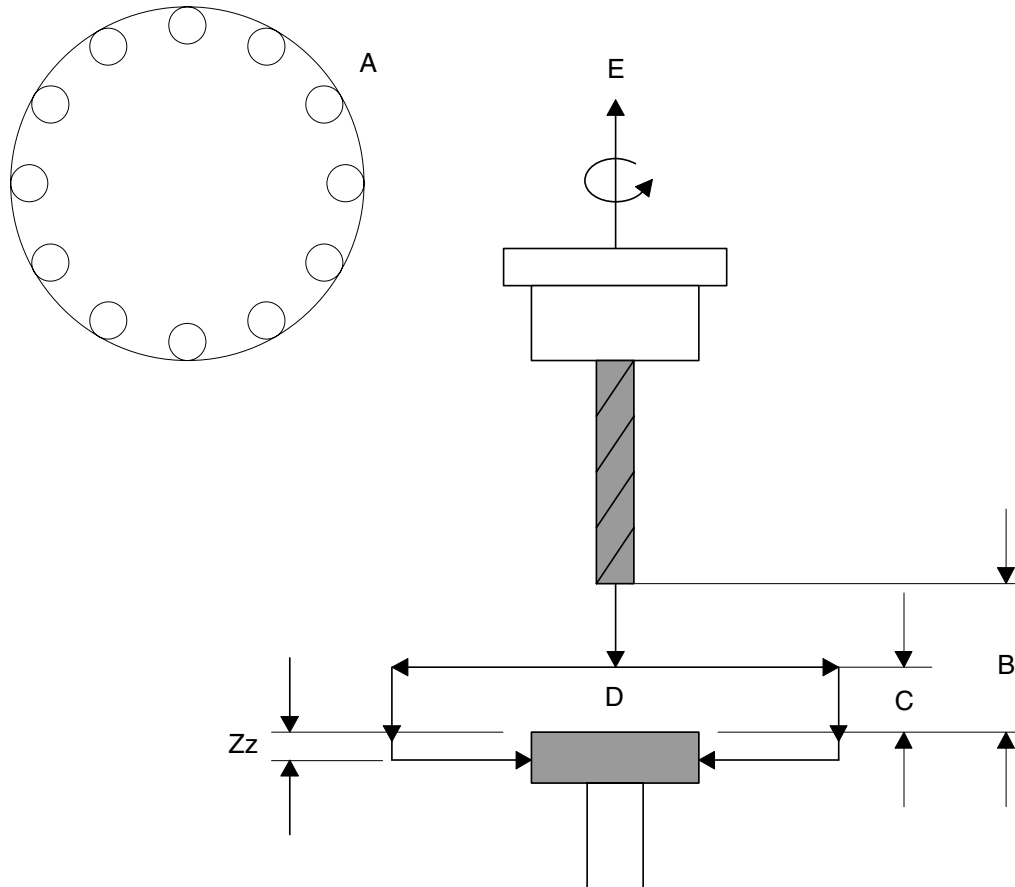
ALARMS

Refer to chapter 7, “macro alarms” for further details of alarms and the actions you should take to correct the fault causing an alarm.

Automatic length and diameter setting – macro O9853



CAUTION: Read the section titled “Renishaw tool select macro” in chapter 3, “macro variables” before running this program.



A = Take tool from tool changer

B = Z approach position (fast)

C = Z clear position (slow)

D = Measure

E = Retract to home position

Figure 5.3 Cutting radius of rotating tool measurement

DESCRIPTION**Radius setting:**

This cycle is used to measure the effective cutting radius of a rotating tool by taking two measurements – one on either side of the tool setting stylus. The cycle selects the tool from the tool changer and moves to the stylus automatically, where the measurement is made.

Length setting:

This cycle is used to measure the effective cutting length of a rotating (or non-rotating) tool by taking one measurement – on the top of the tool setting stylus. The cycle selects the tool from the tool changer and moves to the stylus automatically, where the measurement is made.

Broken tool checking is also possible using this cycle.

APPLICATION

The cycle can be run either by writing a small program to call the macro with suitable inputs or, on some machines, using the manual data input (MDI) method. The cycle selects and measures the chosen tool automatically.

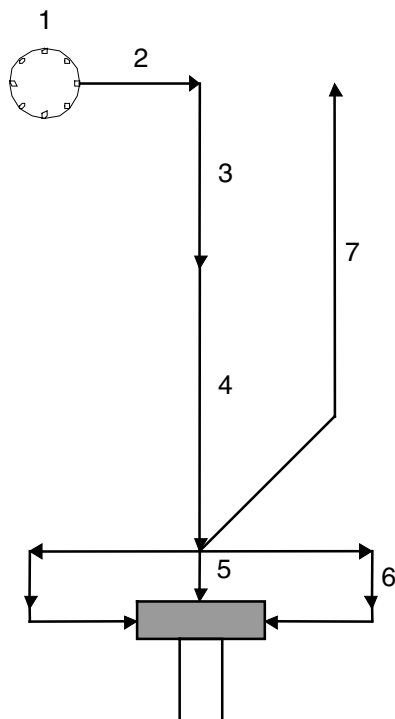


Figure 5.4 Probe movements

NOTE: Approximate tool offset values **MUST** be stored in the tool registers before using the cycle.

The following operations occur, depending on the inputs used:

1. Select the tool from the tool changer.
2. Move in X and Y over the stylus.
3. Fast move down to the approach position and apply the tool offset (protected move).
4. Protected move down to the clearance position.
5. Set the length (rotating or non-rotating) if B1. or B3. input is used.
6. Set the radius (rotating) if B2. or B3. input is used (measurement using both sides of the stylus).
7. Retract to the home position.

FORMAT

G65 P9853 Bb Tt.ttt [Dd SsQq Rr Zz Mm Hh Ii]
 [] Denotes optional inputs

Example: G65P9853B1.T1.D20.S30.Q3.R3.Z-4.M30 H.5I.01

NOTE: 'D' is compulsory if used with B2. or B3.

INPUTS

Bb b = Set as follows:

1. Length set only (default).
2. Diameter set only.
3. Length and diameter set.

Tt t = Tool number and length offset number is assumed the same, e.g. T1 (tool number 1, offset register 1).

Tt.ttt t.ttt = When tool number and length offset number are different, e.g. T1.020 (tool number 1, offset register 20).

NOTE: Note the use of the 3-digit format after the decimal point.

Dd d = Tool radius offset number for update (use for rotating tool setting only).

NOTE: A nominal tool radius must be entered in the tool offset register if the Ss input is not used.

D+d +d = Right-handed cutting tools.

D-d -d = Left-handed cutting tools.

Ss s = Cutter diameter.

This input is not used when the Dd tool offset register contains a nominal tool radius value.

S+s +s = Right-handed cutting tools.

S-s -s = Left-handed cutting tools.
 e.g. S80 = 80 mm (3.15 in) cutter diameter.

Qq	q	= Length overtravel (default 4.0 mm [0.16 in]).
Rr	r	= Overtravel distance, and radial clearance when moving down the side of the stylus (4.0 mm [0.16 in] default).
Zz	z	= Depth for diameter measure from the top of the stylus (default -5.0 mm [-0.20 in]), where Z-ve is down.
Mm	m	= A spare tool offset number to use as a broken tool flag location. Refer to chapter 6, "tool breakage detection".
Hh	h	= The tolerance is set to the $\pm h$ value programmed. Refer to chapter 6, "tool breakage detection". An alarm is generated if it is out of tolerance (see Mm).
li	i	= Size adjustment to compensate for cutting conditions. A positive value sets the tool radius small by the stated amount, e.g. I=.01 sets the cutter radius small by 0.01. It may also be used to create zero nominal tool radius values by entering the nominal radius of the cutter.

NOTE: In the following examples, nominal tool offset length values must be loaded in the tool registers before execution of the cycles.

EXAMPLE 1**B1. tool length setting only – non-rotating, e.g. MDI mode**

G65P9853 B1. T1. T1. = Tool number 1 is selected and offset number 1 is set for length

Alternatively

G65P9853 B1. T1.020 T1.020 = Tool number 1 is selected and offset number 20 is set for length.

EXAMPLE 2**B1. tool length setting only – rotating, e.g. MDI mode**

G65P9853 B1. T1. S80. T1. = tool number 1 is selected and offset number 1 set for length.

Alternatively

G65P9853 B1. T1.020 S80. T1.020 = tool number 1 is selected and offset number 20 set for length.

S80.= 80.0 mm (3.15 in) diameter cutter (will offset by 40.0 mm [1.57 in] and rotate).

EXAMPLE 3**B2. tool diameter setting only, e.g. MDI mode**

G65P9853 B2. T1. D20. [S30.] T1. = tool number 1 is [] optional selected and offset number 1 is applied.

Alternatively

G65P9853 B2. T1.020 D20. [S30.] T1.020 = tool number 1 is [optional] selected and offset number 20 is applied.

D20. = Tool radius offset number to be set (it must already have a nominal value loaded if the Ss input is not used).

S30. = 30.0mm (1.18in) diameter cutter (allows for 15.0 mm [0.6 in] radius cutter and rotate).

EXAMPLE 4

B3. tool length and diameter setting, e.g. MDI mode

G65P9853 B3. T1. D20. [S30.]
[] optional

T1. = tool number 1 is
selected and offset number 1
set for the length.

Alternatively

G65P9853 B3. T1.020 D20. [S30.]
[] optional

T1.020 = tool number 1
is selected and length offset
number 20 is applied.

D20. = tool radius offset number
to be set (must already have
nominal value loaded if the Ss
input is not used).

S30. = 30mm (1.18in) diameter
cutter (allows for 15.0 mm
[0.6 in] radius cutter and rotate).

NOTE: Length setting automatically selects the rotating or non-rotating length setting depending on the size of the cutter (see chapter 3, “macro variables” for setting information).

EXAMPLE 5

O1000(TOOL SET PROGRAM)



CAUTION: Nominal tool offset values must be entered in the relevant tool offset registers before the cycle is run

Full tool setting program (four tools).

A dedicated program for setting four tools.

Set tool 1	N1G65P9853B1.T1.S80 M00
Set tool 2	N2G65P9853B1.T2. M00
Set tool 3	N3G65P9853B2.T3.D23.S30. M00
Set tool 4	N4G65P9853B3.T4.D24.S20. M30

NOTE: Block number 'Nn' and program stop 'M00' are used to aid the setting of individual tools instead of running a full sequence.

Tool 1	80 mm (3.15 in) diameter face cutter. Set length (rotating).
Tool 2	10 mm (0.394 in) diameter slot drill. Set length (non-rotating)
Tool 3	30 mm (1.18 in) diameter slot drill. Set diameter (rotating).
Tool 4	20 mm (0.787 in) diameter end mill. Set length and diameter (rotating).

EXAMPLES 6 AND 7 General tool setting programs



CAUTION: Nominal tool offset values must be entered in the relevant tool offset registers before the cycle is run

Examples showing two general-purpose macros prepared for setting a full suite of 20 tools.

EXAMPLE 6

O7000(LENGTH ONLY)

```
N1G65P9853B1.T1.
M00
N2G65P9853B1.T2.
M00
N3G65P9853B1.T3.
M00
N4G65P9853B1.T4
M00
```

continues

```
N20G65P9853B1.T20.
M00
M30
```

EXAMPLE 7

The tool radius offset register is fixed as (20 + tool number) in the following example.

O7001(LENGTH AND RADIUS)

N1G65P9853B3.T1.D21.

M00

N2G65P9853B3.T2.D22.

M00

N3G65P9853B3.T3.D23.

M00

N4G65P9853B3.T4.D24.

M00

continues

N20G65P9853B3.T20.D40.

M00

M30

NOTE: Use block number 'Nn' search and program stop 'M00' to aid the setting of individual tools

EXAMPLE 8

Tool setting included in the part program

This example shows a complete tool setting sequence within the part program.

The tools must already be approximately set to nominal tool length (as per a tooling sheet) within ± 4 mm. The program example shows how the nominal tool data is automatically loaded to the tool offset registers.



CAUTION: This example uses the Fanuc 0M system variables for loading 'C' type tool offset registers

Tool 1	80 mm (3.15 in) diameter x 120 mm (4.724 in) long face mill
Tool 2	25 mm (0.984 in) diameter x 180 mm (7.086 in) long drill
Tool 3	16 mm (0.629 in) diameter x 100 mm (3.937 in) long end mill
Tool 4	27.300 mm (1.075 in) diameter x 170 mm (6.693 in) long bore bar
O1000	
/M99P20	Use 'Block Delete' to set tools or go to N20
(TOOL SETTING)	
#2001=120.0	Load T1 nominal length
#2002=180.0	Load T2 nominal length
#2003=100.0	Load T3 nominal length
#2203=8.0	Load T3 nominal radius
#2004=170.0	Load T4 nominal length
#2204=13.65	Load T1 nominal radius
N1G65P9853B1.T1.S80	Tool 1 – Offset and set length rotating
N2G65P9853B1.T2.	Tool 2 – On centre, set length non-rotating
N3G65P9853B3.T3.D23.S16.	Tool 3 – Offset and set length and radius
N4G65P9853B3.T4.D24.S27.3	Tool 4 – Offset and set length and radius
(MACHINING)	
N20M06T1(FACE MILL)	continue machining sequence
continue machining program	
M30	

This page is intentionally left blank

Chapter 6

Tool breakage detection

This chapter describes how to use macro cycle O9853 to detect a broken tool.

Contained in this chapter

Tool breakage detection – macro O9853..... 6-2

Tool breakage detection – macro O9853

NOTE: Tool offsets are not adjusted when using the tool breakage detection feature.

DESCRIPTION	<p>The table probe may be used to detect broken tools. This is achieved by using the auto tool setting macro O9853 cycle.</p> <p>The auto tool setting macro O9853 cycle either raises an alarm or sets a flag. This is dependent on the inputs used with the macro program. Raising an alarm stops further execution of the program. The 'flag' method gives the user flexibility in deciding the best course of action that allows for continuous running. This is particularly useful in flexible manufacturing applications.</p> <p>Additional macro logic is required by the application program to make use of the flag method.</p>
APPLICATION	<p>The macro G65P9853 measures a tool independently of any co-ordinate system, making it possible to execute it from within a part program.</p> <p>When a tool is found to be out of tolerance, the program either raises an alarm or sets a flag. When the flag method is used the flag is set to '1' but the 'BROKEN TOOL' alarm is not raised. This gives the user flexibility to decide what is to happen next, e.g. calling of a sister tool.</p>
FORMAT	<p>G65 P9853 Bb Tt.ttt Hh [Dd Ss Qq Rr Zz Mm li] [] Denotes optional inputs</p> <p>Example: G65P9853B1.T1.H.5D8.S30.Q3.R3.Z-4.M30I.01</p>
INPUTS	<p>Input descriptions are the same as described in macro O9853, but the Hh and Mm inputs are specifically for tool breakage detection and are described here. Refer to chapter 5, "tool setting macro cycles" for other input descriptions.</p>

Hh h = Tool breakage tolerance ($\pm h$) value.

Example: H.5 will check whether the tool is within ± 0.5 mm of the existing tool offset value.

Mm m = A spare tool offset number to use as a broken tool flag location. If it is used, a flag is set, but a macro alarm is not generated.
(Only use in conjunction with the Hh input)

OUTPUTS

Mm m = The tool offset store chosen is set to 1 when a broken tool is found, or 0 when in tolerance.

NOTE: The part program must check this flag for corrective action because a macro alarm will not occur.

EXAMPLE 1

Broken Tool – Flag Method

M06T1 Select T1 for machining.

continue machining to end of tool 1 sequence

End of tool 1 sequence

G65P9853B1.T1.H.5M30 Tool breakage check to set a flag.

IF [#2030 EQ1] GOTO ** GOTO block number N** if the
(conditional statement) flag is set to '1'. Otherwise
continue.

M06 T2 Select next tool and continue program.

N**(recover routine) N** is where a sister tool and
pallet change may be invoked.

EXAMPLE 2**Broken tool – alarm method**

M06T1	Select T1 for machining.
continue machining to end of tool 1 sequence	
End of tool 1 sequence	
G65P9853B1.T1.H.5	Tool breakage check. 'BROKEN TOOL' alarm, or continue program.
M06 T2	* Select next tool and continue program.

NOTE: * denotes that this tool change method may not suit all machines.

The 'BROKEN TOOL' alarm is only raised in the example above when the 0.5mm (0.02in) value has been exceeded.

The flag setting method is more suited to users with flexible manufacturing systems, where the raising of the alarm is not suitable.

When it is necessary to incorporate tool breakage detection (using the flag method) with the Fanuc tool life management system, your machine builder must be consulted. This will require work to the PLC by the machine builder.

Chapter 7

Macro alarms

When an error occurs during use of the tool setting software, an alarm number or message is generated. This may be displayed on the screen of the controller. This chapter describes the meaning and likely cause of each alarm message that may be displayed on the screen of the controller. It then describes typical actions that you should take to clear the fault.

Contained in this chapter

Fanuc 0M controller alarms.....	7-2
Alarms	7-2

Fanuc 0M controller alarms

Alarm messages are not displayed on the screen, just the alarm number. The alarm numbers displayed are:

(500 + n) where n is the alarm number

Example: 92(PROBE OPEN) is: alarm 592

Alarms

Format

#500 =

82(TOOL OUT OF RANGE) – Macro O9853 only

Cause

This alarm occurs if the cutter size exceeds the maximum cutter diameter size that has been set in variable #528.

Action

Edit the program.

Format

#500 =

91(FORMAT ERROR)

Cause

The Kk and Hh inputs are mixed or the Ss input is missing.

Action

Edit the program.

This is a reset condition.

Edit and start again from a safe position.

Format

#500 =

92(PROBE OPEN)

Cause

This alarm occurs if the probe is already triggered before a move.

Action

Clear the fault.

This is a reset condition. Clear the fault and start again from a safe start position. The stylus may be in contact with a surface, or the probe has failed to re-seat. This could be due to swarf trapped around the probe eyelid.

Format

#500 =

93(PROBE FAIL)

Cause

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

Action

Edit the program.

This is a reset condition. Edit the program and start again from a safe start position. The surface was not found or the probe has failed.

Format

#500 =

99(BROKEN TOOL)

Cause

This alarm occurs if a tool is out of tolerance and the Mm input is not used.

Action

Replace the defective tool and establish the correct tool offset value.

This page is intentionally left blank

Appendix A

Haas tool setting

This appendix describes the important differences to the software when used in a Haas controller.

The software is based on Renishaw standard Fanuc software disk A-4012-0584.

Contained in this appendix

Introduction	A-2
Changes and differences	A-2
Macro variables.....	A-3
Automatically set variables.....	A-3
Manually set variables	A-4
Preparing a customised installation and calibration program.....	A-6
Sample program.....	A-7

Introduction

Information in this section is specific to the Haas controller and should be used in preference to the general information given in other chapters of this manual.

Changes and differences

This section describes the differences between the tool setting software for the Haas controller and the same software intended for standard Fanuc-based controllers.

- An additional program O9854 is provided which gives a 0.16 second dwell.
- The option G103P1 (two line buffering) is applied in all tool setting programs. It must be cancelled afterwards in the part program (using G103P0) if cutter compensation is to be used.
- It is possible to address either diameter or radius tool offsets by the use of variable #100 in program O9799. This is shown in the section titled “sample program” later in this appendix.
- The software base number for storing set-up and calibration data has been changed to 550 (#550) – previously 520 (#520). This makes the package compatible with the Renishaw vector software.
- M-codes for switching between tool setting and inspection probes. The Haas control requires either M52 or M62 to be active to allow the function of a skip signal into the control. No wiring is required.
 - M-code M52/M62 drives the relay output labelled M22.
 - M62 activates the inspection probe (it is embedded in the inspection software).
 - M52 activates the tool setting probe (it is embedded in the tool setting software, program O9799).
 - Back-off distance #506 is set = .1 as the default value.

Macro variables

Automatically set variables

The following variables are set automatically during complete calibration. It is not necessary for you to pre-set the values.

NOTE: Variables marked * are distances in the machine co-ordinate system, and not the program co-ordinate system.

Setting variable		Internal variable
#550 (550 + 0)	Z calibration value (Non-rotating tools)	#107
#551 (550 + 1)	Z calibration value (Rotating tools)	#113
#552 (550 + 2)	Stylus size for diameter setting	#110
#553 (550 + 3)	X axis stylus centre position (machine co-ordinate)	#111
#554 (550 + 4)	Y axis stylus centre position (machine co-ordinate)	#112

Manually set variables

All of the following variables **MUST** be set before the cycles are used.

Setting variable		Internal variable
#555 (550 + 5)	Z approach position (used in cycle O9853 only). The first fast positioning move from the Z home position where the tool offset is applied (height above stylus).	#115
#556 (550 + 6)	Z clearance position (used in cycle O9853 only). The position above the stylus for clearance moves around the stylus (height above stylus).	#116
#557 (550 + 7)	Tools above this diameter rotate (used in cycle O9853 only)	#117
#558 (550 + 8)	Maximum cutter diameter size	#121
#559 (550 + 9)	Tool offset type (see the section titled “tool offsets” in chapter 3, “macro variables” for the set value) e.g. set = 13. (C type)	#108

Setting variable	Internal variable	
#560 (550 + 10)	Probe orientation. It is necessary to define the diameter measuring axis and the radius offsetting direction for rotating length setting as follows:	#120
	set = 1. Diameter setting:	Along the X axis.
	Rotating length setting:	Cutter radius offset in the Y– direction.
	set = –1. Diameter setting:	Along the X axis.
	Rotating length setting:	Cutter radius offset in the Y+ direction.
	set = 2. Diameter setting:	Along the Y axis.
	Rotating length setting:	Cutter radius offset in the X– direction.
	set = –2. Diameter setting:	Along the Y axis.
	Rotating length setting:	Cutter radius offset in the X+ direction.
#561 (550 + 11)	Flag for inch or metric setting data	
	set = 0. Storing metric data in variables.	
	set = 1. Storing inch data in variables.	
	This flag must be set to represent the units entered in the previous setting variables.	

Preparing a customised installation and calibration program

It is possible to prepare a customised program for installing set-up data and calibrating the probe system. This is useful when the software is to be installed on several similar machines fitted with Haas controllers.

Manually position the reference tool to approximately 10 mm (0.4 in) above the stylus and approximately on the centre-line of the stylus.

When the custom program is run, all macro variable data is set and the probe is fully calibrated.

EXAMPLE 3

Showing a typical program to set up and calibrate on a round stylus

- Tool setting in the Y axis direction.
- Rotating length setting by radius offsetting in the X+ direction.

The reference tool makes the following measurements:

1. Z measure on top of the stylus (three touches).
2. X diameter measure on both sides of the stylus (the direction depends on the #560 setting).
3. Y diameter measure on both sides of the stylus (the direction depends on the #560 setting).
4. Return to 10mm (0.4in) above the stylus and on-centre.


```

%
O8000 (CUSTOM CALIBRATION)
#506= 0.1 (SURFACE BACK-OFF DISTANCE)
#555= 100. (RAPID APPROACH POS. IN Z)
#556= 10. (CLEARANCE POS. IN Z)
#557= 10. (TOOLS ABOVE THIS ROTATE)
#558= 89. (MAX. CUTTER DIAMETER)
#559= 13. (WORK OFFSET TYPE)
#561= 0 (INCH/METRIC STORED DATA)

(LENGTH CALIB)
G65P9851K95.03

(X DIAM CAL)
#560= 1. (X-AXIS SELECT)
G65P9852S10.0K12.7Z-15.(S-TOOL DIA K-STYL SIZE)

(Y DIAM CAL)
#560= -2. (Y-AXIS SELECT)
G65P9852S10.0K12.7Z-15.(S-TOOL DIA K-STYL SIZE)
M30
%
```

Sample program

The following setting program shows the differences from the standard Fanuc settings.

O9799(RENISHAW SETTING)	
(40120634.0A)	
G103P1	Block buffering two blocks.
M98P9854	0.16 second delay macro.
#30=550(EDIT BASE NO.)	Start variable #550
#100=2(2=RADOFF, 1=DIAOFF)	Select radius/diameter offset
(#[#30+11]=0)(1-INCH 0-MET STOREFLAG)	
G90G80G40G00	
M52(TOOLPROBE ON)	Switch on tool setting probe.
(CURRENT METRIC)	
#29=1	

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



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