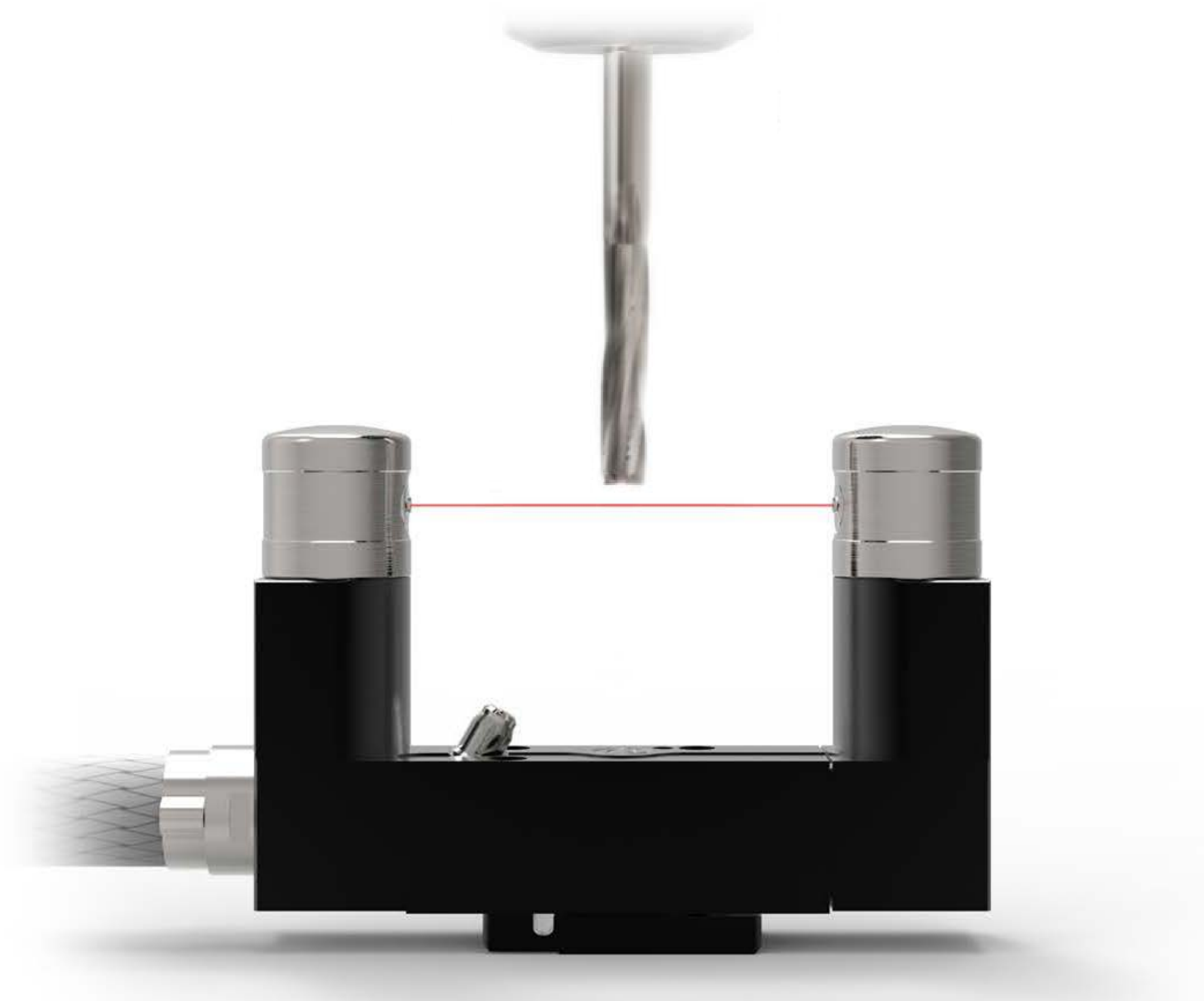
Programming guide
H-2000-6333-0M-B

RENISHAW
apply innovation™

Non-contact tool setting software



Disclaimer

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

Trade marks

RENISHAW and the probe symbol used in the RENISHAW logo are registered trade marks of Renishaw plc in the United Kingdom and other countries. **apply innovation** and names and designations of other Renishaw products and technologies 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.

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

Renishaw apps are available in China via Baidu, Huawei and Tencent.

Renishaw part no: H-2000-6333-0M-B

Issued: 04.2020

Caution – laser safety

Renishaw non-contact tool setting (NCTS) systems emit class 2 laser radiation in the wavelength range 650 nm to 670 nm and have maximum power outputs of less than 1 mW.

Refer to the installation and user guide supplied with the product for the detailed laser safety warnings.

Caution – software safety

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

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

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

IMPORTANT: This software makes use of controller variables in its operation. During its execution, adjustment of these variables, including those listed within this manual, or of tool offsets and work offsets, may lead to malfunction.

Caution – programming manual compatibility

Two measurement methods are available with the NCTS system: Tool Set Mode 1 (TSM1), where the tool is measured as it enters the beam (light-to-dark transition), and Tool Set Mode 2 (TSM2), where the tool is measured as it enters and exits the beam (dark-to-light transition). During system installation, the appropriate method must be chosen. The decision is typically based on M-code availability and measurement conditions (for example, if wet, TSM2 is recommended).

NOTE 1: It is not possible to switch between TSM1 and TSM2 once the system is installed.

NOTE 2: This programming guide covers both modes:

Sections that apply only to Tool Set Mode 1 are marked **TSM1**
Sections that apply only to Tool Set Mode 2 are marked **TSM2**

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.

List of associated publications

When you use the beam alignment cycle (macro O9860), you will also need to refer to the Renishaw publication listed below that is appropriate to the type of NCTS unit to be fitted to your machine. These publications contain instructions that describe how to physically align the beam at the NC transmitter unit.

- *NC4 non-contact tool setting system* installation and maintenance guide (Renishaw part no. H-2000-5230).
- *NC4 non-contact tool setting system (integral air blast)* installation and maintenance guide (Renishaw part no. H-6270-8501).
- *NCi-6 non-contact tool setting interface* installation and user’s guide (Renishaw part no. H-6516-8500).

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 P9862 A1. B1. H0.5 J0.25 M1. Q5. S2500. T20. W5. Y3.

may be entered as:

G65P9862A1.B1.H0.5J0.25M1.Q5.S2500.T20.W5.Y3.

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.

New features

The following new features are provided for both Tool Set Mode 1 and Tool Set Mode 2, unless otherwise stated.

- Calibration data is stored in metric units and converted to imperial units as required when the cycles are run.
- Dedicated base number for partition calibration data (see Appendix B, “Using two measuring positions on the laser beam”).
- Revised long tool/short tool search and sampling method in O9762.
- Selectable air blast duration.
- Revised broken tool detection cycle O9863 improves performance in wet conditions.
- H and E inputs in tool measurement cycle O9862 increase functionality.
- Cutting edge check: B4. option – edge check only; no prior diameter measurement moves.
- Advanced tool cleaning feature.
- Ability to output measured data to the Reporter app (v3.0 or later).

Tool Set Mode 1-related features

- New O9867 cycle with vector moves for improved accuracy and speed.
- New O9868 large tool cycle used for measuring the effective length and radius or diameter of a tool when its diameter is larger than the gap between the NC heads.

Tool Set Mode 2-related features

- Auto-pulse width selection, ensuring the appropriate pulse width is chosen, based on measurement r/min. This is dependent on M-code availability.
- Tool run-out measurement feature of cutting edge checking cycle O9862.
- RPM checking and drip rejection calculations not required.
- New O9867 cycle with vector moves for improved accuracy and speed.
- Known tool length approach: alongside the existing long tool/short tool search method, a rapid approach option is available which uses the existing tool offset value. This is typically used where the machining/measuring process is fixed and conditions are very wet.
- Automatic compatibility when applying the tool offset with parameter settings P5006 and P6006 or P6019.

- New O9762 macro with start and end point options for ease of use.
- Revised scanning cycle O9766, with option for coarse/fine scan or coarse scan only.
- Automatic optimisation of the back-in factor (BIN) in O9762, if selected in the installation wizard.

Reporter

There is a Reporter option in the installation wizard which can be used to display trends of tool measurement. (Reporter app v3.0 or later is required.)

Reporter option

Reporter Application Print option

156 Reporter Part ID variable

157 Reporter Protocol variable

158 Reporter Data variable

This option requires the Reporter app (A-5999-4200) to be installed and connected to the machine tool to receive measured data. If the option is selected and the Reporter app is not connected, the measuring program will continue to run.

Reporter Part ID variable

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

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

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

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

The Part ID is set in the program using the Part ID variable. In the example on page ix a Part ID of 2000 is set to match the program number.

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

Reporter Protocol variable and Reporter Data variable

Protocol variable

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

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

Data variable

The data variable is set in the Reporter app configuration settings and is used to specify the base number for a range of 29 sequential machine variables required to hold data.

For example, enter the value 158 to use machine variable range #158 to #186 (#158 + 28 variables).

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

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

On machine programming

Once tool setting macros have been installed and configured on the CNC, programs can be created to measure tools and the measuring results can be viewed in Reporter.

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

Data Send start and end

Reporting is enabled and disabled using the Data Send start and end macro. The command line should be written as shown below. The example assumes #156 is used for the Part ID.

G65 P9735 A1. B1. C0. I#156

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

G65 P9735 A1. B2. C0. I#156

Description of Data Send inputs

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

U input for measured feature macros

The U input acts as the tool ID number. The tool number that is being measured must be added to the tool setting macro line.

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

Example: Tool length/radius setting with reporting

Assume the tool is a 10 mm (0.394 in) diameter slot drill.

O2000	
M6 T1	
H1	Controller-specific (see the appropriate appendix).
#156=2000	Part ID set to 2000.
G65 P9735 A1. B1. C0. I#156	Report start using #156.
G65 P9862 B3. D21. U1.1	Set the tool length offset (1) and radius offset (21). Report on tool 1.
G65 P9735 A1. B2. C0. I#156	Report end using #156.
M30	

NC4 support

For NC4 customer support, please visit **www.renishaw.com/ncsupport**.

Machine tool apps

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

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

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

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

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

Contents

Renishaw non-contact tool setting system

This guide describes how to use the Renishaw non-contact tool setting (NCTS) system software.

The Renishaw NCTS system comprises laser-based non-contact tool setting units that provide high-speed/high-precision measurement of cutting tools on a machining centre under normal operating conditions.

As a tool is moved through the laser beam, the system detects when the beam is broken. Output signals sent to the controller allow the presence of a tool and the position of the tip, a tooth, or a cutting edge to be established.

The NCTS system allows the following parameters to be established:

- Length and diameter of the cutting tool. For details of the minimum size of tool that can be measured accurately, refer either to the “List of associated publications” on page ii or to the appropriate Renishaw product data sheet.
- Detection of a broken tool.
- Detection of a broken tip or cutting edge, or excessive run-out of a tool.
- Measurement of a corner radius.
- Cutter radius and linear profile checking.
- Oversize tool measurement.
- Compensation for thermal changes in the machine tool.

Features of the NCTS software

The NCTS software provides the following measuring and calibration features:

Measuring macro features

Six measuring macros provide the following features:

- Macro O9862: used for measuring the length and diameter of the cutting tool and for cutting edge checking. Edge checking uses the ‘Latch mode’ feature on the NCi interface.
- Macro O9863: used for broken tool detection by plunge measurement.
- Macro O9865: used for checking the radii and linear profile of the cutting tool. It uses the ‘Latch mode’ feature on the NCi interface.
- Macro O9866: used for broken tool detection of solid tools (drills/taps). It uses the ‘Tool Break Mode’ feature on the NCi interface.
- Macro O9867: used to measure the radius of ball nose or corner radius cutters.
- Macro O9868: used to measure length and diameter of large cutters.

Calibration macro features

Two calibration macros provide the following features:

- Macro O9860: used for aligning the laser beam, setting the provisional positions of the beam in the spindle axis and radial measuring axis and setting the measuring position along the beam. There is also an option to check the alignment of the spindle axis using spindle indexing with an offset pin tool.
- Macro O9861: used for calibrating the positions of the laser beam in the spindle axis and radial measuring axis, calculating the calibration difference for auto-pulse width selection if available, optimisation of the back-in factor (BIN) if selected with TSM2 and for temperature compensation of the spindle axis and radial measuring 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 O9743: used for positioning the tool over the air nozzle.
- Macro O9746: used for vector moves.
- Macro O9747: used for the vector measuring routine.
- Macro O9760: used for the settings data.

- Macro O9761: used for start-up functions.
- Macro O9762: used for the measuring routine.
- Macro O9763: used for the G31 routine.
- Macro O9764: used for the G0 / G1 / G31 routine.
- Macro O9765: used for the G2 / G3 / G28 routine.
- Macro O9766: used for radial scanning.
- Macro O9767: reserved for the angled beam software package only.
- Macro O9768: used for checking the active spindle speed. TSM1
- Macro O9769: used for error messages.
- Macro O9770: used for the Reporter app.

Software memory requirements

The NCTS system software requires approximately 31 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 O9860 (laser beam alignment routine): approximately 2 KB of memory.
- Macro O9861 (laser beam calibration routine): approximately 4 KB of memory.
- Macro O9862 (tool setting routine): approximately 6 KB of memory.
- Macro O9863 (broken tool detection – plunge check): approximately 2 KB of memory.
- Macro O9865 (checking cutter radii and linear profiles): approximately 4 KB of memory.
- Macro O9866 (broken tool detection for solid tools): approximately 2 KB of memory.
- Macro O9867 (corner measurement cycle): approximately 3 KB of memory.
- Macro O9868 (large tool measurement cycle) approximately 4 KB of memory.

Angled beam software package

For information about the angled beam software, see Appendix A, “Angled beam software package”. In general, the cycles supplied with the angled beam software package are similar to those supplied with the standard software, which are described in detail in this publication. However, the following major differences should be noted.

- Beam alignment cycle (macro O9860). An additional input (Aa) is used with this cycle to define the initial approximate angle of the laser beam. The cycle determines an accurate angle and then writes the result to a calibration register that is subsequently used by the other cycles.

- Cutter radius and linear profile cycle (macro O9865). In the angled beam software, profile checking of a corner radius is disabled and cannot be used. Linear profile checking is still possible.

Drip rejection feature TSM1

The NCTS software is compatible with the drip rejection feature of the NCTS system hardware. The drip rejection option provides protection in conditions where there is the possibility of coolant dripping through the laser beam, as this can affect measurement and generally interfere with the operation of the cycles. To use this feature, both the NCTS interface and the software must be set to the drip rejection mode of operation.

The software as supplied is set for use without the drip rejection feature. This state is suitable when the drip rejection feature of the NCTS interface is switched off.

When the software is set to drip rejection mode, the spindle speed is automatically adjusted to the nearest multiple of 500 r/min or 1000 r/min, depending on the setting of #139 in macro O9760.

Macro O9768 is used for calibrating and checking the spindle speed when M-codes are not available to lock the spindle speed to 100%.

Machine spindle speed checking TSM1

A spindle speed (RPM) checking macro (O9768) is used to check the speed of the machine spindle when the drip rejection feature is being used.

There is provision within the macro code to compensate for an incorrect setting of the spindle speed override control within the range of approximately ±50%. This compensation may not work on all machines, so it is best to ensure that the spindle speed override is always set to 100%.

If you notice that the override is not set to 100% whilst a cycle is running, do not change the setting until the cycle has finished.

If the spindle is not running at the required programmed speed, then an RPM OUT OF RANGE error will be generated. When this happens, press program reset, check that the spindle speed override is set to 100%, then restart the cycle.

Auto-pulse width selection TSM2

A pulse width of 20 milliseconds (ms) should be used to achieve optimum performance in very wet conditions, however the minimum measuring spindle speed is then limited to 3000 r/min. The alternative 100 ms pulse width is suitable for normal conditions and has a larger r/min range, as the minimum measuring spindle speed is 600 r/min.

If M-code 3 is connected on the NCi-6 interface, measuring cycles will automatically select the appropriate pulse width based on the measurement spindle speed. Tools that run below 3000 r/min will use a 100 ms pulse width; tools running at 3000 r/min or above will use a 20 ms pulse width.

Optimisation TSM2

Customers who wish to refine the back-in distance (BIN), in order to minimise the tool setting cycle time, can use the automatic optimisation feature while calibrating. This can be selected in the installation wizard and will require three extra macro variables outside of the normal NCTS macro variable range. By default, these variables are #900, #901 and #902. However, they can be changed by use of the installation wizard when compiling the macro software.

If calibration is run a second time, the software will recognise it is optimised and not run optimisation again unless forced to do so with the U1. input.

Air blast option

An air blast option is included with the NCTS software to support NC4 units which have an air blast nozzle fitted. Typically, this is used when coolant is present during the measurement process to remove the coolant from the tool. There are options in the setting data macro (O9760) for selecting continuous air blast or selected air blast.

An advanced tool cleaning option (see Figure 1) is also supplied and can be selected in the installation wizard during software commissioning. This feature enables the tool to move close to the air nozzle and repeatably move across the air stream, offering improved cleaning for very wet and dirty environments. This option is available for the tool setting cycles O9862 and corner radius measuring cycle O9867.

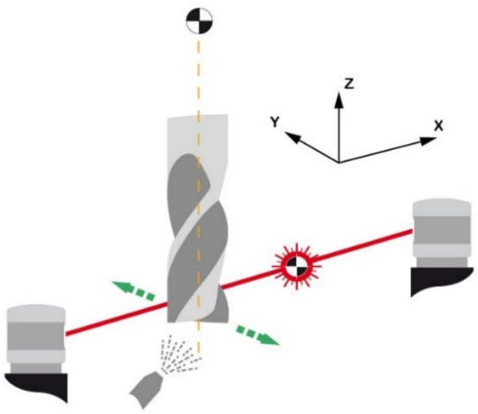


Figure 1 Advanced tool cleaning option

The span across the airstream and the numbers of passes can be adjusted at the top of macro O9743 by changing the following lines:

#2 = **4** *#129 (SPAN*ADJUSTMENT MM)	The value 4 (mm) can be increased or decreased. The total span is the tool diameter + the adjustment value.
#16 = **5** (NO. OF PASSES)	The value 5 can be increased or decreased.

The macro also controls what tool sizes are suitable for cleaning. In the example below, 20 mm (0.787 in) is the maximum tool diameter allowed. This value can be edited to suit different tooling suites.

#30 = **20** * #129 (MAX*DIA*MM)	The value 20 (mm) can be increased or decreased.

The initial positioning above the air nozzle can be adjusted by changing the XYZ positions on the O9743 macro call-up line. These are incremental distances from the measuring position on the beam.

Example:

G65 P9743 F#12 Q#17 R#18 V#29 W#14 **X−10. Y15. Z−8.**

NOTE: Above is an example of the O9743 macro call-up line. This will be slightly different between TSM1 and TSM2, but it is only the XYZ values that require changing.

The software also has an improved measurement cycle that forces the re-measurement of a tool rather than issuing a coolant-related alarm and stopping the process. This feature is included irrespective of whether or not the air blast option is used.

Machine tool controllers supported

For details of the machine tool controllers supported by the NCTS system software, see the appendices at the back of this guide.

Measurement values used in this guide

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

Editing the read-ahead G-code (G53)

This G-code has a role in controlling read ahead. Fast machining or smoothing control options can cause block read-ahead problems when running a cycle.

Read-ahead control, in the form of G53, is called at strategic positions within the cycles. If further code is required to suppress read ahead, add it to the relevant places within the macro code. Adjustment is not normally required, but customisation is possible. Please consult a Renishaw representative for advice if changes are being considered.

Examples of changes

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

Tool offset types supported

Positive tool offset applications

The NCTS 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. These applications are described below.

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.

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

The longest tool for the machine is 200 mm long.

The shortest tool for the machine is 50 mm long.

In the setting data macro (O9760), 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 (but this is not important).

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

The longest tool for the machine is 200 mm long.

The shortest tool for the machine is 50 mm long.

In the setting data macro (O9760), variables #110 and #111 must be set as follows:

#110 = 50.0	Maximum length tool.
#111 = −100.0	Minimum length tool.

Machine axis definitions

The NCTS software is suitable for orthogonal X, Y and Z-axis machines. However, it can also be configured for different orientations of the NCTS hardware on the machine.

Throughout this guide the following axis notations (the standard configuration) have been used:

- Spindle axis (Sp) = Z axis (see the note below).
- Radial measuring axis (Ra) = Y axis.
- Laser beam axis (La) = X axis.

All figures in this guide show this standard configuration.

NOTE: Although the cycles allow the spindle axis to be other than the normal Z axis of the machine, in practice other factors may need to be considered. Usually some additional bespoke changes will be required to complete the installation.

Consider the following points:

- Where is the G28 zero return position of the machine?
- Are there any special M-codes or other preparation details necessary that need to be embedded in the cycles?
- Do any 4th axis positioning issues apply?

Installing the NCTS software

Before installing the NCTS system software, read the guidelines contained in the Readme file on the software media.

Manually loading the software

Renishaw NCTS software is supplied with an installation wizard. In the unlikely event that the wizard cannot be used, you will find all macros stored on the software media in the Macro_A and Macro_P folders.

- Macro_A: macros for angled beam installations.
- Macro_P: macros for parallel beam applications.

Setting data macro O9760 must be edited manually and the appropriate activate and deactivate M-codes must be inserted into the code in all relevant positions.

Example:

#30=#131+#6	
N33	
G65 P9764 Y#30 F#127	
(*LATCH*ON*)	Replace with the appropriate activate M-code.
G53	
G4 X.1	
G65 P9764 Z#28	
(*LATCH*OFF*)	Replace with the appropriate deactivate M-code.
G53	
G4 X.15	

Macro variables

The following variables are used by the NCTS system software:

- #500-series macro variables are used for the calibration data and setting data.
- #100-series macro variables are used for the setting data.
- Macro variables #1 to #32 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.

For details of suitable base number settings, refer to the appropriate appendix at the end of this guide. If the default number is not suitable and needs to be changed, change it as described in “Variables: changing the base number address” on page 21.

Calibration data macro variables

The following variables are set automatically during the calibration cycles:

#520 (520 + 0)	The spindle axis (Sp) position of the beam when measured from the positive side (top) of the beam.
#521 (520 + 1)	The spindle axis (Sp) position of the beam when measured from the negative side (bottom) of the beam.
#522 (520 + 2)	The radial measuring axis (Ra) position of the beam when measured from the positive side of the beam.
#523 (520 + 3)	The radial measuring axis (Ra) position of the beam when measured from the negative side of the beam.
#524 (520 + 4)	The position along the beam axis (La) at which measurements are made.
#525 (520 + 5)	The active dynamic zone value.
#526 (520 + 6)	The spindle (length measuring) axis temperature compensation work offset.
#527 (520 + 7)	The radial measuring axis temperature compensation work offset.
#528 (520 + 8)	The dynamic zone value for 20 ms pulse width. **TSM2**
#529 (520 + 9)	The dynamic zone value for 100 ms pulse width. **TSM2**
#531 (520 + 11)	This is used only with angled beam installations. The angle alignment calibration value for the beam is stored in this variable. The value is used by all other cycles for angular orientation relative to the laser beam (see “Variables: changing the base number address” on page 21).

Compliance with Fanuc parameter P5006.6, P6019.4 and P6006.4 settings [TSM2]

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

Setting data macro variables – O9760

NOTE: The NCTS software should be prepared and installed using the installation wizard. It can subsequently be edited by referring to the descriptions in this manual.

Read the following variable descriptions then edit macro O9760 to suit the requirements of your machine and controller.

#101 A value entered here overrides the normal #113 setting for large tools. A tool having a diameter greater than the specified diameter is set on one side of the beam only.

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

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

Default: 0 (denotes that this option is not used. All tools are set as defined by the #113 value).

#107 Coolant droplets can cause the NC4 to unintentionally trigger. Robustness against false triggers can be improved by increasing this value.[TSM1]

Default: 1

#108 The tool offset type (1 = A, 2 = B, or 3 = C etc.).

For information about setting the tool offset type for the controller you are using, see the appropriate appendix at the back of this guide.

Default: 1

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

1 = Radius.
2 = Diameter.

Default: 1

CAUTION: Variables #110 and #111. Before running any cycle, you must enter valid data that is relevant to the machine tool into variables #110 and #111 (the maximum and minimum tool lengths). There is a danger that the tool will collide with the NCTS unit if these values are incorrect.

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

#111 The minimum length of the tool. This defines the lowest measuring height of the spindle nose above the laser beam.

#112 The maximum diameter of the tool. This value is dependent on the machine tool.

#113 The radial measuring axis options.

1 = measure from the positive side of the beam.
−1 = measure from the negative side of the beam.
2 = measure from both sides of the beam.

Default: 2

#114 The radial calibration options.

1 = measure from the positive side of the beam.
−1 = measure from the negative side of the beam.
2 = measure from both sides of the beam.

Default: 2

#115 NCi-5 / NCi-6 pulse width.

The time the system remains triggered after the laser becomes blocked. This setting ensures that tool edges breaking the beam produces a constant triggered state.

The pulse width is also used during the dark-to-light transition, ensuring the laser is completely unblocked before the system switches to the triggered state. Refer to #138 later in this section if you are installing an NCi-6 unit and using the auto-pulse width select option. **TSM2**

Default: 100 ms

#118 The distance the tool will move in one rotation during the final 'scatter' measures. Increasing the value will reduce measurement cycle time, but create more uncertainty in the measured result.

Default: 0.002 mm (0.0001 in)

#119 The default spindle speed.

Measurement cycles are optimised for a spindle speed of 3000 r/min.

Some tools – for example, those that are unbalanced or large – must be run at speeds less than 3000 r/min. This is the responsibility of the user. Use the S input to set the speed.

Measurement cycle times increase with slower speeds. The minimum speed is 600 r/min depending on the setting of #115 and/or #138. TSM2

Default: 3000 r/min

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

For a description of how to change this number, see "Variables: changing the base number address" on page 21.

Default: #520

#121 The NCTS unit beam axis.

Axis movement along the beam can be inhibited by entering a negative value. This is suitable for applications where it is not possible for the spindle to move along the beam axis.

- If the beam is parallel to the X axis, select 1 (or −1 = locked).
- If the beam is parallel to the Y axis, select 2 (or −2 = locked).
- If the beam is parallel to the Z axis, select 3 (or −3 = locked).

Default: 1

#122 The axis used for radial measurement.

- If the X axis is to be used for radial measurement, select 1.
- If the Y axis is to be used for radial measurement, select 2.
- If the Z axis is to be used for radial measurement, select 3.

Default: 2

#123 The axis used for length measurement (the spindle axis).

- If the X axis is to be used for length measurement, select 1.
- If the Y axis is to be used for length measurement, select 2.
- If the Z axis is to be used for length measurement, select 3.

Default: 3

NOTE: #123 must always define the spindle axis and the direction in which the tool offset is applied. If the spindle is in the negative direction, the value entered must also be negative (−1 = −X axis, −2 = −Y axis, −3 = −Z axis).

#124 The scatter tolerance value.

For a description of this feature, see the figure in “Scatter tolerance checking” on page 18.

Default: 0.005 mm (0.0002 in)

#125 The defines the air blast duration.

0 = Off, no air blast.

1 = Air blast for first two touches; no air blast for scatter measures.

2 = Air blast for complete cycle.

Default: 0.

NOTE: Measurement accuracy can be influenced by the air blast. Option 1 is recommended for typical applications.

#126 The sample size for scatter.

This defines the number of measurement samples to be taken. It must be equal to or less than 5. The number of retry attempts is three times this value.

For a description of this feature, see the figure in “Scatter tolerance checking” on page 18.

Default: 2. TSM2
3. TSM1

#127 The feedrate used for rapid traverse.

Default: 6000 mm/min

#128 The long tool/short tool approach feedrate.

Defines the feedrate for initial long tool/short tool approach move.

Default: 3000 mm/min

#138 The value to reflect the machine parameter P5006.6 and P6006.4/P6019.4, selectable in the wizard. TSM1

#138 Auto-pulse width selection available. Select yes if the auto pulse width is installed and operational. TSM2

0 = No.
1 = Yes.

If #138 = 1 and M-code 3 is ‘active high’, set NCi-6 SW2-4 = OFF
If #138 = 1 and M-code 3 is ‘active low’, set NCi-6 SW2-4 = ON

Default: 0.

#139 Drip rejection and associated spindle speed. [TSM1]

0 = Drip rejection off.

500 = Measurement spindle speed will be rounded to the nearest 500 r/min.

1000 = Measurement spindle speed will be rounded to the nearest 1000 r/min.

This setting must correspond with switches SW1-3 and SW1-4 on the NCi-6 interface unit.

Default: 0.

#141 Approach method. [TSM2]

0 = Long/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, #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 in the beam.

Default: 0.

NOTE: Known tool length approach mode reduces measurement cycle time and is very robust in wet conditions, however, a collision risk exists if the tool offset value is incorrect.

#142 Enter the distance between laser heads of a fixed NC unit. This is only required if you intend running the off-centre measurement cycle (O9868).

Default: vacant (#0).

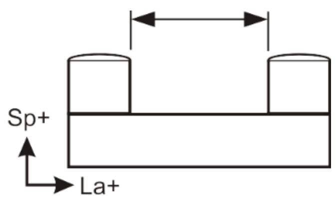


Figure 2 Macro variable #142

#143 Safe distance below laser.

This is the clearance distance from the laser beam to the NC4 housing.

Default: 27 mm (1.063 in).

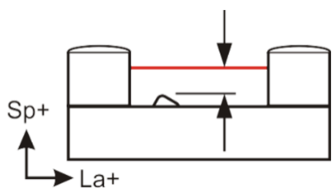


Figure 3 Macro variable #143

#145 The static position zone.

The zone value is used to check the validity of a measurement move. On older controllers this setting may need to be increased. Contact your local Renishaw office for further advice.

If optimisation is selected within the installation wizard then, depending on machine condition, it is recommended that this value is reduced to 0.005 mm (0.0002 in). TSM2

Default: 0.010 mm (0.0004 in).

Editing the setting data macro – O9760

Before running any of the cycles, edit the settings data between block numbers N1 and N2 to suit the application and set-up. This is only required if you do not intend to use the wizard to compile the software.

All calibration data and data at the top of the setting macro O9760 will be written and stored in metric units and converted automatically when the machine is set to run in imperial units.

Next, enter data in the other variables using the same units.

Sample of macro O9760

```
N1
#101 = 80 (ABOVE SET ON ONE SIDE)                                (3.15 in)
#107 = 1 (MAX RETRIES) TSM1
#108 = 1 (TOOL OFFSET TYPE)
#109 = 1 (OFFSET-RADIUS 1/DIAMETER 2)
#110 = 200 (MAX TOOL LENGTH)                                     (8.0 in)
#111 = 70 (MIN TOOL LENGTH)                                      (2.75 in)
#112 = 80 (MAX CUTTER DIAMETER)                                  (3.15 in)
#113 = 2 (TL SET RADIUS MEAS DIR)
#114 = 2 (CALIB RADIUS MEAS DIR)
#115 = 100 (NCi-6 PULSE WIDTH 100 OR 20)
#118 = 0.002 (MEASURE RESOLUTION)                                (0.0001 in)
#119 = 3000 (DEFAULT RPM)
#120 = 520 (BASE NUMBER)
#121 = 0 (BEAM AXIS)
#122 = 2 (RADIAL-MEASURE AXIS)
#123 = 3 (SPINDLE AXIS)
#124 = 0.005(SCATTER TOL)                                        (0.0002 in)
#125 = 0 (AIR BLAST 0=OFF 1=PARTIAL 2=FULL)
#126 = 2 (SAMPLE SCATTER SIZE)
#127 = 6000 (RAPID TRAVERSE)                                     (236.22 in)
#128 = 3000 (LT/ST APPROACH FEED)                                (118.11 in)
#138 = 0 (P5006.6 SETTING) TSM1
#138 = 0 (AUTO PW SELECT AVAILABLE 0=NO 1=YES) TSM2
#139 = 0 (NCi-6 DRIP REJECT 0=OFF/500/1000 RPM) TSM1
#141 = 0 (APPROACH METHOD 0=LT/ST SEARCH,1=KNOWN TL LEN) TSM2
#142 = #0 (DIST BETWEEN NC HEADS FIXED UNIT ONLY)
#143 = 27 (SAFE DISTANCE BELOW LASER)
#145 = 0.010(ZONE CHK)                                           (0.0004 in)
#28 = 0 (M-CODE SP 100 PERCENT 0=NO 1=YES TSM1
(#[#120+10] RPM CHECK) TSM1
(#[#120+11] ANGLE CALIB.)
N2
```

Drip rejection mode settings TSM1

The software supplied can be configured for use with or without drip rejection mode. Set variable #139 as shown:

#139 = 0	Drip rejection option off.
OR	
#139 = 500 OR #139 = 1000	When using drip rejection mode, enter the r/min setting value to match that of the interface unit (currently = 500 or = 1000).

NOTES:

This software package uses an installation wizard to generate the macro code that will be loaded into the machine controller. Variables #[#120 + 8], #[#120 + 9] and #28 are flags that instruct the software if M-codes are available. These variables and the activate and de-activate M-codes are automatically inserted into the generated code.

There is no requirement for an air blast flag. The activate and de-activate M-codes are automatically inserted into the generated code.

CAUTION: When drip rejection mode is used (#139 = 500 or #139 = 1000), the spindle must rotate at the correct spindle speed, otherwise the rotating cutting teeth may not be detected by the NCTS system.

If M-codes are available to lock the spindle speed override to 100%, #28 will be set to 1 after running the installation wizard and M-codes will be inserted in the relevant positions in the macro code.

If M-codes are not available, leave #28 set to 0 so that the default setting will automatically call the RPM checking macro when the cycles are run to verify that the spindle speed is correct (see “Calibrating the spindle speed – O9768 TSM1” on page 26).

NOTE: In the installation wizard, it is possible to avoid the use of O9768 by inputting ‘9’ (M9 coolant off) into the M-codes for setting the spindle speed override to 100%. However, this is not a recommended practice, as it entirely relies on users to check the feedrate overrides are at 100% every time the cycles are run, instead of an automated spindle speed check.

Scatter tolerance checking

The sample size value, defined by #126, must be equal to or less than 5.

In the following example:

- The sample size is 3.
- The maximum number of retries is 9, as it is automatically set to be three times the sample size value.

Sample measurements are taken until either the maximum retries limit is reached, causing an alarm, or a sample of measurements is found to be within limits. In this latter case, the average value is found and measurement is complete.

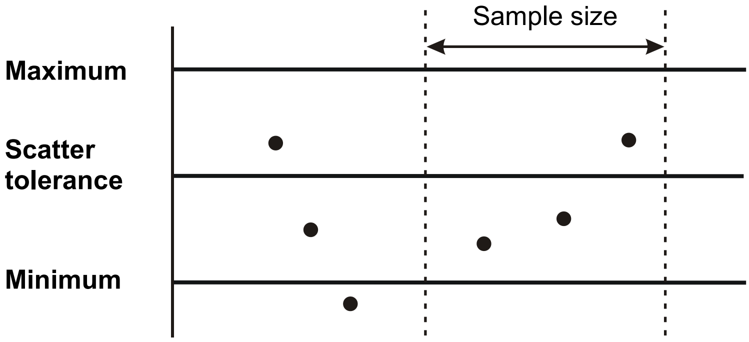


Figure 4 Scatter tolerance checking

Common variables

The following variables are loaded automatically each time a cycle is run:

#116	Active tool length.
#130 to #147	Used for internal calculations.
#148	Tolerance flag output. 0 = In tolerance. 1 = Out of tolerance.
#149	Used for internal calculations.

Variables: changing the base number address

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 setting data macro (O9760) will change the variable range.

Customising the macros

In addition to the information described below, further customisation and installation information is included in the Readme file supplied with the NCTS software.

Editing the beam alignment macro when using an offset pin – O9860

Editing the beam alignment macro to use an offset pin is necessary only if you intend using this method to align the laser beam.

As standard the macro has embedded M0 program stop commands for manual spindle positioning. This can be automated, provided the machine has suitable M-codes or other programmable code for this purpose.

Make edits in the positions described below:

O9860(REN BEAM ALIGN)

( )	
M0	Edit to a suitable M-code (for example, M19).
(ROTATE TO P1 −VE)	
( )	
N1	
N10(LOOP)	
M19	
IF[#13EQ#0]GOTO11	
G65 P9764 Z#31 F#127	
IF[#21NE1]GOTO13	
M5	
( )	
M0	Edit to a suitable M-code (for example, M19).
(ROTATE TO P1 −VE)	
( )	
N13	
#27=#131(Z−)	
IF[#13EQ#0]GOTO14	
G65 P9764 Z#31 F#127	
M5	
( )	
M0	Edit to a suitable M-code (for example, M19).
(ROTATE TO P2 +VE)	
( )	

Editing the measure move macro – O9762

#6 = 1.5 (EDIT*BOF) TSM1

#31 = .6 (EDIT*BIN) TSM2

Before attempting a measurement move, the system must be in the untriggered state. The back-off (BOF) or back-in (BIN) distance positions the tool prior to measurement.

The value can be optimised to reduce measurement cycle time, or increased if a SYSTEM ALREADY TRIGGERED alarm is displayed. TSM2

NOTE: If automatic optimisation has been selected during installation, the optimised BIN value will be stored in #900 by default. TSM2

#30 = 0 (DWELL*BEFORE*EACH*SAMPLE) TSM2

In extreme wet conditions, there is a dwell before each sampling measure which can be increased at the top of the macro. The time delay will help reduce coolant on the tool while performing the sampling measures moves.

Orientation of the NCTS system

Throughout this guide, it is assumed that the NCTS system is installed with the laser beam parallel to the X axis. Length measurements are made from the Z axis, and radial measurements are made from the Y axis.

If your system is installed in a different orientation, necessary adjustments must be made to the axes used for length measurement and radial measurement (for details, see manually set variables #121, #122 and #123).

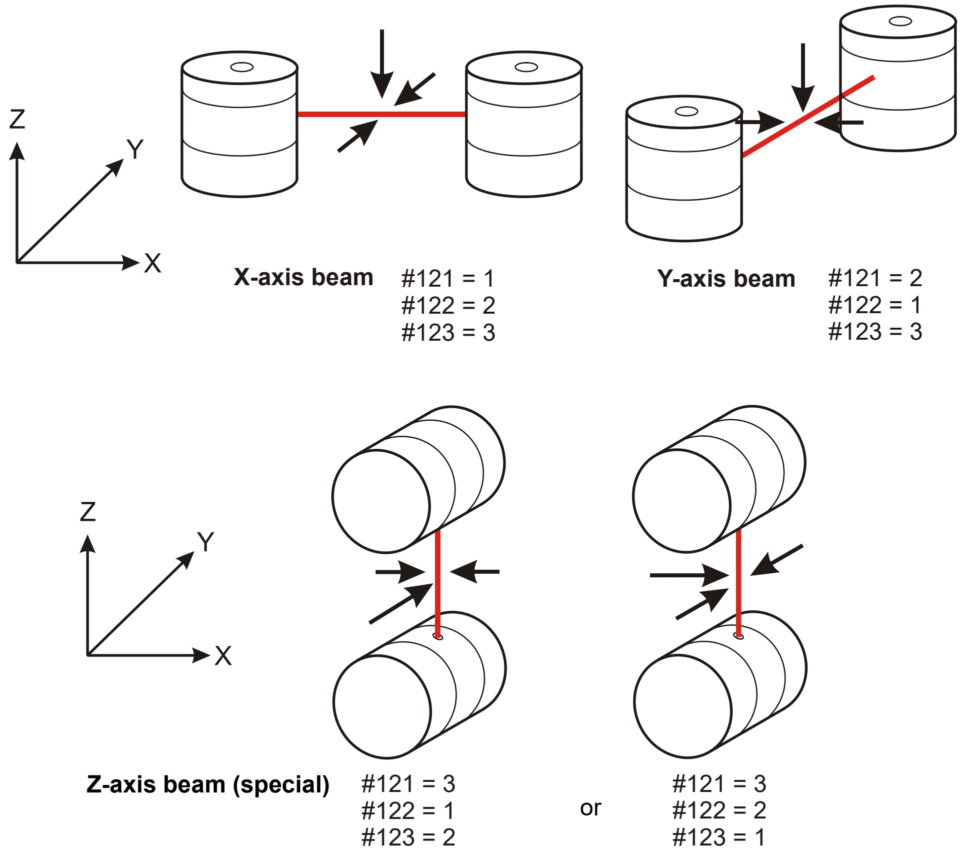


Figure 5 Orientation of the NCTS system

Beam-find and measuring moves

TSM1 – Beam-find moves and measuring moves are all made with the tool moving **into** the laser beam, as shown in Figure 6. Measuring moves are made with the tool rotating.

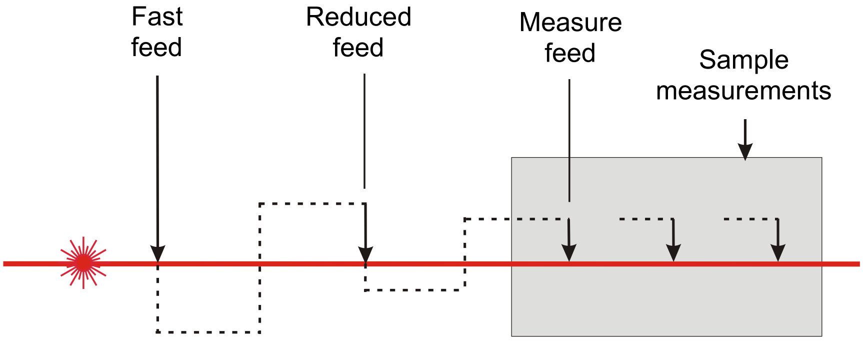


Figure 6 Beam-find and measuring moves TSM1

TSM2 – Beam-find moves and measuring moves are all made as the tool **exits** the laser beam, as shown in Figure 7. Measuring moves are made with the tool rotating.

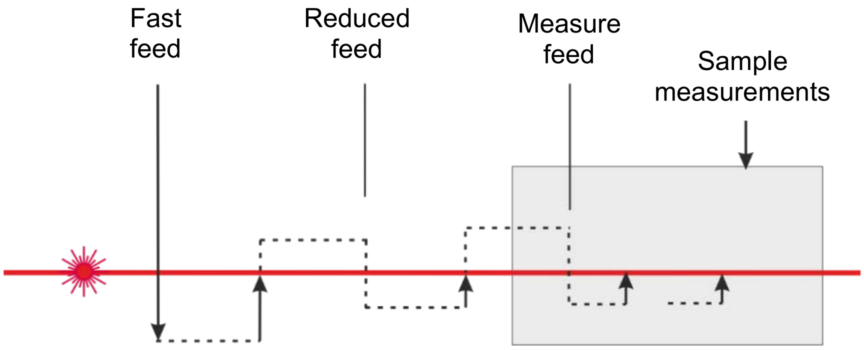


Figure 7 Beam-find and measuring moves TSM2

Calibrating the spindle speed – O9768 TSM1

This cycle automatically runs the spindle at the minimum spindle speed of either 500 r/min or 1000 r/min, with a small XY move of less than 2 mm (0.080 in), depending on the settings in O9760 (see “Editing the setting data macro – O9760” on page 18).

NOTE: It is not necessary to run this cycle if the M-code method of spindle speed control is being used. See “Editing the setting data macro – O9760” on page 18.

This macro is used internally by the software to check the active spindle speed, but it must also be used on its own for RPM calibration before running any other NCTS software cycles.

1. Check that all settings in macro O9760 have been completed. Make any changes necessary as described in “Editing the setting data macro – O9760” on page 18.

2. It is not necessary to have a tool in the spindle to run this cycle, but the machine may require one. Ensure that the tool is suitable to run at the minimum spindle speed as defined by the setting of #139 (500 r/min or 1000 r/min).

3. Jog the spindle nose and tool clear of any obstruction for safety.

4. **Important:** Select 100% on the spindle speed override and verify this setting. In MDI mode, run the spindle at 500 r/min, and check the active screen data to verify the spindle is actually running at 500 r/min.

5. In MDI mode, run macro G65 P9768.

   NOTE: On some machines, it may be necessary to select the tool offset before running G65 P9768.

6. Complete the beam alignment and calibration procedures.

   NOTE: Before running any cycles, check the macro variable assignment used for RPM calibration data. The variable used is #[#120 + 10] – that is, the default is #530. This variable will be available on most machines but, if necessary, it can be changed by either modifying the base number in O9760 or by editing macro O9768 to use another retained variable.

   O9768(REN RPM CHECK)

   #12=#120+10(EDIT-CALIB.VARIABLE NO)   edit: #120+10

   Example: using #519 for RPM calibration

   #12=519(EDIT-CALIB. VARIABLE NO)

Aligning the beam – O9860

NOTE: When installing and setting up the system for the first time, you must run the beam alignment macro **before** you run macro O9861 to calibrate the system.

When using the beam alignment macro, refer to the appropriate Renishaw installation guide in the “List of associated publications” on page ii. These guides contain instructions that describe how to physically align the beam at the NCTS transmitter unit.

Macro O9860 is used during installation of the NCTS system to assist with the alignment of the laser beam. The beam alignment cycle is used for the following tasks:

- To check that the beam is correctly aligned with the machine axis.
- To measure the provisional position of the laser beam in the Sp axis.
- To measure the provisional position of the beam in the Ra axis. Measurements are taken from the positive (using the Rr input) and/or negative (using the −Rr input) sides of the beam.
- To set the measuring point along the beam axis (La) at which the tool is measured.

The provisional values are updated later when the calibration cycle is run.

Although the beam alignment macro is used mainly during installation of the NCTS system, it is also used for routine alignment checking.

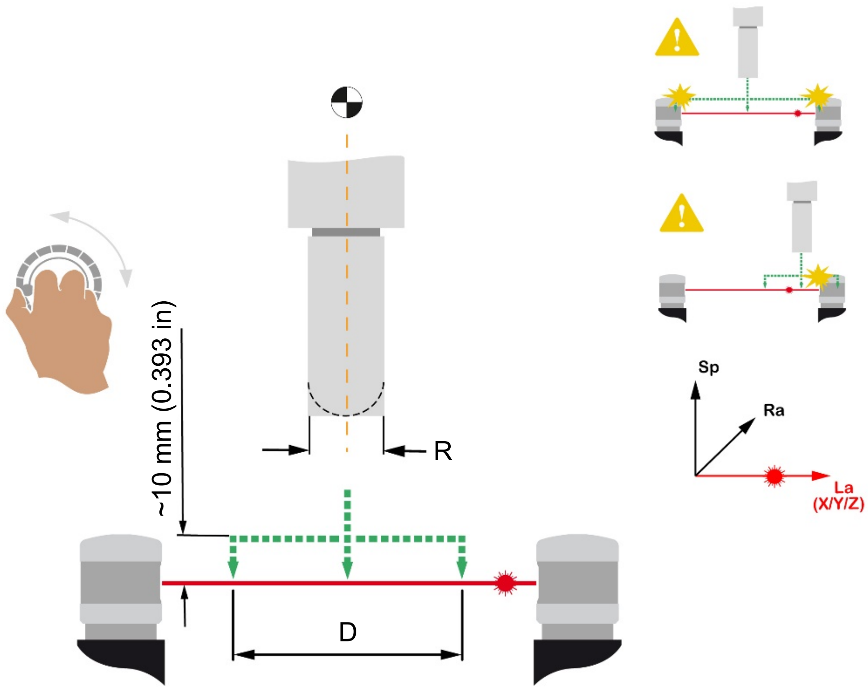


Figure 8 Aligning the beam

Calibration tool required

This cycle requires a calibration tool to be loaded in the spindle of the machine. Ideally, this should be a solid, flat-bottomed or ball-nosed cylinder-type tool having minimal run-out. It is important that you know the exact setting length and diameter of this tool.

NOTE: If a ball-nosed cylinder-type calibration tool has been selected, it is recommended that both sides of the beam can be accessed during calibration.

Description

Load the calibration tool in the spindle of the machine. Using either the jog or handwheel mode, move the tool until the spindle centre line is over the position to be used for tool setting (usually midway along the beam) and approximately 10 mm (0.393 in) above the centre of the beam.

The cycle measures the beam then returns to the centre position and stops on an M00 program stop.

After you have made adjustments to align the beam, restart the cycle to identify new alignment errors.

Setting data

Before running macro O9860, ensure that the settings in macro O9760 are correct. For details, see “Setting data macro variables – O9760” on page 12.

Format

Beam alignment only

G65 P9860 Tt Dd Rr [Kk Qq Zz]
where [ ] denote optional inputs.

Beam alignment and setting provisional beam positions

G65 P9860 B1. Tt Dd Rr [Kk Qq Zz]
where [ ] denote optional inputs.

Example: G65 P9860 B1. T1. D100. K122. Q6. R6. Z15.

Macro inputs

The following inputs are used with this macro:

B1. Provisional calibration of the system.

This is used for checking the alignment of the beam and setting the provisional beam positional data (for details, see “Calibration data macro variables” on page 11).

NOTE: When using the B1. input, the correct length of the calibration tool **must** be entered either in the tool offset register or as a K input value. If you do not wish to update the calibration data, do not use the B1. input.

CAUTION: D input. When specifying the value of the D input, take care that it will not allow any part of the tool holder to make contact with the NCTS system. The projection of the calibration tool should be at least 35 mm (1.38 in) if the default Z input is used for the incremental measuring depth.

Dd d = The span between the reference measuring points.

For greatest accuracy, the span value must be as large as possible, commensurate with the distance between the NCTS system transmitter and receiver units and the size of the calibration tool.

Kk k = Reference length of the calibration tool.

Default: The value in the selected tool offset register.

Qq q = Overtravel distance and radial clearance.

Default: 5 mm (0.197 in)

Rr r = Reference diameter of the calibration tool.

Tt t = Tool length offset number.

This is the offset location in which the master tool length is stored.

Zz z = Incremental measuring depth.

This value determines the depth on the calibration tool at which calibration takes place.

Default: 10 mm (0.394 in)

Outputs

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

#101 Ra-axis beam alignment error over the measured span.

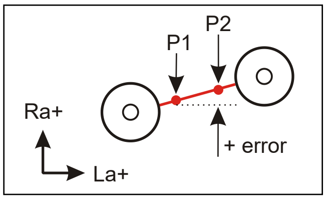


Figure 9 Output #101

#102 Sp-axis beam alignment error over the measured span.

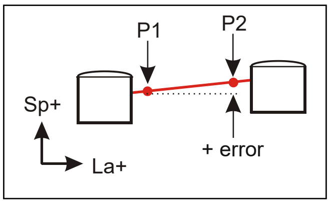


Figure 10 Output #102

If the B1. input is used, the following outputs are also set (it is assumed that the base number is set to 520 in #120).

#520 The provisional tool spindle axis (Sp) position of the beam when measured from the positive side (top) of the beam.

#521 The provisional tool spindle axis (Sp) position of the beam when measured from the negative side (bottom) of the beam.

#522 The provisional radial measuring axis (Ra) position of the beam when measured from the positive side of the beam.

#523 The provisional radial measuring axis (Ra) position of the beam when measured from the negative side of the beam.

#524 The provisional position along the beam axis (La) at which the tool is measured.

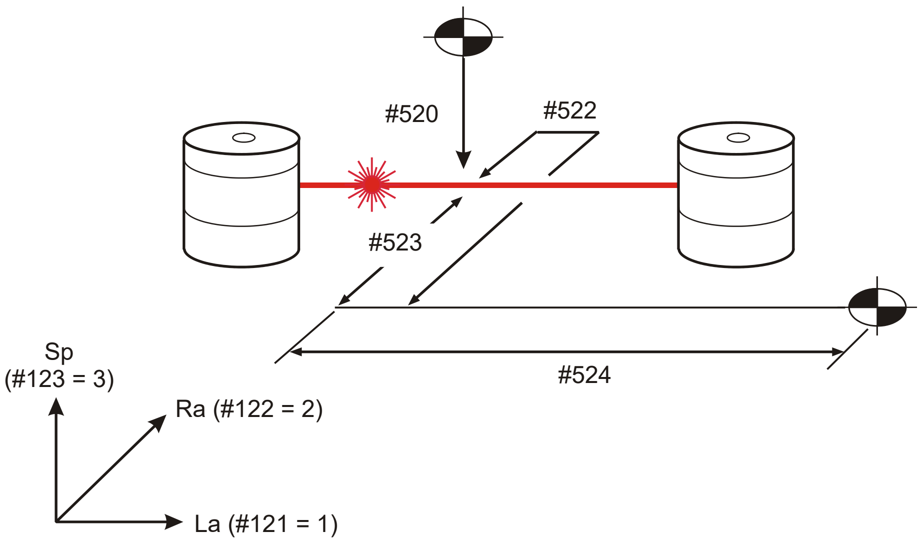


Figure 11 Aligning the beam outputs

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

85 NCI-6/SW2-4 INCORRECT **TSM2**

92 SYSTEM ALREADY TRIGGERED

93 SYSTEM DID NOT TRIGGER

Example: Beam alignment and setting the provisional beam position

O????
(B1. Include approximate set calibration data)
(D Axial distance between measures)
(R Tool diameter)
(Z Search distance)

G21

M6 T1

H1 — Controller-specific. See the appropriate appendix.

G0 G53 X302. Y−236.2

M0 — Handwheel to position.

G65 P9860 B1. T1. D100. R10.

> **NOTE:** Do not use the B1. input if you wish to retain the current calibration data.

M0

M30

Aligning the beam using an offset pin – O9860

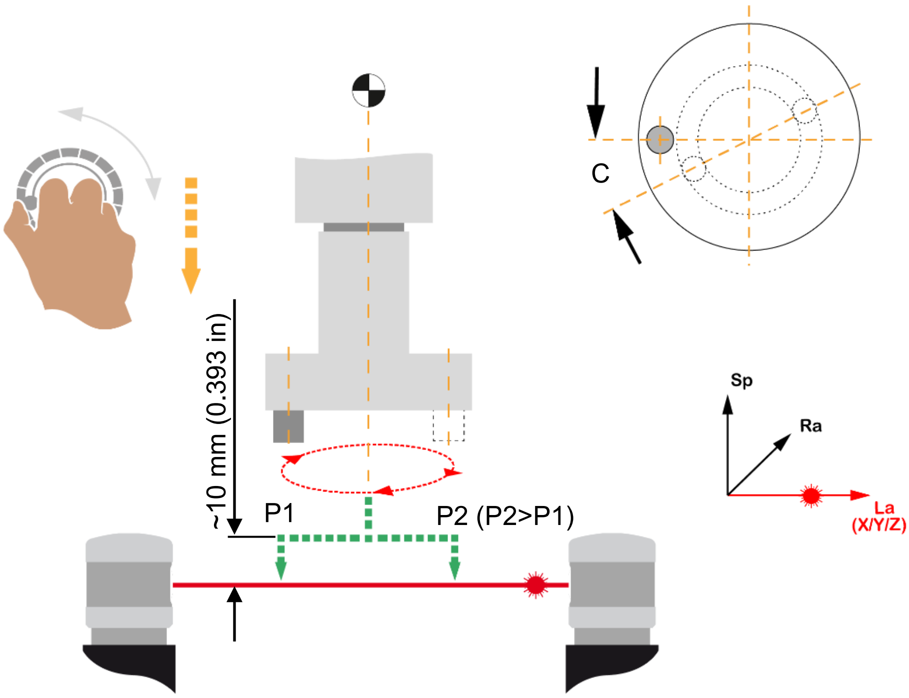


Figure 12 Aligning the beam using an offset pin

Why use this cycle?

On installations where the spindle axis cannot traverse along the beam axis, it is not possible to use the normal method for aligning the beam, as described in “Aligning the beam – O9860” on page 27.

The cycle described here uses an offset pin and rotation of the spindle between two points P1 and P2 along the beam so that a height difference error can be determined to verify that the beam is level. This cycle determines only a height error in the spindle axis. It does not check the squareness of the beam in the radial-measuring axis, but this is not usually critical.

Calibration tool required

This cycle requires a master tool with an offset pin as shown. The tool length must be known and entered in the tool offset register, although the exact size is not important.

Description

NOTE: Not all machines are fitted with spindle positioning as standard. Therefore, the cycle is configured with an M00 program stop at measuring points P1 and P2. The spindle can then be rotated by hand before the cycle continues to run to complete the alignment checks. If spindle positioning is available, the M00 program stop codes can be replaced with programmable positioning commands.

The offset pin must be located so that the measuring positions are as follows:

- P1 < the spindle centre position along the beam; that is, in a more negative direction along the beam from the spindle centre position.
- P2 > the spindle centre position along the beam; that is, in a more positive direction along the beam from the spindle centre position.

Load the calibration tool in the spindle of the machine. Using either the jog or handwheel mode, move the tool until the spindle centre line is over the position to be used for tool setting (usually midway along the beam) and the end of the pin on the tool is approximately 10 mm (0.393 in) above the centre of the beam.

The cycle measures the beam at points P1 and P2, using 180º orientation of the spindle between the two measured points, and stops on M00 program stops.

After you have made adjustments to align the beam, restart the cycle to identify new alignment errors.

Format

Beam alignment only

G65 P9860 Cc Tt [Kk]
where [ ] denote optional inputs.

Beam alignment and setting provisional beam positions

G65 P9860 B1. Cc Tt [Kk]
where [ ] denote optional inputs.

Example: G65 P9860 B1. C180. T1. K122.

Macro inputs

The following inputs are used with this macro:

B1.		Provisional calibration of the system.
		This is used for checking the alignment of the beam and setting the provisional beam positional data (for details, see “Calibration data macro variables” on page 11).

NOTE: When using the B1. input, the correct length of the calibration tool **must** be entered either in the tool offset register or as a K input value. If you do not wish to update the calibration data, do not use the B1. input.

Cc	c =	Used as a flag to select the spindle orientation cycle method.
		It can be set to any value, but a C180. value is recommended.
Kk	k =	Reference length of the calibration tool.
		Default: The value in the selected tool offset register.
Tt	t =	Tool length offset number.
		This is the offset location in which the master tool length is stored.

Calibrating the NCTS system – O9861

Macro O9861 is used to calibrate the NCTS system. It must be used after the laser beam has been aligned with the beam alignment cycle and should also be used for regular calibration of the system. The calibration cycle is used for the following tasks:

- To accurately calibrate the position of the beam in the radial measuring (Ra), beam (La) and spindle (Sp) axes.
- To compensate for variations in the Sp and Ra axes that are caused by temperature changes in the machine tool. Temperature compensation is described in “Temperature compensation tracking – O9861” on page 73.

CAUTION: Before running this cycle or any other cycle (except the beam alignment macro O9860), nominal calibration data **must be** loaded.

This data can be loaded automatically using the beam alignment macro O9860 with the B1. input. Alternatively, approximate data can be entered manually (for details, see “Calibration data macro variables” on page 11).

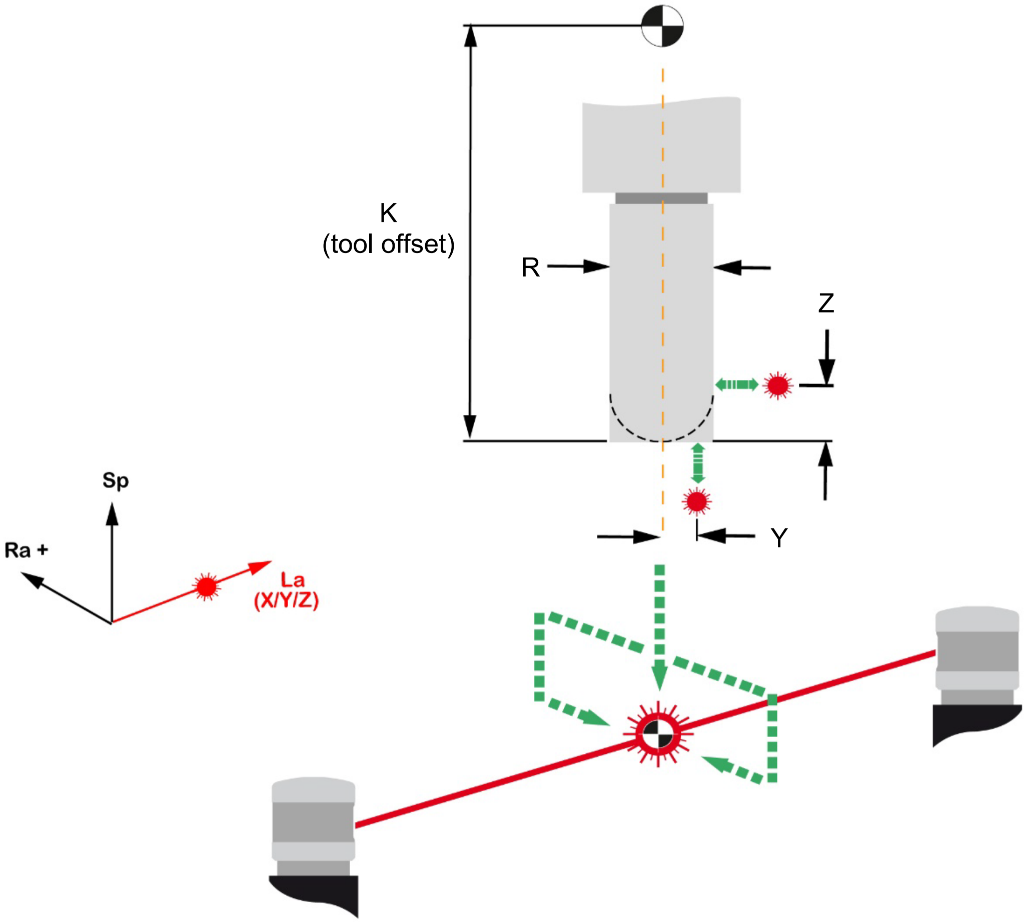


Figure 13 Calibrating the NCTS system

Calibration tool required

This cycle requires a calibration tool to be loaded in the spindle of the machine and the tool number (T) to be active.

Ideally, this should be a solid, flat-bottomed or ball-nosed cylinder-type tool having minimal run-out. It is important that you know the exact setting length and diameter of this tool.

NOTE: If a ball-nosed cylinder-type calibration tool has been selected, it is recommended that both sides of the beam can be accessed during calibration.

Description

Before running this cycle, load the calibration tool in the spindle of the machine and make the tool number (T) active.

Calibration of the position of the beam in the Sp axis and the centre of the beam in the Ra axis occurs while the tool is rotated. The beam width is then calibrated with the tool static. This eliminates run-out errors that may be introduced by the tool.

The cycle always calibrates on the top of the beam and on both sides, provided that #114 = 2 is set in the setting data macro. Additionally, the bottom of the beam position is determined by calculation. When #114 = 1 or −1, the opposite sides of the beam are set to the same value.

Format

G65 P9861 Rr [B1. Cc Kk Qq Ss Tt Uu Yy Zz Ww]
where [ ] denote optional inputs.

Example: G65 P9861 R12. B1. C54. K100.210 Q5. S2500. T1. U1. Y5. Z5. W5.

Macro inputs

The following inputs are used with this macro:

B1.		Final calibration of the system.
Cc	c =	Work offset number that is used to track axis growth. When used with the B1. input, it stores the relevant work offset values as the reference position ready for use later. (See also “Temperature compensation tracking – O9861” on page 73.) Typical: C54 to C59 or C53 (external offset). For details, see the appendix appropriate to your controller at the end of this guide.

Kk	k =	Reference length of the calibration tool. **Default:** Value in the selected tool offset register.
Qq	q =	Overtravel distance and radial clearance. **Default:** 5 mm (0.197 in)
Rr	r =	Reference diameter of the calibration tool.
Ss	s =	Spindle speed at which calibration takes place. For details, see “Setting data macro variables – O9760” on page 12. **Default:** 3000 r/min
Tt	t =	Tool length offset number. This is the offset location for the calibration tool. It is used for machines where the active tool number cannot be read, causing a TOOL OUT OF RANGE alarm.
U1.		Force optimisation. **TSM2** This is only available if optimisation has been selected within the installation wizard when compiling the macro software.
Yy	y =	Radial step-over for length calibration. This is the offset across the beam at which measurement takes place. The tool always comes down first on the beam centre line. **Default:** On centre.
Zz	z =	Measuring height on the tool. This determines the height of the beam on the tool at which diameter calibration takes place. **Default:** 5 mm (0.197 in)

Additional input

Ww input – using a shouldered master tool

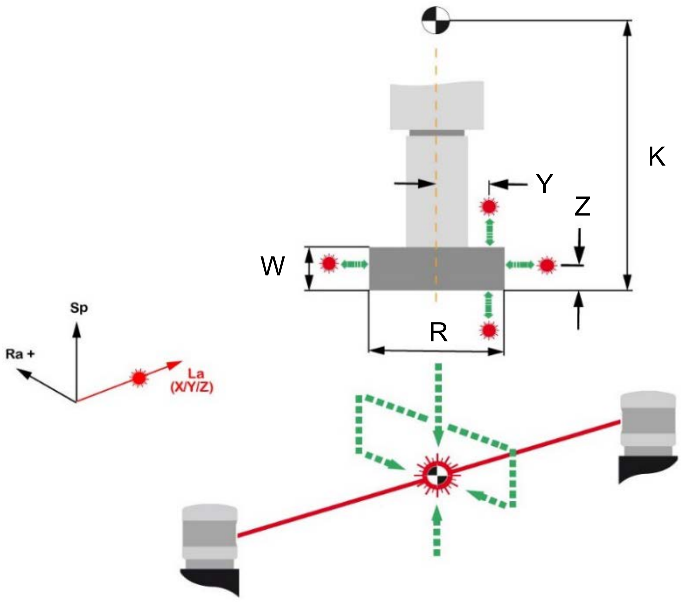


Figure 14 Ww input

The Ww input allows calibration to take place from all four sides of the beam for complete calibration. The cycle always calibrates from the top and bottom of the beam. Provided #114 = 2 is set in the setting data macro, it also calibrates from both sides of the beam. If #114 is set to either 1 or −1, the fourth side is determined by calculation.

Use of the Ww input is valid only when used with the Yy input.

Ww	w =	Reference height of the shoulder of the calibration tool.

Outputs

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

#112	The temperature compensation error for length measurement.
#113	The temperature compensation error for radial measurement.
#520 [520 + 0]	The tool spindle axis (Sp) position of the beam when measured from the positive side (top) of the beam.
#521 [520 + 1]	The tool spindle axis (Sp) position of the beam when measured from the negative side (bottom) of the beam.
#522 [520 + 2]	The radial measuring axis (Ra) position of the beam when measured from the positive side of the beam.
#523 [520 + 3]	The radial measuring axis (Ra) position of the beam when measured from the negative side of the beam.

#524 [520 + 4]	The point along the beam (La) at which measurements are made.
#525 [520 + 5]	The active dynamic zone value.
#526 [520 + 6]	The spindle (length measuring) axis temperature compensation work offset.
#527 [520 + 7]	The radial measuring axis temperature compensation work offset.
#528 (520 + 8)	The dynamic zone value for 20 ms pulse width. **TSM2**
#529 (520 + 9)	The dynamic zone value for 100 ms pulse width. **TSM2**

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

90 OUT OF TOLERANCE

91 FORMAT ERROR

92 SYSTEM ALREADY TRIGGERED

93 SYSTEM DID NOT TRIGGER

Example: Calibration

Assume a calibration tool T1 is used which is 88 mm (3.46 in) long and 6 mm (0.236 in) diameter.

O????	
M6 T1	
H1	Controller specific. See the appropriate appendix.
G65 P9861 B1. K88. R6.	
M30	

IMPORTANT: If you need to track the axis growth caused by temperature variation during the machining operation, use the C input to store the relevant work offset registers as reference values.

Typically, the external work offset G53 is used for temperature compensation tracking. For details, see the appendix appropriate to your controller at the end of this guide.

Example: G65 P9861 B1. K88. R6. C53.

Tool length setting – O9862

Macro O9862 is used to measure the effective length of a cutting tool. The tool length measurement cycle is suitable for on-centre setting of tools such as drills and ball-end mills, and for off-centre setting of tools such as face mills and end mills.

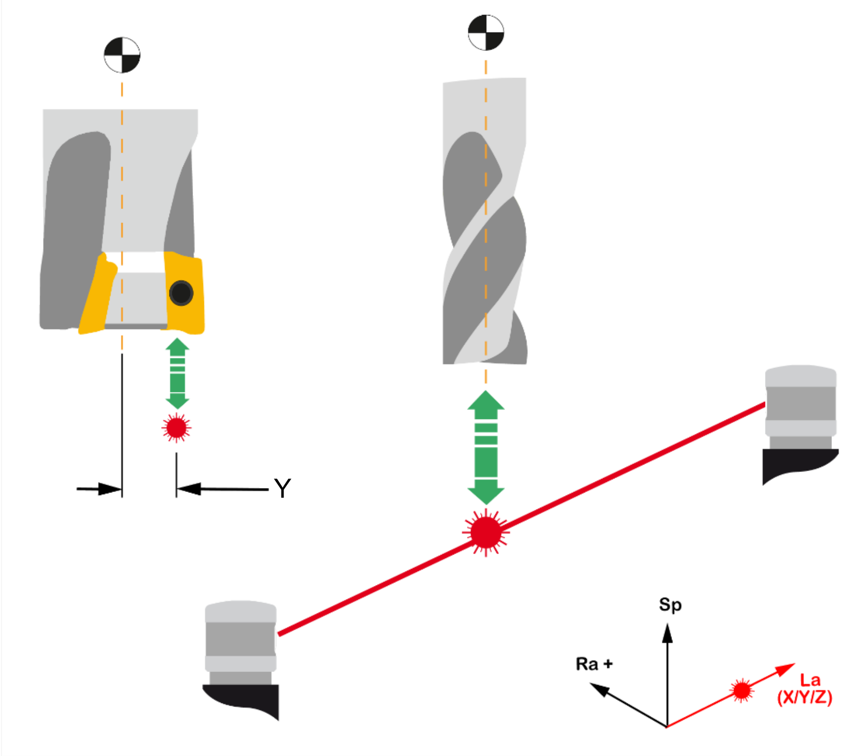


Figure 15 Tool length setting

Description

Tool length is measured while the tool is rotating. The figure shows the two cycle types.

The effective tool length is written into the tool offset register and the wear register is zeroed.

Format

G65 P9862 [A1. B1. Hh Jj Mm Qq Ss Tt Uu Ww Yy]
where [ ] denote optional inputs.

Example: G65 P9862 A1. B1. H0.5 J0.25 M1. Q5. S2500. T20. U1.1 W5. Y3.

Macro inputs

The following inputs are used with this macro:

B1.	Set the length of the tool. B1. is the default B input.
B1.1	Set the length of the tool using the known length tool approach method. B1.1 can be used when #141=0 in setting data macro O9760, but a specific tool requires this type of approach. TSM2

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

MODE	GEOMETRY	WEAR	TOLERANCE CHECK H
No H input	✓	→0	✗
H−	✗	✓	✓
H	✗	✗	✓

Default: No tolerance check.

Jj j = Experience value for the length.

This value is the difference between the measured length of the tool and the effective 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 being used.

Default: Not used.

M1. Tool out of tolerance flag.

Use this flag to prevent a tool OUT OF TOLERANCE alarm from being raised.

Qq q = Overtravel distance.

Default: 5 mm (0.197 in)

Ss s = Spindle speed at which length measurement takes place.

For details, see “Setting data macro variables – O9760” on page 12.

Default: 3000 r/min

CAUTION: T input. 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. When using the known tool length approach method, an approximate length value must be present in the tool register to be updated.

Tt t = Tool 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: Current tool number.

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

Default: Not used.

Yy y = Radial step-over for length setting.

This is the offset across the beam at which length measurement takes place. The value must be less than the radius of the tool. The tool always comes down first on the beam centre line.

Default: On centre.

Additional input

The Aa input is used to inhibit minimum r/min checking for solid tools, where it is not necessary to control the spindle speed. This is particularly useful for long gun drills, where the tool cannot be run unsupported at high spindle speeds.

Solid tool: this is a tool on which the cutting teeth do not protrude below its centre point. Drills, taps and reamers are examples of solid tools. Other tools must be checked above the point at which they become solid; that is, at a position where a good tool would completely block the beam.

A1. Inhibit the minimum r/min checking for solid tools such as drills and taps. This will allow the tool to run using a spindle speed that is less than the minimum r/min checking value.

Default: If this is not programmed, spindle speed checking is performed.

Ww – Upper and lower cutter edge setting

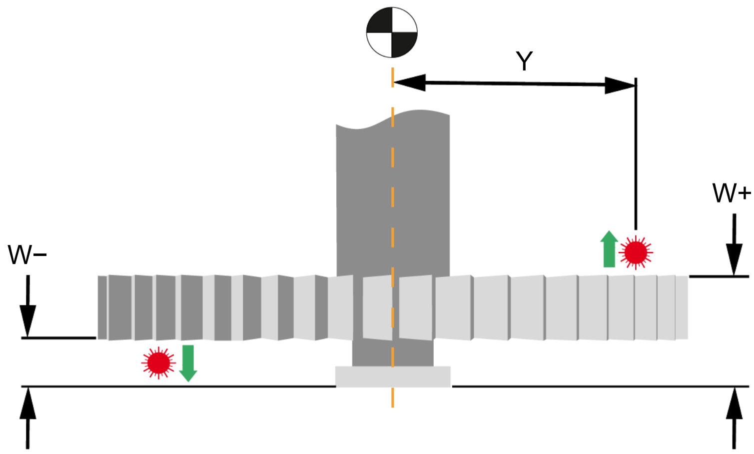


Figure 16 Ww – Upper and lower cutter edge setting

The Ww input is used to set tools on either edge of the cutter. This is ideal for tools such as thin slitting saws (shown above).

TSM1 – The tool measures into the beam with a 2 mm (0.08 in) approach at a reduced feedrate to cater for measuring on thin sections.

TSM2 – The tool measures as it exits the beam using a 2 mm (0.08 in) approach. Programming a Q input will reduce the approach distance to cater for thin sections.

Use of the Ww input is valid only when used with the Yy input.

Ww	w =	The positive value distance from the bottom of the tool to the cutter edge to be set. A positive value means setting takes place on the upper edge of the cutter.
	w− =	The negative value distance from the bottom of the tool to the cutter edge to be set. A negative value means setting takes place on the lower edge of the cutter.

Outputs

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

Set tool length or set the wear.

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

0 = In tolerance.
1 = Out of tolerance.

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

90 OUT OF TOLERANCE

91 FORMAT ERROR

96 RPM OUT OF RANGE

Example 1: Tool length setting using an on-centre tool

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B1.	Set the tool offset (1) on centre.
M30	

A different tool offset can be set by using the T input on the call line as follows:

G65 P9862 B1. S4000. T21.	Set the tool offset (21) on centre. Controlled r/min.

Example 2: Tool length setting using an off-centre tool

Assume the tool is 80 mm (3.15 in) diameter.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B1. Y38. S800	Set the tool offset (1) at 38 mm (1.496 in) radial step-over. Controlled r/min.
M30	

Tool radius/diameter setting – O9862

Macro O9862 is used for measuring the effective radius or diameter of a tool. The tool radius measure cycle allows the radius or diameter to be measured from the positive side of the beam, from the negative side of the beam, or from both sides of the beam.

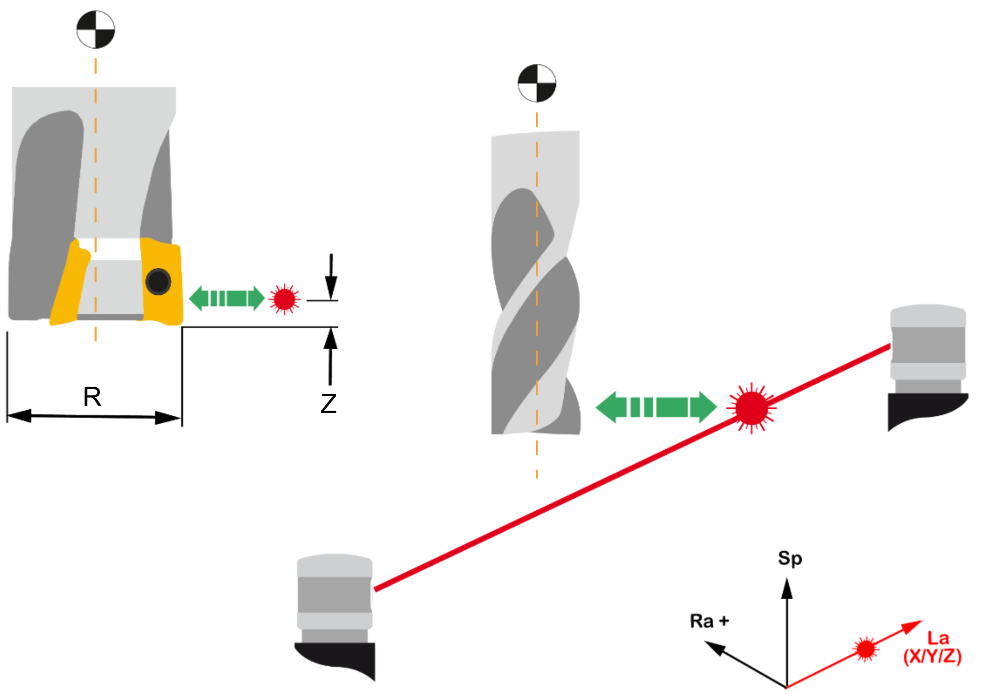


Figure 17 Tool radius/diameter setting

Description

The radius or diameter of a tool is measured while the tool is rotating. Radial measurement is made on either one side or both sides of the beam (see setting #113 in “Setting data macro variables – O9760” on page 12).

The effective radius or diameter is written into the tool offset register. If the controller has separate wear and geometry registers, the wear register is zeroed and the radius/diameter value is placed in the geometry register.

Format

G65 P9862 B2. [Dd Ee Ff Ii Mm Qq Rr Ss Tt Uu Xx Zz]
where [ ] denote optional inputs.

Example: G65 P9862 B2. D5. E0.02 F0.3 I0.2 M1. Q5. R25. S3000. T1. U1.1 X2. Z6.

Macro inputs

The following inputs are used with this macro:

B2. Measure the radius/diameter of the tool.

B2.1 Measure the radius/diameter of the tool using the known tool approach. B2.1 can be used when #141=0 in setting data macro O9760, but a specific tool requires this type of approach. [TSM2]

Dd d = 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.

Ee e = Tolerance value that defines when the tool radius/diameter is out of tolerance.

MODE	GEOMETRY	WEAR	TOLERANCE CHECK E
No E input	✓	→0	✗
E−	✗	✓	✓
E	✗	✗	✓

Default: No tolerance check.

Ff f = The step distance between each radial measurement when using the Xx input.

NOTE: A negative input will set the tool length to the radial high or low point rather than the end of the tool.

Default: 0.3 mm (0.012 in)
Range: 0.1 mm (0.004 in) to 1.5 mm (0.060 in)

Ii i = 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.

M1. Tool out of tolerance flag.

Use this flag to prevent a tool OUT OF TOLERANCE alarm from being raised.

Qq q = Overtravel distance and radial clearance.

Default: 5 mm (0.197 in)

Rr r = Diameter of the tool.

This is the nominal diameter of the tool.

Default: Maximum diameter of tool in #112.

Ss s = Spindle speed at which radius/diameter measurement takes place.

For details, see “Setting data macro variables – O9760” on page 12.

Default: 3000 r/min

CAUTION: T input. 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. When using the known tool length approach method, an approximate length value must be present in the tool register to be updated.

Tt t = Length offset number.

Default: Current tool number.

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

Default: Not used.

Xx x+ = Search distance for a high spot in the spindle axis.

This defines a search distance above the Z input measuring height that is used to find a radial high spot on the cutter. It is suitable for single-point boring bars and cutters with irregular radial profiles.

The step distance between each radial measurement is controlled by the Ff input. A second, finer, scan then takes place at a reduced step distance of F ÷ 4 (F × 0.25).

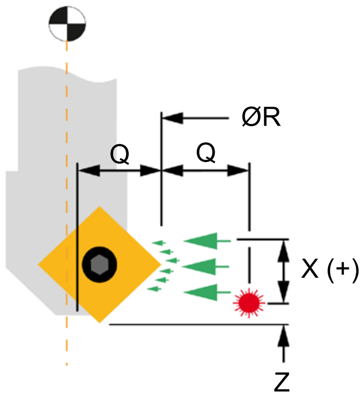


Figure 18 Macro input X+

NOTE: Using this input increases the cycle time, so keep the X distance to a minimum and control the radial moves using the R and Q inputs.

Omitting the fine scan is also possible by editing O9766(REN SCAN). Near the start of the macro there is a line:

#15=4(0=COARSE*SCAN)

Change this to #15=0.

Default: 0

x− = Search distance for a low spot in the spindle axis.

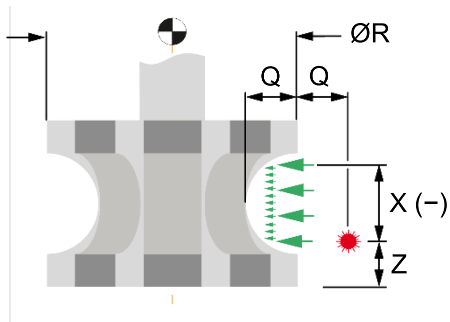


Figure 19 Macro input X−

Zz z = Measuring height of the tool.

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

Default: 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.

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

90	OUT OF TOLERANCE
91	FORMAT ERROR
91	D FORMAT ERROR
96	RPM OUT OF RANGE

Example 1: Tool radius setting

Assume the tool is a 10 mm (0.394 in) diameter slot drill.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B2. R10.	Set the tool radius (H1).
M30	

Example 2: Tool radius setting

Assume the tool is an 80 mm (3.15 in) diameter cutter.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B2. D21. S800. R80.	Set the tool radius offset (21). Controlled radial clearance and r/min.
M30	

Tool length and radius setting – O9862

Macro O9862 is used for measuring the effective length and radius or diameter of a tool. The tool length and radius measure cycle is particularly suitable for tools such as face mills, end mills, slot cutters, disc mill cutters, dovetail cutters and boring tools.

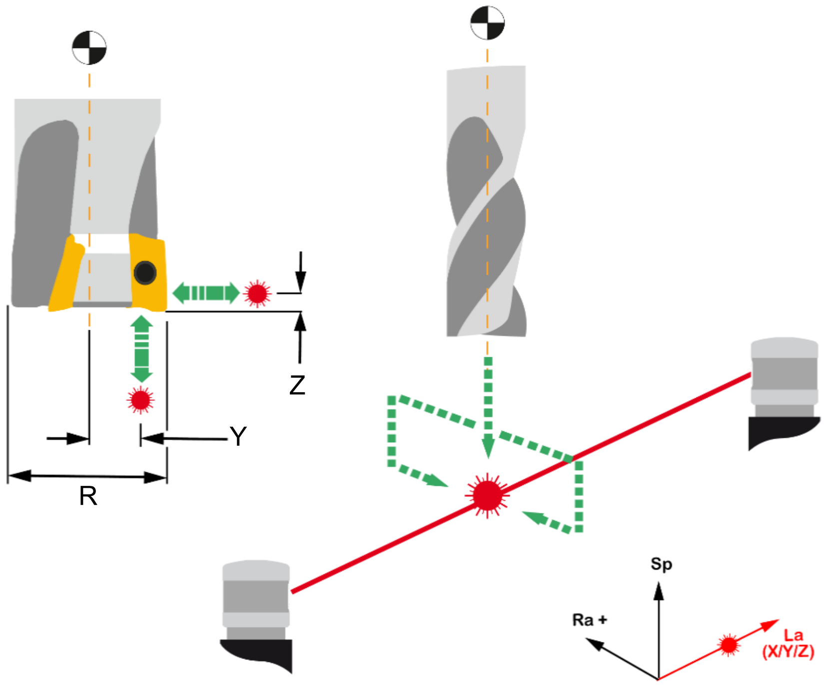


Figure 20 Tool length and radius setting

Description

This single cycle combines the tool length measuring cycle (see “Tool length setting – O9862” on page 40) and the tool radius/diameter measuring cycle (see “Tool radius/diameter setting – O9862” on page 45).

The figure shows the combined cycle moves. Radial measurement can be made on either one or both sides of the beam (see setting #113 in “Setting data macro variables – O9760” on page 12).

Length and radius values are written into the tool offset register. If the controller has separate wear and geometry registers, the wear registers are zeroed and the values are placed in the geometry registers.

Format

G65 P9862 B3. [Dd Ee Ff Hh Ii Jj Mm Qq Rr Ss Tt Uu Ww Yy Xx Zz]
where [ ] denote optional inputs.

Example: G65 P9862 B3. D5. E0.02 F0.3 H0.02 I0.02 J0.025 M1. Q5. R25. S2500 T20. U1.1 W5. Y3. X2. Z6.

Macro inputs

The following inputs are used with this macro:

B3. Measure the length and the radius/diameter of the tool.

B3.1 Measure the radius/diameter of the tool using the known tool approach. B3.1 can be used when #141=0 in setting data macro O9760, but a specific tool requires this type of approach. TSM2

Dd d = Diameter offset number.

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

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

Ee e = Tolerance value that defines when the tool radius/diameter is out of tolerance.

MODE	GEOMETRY	WEAR	TOLERANCE CHECK E
No E input	✓	→0	×
E−	×	✓	✓
E	×	×	✓

Default: No tolerance check.

Ff f = The step distance between each radial measurement when using the Xx input.

NOTE: A negative input will set the tool length to the radial high or low point rather than the end of the tool.

Default: 0.3 mm (0.012 in)
Range: 0.1 mm (0.004 in) to 1.5 mm (0.060 in)

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

MODE	GEOMETRY	WEAR	TOLERANCE CHECK H
No H input	✓	→0	×
H−	×	✓	✓
H	×	×	✓

Default: No tolerance check.

CAUTION: When using I, J and K inputs, it is important that they are specified in alphabetical order.

Ii i = 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.

Jj j = 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.

M1. Tool out of tolerance flag.

Use this flag to prevent a tool OUT OF TOLERANCE alarm from being raised.

Qq q = Overtravel distance and radial clearance.

Default: 5 mm (0.197 in)

Rr r = Diameter of the tool.

This is the nominal diameter of the tool.

Default: Maximum diameter of tool in #112.

Ss s = Spindle speed at which length and radius/diameter measurement takes place.

For details, see “Setting data macro variables – O9760” on page 12.

Default: 3000 r/min

CAUTION: T input. 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. When using the known tool length approach method, an approximate length value must be present in the tool register to be updated.

Tt t = 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: Current tool number.

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

Default: Not used.

Xx x+ = Search distance for a high spot in the spindle axis.

This defines a search distance above the Z input measuring height that is used to find a radial high spot on the cutter. It is suitable for single-point boring bars and cutters with irregular radial profiles.

The step distance between each radial measurement is controlled by the Ff input. A second, finer, scan then takes place at a reduced step distance of F ÷ 4 (F × 0.25).

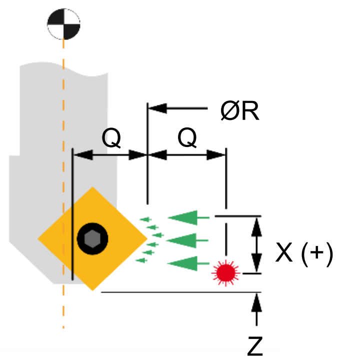


Figure 21 Macro input X+

NOTE: Using this input increases the cycle time, so keep the X distance to a minimum and control the radial moves using the R and Q inputs.

Omitting the fine scan is also possible by editing O9766(REN SCAN). Near the start of the macro there is a line:

#15=4(0=COARSE*SCAN)

Change this to #15=0.

Default: 0

x− = Search distance for a low spot in the spindle axis.

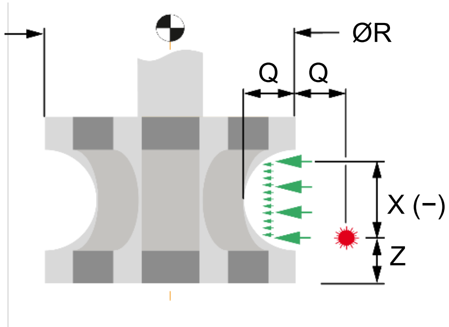


Figure 22 Macro input X−

Yy y = Radial step-over for length setting.

This is the offset across the beam at which length measurement takes place. The value must be less than the radius of the tool. The tool always comes down first on the beam centre line.

Default: On centre.

Zz z = Measuring height of the tool.

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

Default: 5 mm (0.197 in)

Additional input

Ww – Upper and lower cutter edge setting

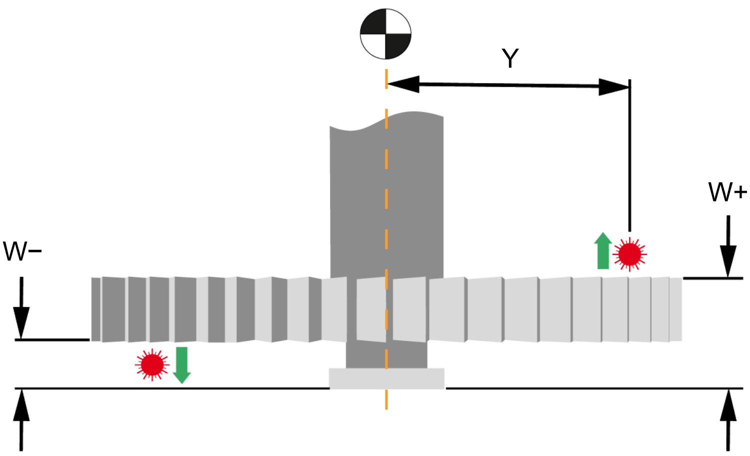


Figure 23 Ww Upper and lower cutter edge setting

The Ww input is used to set tools on either edge of the cutter. This is ideal for tools such as thin slitting saws (shown above).

TSM1 – The tool measures into the beam with a 2 mm (0.08 in) approach at a reduced feedrate to cater for measuring on thin sections.

TSM2 – The tool measures as it exits the beam using a 2 mm (0.08 in) approach. Programming a Q input will reduce the approach distance to cater for thin sections.

Use of the Ww input is valid only when used with the Yy input. The Zz input is not used with this input.

Ww	w =	The positive value distance from the bottom of the tool to the cutter edge to be set. A positive value means setting takes place on the upper edge of the cutter.
	w− =	The negative value distance from the bottom of the tool to the cutter edge to be set. A negative value means setting takes place on the lower edge of the cutter.

Outputs

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

Set tool length.
Set tool 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 = Out of tolerance (length).
2 = Out of tolerance (radius/diameter).

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

90 OUT OF TOLERANCE

91 FORMAT ERROR

91 D FORMAT ERROR

94 SAME T AND D OFFSET

96 RPM OUT OF RANGE

Example 1: Tool length/radius setting

Assume the tool is a 10 mm (0.394 in) diameter slot drill.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B3. D21.	Set the tool length offset (1) and radius offset (21).
M30	

Example 2: Tool length/radius setting

Assume the tool is an 80 mm (3.15 in) diameter cutter.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B3. D21. Y38. T10. S800.	Set the tool length offset (10) at 38 mm (1.496 in) radial step-over and set the radius offset (21). Controlled r/min.
M30	

Cutting edge checking – O9862

NOTE: This cycle can be used only when the latch mode feature of the NCTS system is installed and operational.

Macro O9862 is used for checking the cutting edges of a tool. The diameter cutting edge check cycle checks either for missing or damaged teeth or for excessive run-out of the cutter.

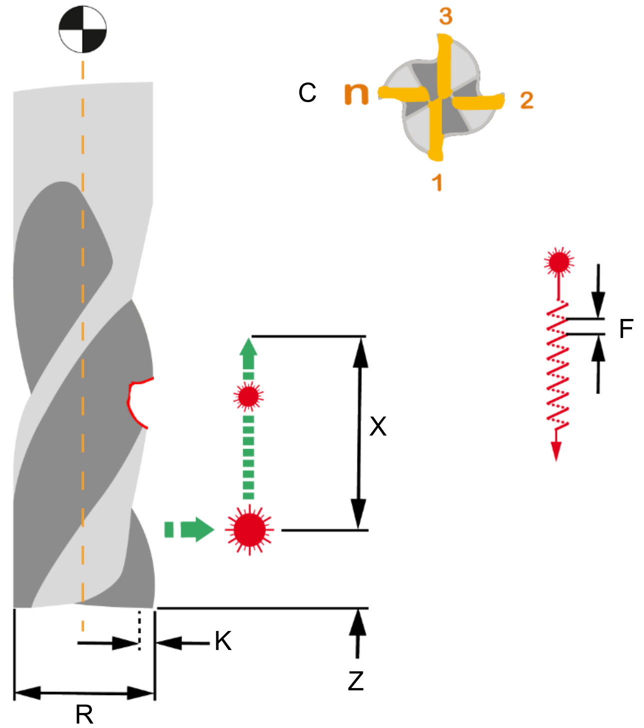


Figure 24 Cutting edge checking

Description

Before a tool is checked for missing teeth or excessive run-out, it is first set for radius/diameter. The diameter cutting edge check cycle then moves the rotating tool into the beam until the teeth interfere with the beam by the cutting edge run-out tolerance value. This value is defined by the K input.

The spindle speed is calculated from the minimum pulse signal delay (#115) of the NCTS system and the number of teeth on the cutting tool. This ensures that when each tooth enters the beam, a permanent beam cut signal is held unless a tooth is either missing or out of tolerance. The beam cut signal is monitored for a minimum of two revolutions.

Format

G65 P9862 B2. or B3. or B4. Cc [Ff Kk Mm Qq Rr Ss Tt Xx Zz]
where [ ] denote optional inputs.

Example: G65 P9862 B2. C6. F0.1 K0.02 M1. Q5. R25. S2500. T1. X5. Z6.

Macro inputs

The following inputs are used with this macro:

B2. B3.		For details of these cycles, see either “Tool radius/diameter setting – O9862” on page 45 or “Tool length and radius setting – O9862” on page 50.
B4.		Cutting edge check without tool offset updating or prior radius/diameter measurement.
B4.1		Cutting edge check using the known tool approach. B4.1 can be used when #141=0 in setting data macro O9760, but a specific tool requires this type of approach. [TSM2]
Cc	c =	The number of teeth on the tool. This automatically selects the cutting-edge check. **Default:** No default.
Ff	f =	Feedrate-per-revolution for cylinder profile checking when using the X input. **Default:** 0.1 mm (0.0394 in)
Kk	k =	Tolerance value that defines when the tool cutting edge run-out is excessive.
Kk−	k− =	Acts as standard K input but additionally measures the tool run-out and stores it in #137. **Default:** 0.025 mm (0.0010 in)
M1.		Tool out of tolerance flag. Use this flag to prevent a tool OUT OF TOLERANCE alarm from being raised.
Qq	q =	Overtravel distance and radial clearance. **Default:** 5 mm (0.197 in)
Rr	r =	Nominal diameter of the tool. **Default:** Maximum diameter of tool in #112.
Ss	s =	Spindle speed at which radius/diameter measurement takes place. For details, see “Setting data macro variables – O9760” on page 12. The spindle speed for cutting edge checking is set automatically and is based on the minimum pulse signal delay of the control and the number of teeth on the tool. **Default:** 3000 r/min

CAUTION: T input. 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. When using the known tool length approach method, an approximate length value must be present in the tool register to be updated.

Tt t = Length offset number.

Default: Current tool number.

Xx x = Cylinder profile checking distance (the spindle axis movement) while edge checking. The value is incremental from the Z input (radial measuring position). It is used in conjunction with the F input (feedrate).

Default: 0

Zz z = Measuring height of the tool.

This is the Sp-axis position from the end face of the tool at which measurement of the cutting edge takes place.

Default: 5 mm (0.197 in)

Outputs

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

#137 Tool run-out **TSM2**

#148 Out of tolerance / missing edge flag.

This is set when either the tool cutting edge run-out is out of tolerance, provided the K input is used, or an edge is missing from the tool.

0 = In tolerance / no edges missing.
2 = Out of tolerance / edge missing.

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

90	OUT OF TOLERANCE
91	FORMAT ERROR
92	SYSTEM ALREADY TRIGGERED
93	SYSTEM DID NOT TRIGGER
94	SAME T AND D OFFSET
96	RPM OUT OF RANGE
98	RUN-OUT/EDGE MISSING

Example 1: Cutting edge checking

Assume a two-flute slot drill and check for a broken edge 0.5 mm (0.02 in) from the end face of the cutter.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B2. C2. D21. Z0.5	

Example 2: Cylinder cutting edge checking

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9862 B3. C2. Z0.5 X5.0	

Broken tool detection: plunge checking – O9863

CAUTION: Before this cycle is run, the current tool offset must be active.

Macro O9863 is used to check for breakage of cutting tools. This cycle uses a plunge check, where the tool is moved into and out of the laser beam in the axis used for length setting. The cycle can also check for a 'long tool' condition, where the tool has possibly pulled out during machining.

Typically, a tool needs to be checked after a machining operation to verify that it is not broken before the next tool is selected.

The cycle can be used for checking most tools, particularly for rotating tool applications where the cutting teeth will intermittently interfere with the beam as it is positioned for tool checking.

This cycle is similar in operation to the broken tool detection for solid tools checking cycle (macro O9866) described on page 65. The differences are internal to the macro, as macro O9866 uses the broken tool latch mode feature of the NCTS system for checking broken tools. This makes it more robust than O9863 in very wet conditions and quicker, but it is suitable only for solid tool applications.

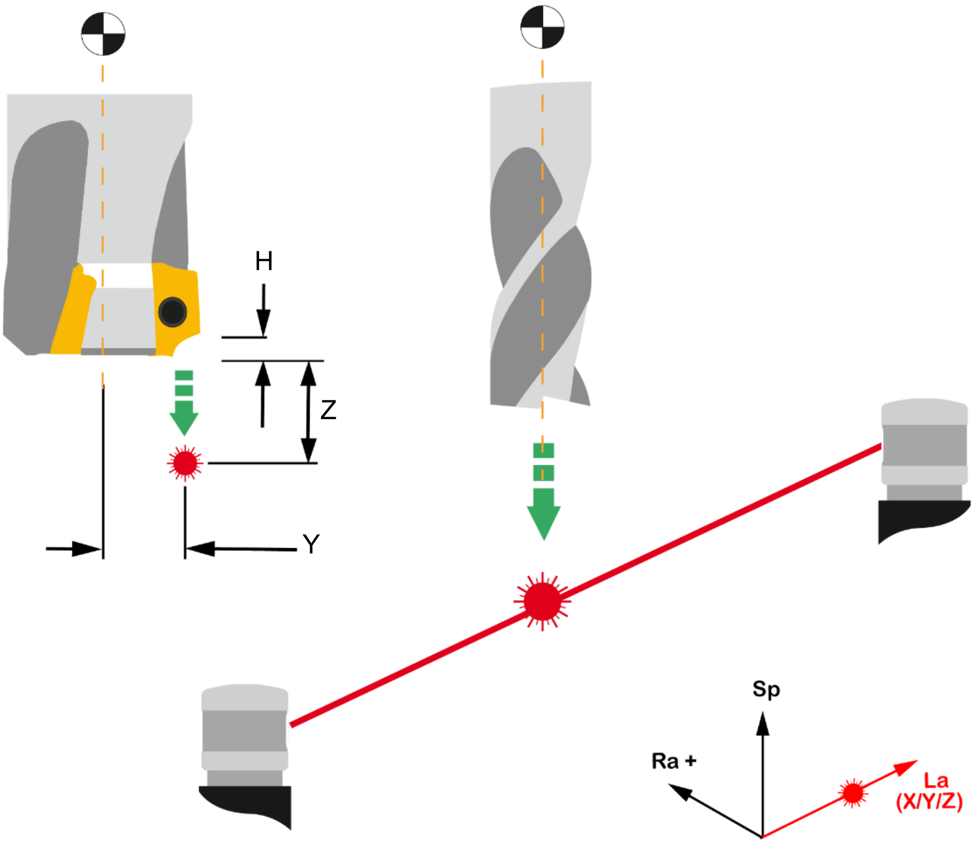


Figure 25 Broken tool detection – plunge checking

Description

Detection of a broken tool occurs while the tool is rotated in the beam. Moves into and out of the beam are at the rapid feedrate.

The tool retracts either to the home position, or to a position in the tool spindle axis (Sp) when the Z input is used. It then moves in the measuring axis (Ra) and laser beam axis (La) until it is above the beam. Finally, it approaches the beam in the spindle axis (Sp).

When a positive H input is used, the tool is checked at the broken tool position only. When a negative H input is used, the tool is checked at both the long tool and broken tool positions.

At the end of the cycle, the tool retracts out of the beam either to a safe position in the spindle axis (Sp) or to the home position, if no Z input is used.

Format

G65 P9863 [Hh Ii Jj Kk Mm Ss Tt Vv Yy Zz Aa]
where [ ] denote optional inputs.

Example: G65 P9863 H5. I12. J4. K2. M1. S2800 T1. V4. Y3. Z6. A1.

Macro inputs

The following inputs are used with this macro:

Hh h = Tolerance value that defines when the tool is defined as broken. A negative value checks the tool for both broken and long tool conditions.

Default: 0.5 mm (0.0197 in)

CAUTION: When using I, J and K inputs, it is important that they are specified in alphabetical order.

Ii i = The number of retries to test the status of the laser before it performs the tool check. Coolant drips could cause false triggers and mask the true laser status. If conditions are very wet, causing excessive false triggers, increase the number of retries.

Default: 5

Jj j = The number of retries to confirm a tool pull-out condition. A higher number of retries should be used when conditions are very wet and coolant masks the true tool condition. The retry logic will only initiate if the beam is broken. **TSM1**

Default: 10

Kk k = The number of retries to confirm a broken tool condition. A higher number of retries should be used when conditions are very wet and coolant masks the true tool condition. **TSM1**

Default: 3

M1. Tool broken flag.

Use this flag to prevent a BROKEN TOOL or OUT OF TOLERANCE alarm from being raised.

Ss s = Spindle speed at which checking for a broken tool takes place.

For details, see “Setting data macro variables – O9760” on page 12.

Default: 3000 r/min

CAUTION: T input. 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.

Tt t = Length offset number.

Default: Current tool number.

Vv v = Additional time to confirm a broken tool or tool pull-out condition. Use this input when conditions are very wet and coolant masks the true tool condition.

Example: V1. increases the checking time by one second.

Yy y = Radial step-over distance.

The offset across the beam at which measurement of the tool length takes place.

Default: On centre.

Zz z = Safety plane.

The distance (in the spindle axis) by which the tool is retracted from the beam.

Default: Retract to the home position.

Additional input

The Aa input is used to inhibit minimum r/min checking for solid tools, where it is not necessary to control the spindle speed. This is particularly useful for long gun drills, where the tool cannot be run unsupported at high spindle speeds.

Solid tool: this is a tool on which the cutting teeth do not protrude below its centre point. Drills, taps and reamers are examples of solid tools. Other tools must be checked above the point at which they become solid; that is, at a position where a good tool would completely block the beam.

A1. Inhibit the minimum r/min checking for solid tools such as drills and taps. This will allow the tool to run using a spindle speed that is less than the minimum r/min checking value.

Default: If this is not programmed, spindle speed checking is performed.

Outputs

The following output is always set when this cycle is executed:

#148 Broken tool flag.

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

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

84 HARDWARE FAULT

90 OUT OF TOLERANCE

91 FORMAT ERROR

96 RPM OUT OF RANGE

99 BROKEN TOOL

Example: Broken tool detection – plunge checking

O????

M6 T1

G0 G43 H1 Z200.

(complete the machining sequence with tool T1)

G0 Z200.	Move to a safe position where the broken tool detection cycle can be called.
X200. Y300.	
G65 P9863 Z100.	Make a broken tool check. Either a BROKEN TOOL alarm is raised and the program stops, or the program continues.
M6 T2	Select the next tool and continue.

(continue machining)

If the broken tool flag method is used, the call cycle is modified as follows:

G65 P9863 Z100. M1.	Make a broken tool check without raising an alarm. The #148 flag is set.
IF[#148EQ1]GOTO100	Go to N100

(continue the program)

Block N100 will contain corrective actions; for example, select a sister tool for use or select a new pallet or component.

Broken tool detection: solid tools – O9866

CAUTION: This cycle can be used only when the Tool Break Mode (NCi-6 M-code 1) feature of the NCTS system is connected.

Macro O9866 is used to check for breakage of cutting tools. It is similar in operation to the broken tool detection plunge checking cycle (macro O9863) described on page 61. The differences are internal to the macro, as it uses the broken tool latch mode feature of the NCTS system for checking broken tools. This makes it more robust than O9863 in coolant conditions and quicker, but it is suitable only for solid tool applications.

Solid tool: this is a tool on which the cutting teeth do not protrude below its centre point. Drills, taps and reamers are examples of solid tools. Other tools must be checked above the point at which they become solid; that is, at a position where a good tool would completely block the beam.

The cycle uses a plunge check, where the tool is moved into and out of the laser beam in the axis used for length setting. The cycle can also check for a ‘long tool’ condition, where the tool has possibly pulled out during machining.

Typically, a tool needs to be checked after a machining operation to verify that it is not broken before the next tool is selected.

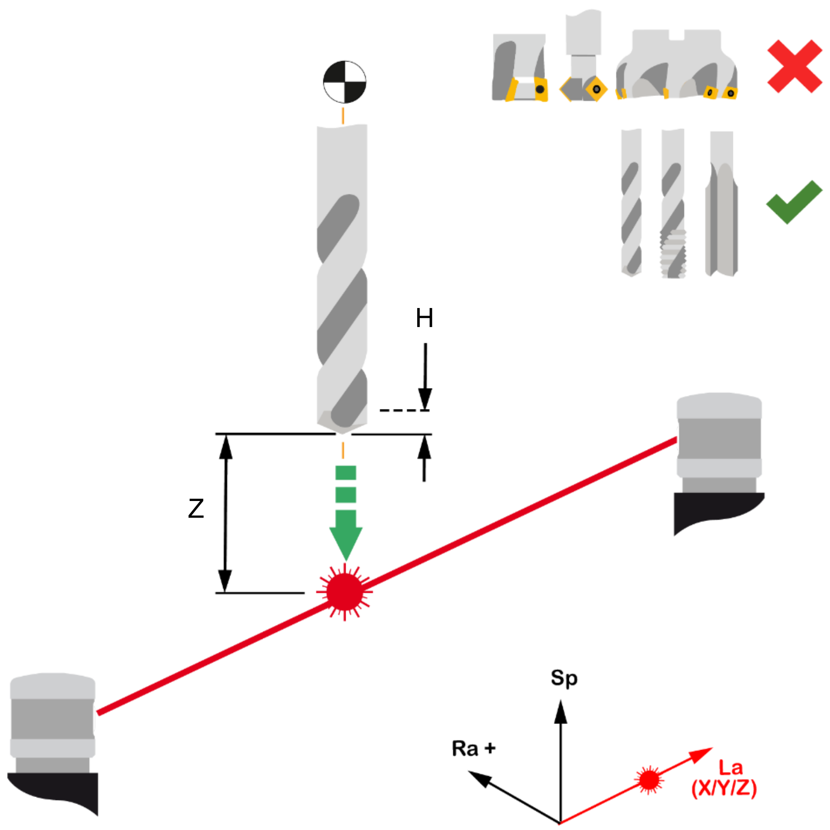


Figure 26 Broken tool detection for solid tools

Description

Detection of a broken tool occurs while the tool is rotated in the beam. Moves into and out of the beam are at the rapid feedrate.

The tool retracts either to the home position, or to a position in the tool spindle axis (Sp) when the Z input is used. It then moves in the measuring axis (Ra) and laser beam axis (La) until it is above the beam. Finally, it approaches the beam in the spindle axis (Sp).

When a positive H input is used, the tool is checked at the broken tool position only. When a negative H input is used, the tool is checked at both the long tool and broken tool positions.

At the end of the cycle, the tool retracts out of the beam either to a safe position in the spindle axis (Sp), or to the home position if no Z input is used.

Format

G65 P9866 [Aa Hh Mm Tt Zz]
where [ ] denote optional inputs.

Example: G65 P9866 A1. H5. M1. T1. Z6.

Macro inputs

The following inputs are used with this macro:

A1. Inhibit the minimum r/min checking for solid tools such as drills and taps. This will allow the tool to run using a spindle speed that is less than the minimum r/min checking value.

Default: If this is not programmed, spindle speed checking is performed.

Hh h = Tolerance value that defines when the tool is defined as broken. A negative value checks the tool for both broken and long tool conditions.

Default: 0.5 mm (0.0197 in)

M1. Tool broken flag.

Use this flag to prevent a BROKEN TOOL or OUT OF TOLERANCE alarm from being raised.

CAUTION: T input. 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.

Tt t = Length offset number.

Default: Current tool number.

Zz	z =	Safety plane.
		The distance (in the spindle axis) by which the tool is retracted from the beam.
		Default: Retract to the home position.

Outputs

The following output is always set when this cycle is executed:

#148 Broken tool flag.

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

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see "Error messages and alarms" on page 85.

91 FORMAT ERROR

96 RPM OUT OF RANGE

99 BROKEN TOOL

Example: Broken tool detection – plunge checking

O????

M6 T1

G0 G43 H1 Z200.

(complete the machining sequence with tool T1)

G0 Z200.	Move to a safe position where the broken tool detection cycle can be called.
X200. Y300.	
G65 P9866 Z100.	Make a broken tool check. Either a BROKEN TOOL alarm is raised and the program stops, or the program continues.
M6 T2	Select the next tool and continue.

(continue machining)

If the broken tool flag method is used, the call cycle is modified as follows:

G65 P9866 Z100. M1.	Make a broken tool check without raising an alarm. The #148 flag is set.
IF[#148EQ1]GOTO100	Go to N100

(continue the program)

Block N100 will contain corrective actions; for example, select a sister tool for use or select a new pallet or component.

Cutter radius and linear profile checking – O9865

NOTE: This cycle can be used only when the latch mode feature of the NCTS system is installed and operational.

This cycle is used to verify the specified form of a profiled cutting tool. It is particularly suitable for ball nose cutters, cutters with a corner radius, and cutters with linear profiles.

The specified form of the profile can be checked either internally, for the detection of missing or broken teeth and inserts, or externally, for detection of misaligned inserts and incorrect forms.

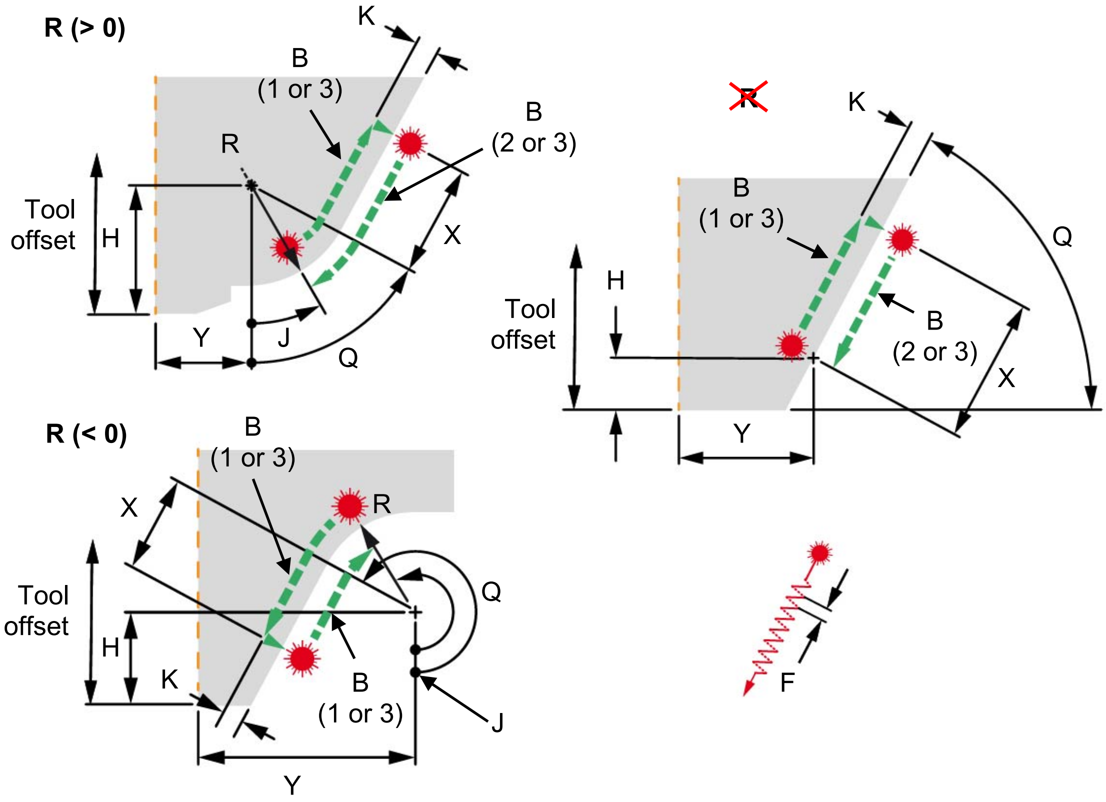


Figure 27 Cutter radius and linear profile checking

Format

Radius profile checking

G65 P9865 Rr Xx [Bb Cc Ff Hh Jj Kk Mm Qq Ss Tt Yy Zz]
where [ ] denote optional inputs.

Example: G65 P9865 B3. C4. F0.3 H1. J1. K0.1 M1. Q90. R10. S4000. T20. X5. Y5. Z50.

Linear profile checking

G65 P9865 Xx [Bb Cc Ff Hh Kk Mm Qq Ss Tt Yy Zz]

Example: G65 P9865 B1. C4. F0.3 H0.5 K0.1 M1. Q90. S4000. T20. X5. Y5. Z50.

Macro inputs

The following inputs are used with this macro:

Bb B1. Check the cutter profile along the negative tolerance profile limit (see the figure on the previous page).

B2. Check the cutter profile along the positive tolerance profile limit (see the figure on the previous page).

B3. Combine both B1. and B2. profile checking in one operation. This is the default if the B input is not used.

B4. B5. B6. These are the same as the B1., B2., and B3. inputs respectively, except that the tool does not retract first. These cycles are suitable for performing secondary profile checks on the same tool.

Cc c = When this input is used, enter the number of cutting edges on the tool. The spindle speed is then automatically adjusted to enable errors on each cutting edge to be checked.

The cycle time using this method is significantly increased, unless the default 0.1 mm/rev (0.004 in/rev) is increased using the F input.

C1. The spindle speed is automatically adjusted for a cutter with a single cutting edge to ensure it is properly checked.

This is also suitable for multiple-tooth cutters, when only the maximum/minimum cutting-edge profile needs to be checked. The cycle time will be faster than checking each individual cutting edge.

Default: The spindle speed is set by either the S input, or by the default value defined in the setting macro O9760 when no S input is used.

Ff f = Feedrate specified as feed/rev for profile checking.

Default: 0.1 mm/rev (0.004 in/rev)

CAUTION: When using I, J and K inputs, it is important that they are specified in alphabetical order.

Kk k = Tolerance value that defines when the cutter profile is out of limits.

Default: 0.025 mm (0.001 in)

M1. Use this input to prevent an alarm being raised when the profile is out of limits.

Ss s = The spindle speed for the cutter.

This value is used for profile checking when no C input is used; otherwise, the spindle speed is adjusted automatically for profile checking.

Default: 3000 r/min

CAUTION: 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.

Tt t = Length offset number.

Default: Current tool number.

Zz z = The retract distance after profile checking.

The tool reference point is retracted to this position.

Default: Retract to the home position.

TIP: To prevent retracting at the end of the cycle, use a Z0. input.

Additional macro inputs for radius profile checking only

Hh h = The height from the tool length offset position to the centre of the radius profile (see the figure on page 68).

Range: ≥ 0

CAUTION: When using I, J and K inputs, it is important that they are specified in alphabetical order.

Jj j = The start angle on the cutter radius for profile checking (see the figure on page 68).

Range: $\geq 0 <$ Q input

Qq	q =	The end angle on the cutter radius. **Range:** ≥ J input ≤ 360°
Rr	r =	The cutter radius value. The R+ value is for convex forms and the R− value is for concave forms.
Xx	x =	Linear distance moved tangentially past the cutter radius profile (see the figure on page 68). **Range:** ≥ 0
Yy	y =	Radial distance to the cutter radius centre. **Range:** ≥ 0

Additional macro inputs for linear profile checking only

Hh	h =	The height to the first profile checking position (see the figure on page 68). This is the height above the tool length offset position. **Range:** ≥ 0
Qq	q =	Angle of the linear profile (see the figure on page 68). **Range:** ≥ 0° ≤ 360°
Xx	x =	Distance along the surface for profile checking (see the figure on page 68). **Range:** ≥ 0
Yy	y =	Radial position to the first profile checking position. **Range:** ≥ 0

Outputs

NOTE: If the B3. or B6. input is used and the tool is found to be out of tolerance during the negative tolerance profile check, the cycle is automatically aborted and does not complete the positive tolerance profile check.

#148 = 0	The profile is in tolerance.
#148 = 1	The profile is out of tolerance during negative tolerance profile checking.
#148 = 2	The profile is out of tolerance during positive tolerance profile checking.

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

91 FORMAT ERROR

96 RPM OUT OF RANGE

97 TOOL OUT OF RANGE

98 RUN-OUT/EDGE MISSING

Example 1: Profile checking a Ø20 mm (0.787 in) ball nose cutter

Assume the profile is to be checked to within ±0.05 mm (0.002 in).

Profile checking starts at 10° on the radius and moves to 90° on the radius. It then moves a further 15 mm (0.591 in) up the diameter of the tool.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9865 R10. X15. J10. K0.05 Q90.	
M30	

Example 2: Profile checking a Ø20 mm (0.787 in) tapered cutter with 10° side taper

Assume the profile is to be checked to within ±0.05 mm (0.002 in).

Profile checking starts at 1 mm (0.0393 in) high and 10.176 mm (0.401 in) radial, and then moves along the taper for 30 mm (1.181 in).

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9865 H1. K0.05 Q80. X30. Y10.176	
M30	

Temperature compensation tracking – O9861

Macro O9861 can be used to calibrate the NCTS system for variations in the spindle axis and/or the radial measuring axis caused by temperature changes in the machine tool.

Run this cycle on a regular basis during machining operations to compensate for growth in the spindle axis and/or radial measuring axis.

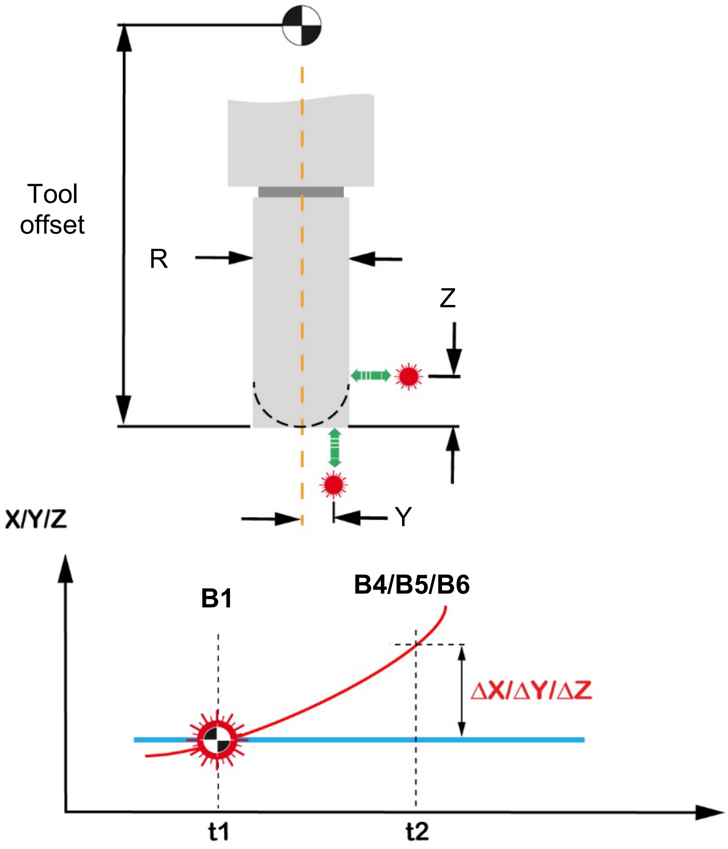


Figure 28 Broken tool detection – plunge checking

Description

The calibration tool must be loaded in the spindle of the machine.

The macro is used as described previously in “Calibrating the NCTS system – O9861” on page 35 but, instead of resetting the calibration data, the beam positions are compared with the original calibration values. The deviation for each axis is then used to adjust the relevant work offset.

NOTE: It is important to use the same calibration tool and input values that were previously used for calibration, except that input B4., B5. or B6. is now used.

After the original reference values have been stored, independent adjustments to the work offset values are ignored and overwritten by the temperature tracking cycle.

Format

Full temperature compensation

G65 P9861 B6. Cc Rr [Hh Kk Qq Ss Tt Yy Zz]
where [ ] denote optional inputs.

Example: G65 P9861 B6. C53. R6. H2. K122. Q5. S2500. T1. Y5. Z5.

Spindle axis temperature compensation

G65 P9861 B4. Cc [Hh Kk Qq Ss Tt Yy]

Example: G65 P9861 B4. C53. H2. K122. Q5. S2500. T1. Y5.

Radial measuring axis temperature compensation

G65 P9861 B5. Cc Rr [Hh Qq Ss Zz]

Example: G65 P9861 B5. C53. R6. H2. Q5. S2500. Z5.

Macro inputs

The following inputs are used with this macro:

B4.		Temperature compensation tracking in the spindle axis. This performs a beam find and length measurement in the spindle axis only.
B5.		Temperature compensation tracking in the radial measuring axis. This performs a beam find and radial measurement only.
B6.		Temperature compensation tracking in both the spindle axis and radial measuring axis. This performs both operations described for inputs B4. and B5.
Cc	c =	Work offset number used to track axis growth due to temperature effects. This must be the same as that used with the B1. input for calibration. For details, see the appendix appropriate to your controller at the end of this guide.
Kk	k =	Reference length of the calibration tool. **Default:** Value in the selected tool offset register.
Hh	h =	Tolerance for the maximum variation of temperature changes. **Default:** No tolerance check.
Qq	q =	Overtravel distance and radial clearance. **Default:** 5 mm (0.197 in)
Rr	r =	Reference diameter of the calibration tool.
Ss	s =	Spindle speed at which calibration takes place. For details, see “Setting data macro variables – O9760” on page 12. **Default:** 3000 r/min

Tt	t =	Tool length offset number. This is the offset location for the calibration tool. It is used for machines where the active tool number cannot be read, causing a TOOL OUT OF RANGE alarm.
Yy	y =	Radial step-over for length calibration. This is the offset across the beam at which measurement takes place. The tool always comes down first on the beam centre line. **Default:** On centre.
Zz	z =	Measuring height on the tool. This determines the height of the beam on the tool at which radial measurement takes place. **Default:** 5 mm (0.197 in)

Outputs

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

#112	Temperature compensation error for length measurement.
#113	Temperature compensation error for radial measurement.
G??	The work offset axis values are corrected.

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see “Error messages and alarms” on page 85.

90 OUT OF TOLERANCE
91 FORMAT ERROR
92 SYSTEM ALREADY TRIGGERED
93 SYSTEM DID NOT TRIGGER

Example

Assume the growth in the spindle axis and radial measuring axis needs to be tracked. The external work offset is used to store the compensations.

```
O????
M6 T1
H1                              Controller-specific. See the appropriate appendix.
G65 P9861 B6. C53. K88. R6.
M30
```

Alternatively, if individual axis tracking is required, use either the B4. or B5. input.

Corner radius measurement cycle – O9867

Macro O9867 is used for measuring the effective tool length, radius/diameter and ball nose or corner radius of a tool.

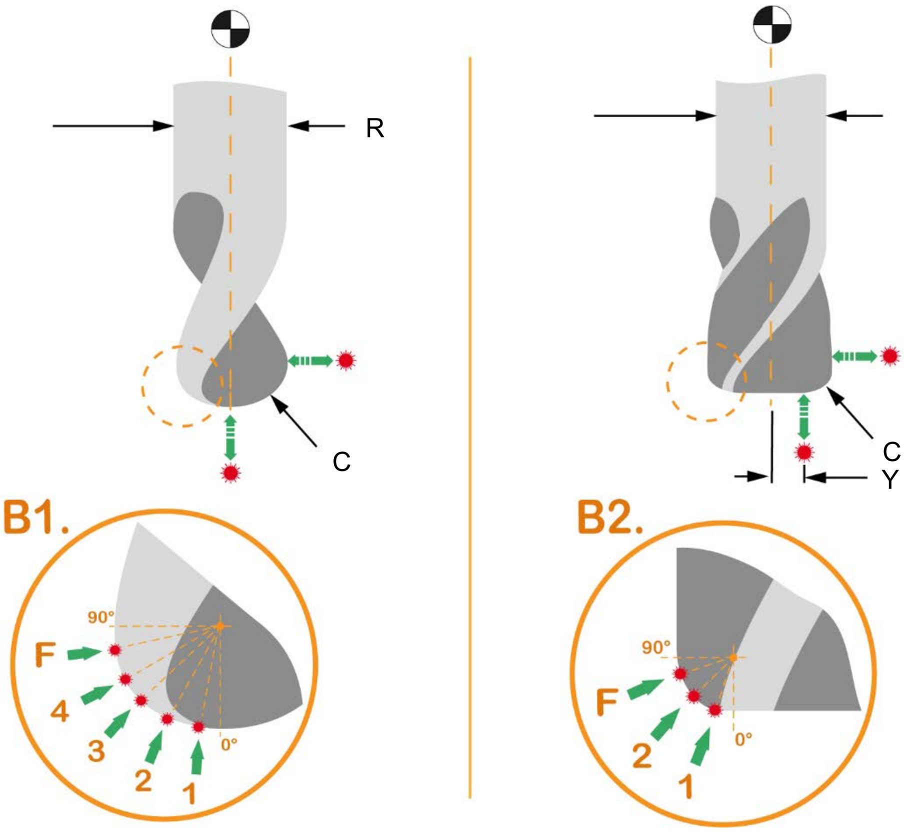


Figure 29 Corner radius measurement cycle

Description

This is the same as the cycle “Tool length and radius setting – O9862”, as described on page 50, except that the corner radius is also measured.

In the case of a ball nose cutter, the effective length and radius/diameter are written into the tool offset register. If the controller has separate wear and geometry registers, the wear register is zeroed and the radius value is placed in the geometry register.

When measuring a cutter with a corner radius, the effective length and radius/diameter of the cutter are written to the tool offset and the measured radius is stored in common variable #102 for reference.

Radial measurement can be made from either side of the beam. The direction is based on the #113 setting in the O9760 macro (see “Setting data macro variables – O9760” on page 12).

Format

G65 P9867 Bb Cc Ff Rr [Dd Ee Hh Ii Jj Mm Qq Ss Tt Uu Vv Yy]
where [ ] denote optional inputs.

Example: G65 P9867 B1. C5. D2. E0.15 F5. H0.2 I0.05 J0.5 M1. Q5. R25. S3000. T1. U1.1 V0.2 Y7.5

Macro inputs

The following inputs are used with this macro:

Bb B1. Measure the length and the radius/diameter of a ball nose cutter. The radius/diameter is calculated from the Ff points.

B1.1 As B1., but using the known tool approach method. This can be used even if #141=0 in setting macro O9760, but an accurate tool length must be present in the appropriate tool offset. **TSM2**

B2. Measure the length and the radius/diameter of a tool with a corner radius. The tool radius/diameter is measured normally, and the corner radius is calculated from the Ff points.

B2.1 As B2., but using the known tool approach method. This can be used even if #141=0 in setting macro O9760, but an accurate tool length must be present in the appropriate tool offset. **TSM2**

Cc c = The nominal corner radius. If using B2., entering an incorrect value will affect measurement accuracy.

Dd d = Diameter offset number.

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

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

Ee e = The tolerance value that defines when the tool radius/diameter is out of tolerance.

MODE	GEOMETRY	WEAR	TOLERANCE CHECK E
No E input	✓	→0	×
E−	×	✓	✓
E	×	×	✓

Default: No tolerance check.

Ff f = The number of corner radius measurements.

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

MODE	GEOMETRY	WEAR	TOLERANCE CHECK H
No H input	✓	→0	✗
H−	✗	✓	✓
H	✗	✗	✓

Default: No tolerance check.

CAUTION: When using I, J and K inputs, it is important that they are specified in alphabetical order.

Ii i = 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.

Jj j = 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.

M1. Tool out of tolerance flag.

Use this flag to prevent a tool OUT OF TOLERANCE alarm being raised.

Qq q = Overtravel distance and radial clearance.

Default: 5 mm (0.197 in)

Rr r = Diameter of the tool.

This is the nominal diameter of the tool.

Ss	s =	The spindle speed at which length and radius/diameter measurement takes place. For details, see “Setting data macro variables – O9760” on page 12. **Default:** 3000 r/min

CAUTION: T input. 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. When using the known tool length approach method, an approximate length value must be present in the tool register to be updated.

Tt	t =	The 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:** Current tool number.
Uu	u =	Reporter printing. This requires the Reporter app (A-5999-4200) to be installed on the machine. Refer to the *Reporter for Fanuc* installation and user guide (Renishaw part no. H-5999-8700) for information on this product and how to use the report feature. **Default:** Not used.
Vv	v =	If using B2., this is the tolerance value that defines when the corner radius is out of tolerance. **Default:** No tolerance check.
Yy	y =	Radial step-over for length setting. This is the offset across the beam at which length measurement takes place. The value must be less than the radius of the tool. The tool always comes down first on the beam centre line. **Default:** On centre.

Outputs

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

- Set tool radius/diameter – ball nose cutter (if using input B1.).
- Set tool length, radius/diameter and corner radius in common variable #102 (if using input B2.).
- #148 out of tolerance flag. This is set when the measured length, radius/diameter or corner radius of the tool is out of tolerance, provided that the Ee, Hh or Vv input is used.

  0 = In tolerance.
  2 = Out of tolerance.

Oversize tool measurement cycle – O9868

Macro O9868 is used for measuring the effective length and radius or diameter of a tool when its diameter is larger than the gap between the NC heads. This cycle is suitable for large or oversize tools such as face mills.

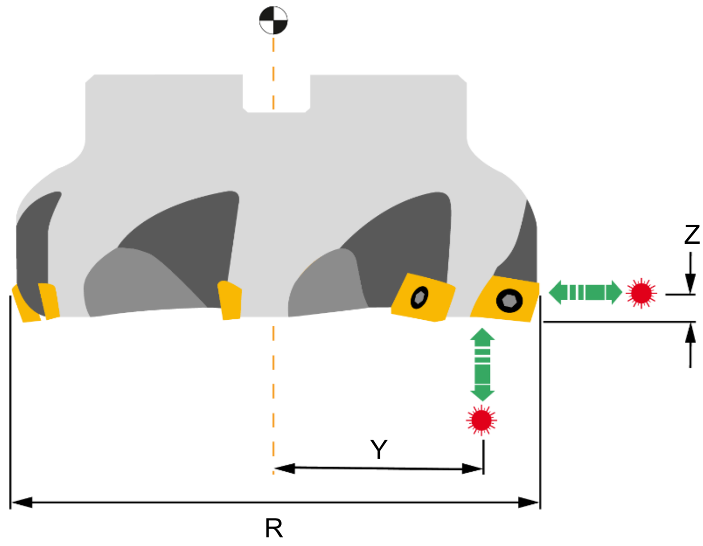


Figure 30 Oversize tool measurement cycle

Description

This is a tool length and radius/diameter measuring cycle for oversize cutters which forces the tool to be measured off-centre. This is controlled by either the diameter input (R) or, if a more specific measuring point is required, the step-over input (Y). At the beginning of the cycle, an internal calculation checks the above inputs and, if deemed safe, the tool will be positioned off-centre at the beginning of the cycle before measurement commences.

Radial measurement is made on the positive side of the laser beam if #113 = 2 or #113 = 1. If #113 = −1, measurement will be made on the negative side of the beam.

Length and radius values are written into the tool offset register. If the controller has separate wear and geometry registers, the wear registers are zeroed and the values are placed in the geometry registers.

NOTE: This cycle is an addition to the standard software package and must be activated in the wizard under the ‘Advanced software settings’ menu.

Format

G65 P9868 A1. Kk Rr Ss [Bb Dd Ee Hh Ii Jj Mm Qq Tt Yy Zz]
where [ ] denote optional inputs.

Example: G65 P9868 A1. B3. D5. E0.02 H0.02 I0.02 J0.025 K157. M1. Q5. R25. S500 T20. Y45. Z6.

Macro inputs

The following inputs are used with this macro:

A1. **Oversize tools.** The Aa input is used to inhibit minimum r/min checking for oversize tools, where it is not necessary to control the spindle speed. This is particularly useful for oversize face mills where the tool cannot be run unsupported at high spindle speeds.

B1. Set the length of the tool.

B2. Measure the radius/diameter of the tool.

B3. Measure the length and the radius/diameter of the tool.

Default: B1.

Dd d = Diameter offset number.

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

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

Ee e = Tolerance value that defines when the tool radius/diameter is out of tolerance.

MODE	GEOMETRY	WEAR	TOLERANCE CHECK E
No E input	✓	→0	×
E−	×	✓	✓
E	×	×	✓

Default: No tolerance check.

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

MODE	GEOMETRY	WEAR	TOLERANCE CHECK H
No H input	✓	→0	×
H−	×	✓	✓
H	×	×	✓

Default: No tolerance check.

CAUTION: When using I, J and K inputs, it is important that they are specified in alphabetical order.

Ii	i =	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.

Jj	j =	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.
Kk	k =	Approximate length of tool.
M1.		Tool out of tolerance flag.
		Use this flag to prevent a tool OUT OF TOLERANCE alarm from being raised.
Qq	q =	Overtravel distance and radial clearance.
		Default: 2 mm (0.0787 in)
Rr	r =	Diameter of the tool.
		This is the nominal diameter of the tool.
Ss	s =	Spindle speed at which length and radius/diameter measurement takes place.
		For details, see “Setting data macro variables – O9760” on page 12.
		Default: Not used.

CAUTION: T input. 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. When using the known tool length approach method, an approximate length value must be present in the tool register to be updated.

Tt t = 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: Current tool number.

Yy y = Radial step-over for length setting.

This is the offset across the beam at which length measurement takes place. The value must be less than the radius of the tool.

Zz z = Measuring height of the tool.

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

Default: 5 mm (0.197 in)

Outputs

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

Set tool length.

Set tool 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 = Out of tolerance (length).
2 = Out of tolerance (radius/diameter).

Alarms

The following alarms may be generated when this cycle is executed. For an explanation of the alarms, see "Error messages and alarms" on page 85.

90 OUT OF TOLERANCE

91 FORMAT ERROR

91 D FORMAT ERROR

91 R FORMAT ERROR

94 SAME T AND D OFFSET

96 RPM OUT OF RANGE

Example 1: Tool length/radius setting

Assume the tool is a 100 mm (3.94 in) diameter face mill.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9868 A1. B3. D21. R100. S400	Set the tool length offset (1) and radius offset (21).
M30	

Example 2: Tool length/radius setting

Assume the tool is an 80 mm (3.15 in) diameter cutter.

O????	
M6 T1	
H1	Controller-specific. See the appropriate appendix.
G65 P9868 A1. B3. D21. R80. T10. S250.	Set the tool length offset (10).
M30	

Error messages and alarms

Error messages are displayed by the error macro program O9769.

When an error state is detected, an error message is displayed on the screen of the controller. Error messages, their meaning and typical actions needed to clear them are described below.

Alarm numbers that are displayed may be controller-specific. The following list shows the default alarm numbers and also the numbers that are displayed when a base number of 500 is added to the default number.

Message 80 MISSING DATA IN O9760
580 MISSING DATA IN O9760

Meaning This alarm is raised if the cycle is run without entering suitable setting data in macro O9760. This usually happens after software installation when the beam alignment macro O9860 is run for the first time.

Action The alarm is raised by monitoring #121 in the setting data macro for a valid value. Check all data at this point to ensure that the default values supplied with the software are suitable for your machine and application.

This is a reset condition.

Message 81 SCATTER TOLERANCE
581 SCATTER TOLERANCE

Meaning At least one measurement in the sample is outside the scatter tolerance limit. The alarm is raised when the retries limit is reached (for details, see “Scatter tolerance checking” on page 18.

Action Review the sample size and scatter tolerance requirements.

Check the reason for poor measurement performance and/or check the sample size and scatter tolerance values. Coolant and swarf on the tool can cause poor measurement performance.

Also check for mechanical movement of the tool/tip insert or NCTS system.

This is a reset condition.

Message 82 TOOL MEASUREMENT ERROR
582 TOOL MEASUREMENT ERROR

Meaning This alarm is raised if the laser system cannot measure the tool successfully. Typically, the software will attempt to measure the tool five times before issuing this alarm. The reason for an unsuccessful measurement could be a system fault or coolant obstructing the measuring process.

Action Reset the alarm.

Check that the system is functioning correctly and increase the number of measurement retries if the system is exposed to high levels of coolant.

This is a reset condition.

Message 83 Z/X/W INPUT TOO LARGE
583 Z/X/W INPUT TOO LARGE

Meaning The tool cannot be measured because the Z, X or W inputs are too large, based on the safe distance below laser (#143 in setting data macro O9760).

Action Investigate/correct #143 in the setting data macro O9760 or reduce the Z, X or W values.

Message 84 HARDWARE FAULT
584 HARDWARE FAULT

Meaning The NCTS system is not working. Either the beam is not transmitting because of a power supply failure, the beam has become blocked or coolant has continuously blocked the beam and a change of state cannot be detected. If the system has just been installed, it is worth checking that the M-codes are working properly and the signals are not inverted.

Action Investigate and correct the fault.

This is a reset condition.

Message 85 NCI-6/SW2-4 INCORRECT
585 NCI-6/SW2-4 INCORRECT

Meaning This alarm is raised if the auto-pulse width option is used and SW2-4 on the NCi-6 interface is set incorrectly.

Action Set switch SW2-4 to the opposite position.

Message 87 FALSE TRIGGER
587 FALSE TRIGGER

Meaning The beam has been cut several times during a measurement move. This may be caused by coolant triggering the beam.

Action Identify and correct the cause of the false triggers. The default number of retries allowed is set to 1. If necessary, change this number in macro O9762 (for details, see “Editing the measure move macro – O9762” on page 23).

This is a reset condition.

Message 90 OUT OF TOLERANCE
590 OUT OF TOLERANCE

Meaning 1 Macro O9862: The measured length or diameter of the tool is out of tolerance (either a positive or negative limit is exceeded). A broken tool or the tool being pulled out of its holder may cause this.

Action 1 Reset and replace the tool, or adjust the tool then reset it.

Meaning 2 Macro O9861: The temperature compensation drift values have exceeded the allowable tolerance.

Action 2 Investigate the cause of excessive movement. The work offset registers used for temperature compensation may have been wrongly adjusted since storing reference values.

This is a reset condition.

Message 91 FORMAT ERROR
591 FORMAT ERROR

Meaning Either a macro input is missing or the value entered is incorrect.

Action Correct the macro input line then run again.

This is a reset condition.

Message 91 D FORMAT ERROR
591 D FORMAT ERROR

Meaning The diameter/radius offset number has not been included in the macro statement.

Action Edit the program.

Message 91 F FORMAT ERROR
591 F FORMAT ERROR

Meaning Either the F input is missing or the value entered is incorrect.

The radial scanning cycle macro O9862 (using the X input) requires a search step size value which is less than half the X search distance.

Action Correct the macro input line then run again.

This is a reset condition.

Message 91 R FORMAT ERROR
591 R FORMAT ERROR

Meaning Either the R input is missing or the value entered is incorrect.

Action Correct the macro input line then run again.

Message 91 Y FORMAT ERROR
591 Y FORMAT ERROR

Meaning The Y step-over has not been included in the macro statement or it has an invalid value.

Action Edit the program.

Message 91 R INPUT TOO LARGE
591 R INPUT TOO LARGE

Meaning The tool cannot be measured because the tool diameter (R input) is too large, based on the distance between the NC heads (#142 in setting data macro O9760).

Action Investigate/correct #142 in the setting data macro (O9760).

Message 92 SYSTEM ALREADY TRIGGERED
592 SYSTEM ALREADY TRIGGERED

Meaning The system was in the triggered state at the start of a measuring move. This may be caused by swarf or coolant interfering with the beam, or the back-off / back-in distance could be too small.

Action Remove the swarf or increase the back-off / back-in distance in O9762 (see page 23).

This is a reset condition.

Message 93 SYSTEM DID NOT TRIGGER
593 SYSTEM DID NOT TRIGGER

Meaning No trigger was registered during a measuring move. This may be caused by swarf or coolant interfering with the beam.

Action Remove the swarf or increase the overtravel distance using the Q input.

This is a reset condition.

Message	94 SAME T AND D OFFSET 594 SAME T AND D OFFSET
Meaning	The same tool offset number has been used for the length and the diameter/radius.
Action	Correct the macro input line then run the macro again. This is a reset condition.

Message	96 RPM OUT OF RANGE 596 RPM OUT OF RANGE
Meaning	The S input entered is less than the permissible value. TSM1 – 800 r/min. TSM2 – 600 r/min if pulse width is 100 ms. TSM2 – 3000 r/min if pulse width is 20 ms.
Action	Correct the macro input line then run the macro again.

NOTE: If drip rejection is switched on, this alarm may also be raised when the spindle speed override is not set to 100%, or when the programmed spindle speed is less than the value specified in #139 in setting data macro O9760.

Message	97 TOOL OUT OF RANGE 597 TOOL OUT OF RANGE
Meaning	Either the size of the cutting tool exceeds the size that is set in variables #110 to #112 inclusive, or the tool offset number/tool number is wrong.
Action	Edit the program. Check the cutter size.

Message	98 RUN-OUT/EDGE MISSING 598 RUN-OUT/EDGE MISSING
Meaning	A cutting edge is missing or damaged, or the tool is eccentric.
Action	Replace or adjust the defective tool, or modify the tolerance value.

Message	99 BROKEN TOOL 599 BROKEN TOOL
Meaning	The tool is out of tolerance.
Action	Replace the defective tool and establish the correct tool offset value.

Appendix A Angled beam software package

About the angled beam software

The optional angled beam software is designed for applications where the laser beam of the NCTS system cannot be mounted parallel to a machine axis. Otherwise, it is similar in operation to the standard software described in this publication. All XY plane moves will be orientated to align with the laser beam.

Additional calculations relating to the angled orientation of the laser beam mean that the macros require more memory and extra calculation time, making the overall cycle times longer when compared to the standard software.

Orientation of the NCTS system

During installation of the NCTS system the orientation of the laser beam must be defined as described below. The angle of the laser beam is automatically set when the beam alignment macro is run.

Available orientations are restricted as follows:

- The Sp (spindle) axis must always be the Z axis (#123 = 3).
- X-axis beam (#121 = 1, #122 = 2).
- Y-axis beam (#121 = 2, #122 = 1).

The beam can be set at an angle to either the X axis or Y axis as follows:

X-axis angle orientation

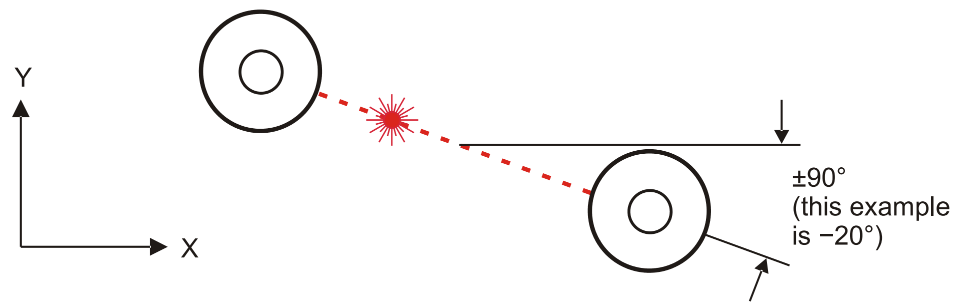


Figure A1 X-axis angle orientation

Y-axis angle orientation

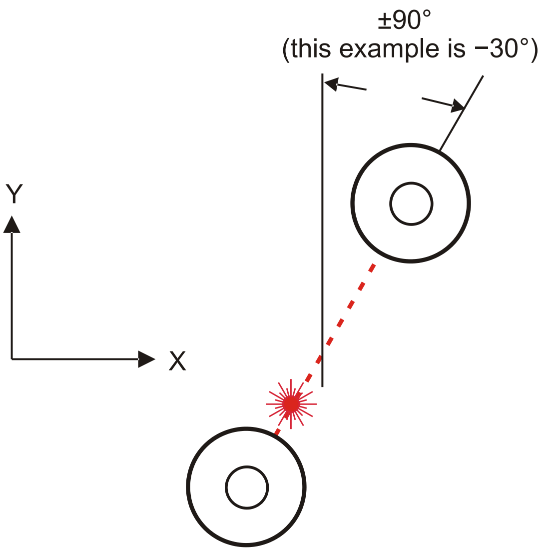


Figure A2 Y-axis angle orientation

Beam alignment cycle – O9860

Additional A input

Aa a = This specifies the approximate angle of the laser beam (see the figure on this page and on the previous page). It is a compulsory input.

Default: 0

TIP: It is recommended that, if you are unsure of the exact angle of the beam, you should enter an estimated A input value and set a small D input span value (say 10 mm [0.394 in]), then run the cycle. The correct angle will be stored in variable #100 at the end of the cycle. A revised angle can then be used and the span increased before running the cycle again.

Outputs

#100 The correct angle of the beam is output to this variable. Other beam positions can be seen in #101 and #102.

#531 (520+11) The angle alignment calibration value for the beam is stored in this variable. The value is used by all other cycles for angular orientation relative to the laser beam (see “Variables: changing the base number address” on page 21).

Cutter radius and linear profile cycle – O9865

Radius profile checking (not fitted)

Radius profile checking has been inhibited, as it is not possible to use the G2/G3 commands for profiling in non-orthogonal planes. Other methods are less practical due to the calculation overhead.

A FORMAT ERROR alarm is raised if the R input is used.

Linear profile checking (fitted)

Linear profile checking is provided. For information about using this cycle, see “Cutter radius and linear profile checking – O9865” on page 68.

Appendix B Using two measuring positions on the laser beam

Renishaw's parallel beam laser technology allows tool measurement to take place at any position along the beam. This feature allows all cycles described in this guide to be used at either of two pre-set locations on the laser beam. This is useful on machines with a partitioned axis or on large machines when it is more convenient to traverse to a local measuring position instead of traversing large distances along the laser beam axis.

The partition value (#[#120+14]) is used to select this option. Typically, the value is a "machine" position as shown in the figure below, but with further software edits this variable can be replaced by a flag or marker supplied by the machine tool builder.

Flag or marker examples include:

- When tool measurement is required in two different machine axes (5-axis machine).
- On a pallet machine with lasers mounted on each pallet.

For further information about two-point measurement, contact your nearest Renishaw office.

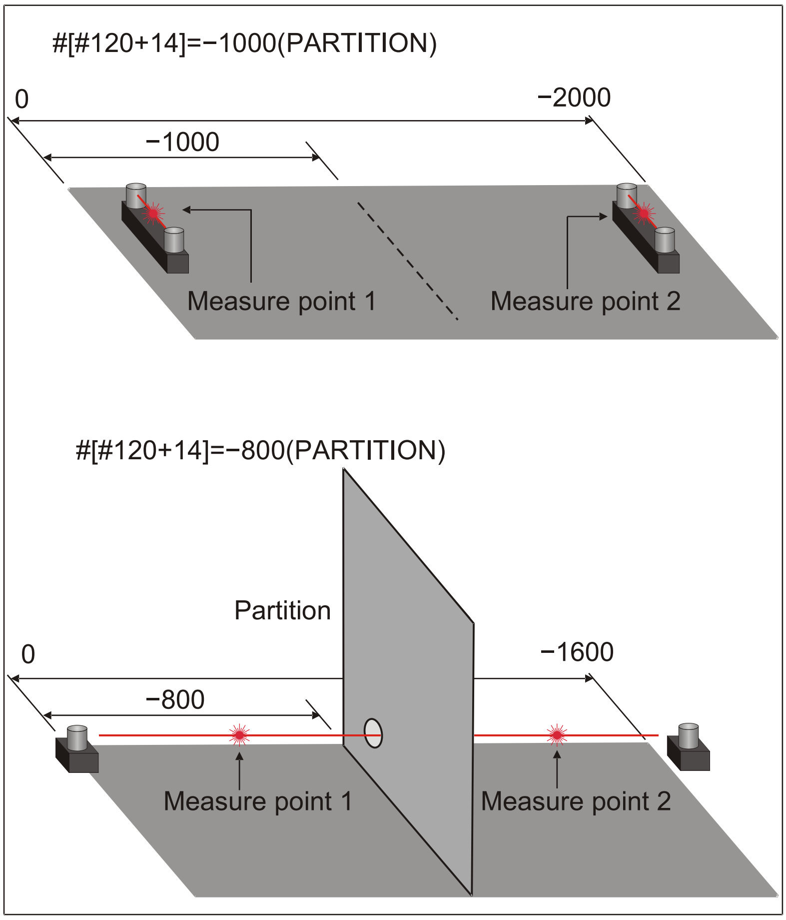


Figure B1 Using two measuring positions on the laser beam

Application

1. Ensure that the axis partition variable has been set to a suitable position along the laser beam axis. This value determines the traverse range assigned for each location on the laser beam.

2. Run the alignment cycle.

   To set up a measuring location, it is necessary to run a normal beam alignment cycle at the first measuring position. Move the master tool/spindle to the position where you want to set tools and then run the alignment cycle. This position is automatically stored and will be used as the measuring position for tools when the spindle/tool is in this traverse range.

3. Now repeat the alignment and calibration cycles at the second measuring position. Note that this must be done in the second traverse range to complete the set-up of two measuring positions.

4. The system is now ready to use. You can run the cycles in the normal way. It is only necessary to ensure that the spindle is in the correct axis range before running a cycle. The cycles will automatically go to the correct place on the laser beam.

Additional variables required to support this application

A range of extended #500 macro variables is needed for this application, so check their availability prior to setting the partition base number. The information described below is based on a typical customisation.

Example: setting macro O9760

#120=520(BASE*NUMBER)	Used to store calibration data for measure point 1 #520 to #534.
IF[#[#120+14]EQ#0]GOTO16	
IF[#[5020+#121]GE#[#120+14]]GOTO16	
#120=535	Used to store calibration data for measure point 2 #535 to #546.

Appendix C Fanuc controller settings

Machine tool controllers supported

The Renishaw NCTS software described in this programming guide is suitable for use on the following controllers:

- Fanuc Series 0, 6, 10–15, 16–21, 30*i* / 31*i* / 32*i* / 300*i* fitted with the macro ‘B’ option.

Type A offsets	One register per tool – no choices	#108 = 1
Type B offsets	Two registers per tool – geometry/wear	#108 = 2
Type C offsets	Four registers per tool – length geometry/wear and radius geometry/wear	#108 = 3

Variables: changing the base number address

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 setting data macro (O9760) will change this.

Why change the base number?

The default setting uses variables #520 to #529 inclusive. This range suits all listed controllers except the Fanuc 6.

When these variables are already used for other purposes, you will need to define a different range. The following suggestions may be helpful:

- Use an additional retained common macro variable option.
- Use spare tool offsets. Use a 2000-series system variable base number; for example, #120 = 2090, to use offsets 90 to 99.

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

If the NCTS software is to be used without any other Renishaw inspection software present, use the default settings, unless #520 to #531 are used for other purposes.

If the NCTS software is to be used in conjunction with other Renishaw software, avoid macro #500-series clashes by changing the base number.

Fanuc 6 systems

NOTE: Fanuc 6 systems are now very dated and are not really suitable for this package, particularly because of the lack of variables and processing speed.

Recent changes have also included the use of G53 codes within the macros to stop read-ahead problems. It will be necessary to strip these manually for this application.

On later software versions, variables #500 to #511 are available.

Set the base number to 500 to use #500 to #509 inclusive.

Alternatively, use tool offsets. Set the base number to 2090 to use offsets 90 to 99 inclusive.

Renishaw vector software packages

Variables #500 to #549 are used, so the base number must be changed.

Renishaw Inspection Plus software

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

Calibrating the NCTS system – O9861

Macro inputs

Cc c = Work offset number used to track axis growth.

When used with the B1. input, it stores the relevant work offset values as the reference position ready for use later.

(See also “Temperature compensation tracking – O9861” on page 73.)

C54 to C59 (G54 to G59)

C53 external work offset

Additional offsets

C101 to C148 (G54.1P1 to G54.1P48)

Extended additional offsets

C101 to C400 (G54.1P1 to G54.1P300)

Example: typical program format

M6 T1

H1 This line is not usually required.

G65 P9861 B1. K88. R6.

M30

Fanuc 0M control and alarm numbers

In “Error messages and alarms”, the list shows the default alarm numbers and the numbers that are displayed when a base number of 500 is added to the default number.

All Fanuc controls, with the exception of the type 0M, display the default alarm numbers. The Fanuc 0M control displays alarm numbers to which a base number of 500 have been added.

Appendix D Yasnac controller settings

For Yasnac controller types MX3, J50, I80 and J300, use the following tool offset options. Set the tool offset type in the setting data macro (O9760) as follows:

MX3 and J50 controllers

Basic 99 offsets	H and D common	#108 = 1
299 Pair option	H and D common	#108 = 1
Basic 99 offsets	H and D separate	#108 = 2
299 Pair option	H and D separate	#108 = 3

I80 and J300 controllers

Basic 99 offsets	H and D common	#108 = 1
299 Pair option	H and D common	#108 = 1
999 Pair option	H and D common	#108 = 1
1199 Pair option	H and D common	#108 = 1
Basic 99 offsets	H and D separate	#108 = 2
299 Pair option	H and D separate	#108 = 3
999 Pair option	H and D separate	#108 = 4
1199 Pair option	H and D separate	#108 = 5

Calibrating the NCTS system – O9861

Macro inputs

Cc c = Work offset number used to track axis growth.

When used with the B1. input, it stores the relevant work offset values as the reference position ready for use later.

C54 to C59 (G54P1 to G59P1)

Additional offsets

C54.02 to C59.05 (G54P2 to G59P5)	MX3 and J50 series.
C54.02 to C59.27 (G54P2 to G59P27)	I80 and J300 series.

Example: typical program format

```
M6 T1
H1                      This line is not normally required.
G65 P9861 B1. K88. R6.
M30
```

Appendix E Meldas controller settings

For Meldas controller types M3, M32, M300-series and M500-series, use the following tool offset options.

Set the tool offset type in the setting data macro (O9760) as follows:

Type A offsets (type 1)	Common type	#108 = 1
Type B offsets (type 2)	Separate type Length – geometry Length – wear Radius – geometry Radius – wear	#108 = 2

Calibrating the NCTS system – O9861

Macro inputs

Cc c = Work offset number used to track axis growth.

When used with the B1. input, it stores the relevant work offset values as the reference position ready for use later.

(See also “Temperature compensation tracking – O9861” on page 73.)

C54 to C59 (G54 to G59)

Additional offsets

C101 to C148 (G54.1P1 to G54.1P48)

Example: typical program format

M6 T1	
H1	This line is not normally required.
G65 P9861 B1. K88. R6.	
M30	

Renishaw plc

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

T +44 (0)1453 524524
F +44 (0)1453 524901
E uk@renishaw.com
www.renishaw.com

apply innovation™

For worldwide contact details, visit www.renishaw.com/contact

H-2000-6333-0M