Programming manual
H-2000-6826-0A-A

RENISHAW
apply innovation™

Renishaw GoProbe cycles for contact tool setting

For use with Fanuc and Meldas controllers

Disclaimer

THIS DOCUMENT IS SUPPLIED AS AN AID FOR USING GOPROBE CYCLES WITH A RENISHAW MACHINE TOOL PROBE. IT IS PROVIDED FREE OF CHARGE OR AT A NOMINAL COST. YOU MUST ENSURE THAT YOUR PROBE IS SET UP CORRECTLY IN ACCORDANCE WITH RENISHAW INSTRUCTIONS AND THAT DATA IS ENTERED ACCURATELY INTO THE MACHINE TOOL CONTROL. RENISHAW PLC MAKES NO WARRANTIES OR REPRESENTATIONS, AND EXCLUDES LIABILITY HOWSOEVER ARISING, REGARDING THE USE OF THIS DOCUMENT.

Trade marks

Renishaw plc. Registered in England and Wales.

Company no: 1106260.

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

Renishaw part no: H-2000-6826-0A-A

Issued: 12.2020

Renishaw customer services – calling a Renishaw subsidiary office

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

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

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

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

EQUIPMENT REGISTRATION RECORD

Please complete this form after the Renishaw equipment has been installed on your machine. Keep one copy yourself and return a copy to your local Renishaw Customer Support office (for contact details, see **www.renishaw.com/contact**). The Renishaw Installation Engineer should normally complete these forms.

MACHINE DETAILS

Machine description .....

Machine type .....

Controller .....

Special control options .....

.....

.....

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

SPECIAL SWITCHING M-CODES (OR OTHER) WHERE APPLICABLE

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

ADDITIONAL INFORMATION

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

CAUTION – Software safety

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

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

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

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

Contents

Before you begin

About this manual. iv
Assumptions. v
Selecting between manual and automated mode vi

Chapter 1 Tool setter set-up

Introduction. 1-1
Five steps of tool setting. 1-2
Constructing a manual single-line command 1-3
Cycle summary 1-4
Tool setter set-up cycles. 1-5
M200: Tool setter check 1-5
M201: Tool setter calibration. 1-6

Chapter 2 Tool setter set-up optional inputs

Introduction. 2-1
Tool setter set-up optional inputs 2-2

Chapter 3 Manual tool setting

Introduction. 3-1
Five steps of tool setting. 3-2
Constructing a manual single-line command 3-3
Input summary 3-4
Output summary. 3-4
Off-centre measurement 3-5
Cycle summary 3-5
Tool setting cycles 3-6
M21: Tool length 3-6
M22: Tool length and diameter 3-7

Chapter 4 Automated tool setting

Introduction. 4-1
Constructing an automated single-line command 4-2
Input summary 4-3
Output summary. 4-3
Off-centre measurement 4-4
Cycle summary 4-4
Tool setting cycles 4-5
M21: Tool length 4-5
M22: Tool length and diameter 4-6
M23: Broken tool detection. 4-7

Chapter 5 Tool setting optional inputs

Introduction. 5-1
Input matrix 5-2
Tool setting optional inputs 5-3

Chapter 6 Alarms

Introduction. 6-1
GoProbe alarms 6-2
Fanuc/Meldas controller alarms 6-3
General alarms. 6-3

Before you begin

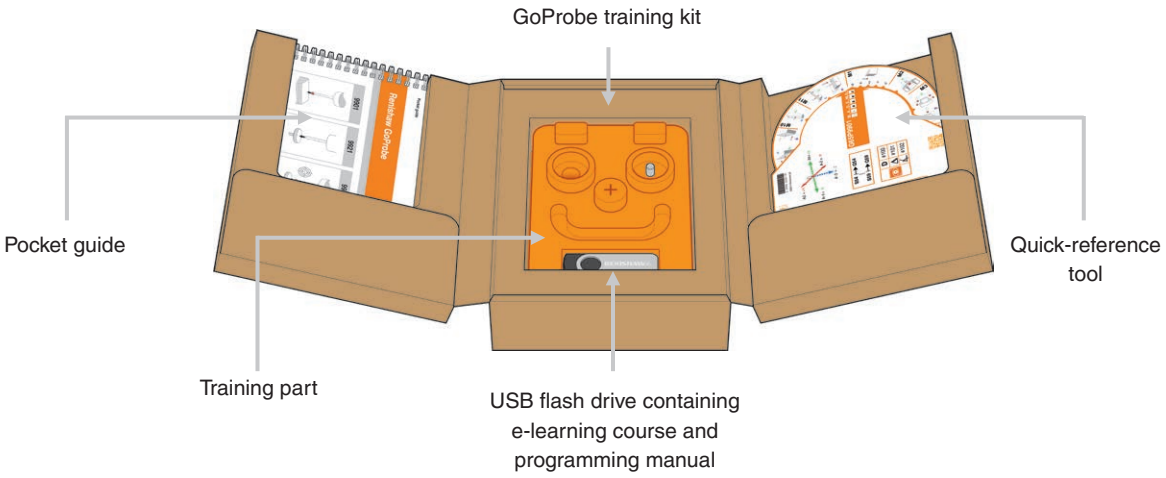
About this manual

This manual contains detailed information on the Renishaw GoProbe tool setting cycles for use on Fanuc controllers. GoProbe provides an easy and intuitive way for customers who are new to probing to get started by using single-line commands for contact tool setting. The software is designed to work with a range of Renishaw contact tool setting probes and to be compatible with a range of Renishaw macro software programs. Use of non-Renishaw contact tool setting probes is not supported.

GoProbe uses easy-to-follow steps to deliver simplified and consistent instructions for contact tool setting cycles. The aim is to guide the user through the basic steps of probing and generating single-line commands for tool setting. Both manual and automated cycles are covered in this manual.

Tool setter set-up cycles include a tool setter check cycle and a tool setter calibration cycle. The tool setter check cycle can be used to establish if the tool setter is set up and ready to use.

This manual should be used in conjunction with the other components of the GoProbe training kit and the GoProbe app. It is recommended that users complete the self-study e-learning course before using the GoProbe cycles.



Download on the App Store

GET IT ON Google Play

Assumptions

It is assumed that users of this manual have previous experience of using Fanuc or Meldas CNC machine tool controllers and are familiar with the controller interface.

NOTE: All examples in this programming manual use metric values.

Selecting between manual and automated mode

The differences between running tool setting cycles in manual and automated mode are highlighted below. The aim of this section is to help you decide which scenarios are best suited to using the contact tool setter in manual or automated mode.

Manual mode:

- The user manually moves (jogs) the tool to the start position, approximately 10 mm above the tool setter.
- The single-line command is manually entered in MDI mode.
- The tool offset for the selected tool is updated.
- At the end of the cycle the tool returns to the spindle axis start position.

NOTE: Tool setter set-up cycles are only available in manual mode.

Automated mode:

- The cycle automatically moves the tool to the start position.
- The single-line command can be used in MDI mode, or it can be embedded into a cutting or probing program.
- The tool offset for the selected tool is automatically updated.
- At the end of the cycle the tool automatically returns to the reference (home) position.
- Additional, optional inputs provide a range of in-process tool setting options.

Chapter 1

Tool setter set-up

Introduction

This chapter describes the five steps of tool setting and the single-line command for each tool setter set-up cycle. These cycles include a tool setter check and a calibration cycle. Calibration data storage is also covered. This chapter also introduces the optional inputs that are available within the tool setter set-up program.

Key points for tool setter set-up cycles:

- The tool setter check cycle enables the user to check that the tool setter is ready for use.
- The tool setter calibration cycle enables the user to set up their tool setter so that it is ready for use.
- The user manually moves (jogs) the tool to the start position.
- The single-line command is manually entered in MDI mode.
- At the end of the cycle the tool returns to the start position.

Calibration data storage:

Calibration data for the tool setter is stored in machine common variables. The exact location depends on the setting of #120, the variable base number defined during software installation.

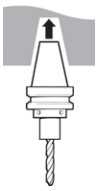
Example: If #120 is set to 520

#520 = Stylus Z height

Five steps of tool setting

The tool setting process is split into five easy steps.

1



Bring the tool into the spindle.

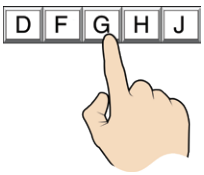
2

Manually jog the tool to the start position (typically 10 mm above the tool setter disc).

3

Press MDI.

4



Enter the single-line command.

5

Press cycle start.

Constructing a manual single-line command

G65P99□□	M□.D□.;

1. Choose program (G65P9921 for tool setting).
2. Choose and insert cycle number (M).
3. Choose and insert relevant inputs. Inputs will vary depending on the cycle.

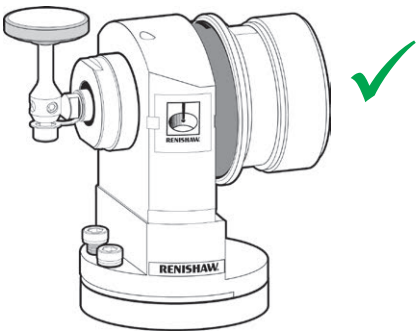
Example: To check that the tool setter is ready for use, the single-line command is:

G65 P9921 M200.;

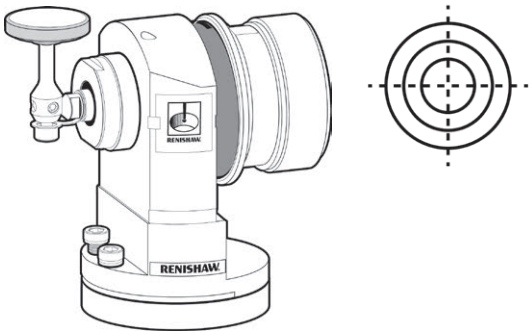
Program Cycle

NOTE: The single-line command shown above includes a space between each input for visual clarity. This space should be omitted when entering the command into the CNC machine tool controller.

Cycle summary



M200: Tool setter check



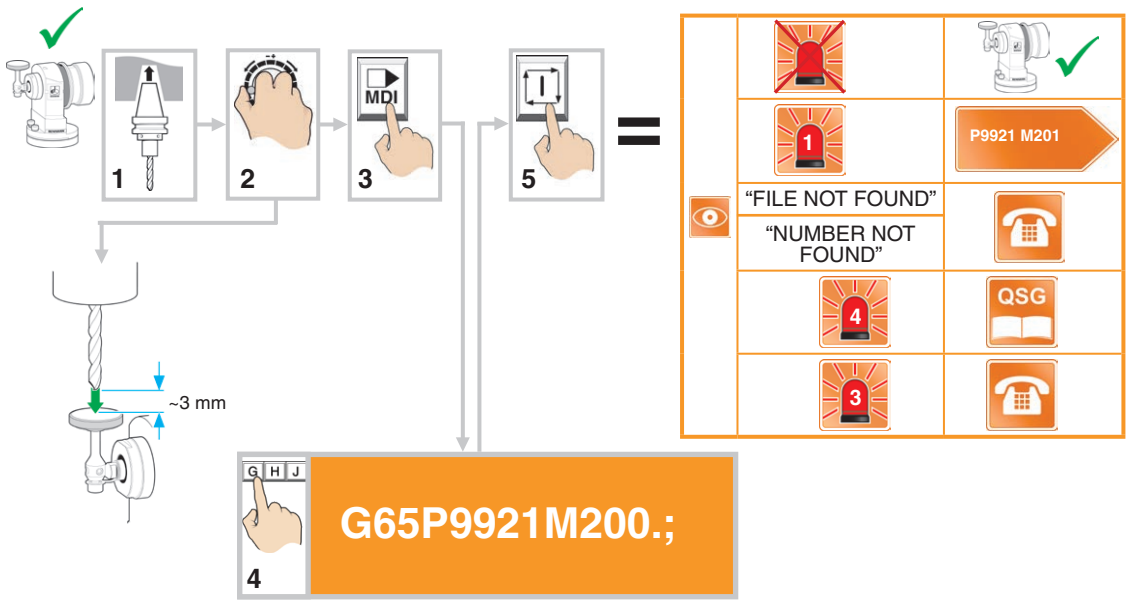
M201: Tool setter calibration

Tool setter set-up cycles

M200: Tool setter check

- Checks that the tool setter is ready for use
- The tool start point and path are shown in the diagram below

Standard – manual mode – checking the tool setter is ready for use



Advanced – manual mode – optional inputs

F = First touch feedrate

The addition of further, optional, inputs enhances the functionality of tool setting set-up cycles. For more information on these inputs, see Chapter 2, "Tool setter set-up optional inputs".

Tool setter system check ensures that:

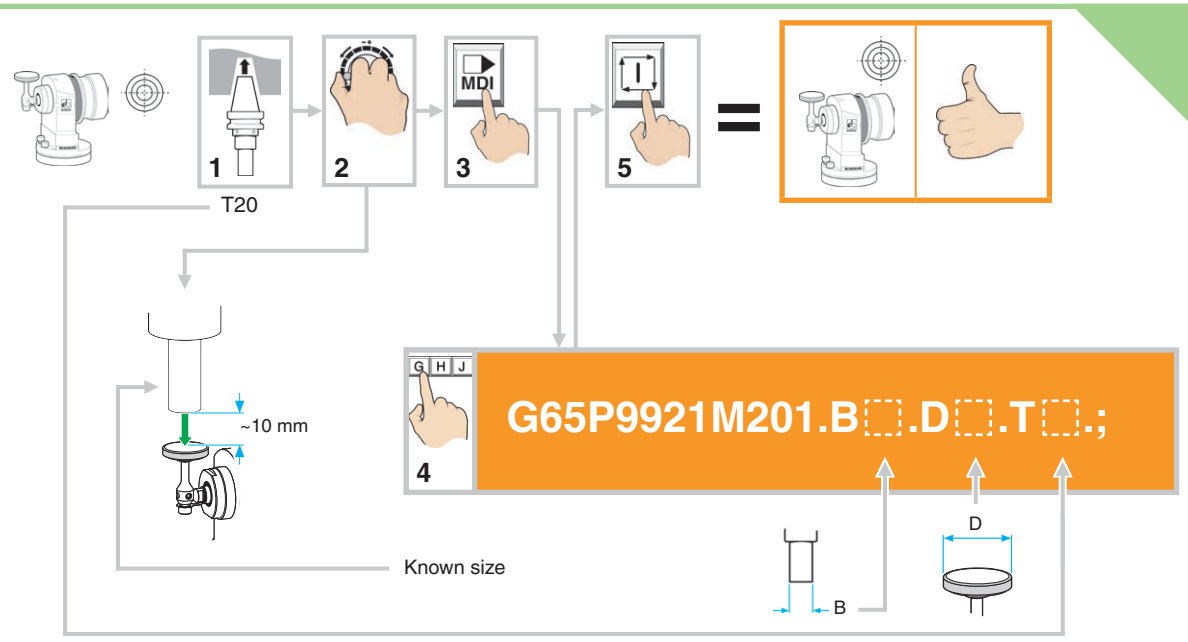
- GoProbe cycles are loaded.
- The tool setter can be switched on and off.
- The interface is working.
- SKIP is working (machine stops when the stylus is deflected and the tool setter is triggered).
- Calibration data exists.
- The tool setter is level.

NOTE: Upon completion of the cycle, the message "Probe basic check passed" is displayed. If the cycle fails, an alarm will be raised. Please refer to Chapter 6, "Alarms" for more information.

M201: Tool setter calibration

- Calibrates the tool setter
- The calibration tool start point and calibration tool path are shown in the diagram below

Standard – manual mode – calibrating the tool setter



Compulsory inputs

B = Diameter of calibration tool or tool of known size

D = Diameter of tool setter disc stylus

T = Tool number for calibration tool or tool of known size

NOTE: The tool should be of known length and diameter.

Advanced – manual mode – optional inputs

A1. = Z calibration only

F = First touch feedrate

Q = Overtravel distance

Z = Distance from the top face of the tool setter stylus disc for radius measure

The addition of further, optional, inputs enhances the functionality of tool setting set-up cycles. For more information on these inputs, see Chapter 2, “Tool setter set-up optional inputs”.

Chapter 2

Tool setter set-up optional inputs

Introduction

This chapter describes the optional inputs that are available for tool setter set-up cycles.

Tool setter set-up optional inputs

A1. = Z calibration only

Applies to cycle M201 only.

Calibrate only in Z.

F = First touch feedrate

Applies to cycles M200 and M201.

This is the speed at which the tool is moved onto the tool setter stylus. The default speed is 1000 mm/min.

Example: F500. changes the speed to 500 mm/min.

Q = Overtravel distance

Applies to cycle M201 only.

Defines the maximum distance to travel past the target surface.

The default overtravel distance is 5 mm in all axes. If these values are unsuitable, then using the Q input will allow the probe to adjust the distance it travels whilst searching for a surface.

Example: Q20. sets an overtravel distance of 20 mm.

Z = Distance from the top face of the tool setter stylus disc for radius measure

Applies to cycle M201 only.

This is the distance between the top face of the stylus disc and the point at which the probe touches the side of the stylus disc when performing a radius measurement. The default value is 5 mm.

Example: Z10. changes the distance the probe moves from the top face to the point where it touches the side of the stylus disc to 10 mm.

Chapter 3

Manual tool setting

Introduction

This chapter describes the five steps of tool setting and the single-line command for each of the available manual tool setting cycles. It also introduces the optional inputs and cycle outputs available within the manual tool setting program.

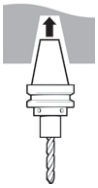
Key points for manual tool setting cycles:

- The user manually moves (jogs) the tool to the start position, approximately 10 mm above the tool setter.
- The single-line command is manually entered in MDI mode.
- The tool offset for the selected tool is automatically updated.
- At the end of the cycle the tool returns to the spindle axis start position.

Five steps of tool setting

The tool setting process is split into five easy steps.

1



Bring the tool into the spindle.

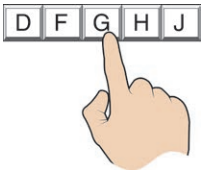
2

Manually jog the tool to the start position (typically 10 mm above the tool setter disc).

3

Press MDI.

4



Enter the single-line command.

5

Press cycle start.

Constructing a manual single-line command

G65P99□□	M□.D□.;

1. Choose program (G65P9921 for tool setting).
2. Choose and insert cycle number (M).
3. Choose and insert relevant inputs. Inputs will vary depending on the cycle.

Example: To measure the length and diameter of a tool with a nominal diameter of 5 mm, the single-line command is:

G65P 9921 M22. D5.;

Program Cycle Diameter

NOTES:

The single-line command shown above includes a space between each input for visual clarity. This space should be omitted when entering the command into the CNC machine tool controller.

Within this programming manual, all whole numerical values contained within single-line command examples end with a decimal point to ensure that the CNC controller reads these values as mm (or inch) rather than µm (or µin). 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. Where the numerical is a decimal fraction, for example 50.002, there is no need to include a further decimal point at the end of the value.

Further examples:

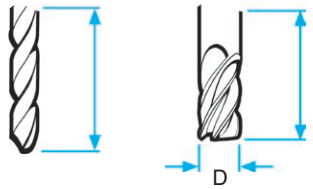
D = 5.5, the single-line command is G65P9921M22.D5▪5;

D = 5.05, the single-line command is G65P9921M22.D5▪05;

Input summary

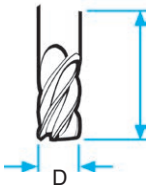
G65P9921	M□.D□.;

M = Cycle selection



M21 M22

D = Tool diameter



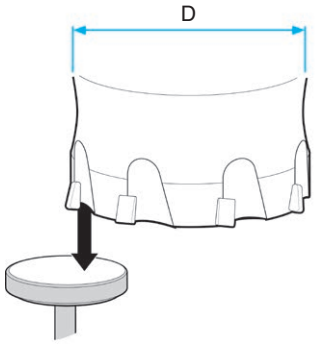
Output summary

OFFSET

NO.	(LENGTH) GEOM	(LENGTH) WEAR	(RADIUS) GEOM	(RADIUS) WEAR
001	72.084	0.125	0.000	0.000
002	76.397	0.000	0.000	0.000
003	86.356	0.000	24.893	0.000
004	74.383	0.000	4.033	0.000
005	230.114	0.000	6.600	0.000
006	105.732	0.000	4.859	0.000
007	101.724	0.000	4.998	0.000
008	145.000	0.000	0.000	0.000
009	163.455	0.000	0.000	0.000

After the cycle is completed, the tool offset value is automatically updated.

Off-centre measurement



This method should be used in situations where the diameter of the tool is greater than the stylus disc diameter of the tool setter. The D input is used in the single-line command for both the M21: Tool length cycle and the M22: Tool length and diameter cycle.

If D is greater than the diameter of the stylus disc:

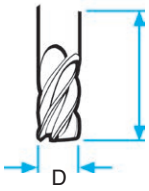
- Position (jog) the edge of the tool to a position approximately 10 mm above the centre of the tool setter disc.
- The tool will spin. (The appropriate spin speed is automatically determined by the D value.)

NOTE: Although the D input is compulsory in most scenarios when measuring off centre, it can be omitted during a length measurement cycle (M21) in order to stop the tool from spinning when required.

Cycle summary

Tool setting cycles

M21: Tool length measure



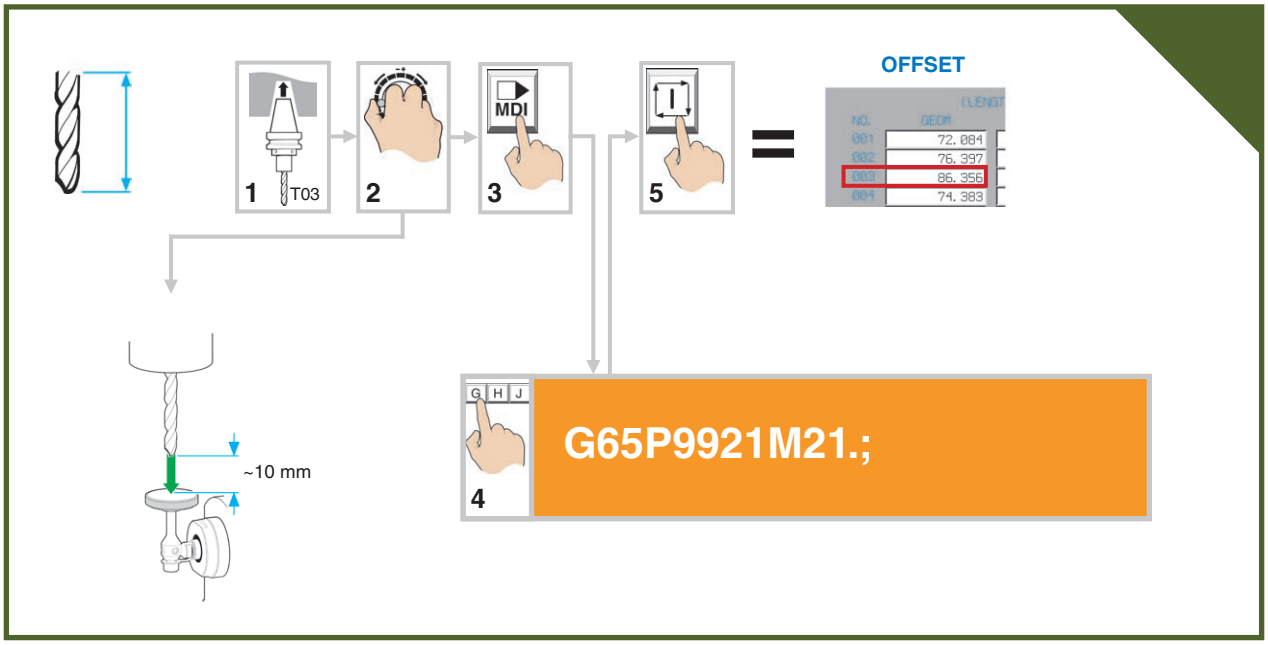
M22: Tool length and diameter measure

Tool setting cycles

M21: Tool length

- The tool offset value for length is updated for the selected tool
- The probe start point and probe path are shown in the diagram below

Standard – manual mode – measuring the tool length



Advanced – manual mode – optional inputs

D or D– = Tool diameter. Use if the tool diameter is greater than the stylus disc.
(See “Off-centre measurement” on page 3-5.)
F = First touch feedrate
K = Experience value (length)
Q = Overtravel distance
T = Tool number to be updated

The addition of further, optional, inputs enhances the functionality of tool setting cycles.
For more information on these inputs, see Chapter 5, “Tool setting optional inputs”.

NOTE: Using the D– input will rotate the tool in an M3 direction if required.

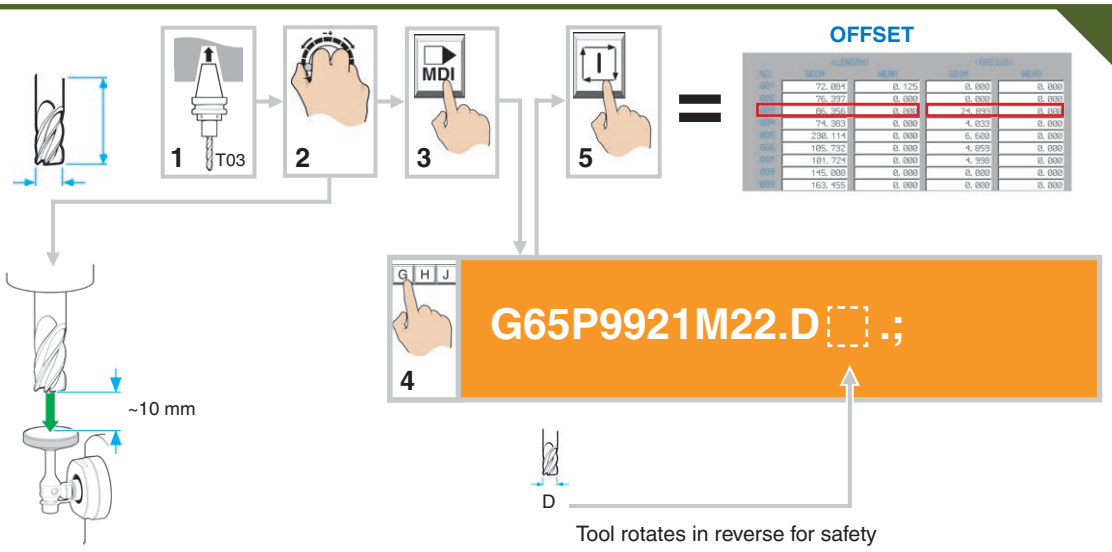
Automated

For information on how to automate this cycle, see Chapter 4, “Automated tool setting”.

M22: Tool length and diameter

- The tool offset values for length and diameter are updated for the selected tool
- The probe start point and probe path are shown in the diagram below

Standard – manual mode – measuring the tool length and diameter



Compulsory inputs

D or D– = Tool diameter. If the tool diameter is greater than the stylus disc, see “Off-centre measurement” on page 3-5.

NOTE: Using the D– input will rotate the tool in an M3 direction if required.

Advanced – manual mode – optional inputs

E = Diameter input for type A tool offset
F = First touch feedrate
J = Experience value (radius)
K = Experience value (length)
Q = Overtravel distance
R = Radial clearance
T = Tool number to be updated
Z = The measuring position from the stylus disc top face

The addition of further, optional, inputs enhances the functionality of tool setting cycles. For more information on these inputs, see Chapter 5, “Tool setting optional inputs”.

Automated

For information on how to automate this cycle, see Chapter 4, “Automated tool setting”.

This page is intentionally left blank.

Chapter 4

Automated tool setting

Introduction

This chapter describes how to create the single-line command for each automated tool setting cycle. It also introduces the optional inputs and cycle outputs available within the automated tool setting program.

Key points for automated tool setting cycles:

- The cycle automatically moves the tool to the start position.
- The single-line command can be used in MDI mode, or it can be embedded into a cutting or probing program.
- The tool offset for the selected tool is automatically updated.
- At the end of the cycle the tool automatically returns to the reference (home) position.
- Additional, optional inputs provide a range of in-process tool setting options.

Constructing an automated single-line command

G65P99□□	M□.C0.D□.;

1. Choose program (G65P9921 for tool setting).
2. Choose and insert cycle number (M).
3. Insert C0. (automated mode).
4. Choose and insert relevant inputs. Inputs will vary depending on the cycle.

Example: To measure the length and diameter of a tool with a nominal diameter of 5 mm, the single-line command is:

G65P 9921 M22. C0. D5.;

Program Cycle Automate Diameter

NOTES:

The single-line command shown above includes a space between each input for visual clarity. This space should be omitted when entering the command into the CNC machine tool controller.

Within this programming manual, all whole numerical values contained within single-line command examples end with a decimal point to ensure that the CNC controller reads these values as mm (or inch) rather than µm (or µin). 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. Where the numerical is a decimal fraction, for example 50.002, there is no need to include a further decimal point at the end of the value.

Further examples:

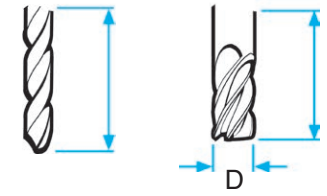
D = 5.5, the single-line command is G65P9921M22.C0.D5▪5;

D = 5.05, the single-line command is G65P9921M22.C0.D5▪05;

Input summary

G65P9921	M□.C0.D□.;

M = Cycle selection

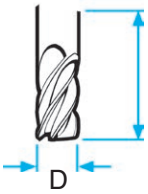


M21 **M22**

M23

C0. = Automated

D = Tool diameter



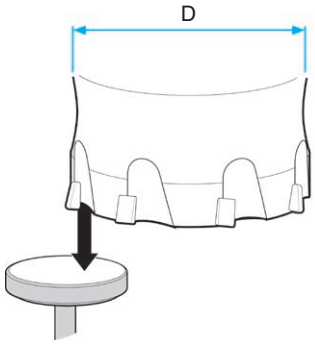
Output summary

OFFSET

NO.	(LENGTH) GEOM	(LENGTH) WEAR	(RADIUS) GEOM	(RADIUS) WEAR
001	72.084	0.125	0.000	0.000
002	76.397	0.000	0.000	0.000
003	86.356	0.000	24.893	0.000
004	74.383	0.000	4.033	0.000
005	230.114	0.000	6.600	0.000
006	105.732	0.000	4.859	0.000
007	101.724	0.000	4.998	0.000
008	145.000	0.000	0.000	0.000
009	163.455	0.000	0.000	0.000

After the cycle is completed, the tool offset value is automatically updated.

Off-centre measurement



In situations where the diameter of the tool is greater than the stylus disc diameter of the tool setter, the D and Y inputs must be used in the single-line command for both the M21: Tool length cycle and the M22: Tool length and diameter cycle. The M23: Broken tool detection cycle only requires a D input.

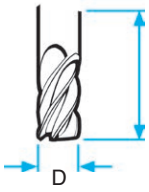
If D is greater than the diameter of the stylus disc:

- The tool is automatically driven into position. The Y input is the nominal tool length. It is used to position the tool above the stylus prior to measurement.
- The tool will spin. (The appropriate spin speed is automatically determined by the D value.)

Cycle summary

Tool setting cycles

M21: Tool length measure



M22: Tool length and diameter measure

M23: Broken tool detection

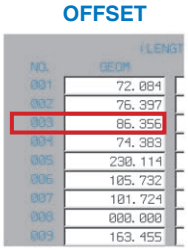
Tool setting cycles

M21: Tool length

- The tool offset value for length is updated for the selected tool

Standard – automated mode – measuring the tool length

=



Compulsory inputs

C0. = Automates the cycle

Advanced – automated mode – optional inputs

A = Number of teeth

D or D– = Tool diameter. Use if the tool diameter is greater than the stylus disc. (See “Off-centre measurement” on page 4-4.)

F = First touch feedrate

K = Experience value (length)

Q = Overtravel distance

T = Tool offset number to be updated (use if pre-select system is present)

V = Long tool / short tool feedrate override

Y = Nominal tool length

The addition of further, optional, inputs enhances the functionality of tool setting cycles. For more information on these inputs, see Chapter 5, “Tool setting optional inputs”.

NOTE: Using the D– input will rotate the tool in an M3 direction if required.

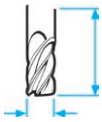
Manual

For information on manually operating this cycle, see Chapter 3, “Manual tool setting”.

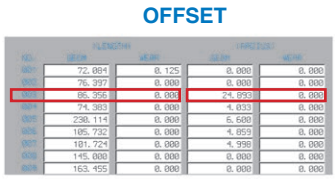
M22: Tool length and diameter

- The tool offset value for length and diameter are updated for the selected tool

Standard – automated mode – measuring the tool length and diameter



=



Tool rotates in reverse for safety

Compulsory inputs

D or D– = Tool diameter. If the tool diameter is greater than the stylus disc, see “Off-centre measurement” on page 4-4.

NOTE: Using the D– input will rotate the tool in an M3 direction if required.

Advanced – automated mode – optional inputs

A = Number of teeth
E = Diameter input for type A tool offset
F = First touch feedrate
J = Experience value (radius)
K = Experience value (length)
Q = Overtravel distance
R = Radial clearance
T = Tool offset number to be updated (use if pre-select system is present)
V = Long tool / short tool feedrate override
Y = Nominal tool length
Z = The measuring position from the stylus disc top face

The addition of further, optional, inputs enhances the functionality of tool setting cycles. For more information on these inputs, see Chapter 5, “Tool setting optional inputs”.

Manual

For information on manually operating this cycle, see Chapter 3, “Manual tool setting”.

M23: Broken tool detection

- An alarm will be raised if the tool is broken or pulled out from its holder (if the tool length value is shorter or longer than the stored value)
- This cycle is only available in automated mode

Standard – automated mode – broken tool detection

Compulsory inputs
C0. = Automates the cycle
T = Tool offset number

Advanced – automated mode – optional inputs

D or D– = Tool diameter. Use if the tool diameter is greater than the stylus disc.
(See “Off-centre measurement” on page 4-4.)
F = First touch feedrate
H = Size tolerance
Q = Overtravel distance
U1. = Alarm suppression flag (can be used with H input)

The addition of further, optional, inputs enhances the functionality of tool setting cycles.
For more information on these inputs, see Chapter 5, “Tool setting optional inputs”.

NOTE: Using the D– input will rotate the tool in an M3 direction if required.

This page is intentionally left blank.

Chapter 5

Tool setting optional inputs

Introduction

This chapter describes the optional inputs that are available for use in tool setting cycles.

Input matrix

G65P9921		A Number of teeth	D/D– Tool diameter	E Diameter (type A tool offset)	F First touch feedrate	H Size tolerance (automated only)	J Experience value (diameter or radius)	K Experience value (length)	Q Overtravel	R Radial clearance	T Tool offset number	U1. Alarm suppression flag	V Long tool / short tool feedrate override	Y Nominal tool length (automated only)	Z Distance from top of tool setter disc stylus for diameter/radius measure
		Tool setting optional inputs													
M21	Length measure	✓	✓ (use when the value of D or D– is greater than the diameter of the tool setter stylus disc)		✓			✓	✓		✓		✓	✓	
M22	Length and diameter measure	✓		✓	✓		✓	✓	✓	✓	✓		✓	✓	✓
M23	Broken tool detection		✓ (use when the value of D or D– is greater than the diameter of the tool setter stylus disc)		✓	✓ (default 0.5 mm)			✓			✓			

Tool setting optional inputs

A = Number of teeth

Applies to cycle M21 and M22.

If the approach method (#141) is set to 0 or 2, this input can be used to optimise the cycle time.

Default value: 1.

CAUTION: Do not exceed the number of teeth present on the tool, as damage to the stylus or tool could occur.

Example: A4. optimises the search feedrate for a 4 toothed tool.

D = Tool diameter (tool spins in M4 direction)

Applies to cycles M21 and M23.

The diameter of the tool. This optional input is used when the tool diameter is greater than the diameter of the tool setter stylus disc.

Example: D30. indicates a tool diameter of 30 mm and will spin the tool in an M4 direction.

D– = Tool diameter (tool spins in M3 direction)

Applies to cycles M21 and M23.

The diameter of the tool. This optional input is used when the tool diameter is greater than the diameter of the tool setter stylus disc, and in instances where the tool is required to spin in an M3 direction.

Example: D–30. indicates a tool diameter of 30 mm and will spin the tool in an M3 direction.

E = Diameter input for Fanuc with type A tool offsets

Applies to cycle M22 only.

Fanuc controllers with type A tool offsets do not have a separate tool offset column for diameter/radius. When using this type of controller, the E input is used to define the tool offset number to be updated. For controllers with type B or type C tool offsets, the current tool is set as default.

Example: E32. stores the diameter/radius in tool offset number 32 for type A tool offsets.

F = First touch feedrate

Applies to all cycles.

This is the feedrate for the tool search function. The default feedrate is 1000 mm/min.

Example: F500. changes the feedrate to 500 mm/min.

H = Size tolerance

Applies to cycle M23.

This is the tolerance value of the broken tool detection cycle. If the measured value is outside the tolerance, an alarm will be raised. The default tolerance value is 0.5 mm.

Example: H0.2 sets a tolerance value of 0.2 mm.

J = Experience value (diameter or radius)

Applies to cycle M22 only.

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

Example: J0.25 subtracts 0.25 mm from the diameter/radius of the measured tool size.

K = Experience value (length)

Applies to cycles M21 and M22.

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.

Example: K0.15 subtracts 0.15 mm from the measured length of the tool.

Q = Overtravel distance

Applies to all cycles.

Defines the maximum distance to travel past the target surface.

The default overtravel distance is 5 mm in all axes. If these values are unsuitable, then using the Q input will allow the probe to adjust the distance it travels whilst searching for a surface.

Example: Q20. sets an overtravel distance of 20 mm.

R = Radial clearance

Applies to cycle M22 only.

This is the clearance value when moving down the side of a stylus prior to measurement. The default value is 5 mm.

Example: R7.5 sets a radial clearance of 7.5 mm.

T = Tool offset number

Applies to cycles M21 and M22.

This is the tool offset number to be updated. If the machine is fitted with a pre-select function, a T input is advisable.

Example: T10. will update the tool offset for tool number 10.

U1. = Alarm suppression flag

Applies to cycle M23.

This is used with an H input and prevents an out-of-tolerance alarm being raised.

The following output is always set:

#146 = broken tool flag for tool setter

#[base number + 3] = broken tool flag for LTS

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

Example: U1. will suppress the alarm when the measured tool is out of the tolerance value set by the H input.

V = Long tool / short tool feedrate override

Applies to cycle M21 and M22.

This option is only available when the approach method (#141) is set to 2. It overrides the calculated long tool / short tool feedrate when the tool is rotating.

CAUTION: The long tool / short tool feedrate for rotating tools is calculated by the software to protect the tool and stylus. Increasing this feedrate could result in damage to the system.

Example: V500. changes the long tool / short tool feedrate to 500 mm/min.

Y = Nominal tool length

Applies to cycles M21 and M22.

This is used with the D input during off-centre measurement or when the tool diameter is greater than the stylus disc diameter. The value entered is used when positioning the tool above the stylus prior to measurement. An incorrect input could cause a collision between the tool and tool setter.

Example: Y110. indicates the nominal tool length is 110 mm.

Z = The measuring position from the stylus disc top face

Applies to cycle M22 only.

This is the Z-axis measurement position – measured from the top face of the stylus disc – at which diameter/radius measurement takes place. The default position is 5 mm below the stylus disc face.

Example: Z20. changes the measuring position to 20 mm below the stylus disc face.

Chapter 6

Alarms

Introduction

This chapter contains information on how to identify an alarm number that may be displayed on a Fanuc or Meldas controller, the meaning and likely cause of each alarm, and the typical actions you need to take to clear the fault. Alarms include:

- GoProbe alarms
- Fanuc/Meldas controller alarms
- General alarms

GoProbe alarms

The GoProbe alarms are shown below.

CALIBRATION REQUIRED

Calibration required

SKIP FUNCTION TEST FAILED

SKIP function test failed

DISC STYLUS NOT LEVEL

Disc stylus not level (> 15 µm)

FILE*NOT*FOUND/NUMBER*NOT*FOUND

File not found (software not loaded)

Fanuc/Meldas controller alarms

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

Example: 92(PROBE*OPEN) is alarm 592.

General alarms

Format: #3000 = 80 (MISSING*DATA*IN*O9750)

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

Action: Edit the settings data macro O9750 and restart the cycle.

Format: #3000 = 81 (TOOL*PULL*OUT)

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

Action: Inspect, adjust and remeasure the tool.

Format: #3000 = 90 (OUT*OF*TOLERANCE)

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

Action: Inspect and replace the tool if necessary and remeasure the tool.

Format: #3000 = 91 (FORMAT*ERROR)

Cause: B, or D and T inputs are missing from the call line for calibration cycle M201.

Action: Edit the input line then run the cycle again.

Format: #3000 = 92 (PROBE*OPEN)

Cause: The probe is triggered unexpectedly.

Action: Edit the back-off factor in #(#120+6).

Format: #3000 = 93 (PROBE*FAIL)

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

Action: Correct the error and restart the program.

Format: #3000 = 95 (Y*INPUT*OUT*OF*RANGE)

Cause: The specified Y input value is outside the “Long tool/Short tool” range set in settings data macro O9750.

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

Format: #3000 = 97 (TOOL*OUT*OF*RANGE)

Cause: This alarm is generated if the T input has a negative value or if there is no T input during an M201 cycle.

Action: Edit the input line then run the cycle again.

Format: #3000 = 98 (D*INPUT*MISSING)
#3000 = 82 (Y*INPUT*MISSING)

Cause: A compulsory D or Y input is missing.

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

Format: #3000 = 99 (BROKEN*TOOL)

Cause: This alarm is generated if the tool is broken or has been pulled out of the tool holder. The following output is always set:

#146 = broken tool flag for 3D tool setter
#[base number + 3] = broken tool flag for LTS
1 = Broken tool
2 = Long tool

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

This page is intentionally left blank.

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

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

www.renishaw.com

RENISHAW
apply innovation™

For more information on GoProbe, visit
www.renishaw.com/goprobe

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

H-2000-6826-0A

Issued: 12.2020

Part no. H-2000-6826-0A-A