

RENISHAW PRODUCTIVITY+ FOR MASTERCAM TUTORIAL

June 2018

RENISHAW PRODUCTIVITY+ FOR MASTERCAM TUTORIAL

June 2018

© 2018 CNC Software, Inc. – All rights reserved.

Software: Mastercam 2019

Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

<http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx>

Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.PDF) – installed with each release – includes the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

Introduction	5
General Tutorial Requirements	5
Installation Requirements	5
Minimum Requirements	5
Usage Prerequisites	5
Support	7
When to Use Productivity+	7
Post Processing Requirements and Installs	7
Productivity+ in Mastercam 2019	9
Features and Benefits	9
Tool height setting	9
Tool breakage checking	9
Part setup	9
Inspection/reporting of basic geometric features	9
Remachining	9
Associativity	9
Using Productivity+ in Mastercam	9
Probe Operations	10
Posts	12
Generating Probe Operations	13
Setting Up Your Probing Module and Stylus	14
Tool Setting	15
Measuring	18
Machine Updates	18
Remachining	21
The Logic Statement	22
Reporting	24
Productivity+ and Mastercam Posts	27
Posting Probe Operations	27

Conclusion	31
Mastercam Resources	31
Contact Us	31

INTRODUCTION

This introduction to Mastercam's probing application using Renishaw's Productivity+™ in Mastercam to combine probing with machining. It introduces you to some sample operations, addresses requirements, and discusses how probe operations and posts work in Mastercam.

WARNING: Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.

Estimated time to complete this tutorial: 3 hours

General Tutorial Requirements

All Mastercam 2019 tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel:
www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

Installation Requirements

To take advantage of Productivity+ in Mastercam, you must have a Renishaw probe, which is operational and accurate, installed on your machine tool, as well as functioning Inspection+ software. Productivity+ software generates the necessary NC code.

Minimum Requirements

To run Mastercam 2019 with Renishaw Productivity+, you must have at least Mastercam 2019 Mill (Renishaw Productivity+ is not compatible with Mastercam Lathe, or Mastercam Mill-Turn. For more information on Mastercam product levels, please visit <http://www.mastercam.com/Solutions>.

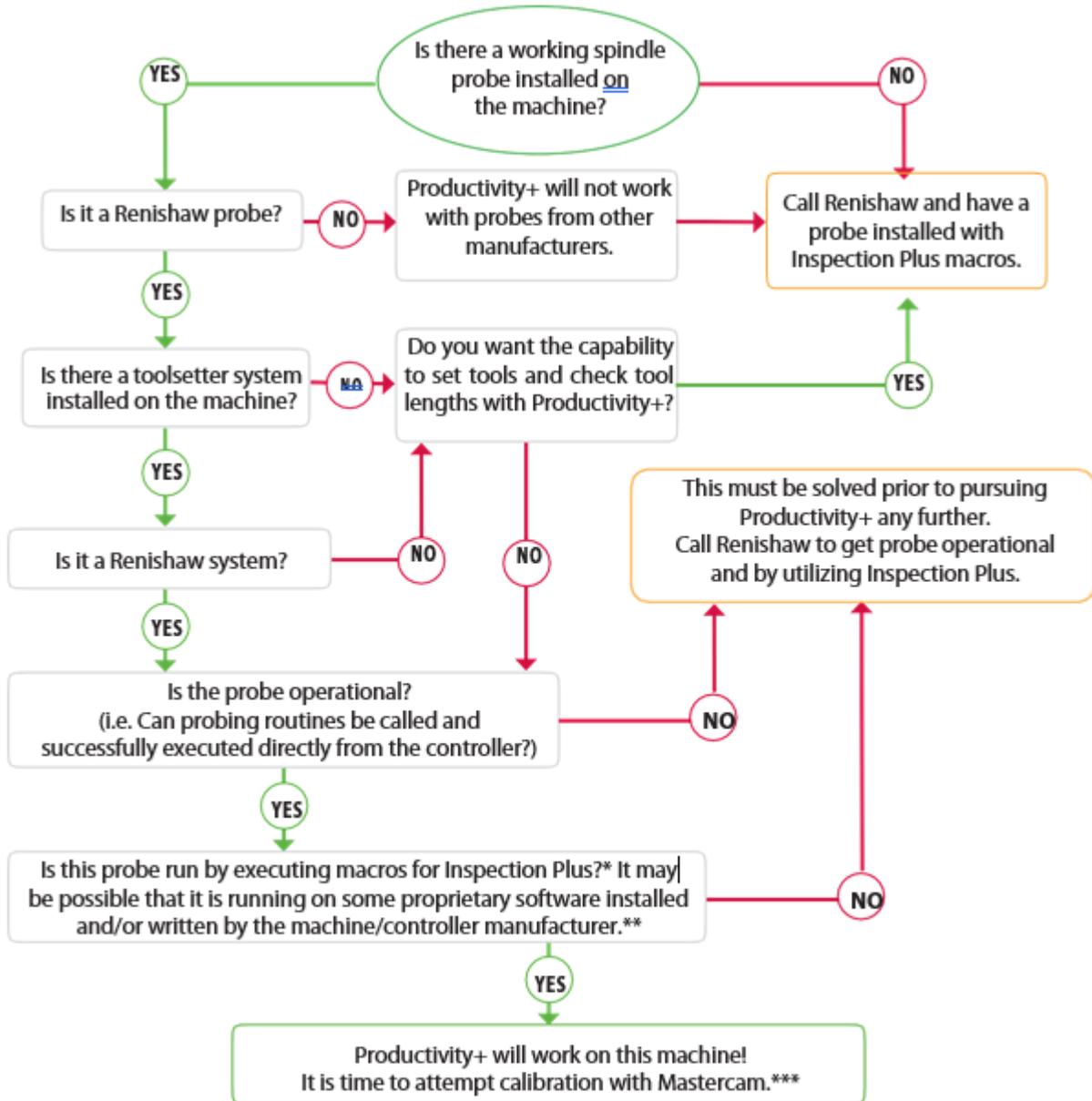
Usage Prerequisites

In addition to experience working in Mastercam 2019, it is helpful to have a working knowledge of the Renishaw Productivity+ suite.

When you use Productivity+, you need to alter the MP post and then post that output with a Renishaw post. CNC Software, Inc. provides a HAAS post and supporting files with the install. Please contact your local Mastercam Reseller about post customization for other machine tools.

Use Renishaw Productivity+ with only Mill machine definitions.

Can My Machine Run PRODUCTIVITY+?



* If both the work probe and toolsetter are to be used then both must be operational with Inspection Plus in order for both to work.

** Okuma, Hurco & Mori (possibly others) are all available with Renishaw probes that will operate with the manufacturer's software and Inspection Plus is not required. It must be purchased and integrated as an option either with the purchase of the machine or after-the-fact.

*** Ensure that probes have been properly setup to Renishaw specifications (stylus indicated in within tolerance) and the probes are reading accurately using the Inspection Plus macros.

Support

Productivity+ is a product of the Renishaw company. CNC Software, Inc. and its approved Resellers support only the integration and usage of Renishaw Productivity+ within Mastercam 2019.

When to Use Productivity+

Use Renishaw Productivity+ to take simple measurements and alignments. This will allow you to set up and validate your parts at the machine tool. This can lead to better process control and reporting of the performance of your machine tool. Productivity+ can be used with 3- and 4-axis mills (vertical and horizontal) and multiaxis (3+2) machines.

You can use Productivity+ to perform such automated functions as:

- Part setup
- Tool height and diameter settings
- Adjust wear and geometry values of tools in the control
- Tool breakage checking
- Inspection/reporting of basic geometric features (points, lines, planes, circles, web, and pocket sizes)
- Measuring machined areas and dynamically setting offsets to remachine

Notes:

- Productivity+ supports only Renishaw probes.
- Productivity+ will not probe complex free-flowing surfaces.
- The current version does not support continuous 4- or 5-axis motion.

Post Processing Requirements and Installs

To use Renishaw Productivity+, a Renishaw machine file (`.RenMF`) must be installed on your workstation. Additionally, your Mastercam 2019 post (`.PST`) must be configured to support Productivity+.

When you post a part with a properly configured Mastercam 2019 post, it interacts with the `.RenMF` file to create both probing and metal-cutting code. If your Mastercam 2019 post processor has not been properly configured, probing operations will not be incorporated in the NC output.

Please see [Productivity+ and Mastercam Posts](#) for more information.

PRODUCTIVITY+ IN MASTERCAM 2019

Renishaw Productivity+ allows you to use to program and simulate a Renishaw probe for your machine tool. It performs simple measurements and alignments, and allows the set up and validation of your parts at the machine tool. This can lead to better process control and reporting of the performance of your machine tool.

Features and Benefits

The following lists the key benefits of using the automated features of Productivity+ in Mastercam 2019:

Tool height setting

Productivity+ can measure your tool height and automatically update your tool parameters.

Tool breakage checking

Productivity+ allows you to program your tool setter for tool breakage detection. If breakage is detected, an error is automatically output to the control.

Part setup

Probing operations can calculate the position of part components and features and automatically update and/or set your machine offsets.

Inspection/reporting of basic geometric features

Productivity+ can provide automatic inspection and reporting of basic/prismatic geometric features (i.e., points, lines, planes, circles, web and pocket sizes), which can be easily set up as an operation.

Note: At this time, Productivity+ in Mastercam 2019 cannot be used on types of features with free-flowing surfaces, such as ports.

Remachining

Productivity+ in Mastercam 2019 provides automated probing of your part during the machining process. This can update your tooling parameters on features that might be undersized, set the offsets (i.e. diameter and wear offsets), and remachine the feature. This functionality also detects features that are oversized (out of spec) and cannot be remachined, which can be set to end the process, eliminating wasted machine time.

Associativity

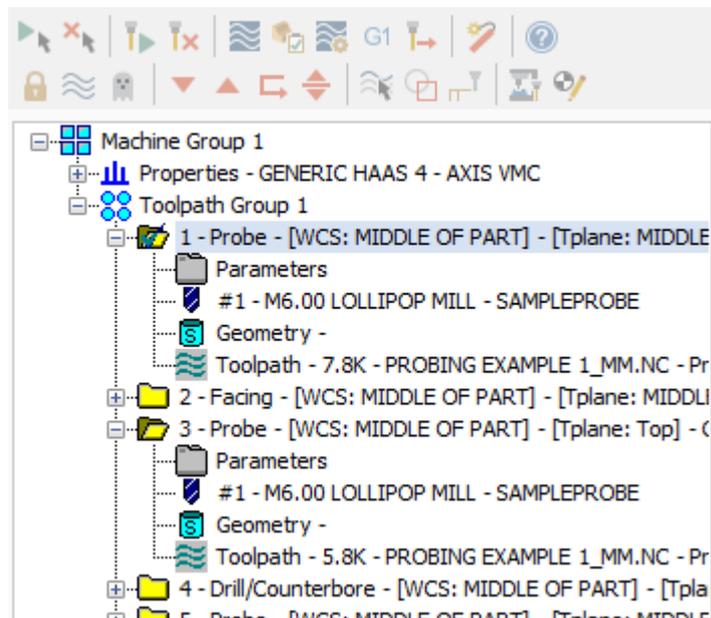
Probe toolpaths are associative to part geometry. If a part's geometry is moved or changed, the probe toolpaths will update accordingly. Editing part geometry or moving a part in space, may result in dirty toolpaths. Regenerate the dirty operations to update the toolpaths.

Using Productivity+ in Mastercam

CNC Software, Inc. has incorporated Renishaw's Productivity+ interface into Mastercam 2019, allowing you to integrate process control into your CNC programs. To generate a probe cycle, you create and edit a probe operation, just like you would any other toolpath operation in Mastercam 2019. Then, like any other toolpath, Mastercam outputs the probe operation to your machine. However, unlike other Mastercam toolpath operations, probe output requires a two-stage posting process.

Probe Operations

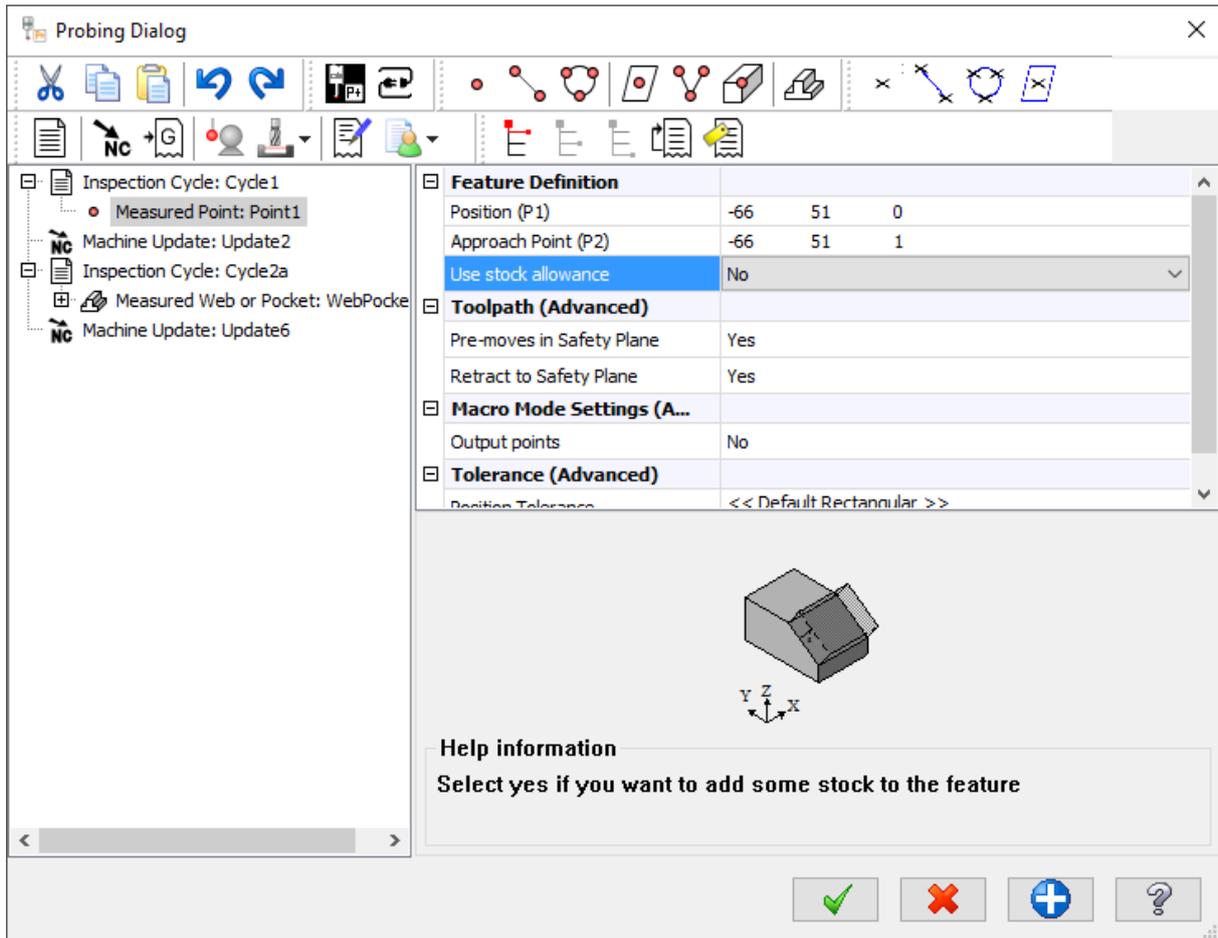
The probe operation is stored in Mastercam 2019's Toolpaths Manager. Edit the operation, as needed, to make adjustments. The operation is a snapshot of your model at the time the operation was created.



To create a probe operation, select **Probe** from the **Mill Toolpaths** or **Router Toolpaths** contextual tab.

Probe operation parameters are set in the **Probing Dialog** box. Your selections generate the following:

- Probe calibration
- Tool settings
- Inspection cycles for measuring different aspects of your part
- Machine update cycles to adjust to the recorded measurements
- Reports
- Logic statements to vary the response based on measurements
- Label statements to use as reference points in conjunction with the probe operation tree



See Mastercam's Help for more information on the functions and options available in this dialog box.

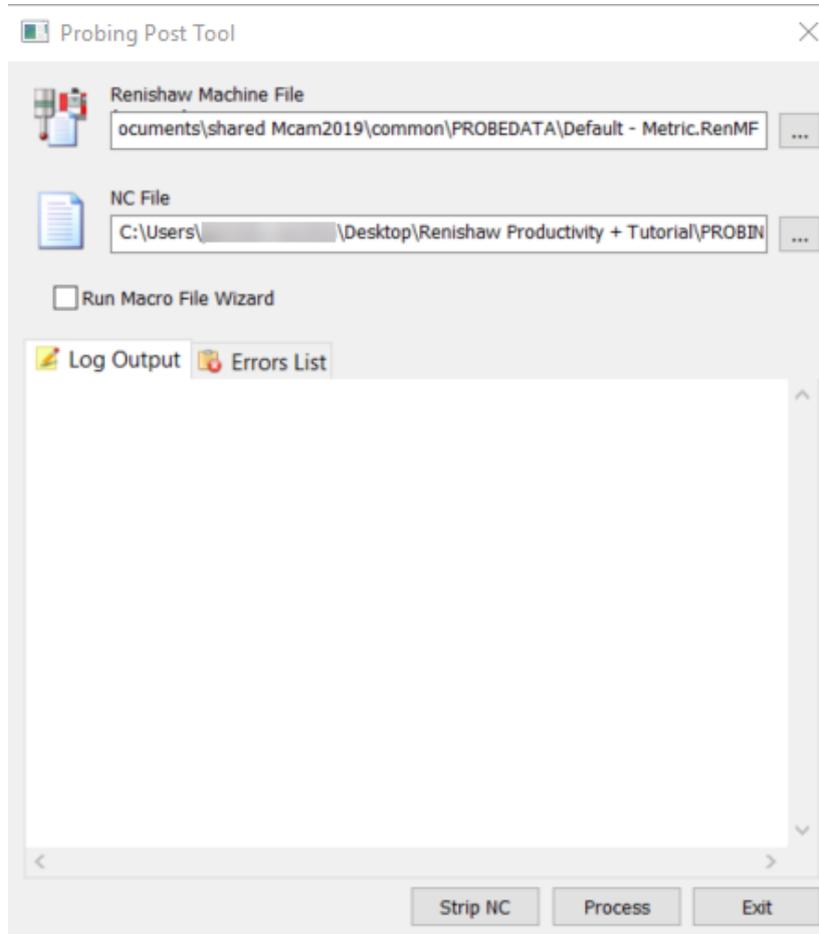
Use Renishaw's embedded Help for more detailed information on the parameters you set for each function. Hover over a setting, or click it to display Help information at the bottom of the dialog box. If the Help topic contains more information than will fit in that section, you can either use the vertical scrolling bar, or resize the Probing Dialog window.

Note: A warning symbol or a red X to the left of a setting indicates a missing or erroneous value/parameter. Hover over the setting for more information.

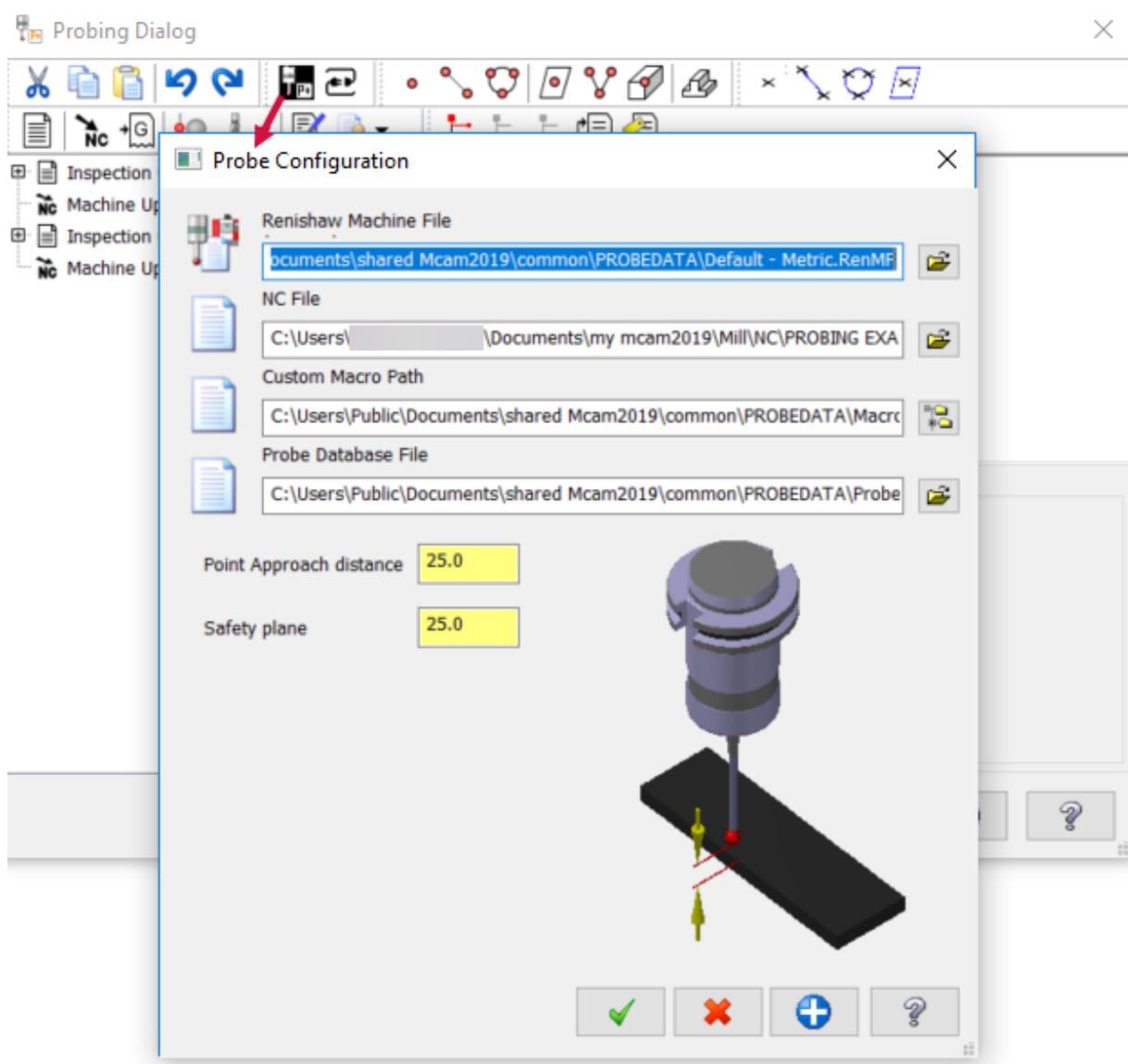
Feature Definition	
Point 1 (P1)	-71.7321 0 0
Point 2 (P2)	71.7321 0 0
X Width (w)	0
Inside/Outside	Outside
Use stock allowance	No
Toolpath	
Start Clearance (I1)	0
End Clearance (I2)	0
Inspection Depth (d)	-5
Measure Ends	Yes

Posts

Probing operations require an additional processing step before they are posted to your machine. This additional processing incorporates instructions from the selected Renishaw machine file (.RenMF) and is initiated from the Probing Post Tool dialog box which displays after you accept the settings in Mastercam's standard Post processing dialog box. (See Mastercam's Help for more information on the Post Processing dialog box.)



In order to successfully create probing output, you must have a post processor with probing enabled and a .RenMF customized for your specific machine. Select the .RenMF after you have initiated the post from the Probing Post Tool dialog box, or when you configure your probe operation from the Probe Configuration dialog box.



For more information on post files and posting probe operations in Mastercam 2019, see [Productivity+ and Mastercam Posts](#).

Generating Probe Operations

Productivity+ is used for tool and part setup, in-process gauging, and post process inspection. It uses a measuring probe—via a Mastercam probe operation—on a machine tool to determine fixture offsets, orientation, and critical dimensions.

This guide uses the part file `PROBINGEXAMPLE`, which is provided with the tutorial, to demonstrate some of the available probe operations. The post file for the Generic Haas 4-axis machine (`Generic Haas 4X Mill.pst`) and the mill machine definition (`Generic Haas 4X Mill MM.mcam-mmd`) is also provided.

The post file must be placed in the following file location:

C:\Users\Public\Public Documents\shared Mcam2019\mill\Posts

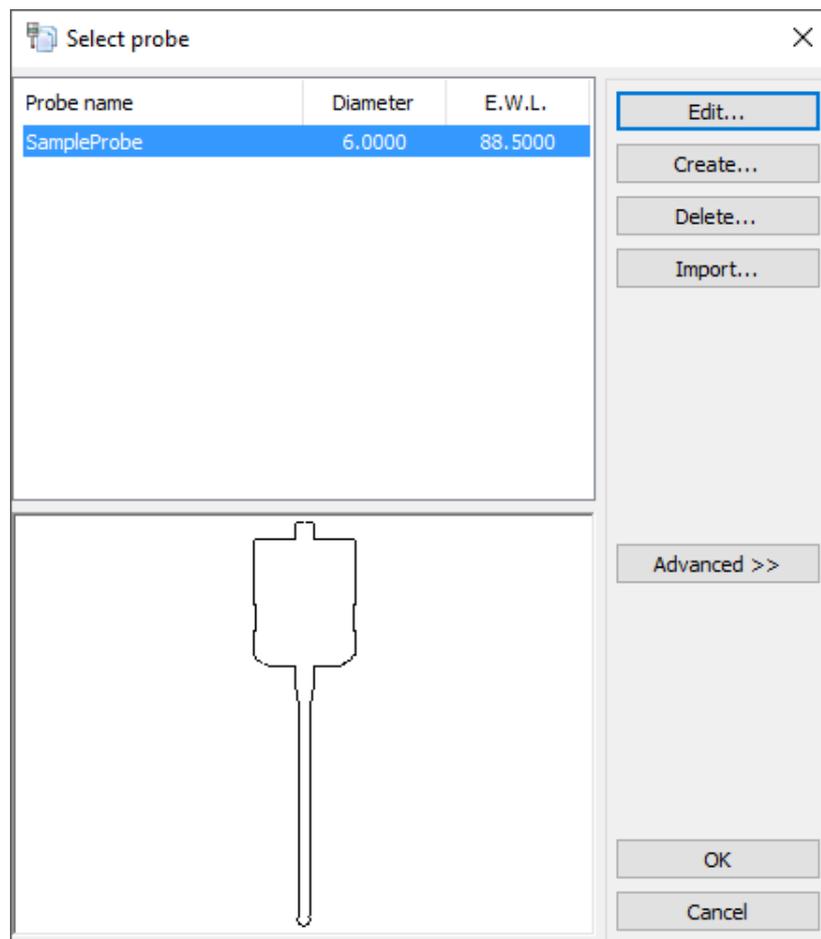
The mill machine definition must be placed in the following file location:

C:\Users\Public\Documents\shared Mcam2019\CNC_MACHINES

Setting Up Your Probing Module and Stylus

Begin by opening the part `ProbingExample`. Save the part as `ProbingExample_XXX`, where `XXX` are your initials.

Before you can create a probing operation, your probe and stylus information needs to be communicated to the Renishaw software within Mastercam 2019. The **Select probe** dialog box opens the first time you create a probe operation.



The buttons on the right-side of the dialog box do the following:

- **Edit:** Opens the **Edit Probe** dialog box, where you can edit the properties of the current probing module and stylus.
- **Create:** Opens the **Edit Probe** dialog box, where you can create a new probing module and stylus.
- **Delete:** Deletes the selected probing module from the database (completing this action requires confirmation).
- **Import:** Imports a probing database (.RenPD) from another source.

- **Advanced:** Expands the **Select probe** dialog box to display properties of the current probing module and stylus.

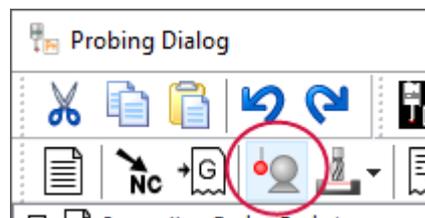
Configure your probe and stylus by editing the following parameters. Be sure to change the Tool change ID, which is intentionally set to require input. (You do not need to do this as part of the tutorial. This is a general procedure that you can refer to as needed.)

☐ General	
Probe name	SampleProbe
Tool change ID	2
Tool offset	1
Probe module	OMP40 ▾
Probe type	Touch Trigger
Transmission type	Optical
Stylus	A-5000-3709
Extension total length	0
Probe switch-on alarm	Yes
☐ Stylus Parameters	
Ball diameter (A)	6
Length (B)	50
Stem diameter (C)	4.5
Holder straight taper length (E)	7
Holder length (F)	17
Thread type	M4

1. Change the default **Probe name** to more easily identify your machine/probe installation.
2. Set the **Tool change ID** to the tool slot that your machine uses for the probe.
3. Click the **Probe module** drop-down to choose the probing module installed on your machine.
4. Click the **Stylus** drop-down to choose the stylus attached to your probe module.
5. Click **OK** to save your probe information to the database (.RenPD).

The **Probing Dialog** dialog box opens after you have configured your probe and stylus information.

Note: Calibrate your probe with your controller as well as with your Productivity+ software before attempting to use it in production. Create a calibration cycle from within Mastercam 2019 by selecting **Probe Calibration** on the Probing Dialog toolbar.



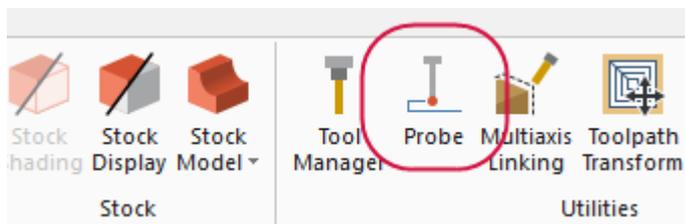
Tool Setting

Prior to machining, if you have a tool setter on the machine, use Productivity+ to check tool offsets. Use the probe's breakage detection during processing to determine aberrations in tool diameter and length.

Measuring Tool Length and Diameter

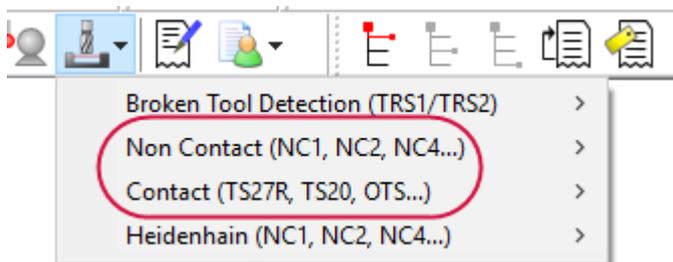
In one probe operation, you can establish tool offsets for all tools in the tool magazine. The procedure below describes the basic steps that are required to create a tool setting cycle. Specific settings are determined by your machine control and tool setter.

1. To create a probe operation, select **Probe** from the **Mill Toolpaths** or **Router Toolpaths** contextual tab.

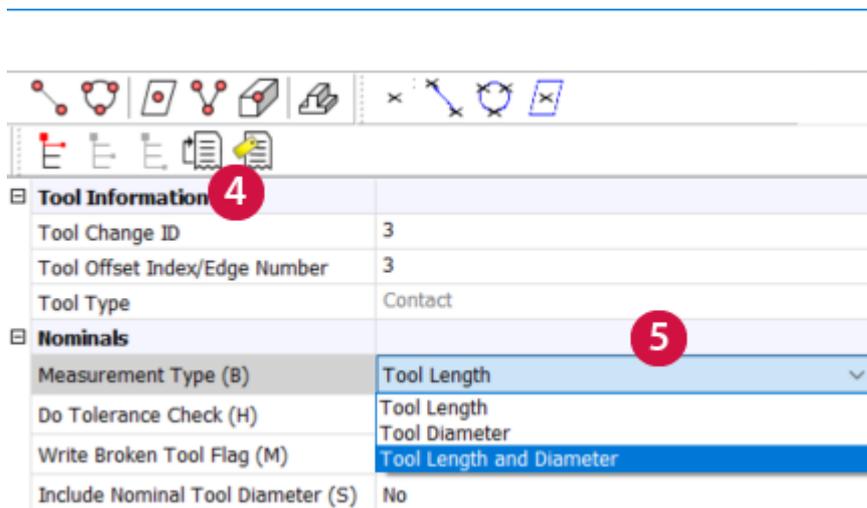


The **Probing Dialog** box opens.

2. Select your tool setter type (contact or non-contact) from the **Tool Setting** menu. The most common tool setter is the contact OTS.



3. From the fly-out menu, choose a tool length/diameter for your control, or select the generic option.
4. Enter the necessary tool information. (Tool change ID.)
5. Enter the remaining parameters as necessary.

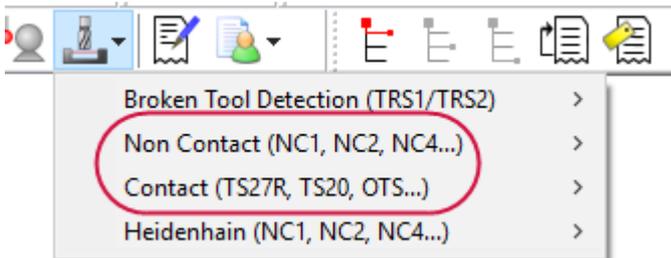


6. Click **OK**.

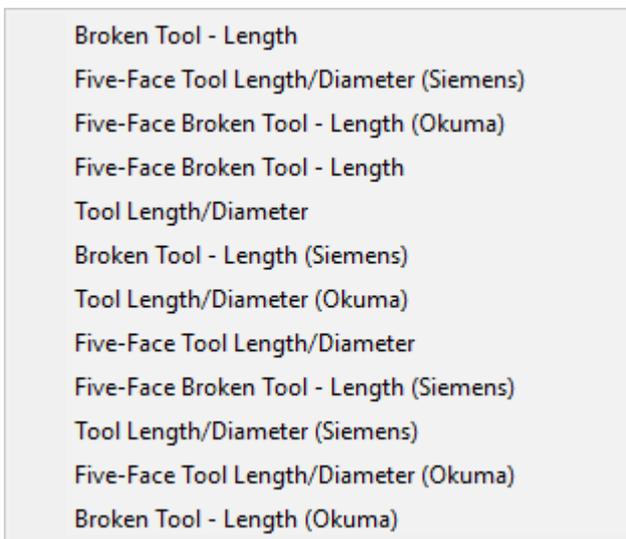
Breakage Detection

Probe operations can monitor tool integrity during the cutting process. The procedure below describes the basic steps that are required to create a tool inspection cycle. Specific settings are determined by your machine control and tool setter.

1. Create a probe operation.
2. Choose **Contact** or **Non-Contact**, based on the probe you have.



3. From the fly-out menu, select the appropriate option. Choose the cycles that correspond to the macros that are loaded on your control.



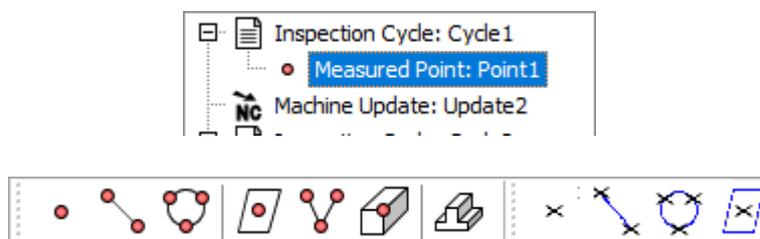
4. Enter the necessary tool information. (Tool change ID.)

Tool Information	
Tool Change ID	3
Tool Offset Index/Edge Number	3
Tool Type	Contact
Nominals	
Measurement Type (B)	Tool Length
Do Tolerance Check (H)	No
Write Broken Tool Flag (M)	No
Include Nominal Tool Diameter (S)	No

5. Set the remaining parameters, as necessary.
6. Click **OK**.

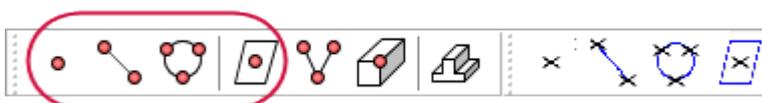
Measuring

Probing operations use measurements obtained from selected or constructed entities to execute process control tasks. Select a probe method from the options on the Probing Dialog toolbar and enter data, as necessary, to create an inspection cycle.

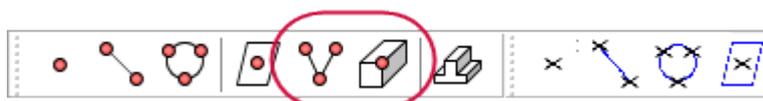


You can probe the following features:

- Points, Lines, Circles/Arcs, and Planes



- 2D Corners and 3D Corners



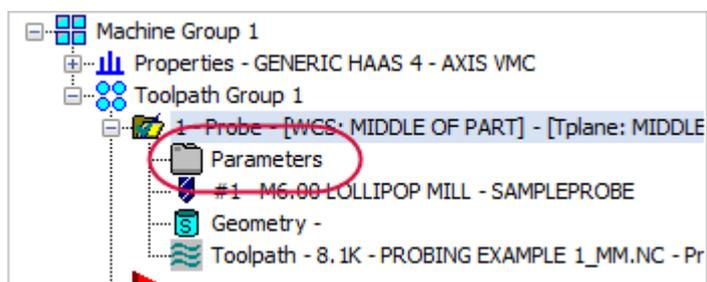
- Webs or Pockets



Machine Updates

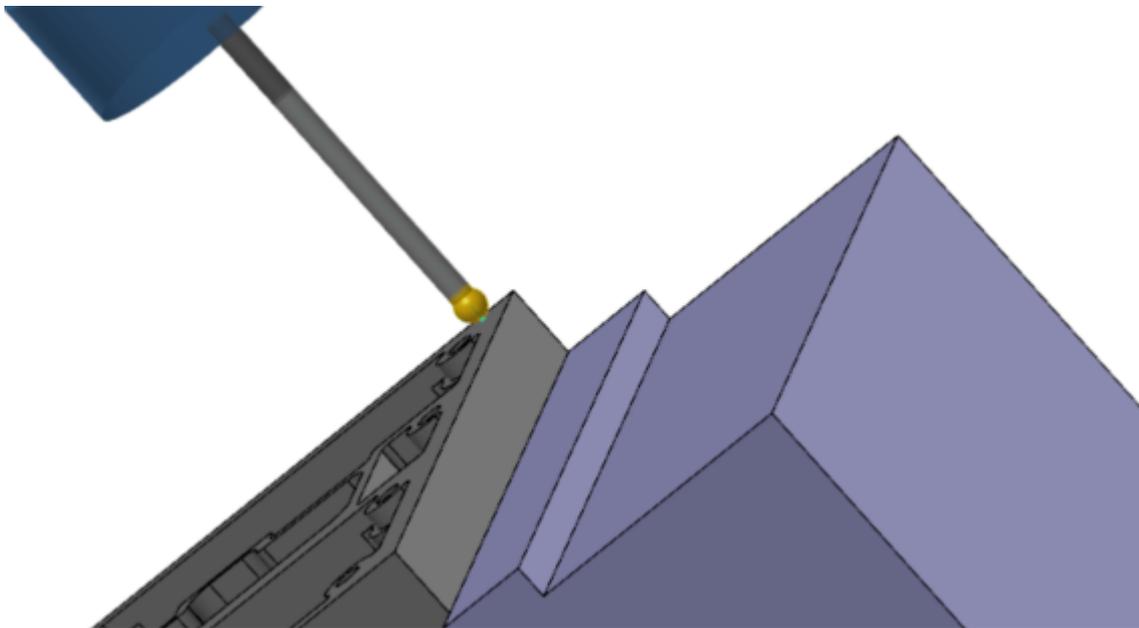
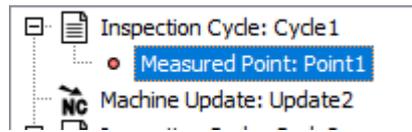
Operation 1 in the sample part shows an example of using a probe operation to establish the part position and then updating the machine settings based on the measurements.

As with other Mastercam toolpaths, you can click in the Toolpaths Manager to view the operation's settings. Open the Operation 1 parameters to see how each inspection cycle is used to update the machine.

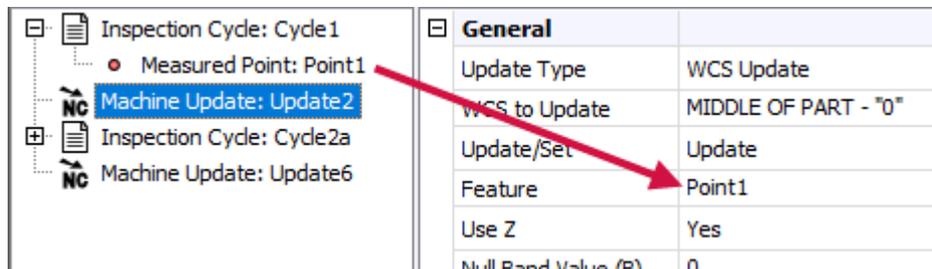


Cycle1/Update2 – Establish Top of Part

Cycle1 probes a point at the part top.



Update2 redefines the selected WCS by changing the Z coordinate to equal the measured point in Cycle1.

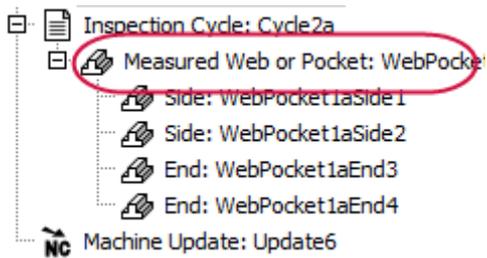


You can make the following machine updates with Productivity+ by selecting an option from the **Update Type** drop-down list. The availability of these options depends on the type of cycle you are referencing.

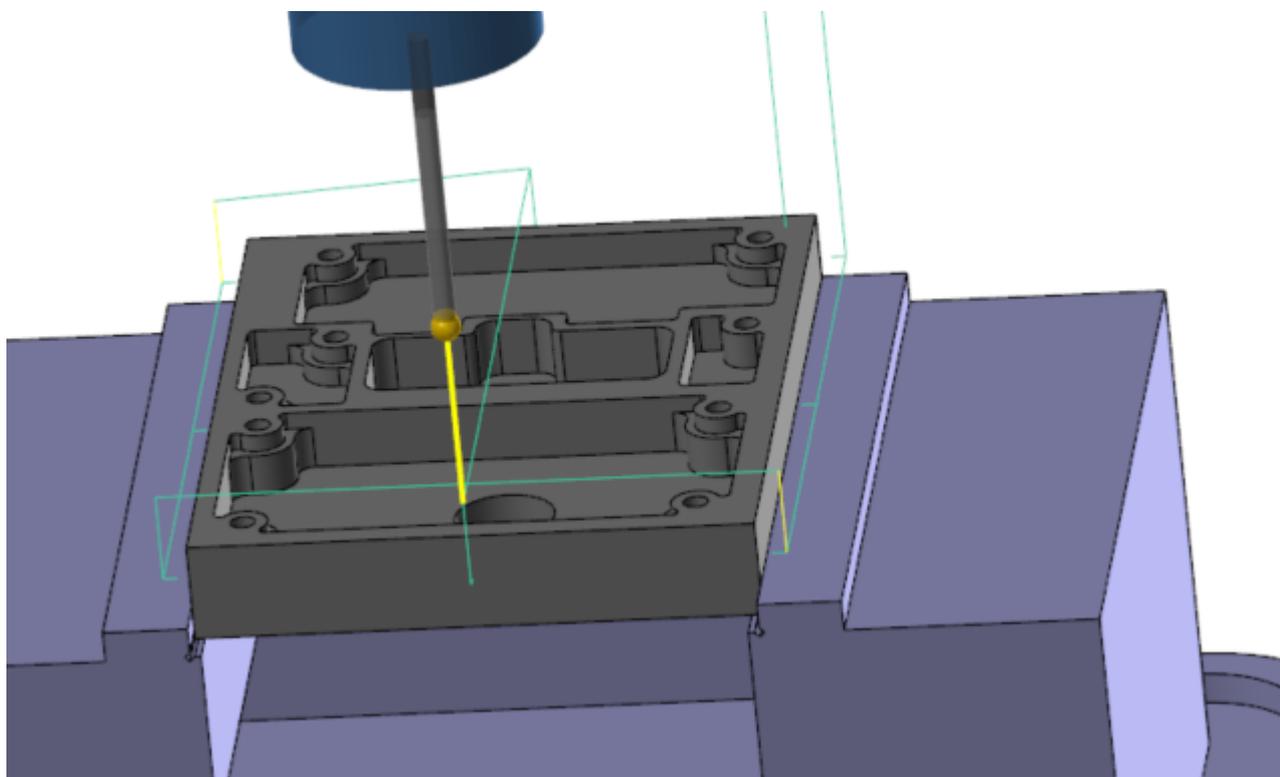
- WCS Update
- Tool Diameter
- Rotation Update
- Tool Length
- Machine Variable

Cycle2a/Update6 – Establish Part Center

Cycle2a probes a series of points on all four sides of the part. This computes the location and dimensions of the Pocket feature.



Feature characteristics obtained from a probe can be reported to the user, used in calculations, or used in logic statements.



Update6 redefines the selected WCS by changing the X and Y coordinates to equal those measured in Cycle2a, establishing the part center.

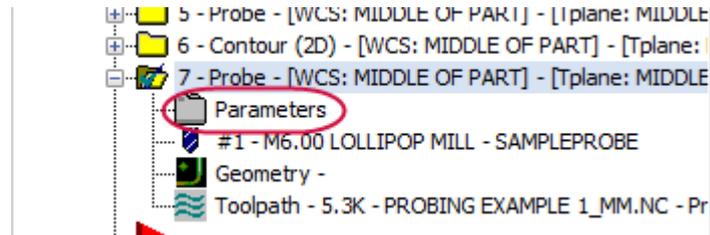
<ul style="list-style-type: none"> [-] Inspection Cycle: Cycle1 <ul style="list-style-type: none"> ● Measured Point: Point1 [-] Machine Update: Update2 [-] Inspection Cycle: Cycle2a <ul style="list-style-type: none"> [-] Measured Web or Pocket: WebPocket1a <ul style="list-style-type: none"> [-] Side: WebPocket1aSide1 [-] Side: WebPocket1aSide2 [-] End: WebPocket1aEnd3 [-] End: WebPocket1aEnd4 [-] Machine Update: Update6 	<table border="1"> <thead> <tr> <th colspan="2">General</th> </tr> </thead> <tbody> <tr> <td>Update Type</td> <td>WCS Update</td> </tr> <tr> <td>WCS to Update</td> <td>MIDDLE OF PART - "0"</td> </tr> <tr> <td>Update/Set</td> <td>Update</td> </tr> <tr> <td>Feature</td> <td>WebPocket1a</td> </tr> <tr> <td>Use X</td> <td>Yes</td> </tr> <tr> <td>Use Y</td> <td>Yes</td> </tr> <tr> <td>Null Band Value (B)</td> <td>0</td> </tr> <tr> <td>Percentage Feedback</td> <td>100</td> </tr> </tbody> </table>	General		Update Type	WCS Update	WCS to Update	MIDDLE OF PART - "0"	Update/Set	Update	Feature	WebPocket1a	Use X	Yes	Use Y	Yes	Null Band Value (B)	0	Percentage Feedback	100
General																			
Update Type	WCS Update																		
WCS to Update	MIDDLE OF PART - "0"																		
Update/Set	Update																		
Feature	WebPocket1a																		
Use X	Yes																		
Use Y	Yes																		
Null Band Value (B)	0																		
Percentage Feedback	100																		

Update2 redefines the selected WCS by changing the X and Y coordinates to equal those measured in Cycle2a, establishing the part center.

Note: The options listed for **WCS to Update** are populated by the planes that are created in Mastercam's Planes Manager. Work offset numbers assigned in the Planes Manager will automatically be used when the plane is changed in the probing operation. See Mastercam's Help for more information about the Planes Manager.

Remachining

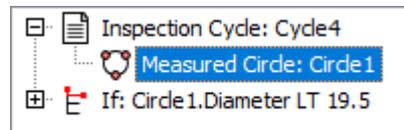
Operation 7 in the sample part shows an example of creating a probe operation to check the feature for out-of-tolerance measurements. If the probing cycle picks up out-of-tolerance measurements, you can program Productivity+ to modify the tool offsets (length, diameter, or wear) and remachine the feature. In cases where remachining cannot repair the feature (i.e., oversized holes), you can also program Productivity+ to stop the machine.



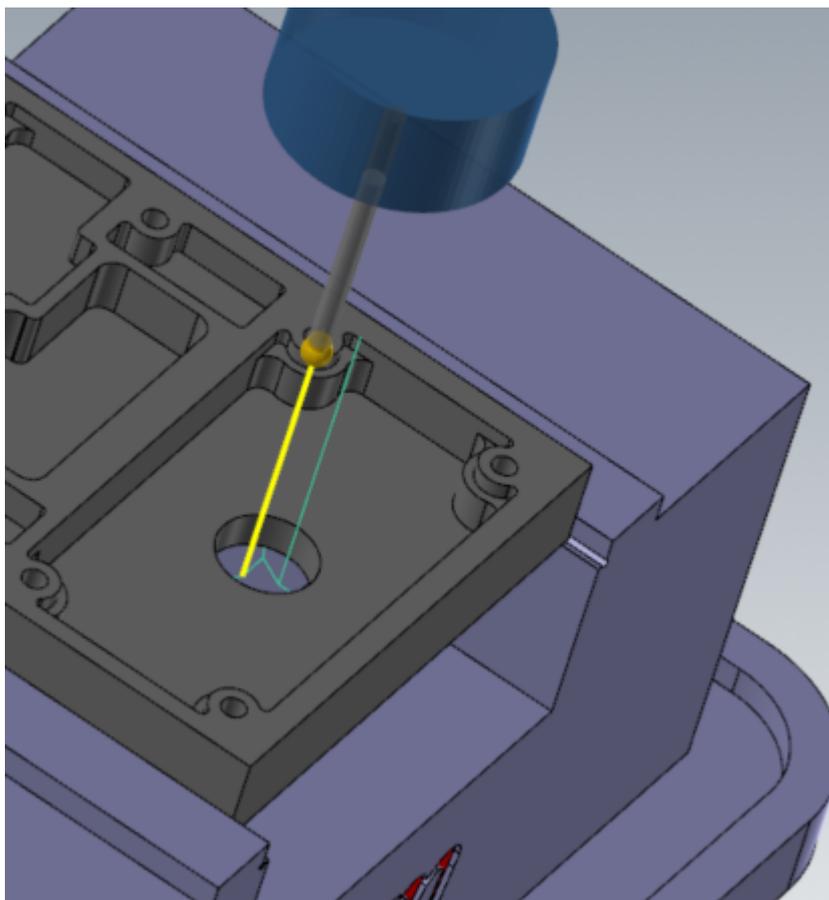
Open the Operation 7 parameters to see how Productivity+ uses logic statements to remachine features.

The Inspection Cycle

Cycle4 probes the selected circle, checking its center point and diameter.

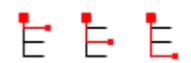


The **Measure Circle** function probes points within the hole to establish its inner diameter (ID).



The Logic Statement

Build logic statements by creating conditions and actions from the options in the Probing Dialog toolbar:

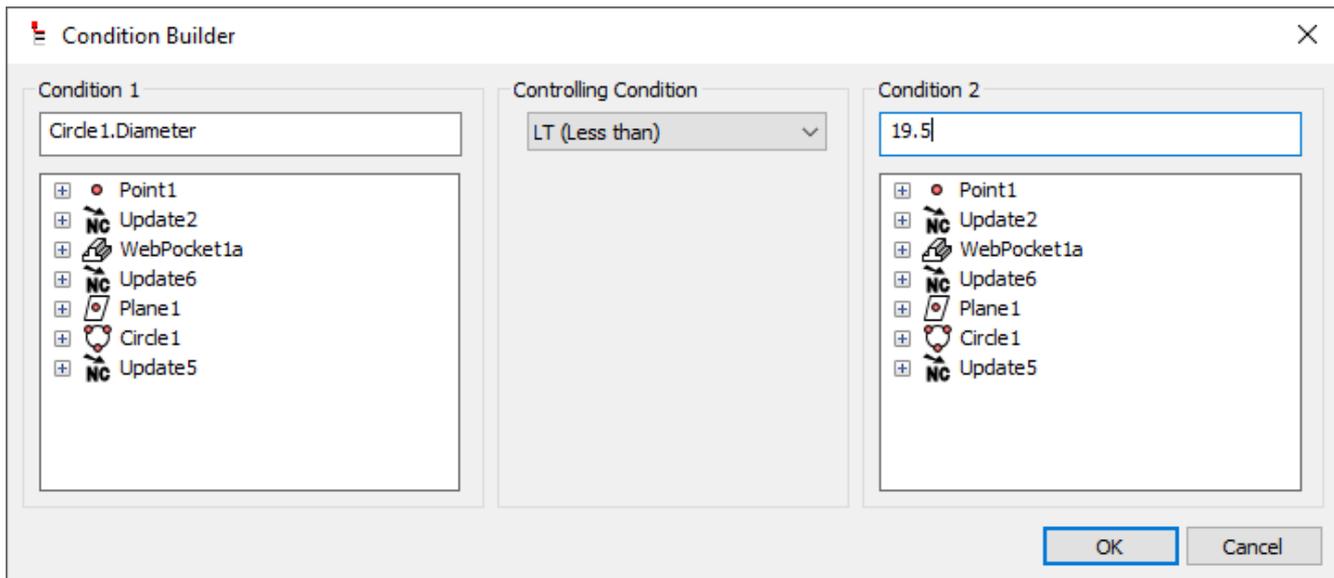


- **If...Then**
- **Else If**
- **Else**

To enter or edit a condition statement—the If or Else If portions of the logic branch—open the Condition Builder dialog box. From the Probing Dialog click the condition field, and then click **Browse** on the right side of the field. A hard value may also be entered here.



The **Condition Builder** dialog box:



In the **Condition Builder** dialog box, a **Controlling Condition** defines the relationship between any two of the following:

- Measured entities (defined in any probing cycle)
- Constructed entities (defined in any probing cycle)
- Nominal values

Condition options (Condition 1 and Condition 2) can refer to any feature you have defined in any probing cycle or to a nominal value that you enter.

Note: In the Condition Builder, you can right-click on a feature to quickly select an entity attribute or the nominal value of a entity.

In Operation 7, the **If** statement checks the probed diameter of Circle1 to a nominal value of 19.5 mm.

Condition Circle1.Diameter LT 19.5

Add the actions you would like performed to the Then or Else portions of the logic branch. Action statements would typically be a machine update, g-code block, or GoTo statement.

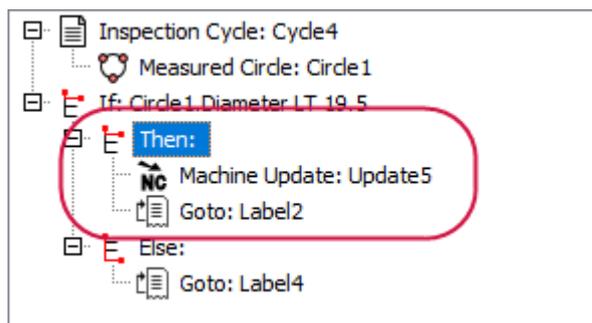
Labels and GoTo Statements

A **Label** creates a reference point in a probing cycle or operation. A **GoTo** statement programs the machine to jump forward or back to that reference point to skip operations or repeat them. All GoTo statements must reference a label, which is why a label is created every time you add a GoTo. You can delete the created label and reference one of your choice.

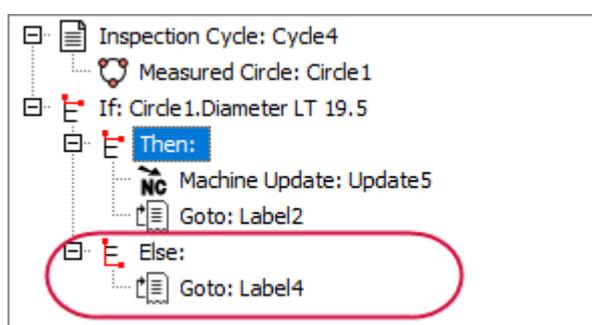


Operation 7 lists two different courses of action based on whether the measured diameter is less than 19.5 mm or not:

If the diameter of Circle1 is less than 19.5 mm, the **Then** statement programs a machine update that modifies the tool diameter offset and then returns the process to a label (Label2) established in Operation 5, so that the part can be remachined with the Contour toolpath (Operation 6).



Else (if the diameter is not less than 19.5 mm), the process should proceed to Label 4, which is located in Operation 8.



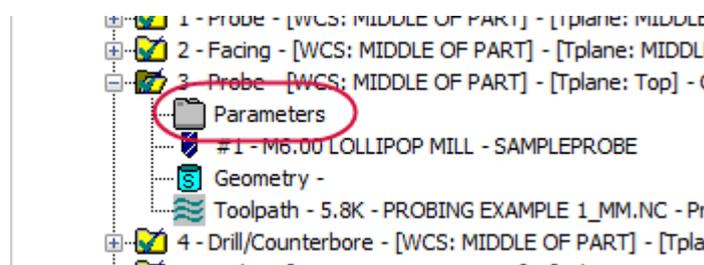
Note: Open the parameters for Operations 5 and 8 to view Label2 and Label4.

Reporting

With any probe cycle, you have the option to output measured data into a report to a file or RS232 port. Operation 3 in the sample part uses a probe operation to inspect part features after machining and generate a report. This reporting happens by whichever means your machine currently dprints (for example, via RS232 or to a file).

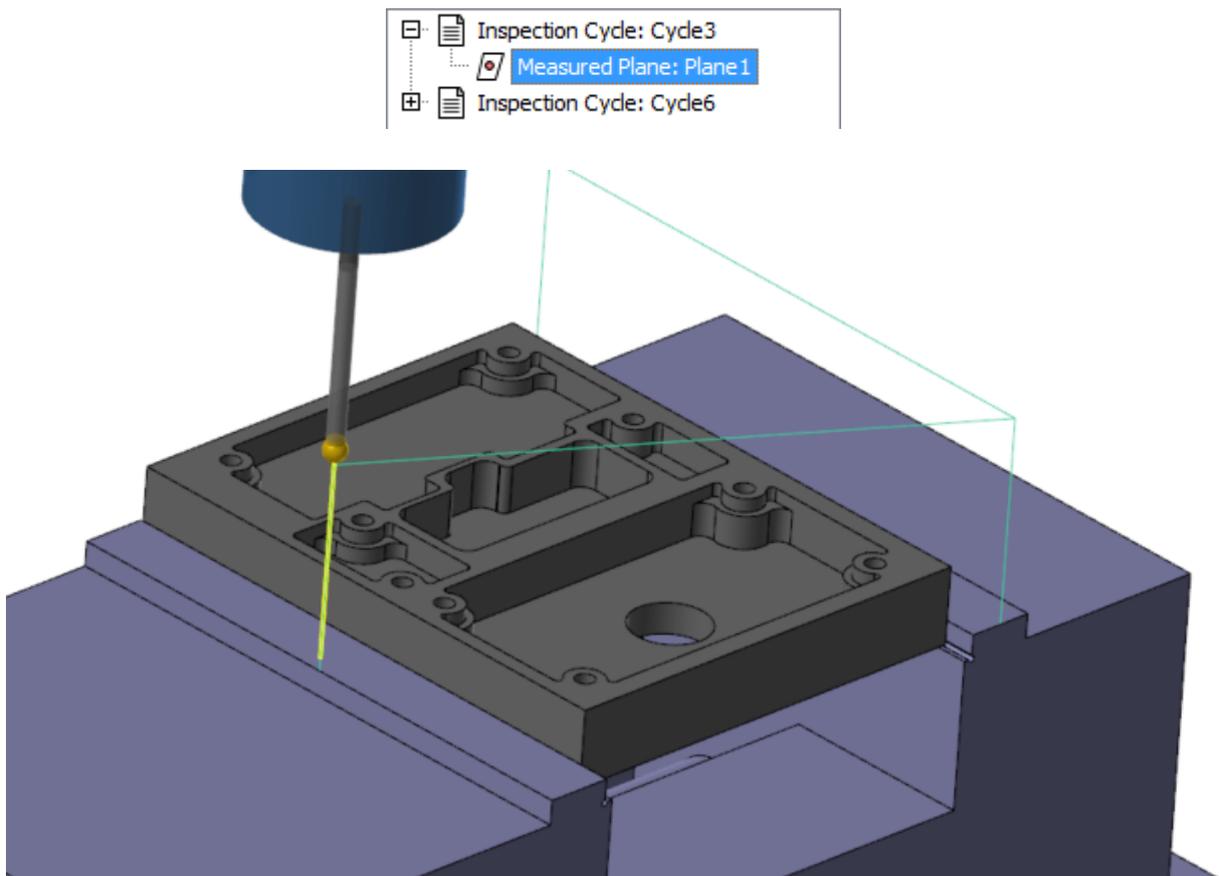
Note: The report format is generated by your control's library macros and depends on your machine control. The current DPrint setting is used to output the report.

Open the Operation 3 parameters to see how Productivity+ generates reports.



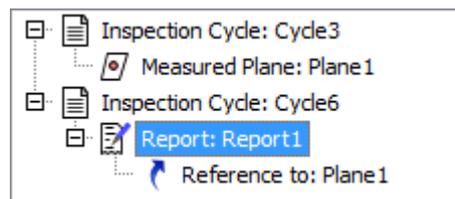
Cycle3 – Inspect

Cycle3 inspects three selected points to establish a plane.



Cycle6 – Report

Cycle6 generates a report that lists the outcome of Cycle3.

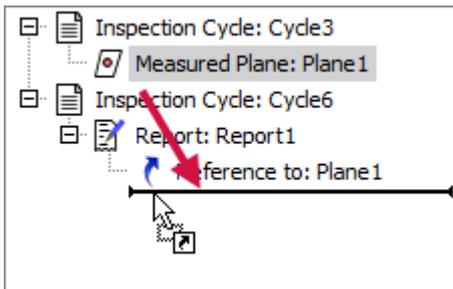


Follow these steps to create a report:

1. Select **Reporting** to create a reporting cycle.



2. Drag measured or constructed entities to be referenced on to the report statement.



The sample part generates a report fairly early in the cutting process. However, you can generate reports at any time: before, during, or after machining. If you want to show Tolerancing, you must turn tolerancing on in the report cycle.

PRODUCTIVITY+ AND MASTERCAM POSTS

When you post a part with a properly configured Mastercam 2019 post, it interacts with the `.RenMF` file to create both probing and machining code. If your Mastercam 2019 post processor has not been properly configured, probing operations will not be incorporated in the NC output.

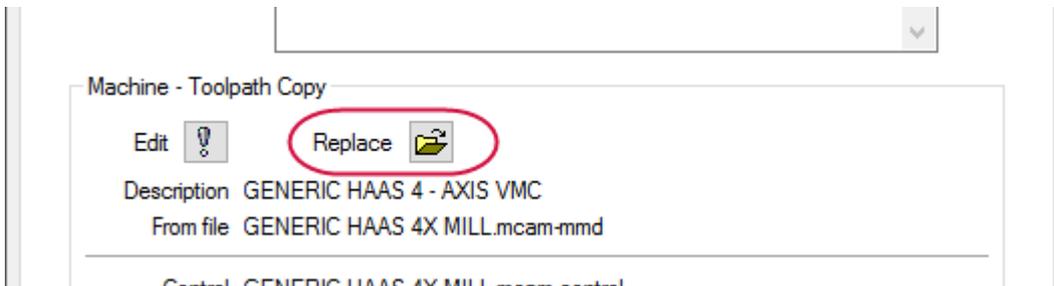
Please contact your local Mastercam 2019 Reseller about post customization for other machine tools.

The following exercises require both a Productivity+ enabled machine definition and post processor.

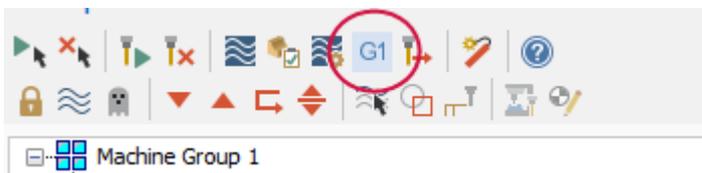
Posting Probe Operations

1. Create a part using one of the Productivity+ enabled machine definitions, or load `ProbingExample`, which was provided with this tutorial.

Note: If your part was *not* created with a Productivity+ enabled machine definition, replace its current machine definition with one that points to a properly configured post in the Machine Group Properties dialog box.

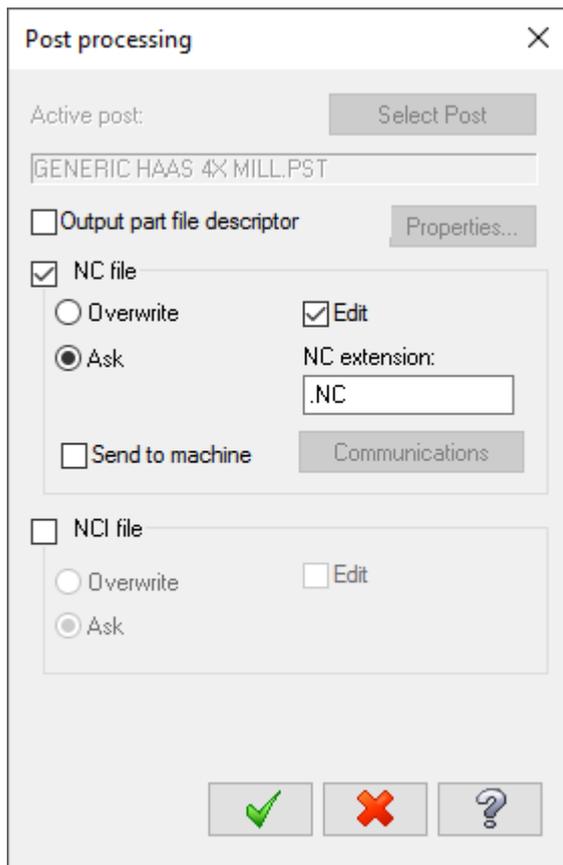


2. Select the desired operations and select **Post selected operations**.

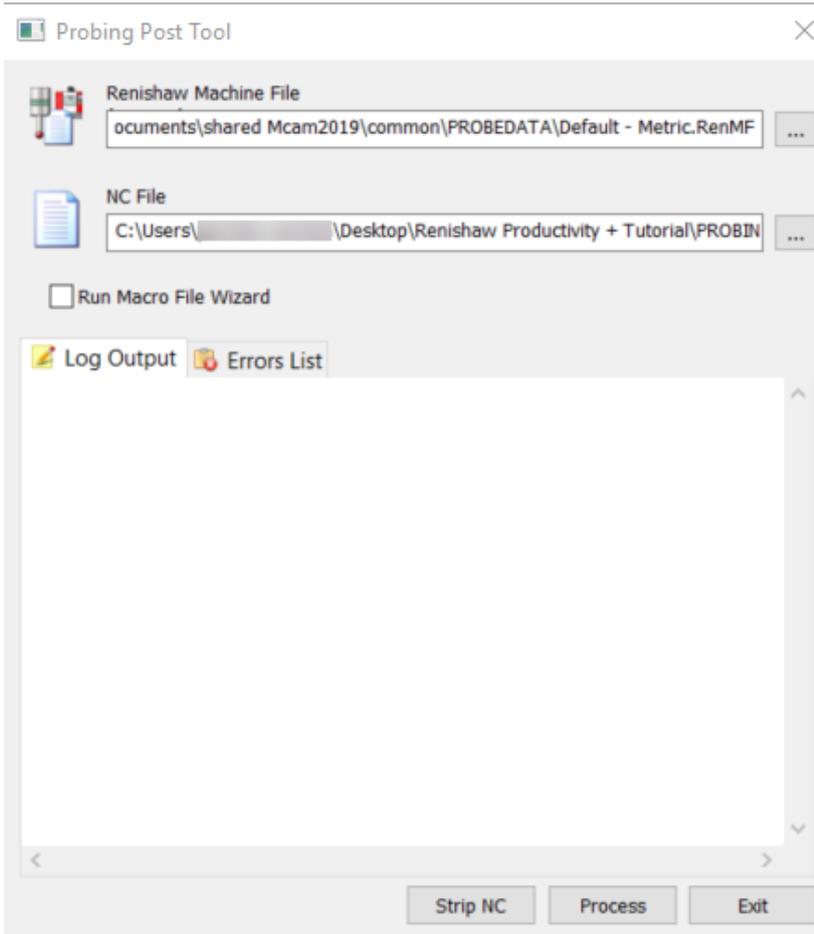


The **Post processing** dialog box displays.

3. Click **OK**. Also, click **Save** in the **Save As** dialog box that appears.



4. The posting process begins and the **Probing Post Tool** dialog box displays.



5. Click **Browse** next to the **Renishaw Machine File** path.
6. Select the desired Renishaw machine file.

Note: Select **Strip NC** to remove all probing calls from the posted NC code. Remember that the Renishaw machine file must be customized for your specific machine before you can run the posted code. Refer to your Renishaw product documentation.

7. Select **Process**. When processing completes, the Log Output tab displays the processing results.
8. Select **Done**. Mastercam 2019 completes the posting process.

The following picture shows how some sample posted code looks in Mastercam's Code Editor.

```

-- -----
84 G65P2008A113.B114.C115.D-71.732E-56.F0.H1.I0.J106.K107.M108.Q112.
85 G65P2001X-68.732Y-62.Z-8.F5000.
86 G65P2101A0.B1500.C0.I100.J3.KD.05I-68.732J-62.K-8.I-68.732J-54.K-8.I113.J114.F
87 G65P2008A113.B114.C115.D-71.732E-56.F0.H1.I0.J106.K107.M108.Q112.
88 G65P2009A106.B107.C108.I112.J3.K-71.732I-56.J143.464KD.I1.J0.KD.I71.732J-56.KI
89 G65P2001Z10.
90 G65P2107A104.B102.C100.I111.J109.K101.I106.J107.K105.I108.J112.K103.I3.J3.K5.1
91 G65P2002Z50.
92 G65P2108B5221.I0.J1.C0.D0.F1.H1.K5221.Q0.R100.S101.U1.V2.W1.X106.Y107.
93 G65P2007
94 (PRODPLUS_END0011)
95 (_RENGCODE_START0012)
96
97 N560 M01
98 N570 T22 M6
99 N580 G0 G90 G54 X-111.901 Y-63.43 A0. S1100 M3
100 N590 G43 H1897 Z50.
101 N600 Z10.
102 N610 G1 Z7.3 F680.
103 N620 X-96.12 Y-70.366

```

CONCLUSION

Congratulations! You have completed the *Renishaw Productivity+ for Mastercam Tutorial!* Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select **Help, Tutorials** from the **File** tab.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's **File** tab or by pressing **[Alt+H]** on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help, Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com. For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)! Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledgebase at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

**ATTENTION! UPDATES MAY BE AVAILABLE.
PLEASE REFER TO [MASTERCAM.COM/SUPPORT](http://mastercam.com/support)
FOR THE LATEST DOWNLOADS.**



CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084



www.mastercam.com

Mastercam

Mastercam® is a registered trademark of CNC Software, Inc. ©1983-2018. All rights reserved.
All other trademarks are property of their respective owners.