Getting Started with Renishaw Productivity+ ™

April 2015

Mastercam X9

Mastercam® X9 Getting Started with Renishaw Productivity +™

Date: April 2015

Software: Mastercam X9

TERMS OF USE

Be sure you have the latest information!

Information might have been changed or added since this document was published. The latest version of this 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.

Contents

Introduction

This introduction to Mastercam's probing application is designed to get you started using Renishaw's Productivity+™ in Mastercam to combine probing with machining. It introduces you to some sample operations, as well as addressing requirements and discussing how probe operations and posts work in Mastercam.

Installation Requirements

To take advantage of Productivity+ in Mastercam, you must have a Renishaw probe installed on your machine tool. Productivity+ software generates the necessary NC code.

Please see the *Mastercam X9 Installation Guide* for general system requirements.

Minimum Mastercam Requirements

To run Mastercam with Renishaw Productivity+, you must have at least Mastercam Mill. For more information on Mastercam product levels, please see Mastercam Help or visit www.mastercam.com/solutions.

Usage Pre-requisites

In addition to experience working in Mastercam, 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 Mill machine definitions.

Support

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

When to Use Productivity+

Use Renishaw Productivity+ in Mastercam 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 settings
- 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 and in the proper location. Additionally, your Mastercam post (`.PST`) must be configured to support Productivity+.

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

Please see “Productivity+ and Mastercam Posts” on page 21 for more information.

Productivity+ in Mastercam

About Renishaw Productivity+

Renishaw Productivity+ allows you to use Mastercam 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. Productivity+ can be used with 3- and 4-axis mills (vertical and horizontal) and multi-axis (3+2) machines.

Features and Benefits

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

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 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 cannot be used on types of features with free-flowing surfaces such as ports.

Remachining

Productivity+ in Mastercam 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.

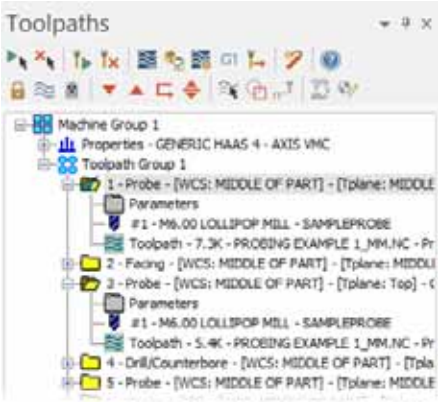
Using Productivity+ in Mastercam

CNC Software, Inc. has incorporated Renishaw's Productivity+ interface into Mastercam, 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. 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'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. Updating your part model will not mark a probe operation dirty.

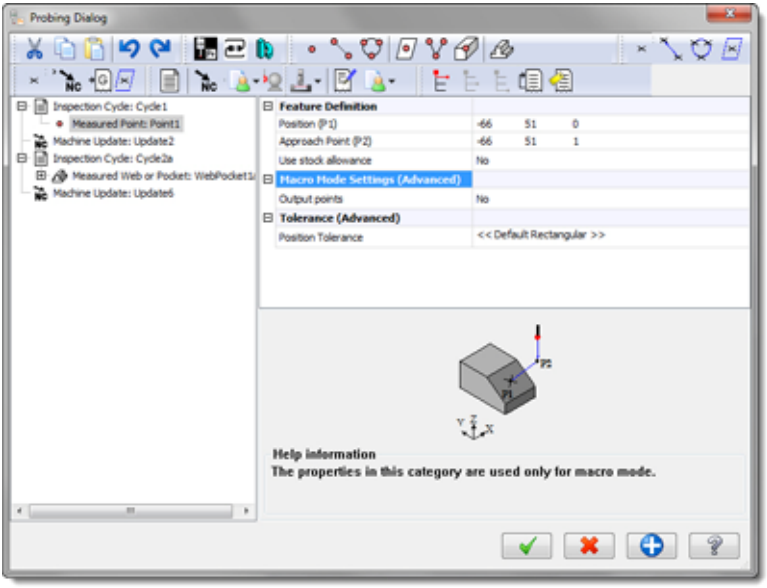
To create a probe operation, select **Probe** from the Toolpaths menu.



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 taken measurements
- Reports

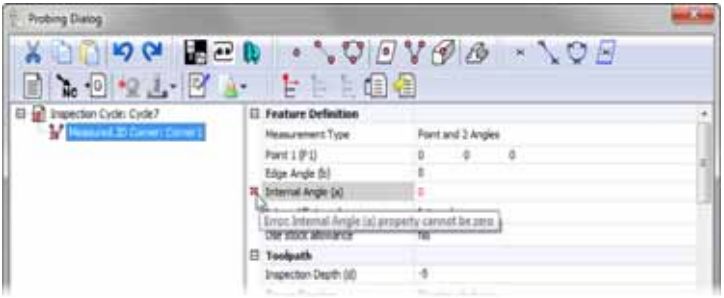
- Logic statements to vary the response based on measurements
- Label statements to use as reference points within the probe operation tree



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

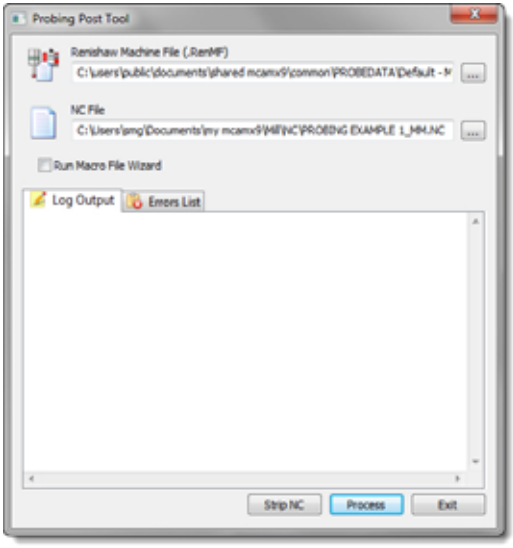
Use Renishaw's embedded Help for more detailed information on the parameters you set for each function. Hover over a setting, or click it to display Help information at the bottom of the dialog box. If the Help topic contains more information than will fit in that section, you can either use the vertical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

TIP: 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.

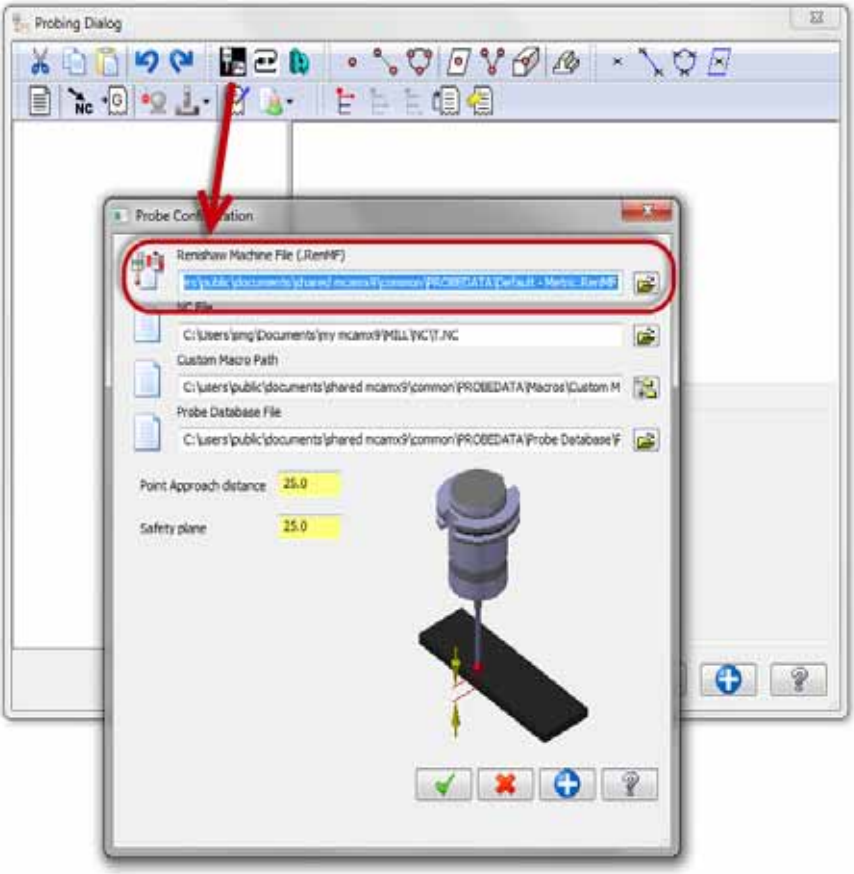


Posts

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 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, see page 21.

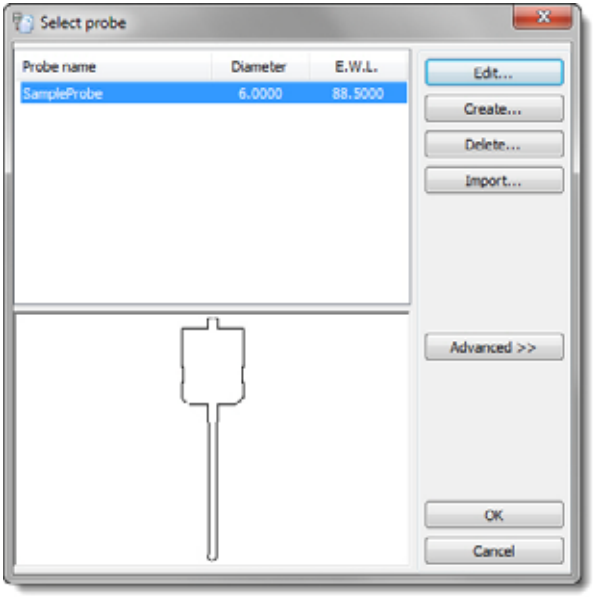
Generating Probe Operations

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 `PROBINGEXAMPLE.MCX-9`, a part which was provided with the tutorial to demonstrate some of the available probe operations.

Setting Up Your Probing Module and Stylus

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



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

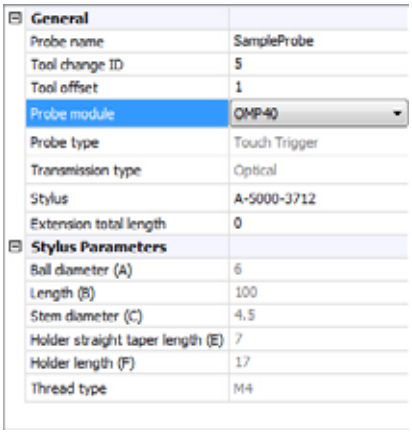
- **Edit** - Opens the **Edit Probe** dialog box, where you can edit the properties of the current probing module and stylus.
- **Create** - Opens the **Edit Probe** dialog box, where you can create a new probing module and stylus.

- **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.

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** opens after you have configured your probe and stylus information.



CAUTION: 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 by selecting the **Probe Calibration** button in the Probing Dialog toolbar.

Tool Setting

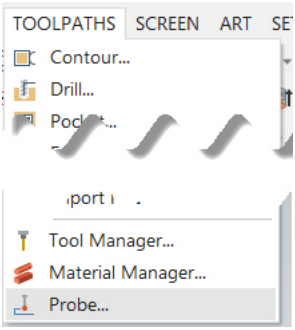
Prior to machining, use Productivity+ to establish tool offsets. Use the probe's breakage detection during processing to determine aberrations in tool diameter and length.

Measuring Tool Length and Diameter

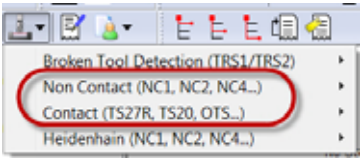
In one probe operation, you can establish tool offsets for all tools in the tool magazine. The procedure below describes the basic steps that are required to create a tool setting cycle. Specific settings are determined by your machine control and tool setter.

1 Create a probe operation by selecting **Probe** from the Toolpaths menu.

The Probing Dialog box opens.

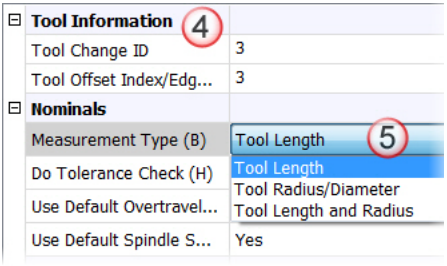


2 Select your tool setter type (contact or non-contact) from the **Tool Setting** menu.



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 and tool offset index number.)



TIP: Enter 0 to match the tool offset index to the tool change ID for any value greater than 9.

5 Set the remaining parameters as necessary.

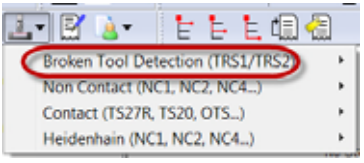
6 Click **OK**.

Breakage Detection

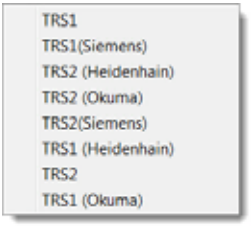
Probe operations can monitor tool integrity during the cutting process. The procedure below describes the basic steps that are required to create a tool inspection cycle. Specific settings are determined by your machine control and tool setter.

1 Create a probe operation.

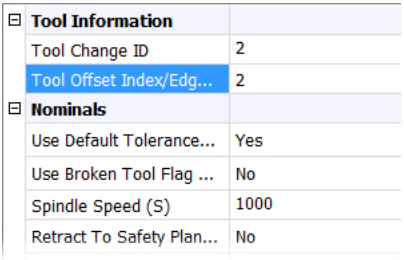
2 Choose the **Broken Tool Detection** option from the **Tool Setting** menu.



3 From the fly-out menu, select a tool setter for your control, or select the generic option.



4 Enter the necessary tool information. (Tool change ID and tool offset index number.)

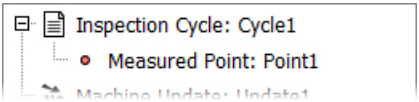


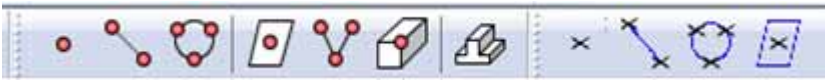
5 Set the remaining parameters, as necessary.

6 Click **OK**.

Measuring

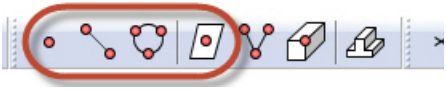
Probing operations use measurements obtained from selected or constructed entities to execute process control tasks. Select a probe method from the options on the Probing Dialog toolbar and enter data, as necessary, to create an inspection cycle.





You can probe the following features:

- Points, Lines, Circles/Arcs, and Planes



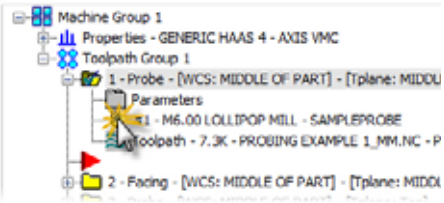
- 2D Corners and 3D Corners

- Webs or Pockets

Machine Updates

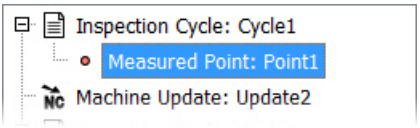
Operation 1 in the sample part shows an example of using a probe operation to establish the part position and then updating the machine settings based on the measurements.

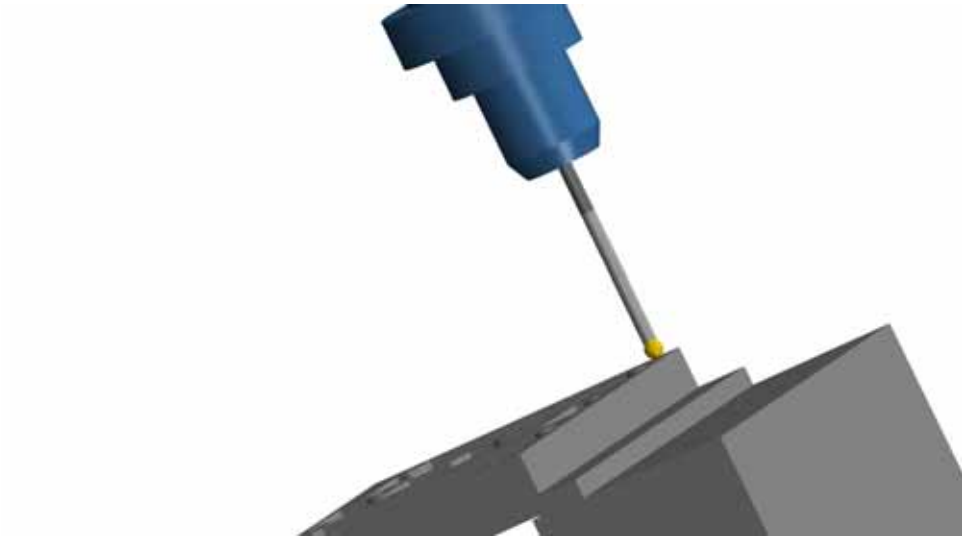
As with other Mastercam toolpaths, you can click in the Toolpaths Manager to view the operation's settings. Open the Operation 1 parameters to see how each inspection cycle is used to update the machine.



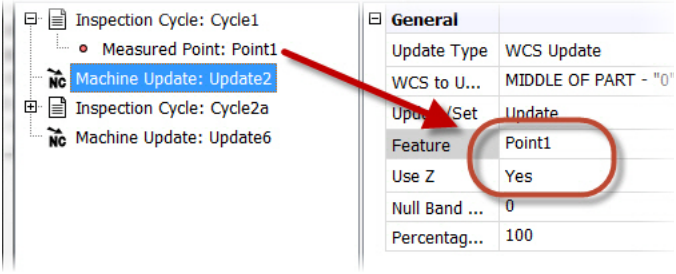
Cycle1/Update2 – Establish Top of Part

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.

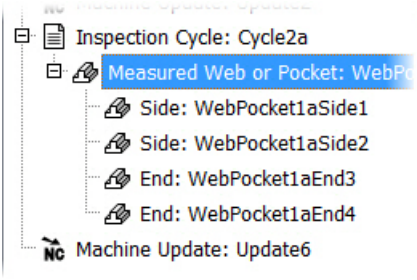


TIP: You can make the following machine updates with Productivity+ by selecting an option from the **Update Type** drop-down list:

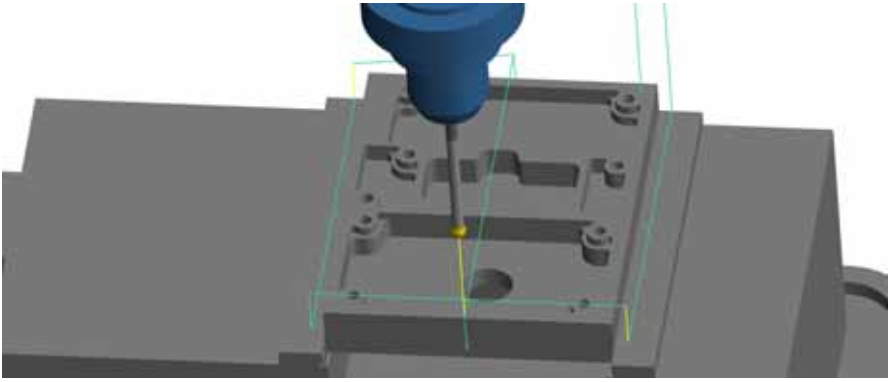
- WCS Update
- Tool Diameter
- Rotation Update
- Tool Length
- Machine Variable

Cycle2a/Update6 – Establish Part Center

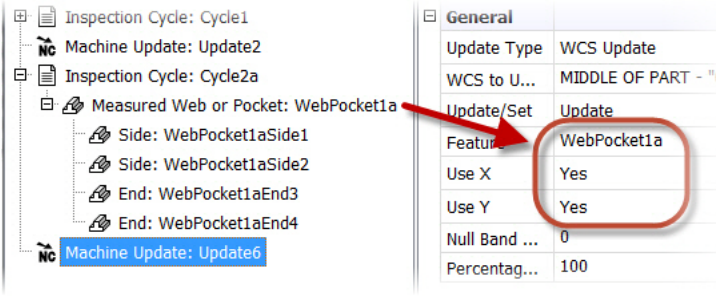
Cycle2a probes a series of points on all four sides of the part. This computes the location and dimensions of the Pocket feature.



TIP: 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.

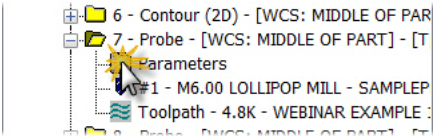


NOTE: The options listed for **WCS to Update** are populated by the planes that are created in Mastercam's Plane Manager. Work offset numbers assigned in the Plane Manager will automatically be used when the plane is changed in the probing operation. See Mastercam Help for more information about the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

Remachining

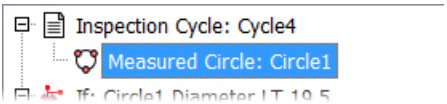
Operation 7 in the sample part shows an example of creating a probe operation to check the feature for out-of-tolerance measurements. If the probing cycle picks up out-of-tolerance measurements, you can program Productivity+ to modify the tool offsets (length, diameter, or wear) and remachine the feature. 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.

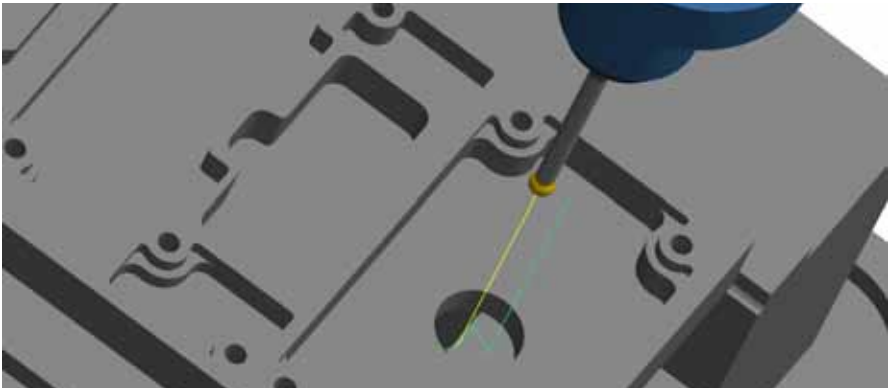


The Inspection Cycle

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

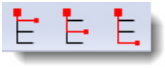


The **Measure Circle** function probes points within the hole to establish its inner diameter (ID).



The Logic Statement

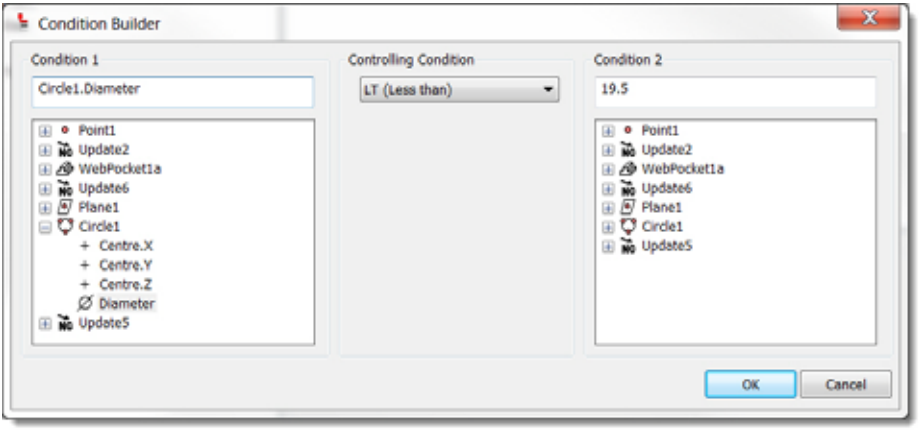
Build logic statements by creating conditions and actions from the options in the Probing Dialog toolbar:



- **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 the **Browse** button on the right side of the field.

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.

TIP: 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

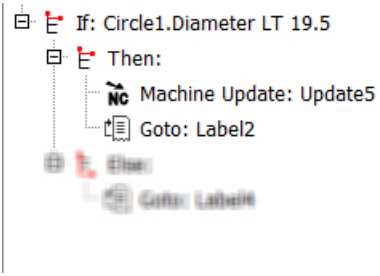
Add the actions you would like performed to the Then or Else portions of the logic branch. Action statements would typically be a machine update, g-code block, or GoTo statement.

Labels and GoTo Statements

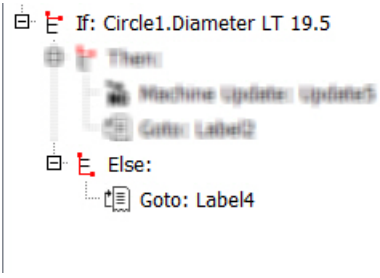
A ***Label*** creates a reference point in a probing cycle or operation. A ***GoTo*** statement programs the machine to jump forward or back to that reference point to skip operations or repeat them. All GoTo statements must reference a label.

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.



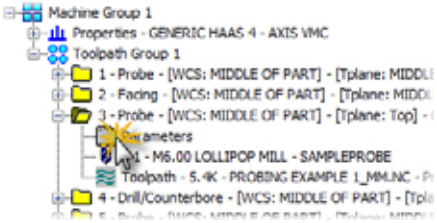
TIP: Open the parameters for Operations 5 and 8 to view Label2 and Label4.

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.

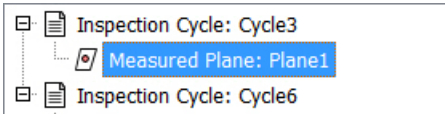
NOTE: The report format is generated by your control's library macros and depends on your machine control.

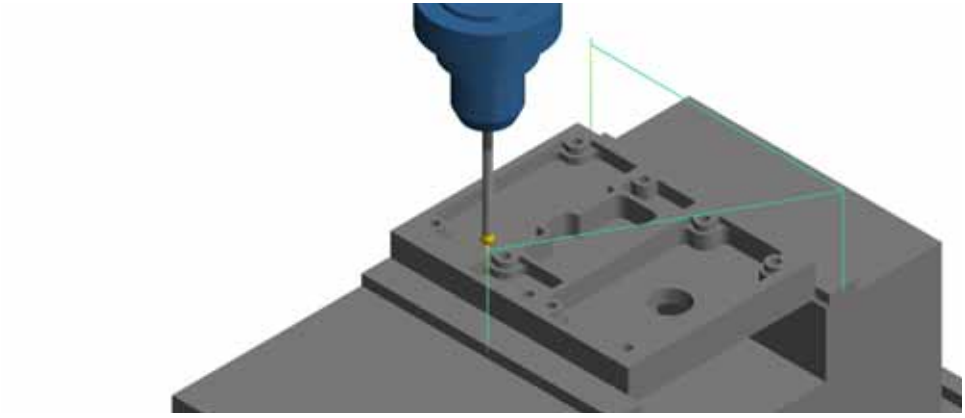
Open the Operation 3 parameters to see how Productivity+ generates reports.



Cycle3 – Inspect

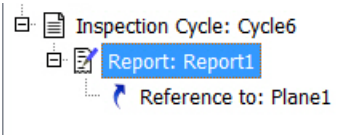
Cycle3 inspects three selected points to establish a plane.





Cycle6 – Report

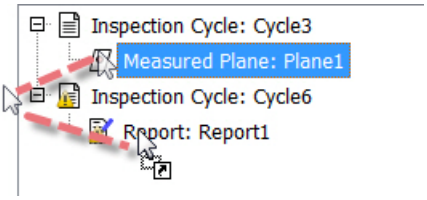
Cycle6 generates a report that lists the outcome of Cycle3.



Follow these steps to create a report.

1 Select the **Reporting** button 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.

Productivity+ and Mastercam Posts

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

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

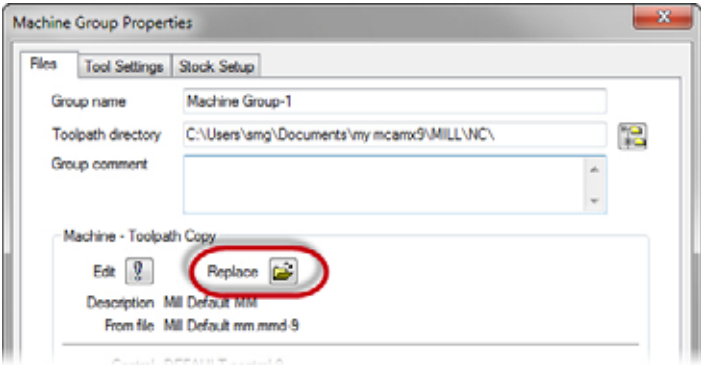
The following exercises require both a Productivity+ enabled machine definition and post processor.

IMPORTANT: The Productivity+ feature has only been tested with Mill posts.

Posting Probe Operations

1. Create a part using one of the Productivity+ enabled machine definitions (`.MMD-9`), or load `ProbingExample.mcx`, which was provided with this tutorial.

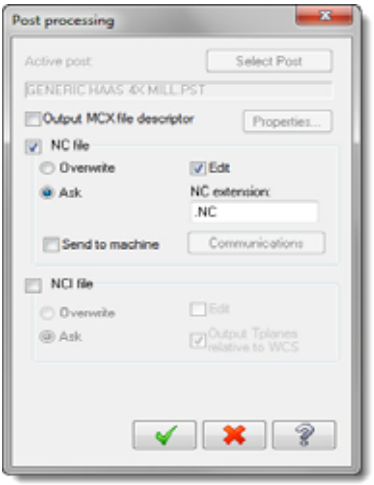
TIP: If your part was *not* created with a Productivity+ enabled machine definition, replace its current machine definition with one that points to a properly configured post.



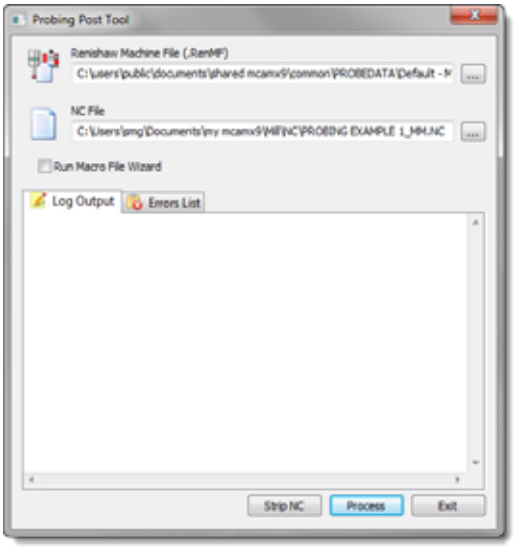
2 Select the desired operations and click the **Post** button.

The Post processing dialog box displays.

3 Click **OK**.



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



4 Click the **Browse** button next to the Renishaw Machine File path.

5 Select the desired Renishaw machine file.

TIP: Click the **Strip NC** button to remove all probing calls from the posted NC code.

IMPORTANT: 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.

6 Click the **Process** button.

When processing completes, the Log Output tab displays the processing results.

7 Click the **Done** button.

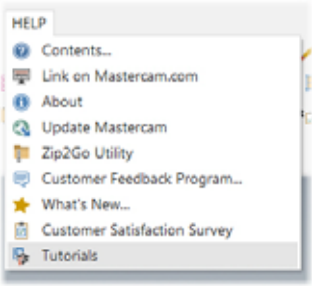
Mastercam completes the posting process.

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

Conclusion

Congratulations! You have completed the *Getting Started with Renishaw Productivity+™* 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. The Mastercam tutorial series is in continual development, and we will add modules as we complete them. Visit our website, or select **Tutorials** from the **Help** menu to see the latest publications.



Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's menu bar or by pressing [**Alt+H**] on your keyboard. Also, most dialog boxes, function panels, and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—CNC Software's Technical Support department (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*—CNC offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. The Mastercam tutorial series is in continual development, with new modules added as we complete them. Visit our website, or select **Tutorials** from the **Help** menu to see the latest publications.
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your

skills at your own pace and help prepare yourself 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, including many videos, 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 knowledge base at kb.mastercam.com. To register, select **Help**, **Link on Mastercam.com** from the Mastercam menu and follow the instructions.

Mastercam Documentation

Mastercam installs the following documents in the `\Documentation` folder of your Mastercam installation:

- *What's New in Mastercam X9*
- *Mastercam X9 Installation Guide*
- *Mastercam X9 Administrator Guide*
- *Mastercam X9 Transition Guide*
- *Mastercam X9 Quick Reference Card*
- *Mastercam X9 ReadMe*

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.
Go to Mastercam.com/Support for the latest downloads.

CNC software, inc.

671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com