How to enable Productivity Plus in Mastercam probing: update fixture and tool diameter offsets

This guide shows the Mastercam probing workflow where Productivity+ is active. You build a probe routine, post it, and check the updated offsets on the CNC control. In the post log, you see Mastercam extract, initialize, and load Productivity+ data. If you want to know how to enable Productivity Plus in Mastercam for probing, this workflow shows where Productivity+ comes into play. It does not cover installation or licensing.

You first update a fixture offset, then compare work offset values, and finally update a tool diameter offset.

Prerequisites

Before you start, make sure you have the following:

- A Mastercam part file with a probing toolpath. The tool diameter example uses the file "Update Tool Dia Offset Ready for Probe.emcam".
- A probe in your tool list. The example uses the "VF2 Probe" (OMP-40 module, touch trigger, optical transmission).
- The Renishaw machine file, which the "Probing Post Tool" dialog uses for post processing.
- A CNC control where you can view work offsets and tool offsets.

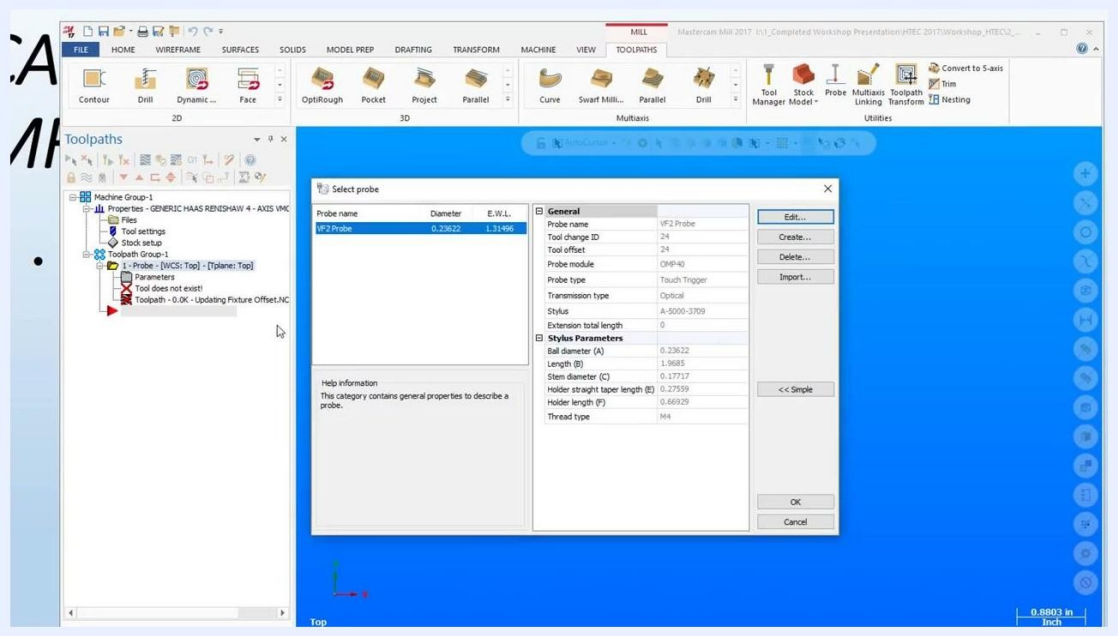
Update a fixture offset

Step 1: Select the probe tool

Open the **Select probe** dialog and choose the probe for the toolpath. The example uses the "VF2 Probe" with these parameters:

- Diameter: 0.23622
- E.W.L.: 1.31496
- Tool change ID: 24
- Probe module: OMP-40
- Probe type: Touch Trigger
- Transmission type: Optical

The dialog also lists the stylus parameters, such as ball diameter, length, and stem diameter.



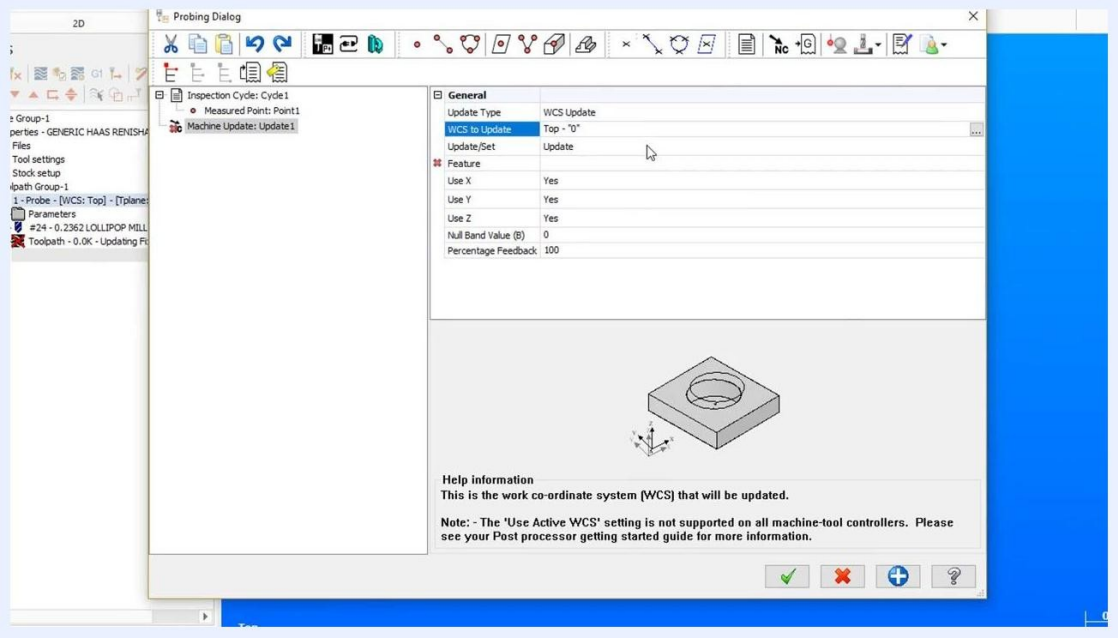
Mastercam interface with the probe selection dialog and toolpath setup

Step 2: Configure the feature definition

In the **Probing Dialog**, set the following under **Feature Definition**:

- Position (P1): 0, 0, 0
- Position (P1 including stock): 0, 0, 0
- Approach Point (P2): 0, 0, 1
- Approach Point (P2 including stock): 0, 0, 1
- Use stock allowance: Yes
- Stock allowance (s): 0

Under **Macro Mode Settings (Advanced)**, set Output points to No. Under **Tolerance (Advanced)**, leave Position Tolerance at << Default Rectangular >>. A 3D preview of the probing feature appears at the bottom of the dialog.



Probing dialog with feature definition and stock allowance settings

Step 3: Add a machine update to the first inspection cycle

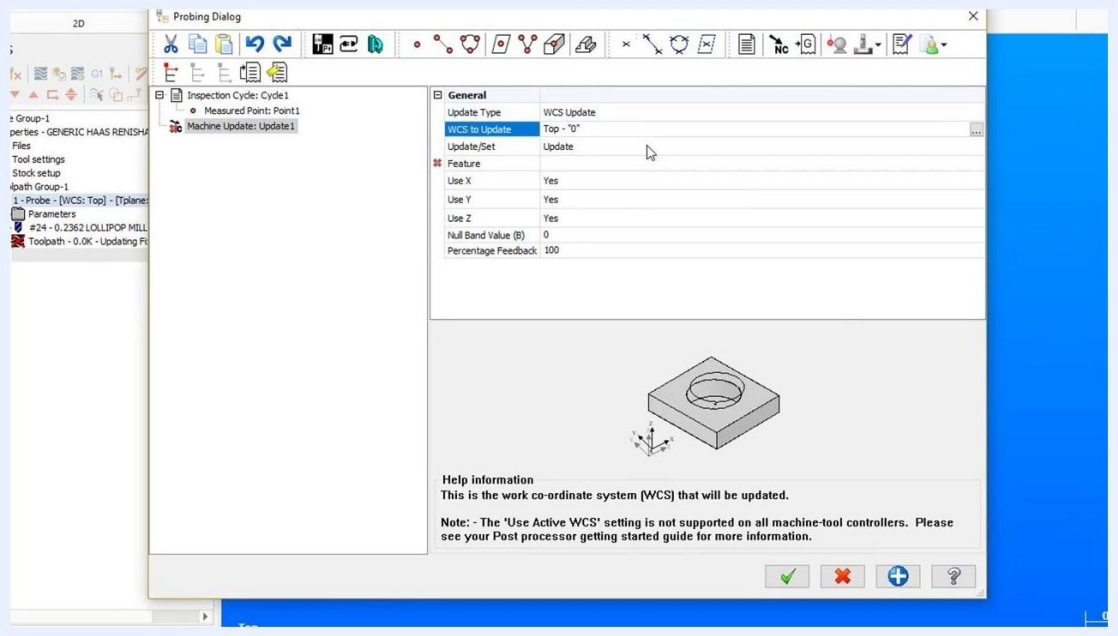
Under **Inspection Cycle: Cycle 1**, add a **Machine Update: Update 1** step. Under **General**, set:

- Update Type: WCS Update
- WCS to Update: Top - "0"
- Update/Set: Update

Under **Feature**, set:

- Use X: Yes
- Use Y: Yes
- Use Z: Yes
- Null Band Value (B): 0
- Percentage Feedback: 100

The preview shows a part with a bore, and the help text at the bottom explains the WCS update.



Probing dialog with WCS update parameters and inspection cycle details

Step 4: Configure a second inspection cycle for a web or pocket

Add a second cycle for measuring a web or pocket. **Inspection Cycle: Cycle2** contains these steps:

- Measured Web or Pocket: WebPocket1
- Side: WebPocket1Side1, WebPocket1Side2
- End: WebPocket1End3, WebPocket1End4

Start with these **Feature Definition** values:

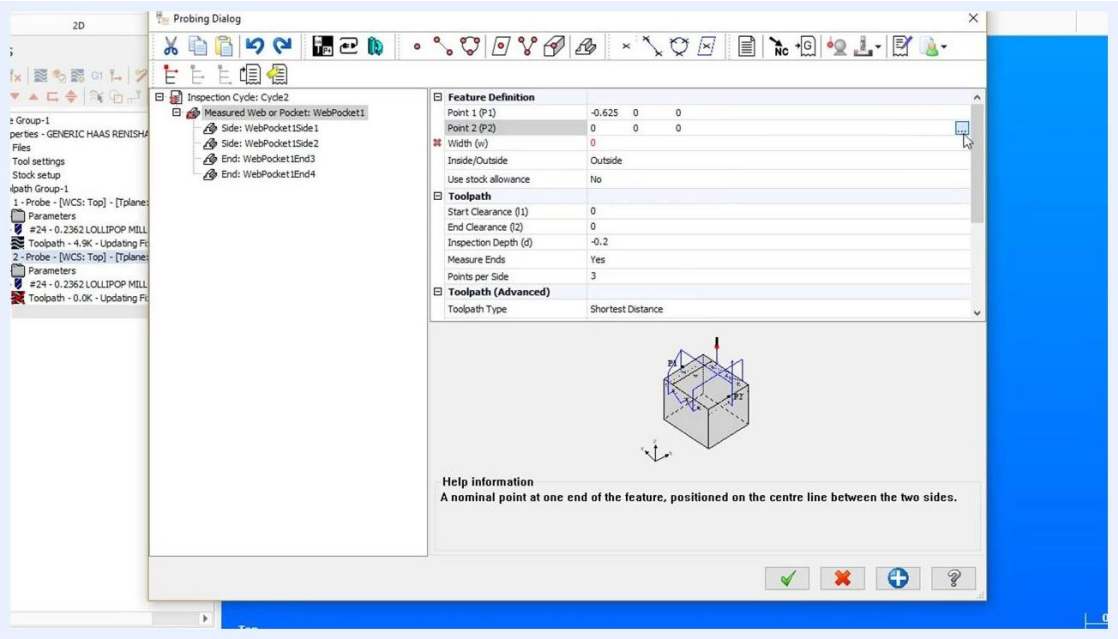
- Point 1 (P1): -0.625, 0, 0
- Point 2 (P2): 0, 0, 0
- Width (w): 0
- Inside/Outside: Outside
- Use stock allowance: No

In the **Toolpath** section, the values are:

- Start Clearance (1): 0
- End Clearance (2): 0

- Inspection Depth (d): -0.2
- Measure Ends: Yes
- Points per Side: 3

Under **Toolpath (Advanced)**, the Toolpath Type is Shortest Distance. The 3D preview shows the probing path on a box-shaped part.

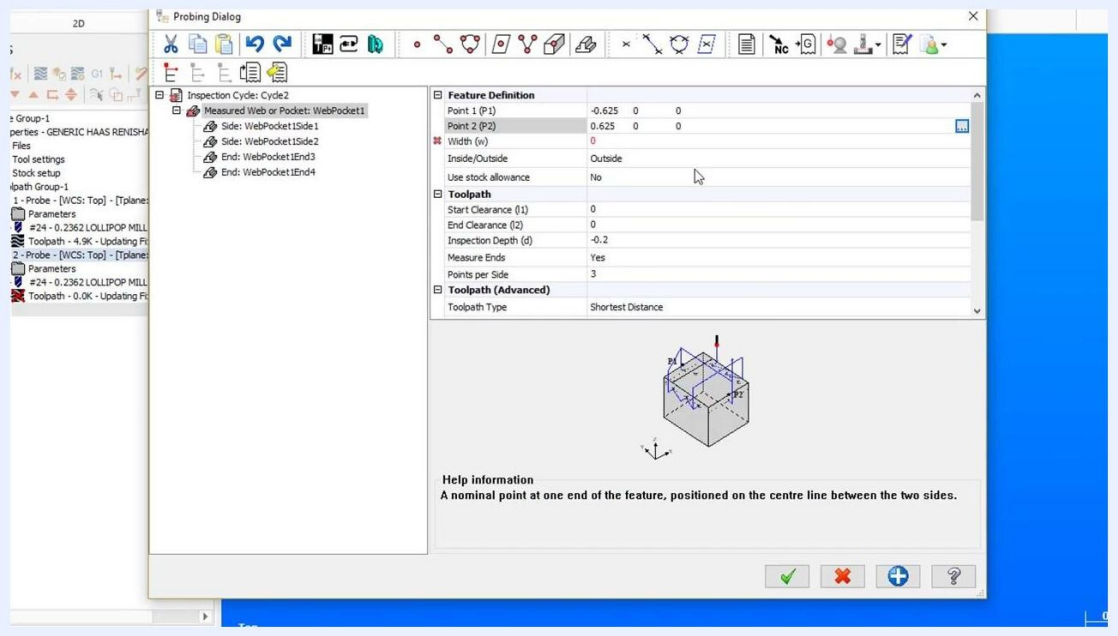


Probing dialog with web/pocket feature definition and toolpath setup

Next, update the feature definition to match the part:

- Point 2 (P2): 0.625, 0, 0
- Width (w): 0.75
- Points per Side: 2

The other values stay the same, and the preview updates to show the new probing points.

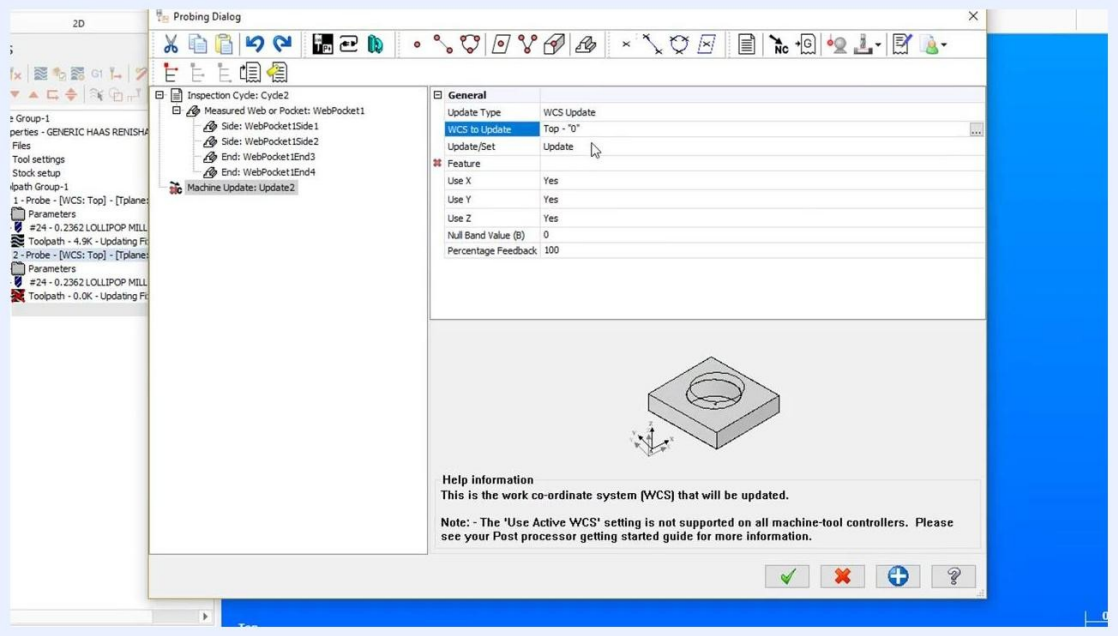


Probing dialog with updated feature definition and toolpath for web/pocket

Finally, set both clearances in the **Toolpath** section:

- Start Clearance (1): 0.1
- End Clearance (2): 0.1

The preview shows the new clearance distances. The help text now describes the minimum distance between the end point and the stylus ball.



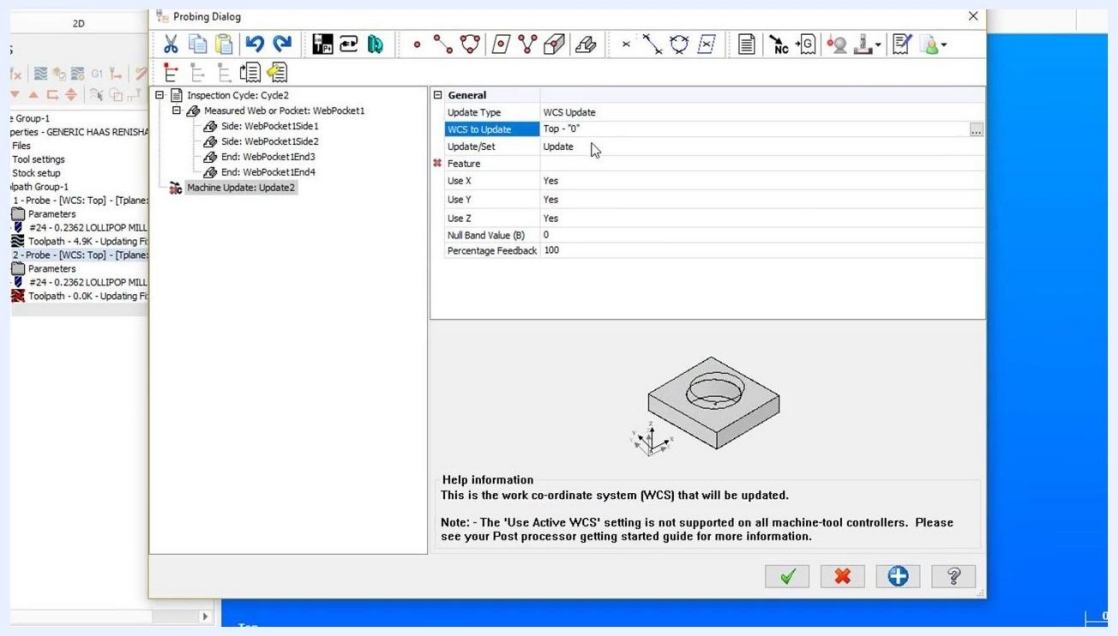
Probing dialog with finalized toolpath clearances for web/pocket measurement

Step 5: Add a machine update to the second inspection cycle

Under **Inspection Cycle: Cycle2**, add a **Machine Update: Update2** step. The right panel shows these settings:

- **General**: Update Type (WCS Update), WCS to Update (Top - "0"), Update/Set (Update)
- **Feature**: Use X (Yes), Use Y (Yes), Use Z (Yes), Null Band Value (0), Percentage Feedback (100)

The preview shows a part with a bore, and the help text explains the WCS update.



Probing dialog with the machine update step for Cycle2

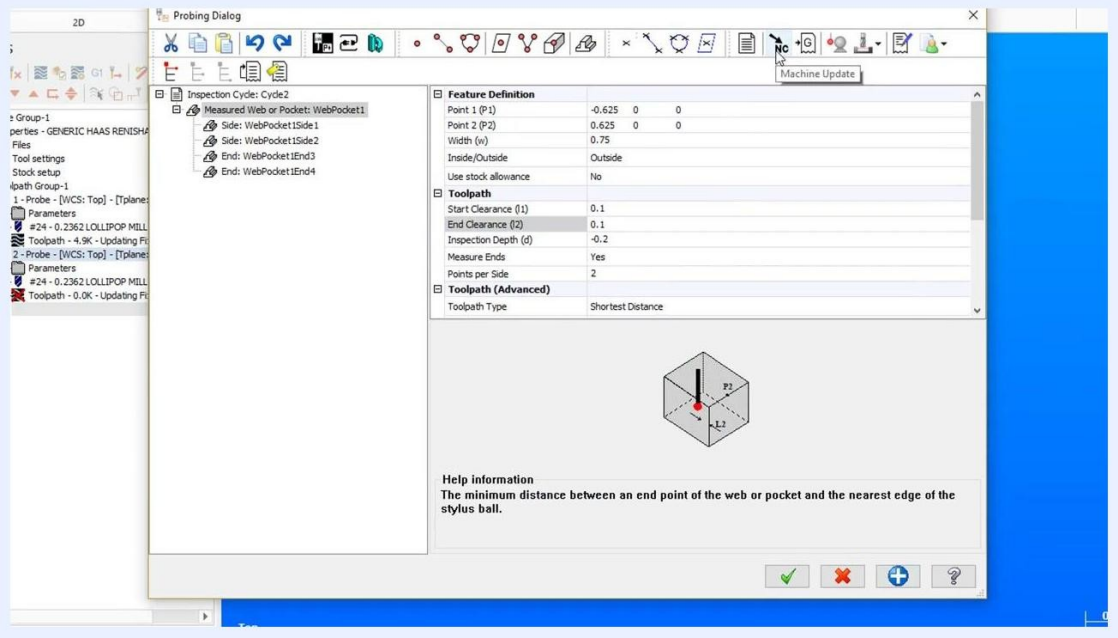
Step 6: Simulate the probe routine

Open the Mastercam Simulator. The **Verify** tab offers toolpath verification, collision checking, and material cutting. The 3D view shows the VF2 Probe approaching a red workpiece.

The **Move List** panel on the right shows move details, including:

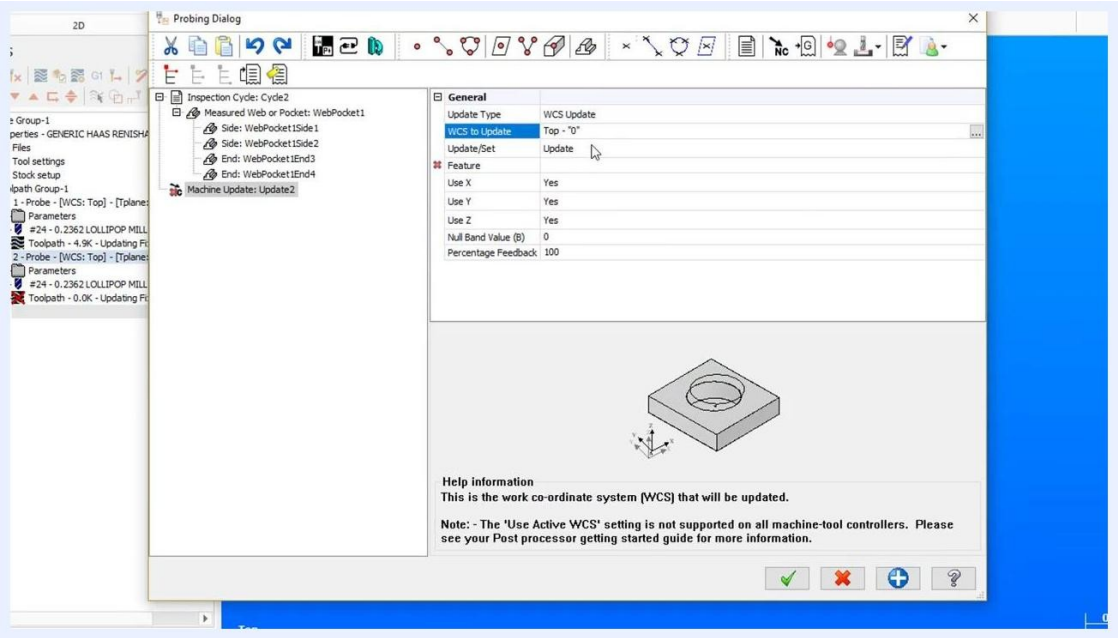
- Move Type: Rapid Linear Move
- Operation: N Probe
- Tool Number: #24 - VF2 Probe

- Feed length, time, rapid length, and total time



Mastercam simulation showing the probe approaching the workpiece with move details

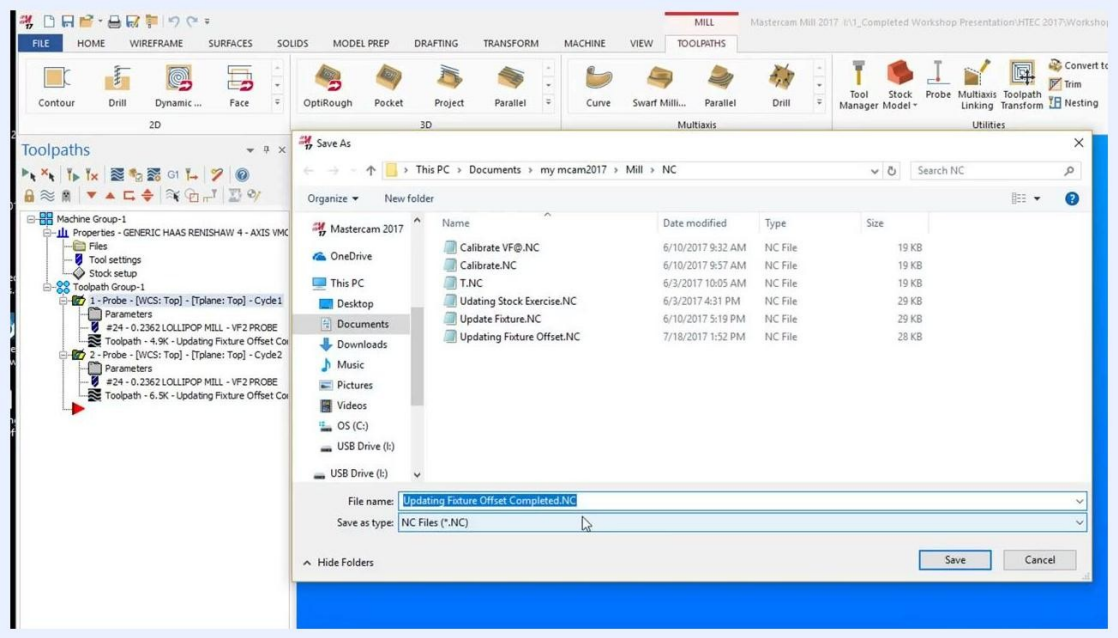
Run the simulation to the end. The **Move List** shows the final move (Move ID 32 of 32), and the 3D view shows the probe path as toolpath lines. The status bar shows "Enhancing Model 100%" and "Enhancing View 100%".



Mastercam simulation with the completed probe toolpath and move list

Step 7: Save the NC file and review the post log

In the **Save As** dialog, save the file as "Updating Fixture Offset Completed.NC". The example folder is "This PC > Documents > my mcam2017 > Mill > NC", and the file type is **NC Files (*.NC)**.

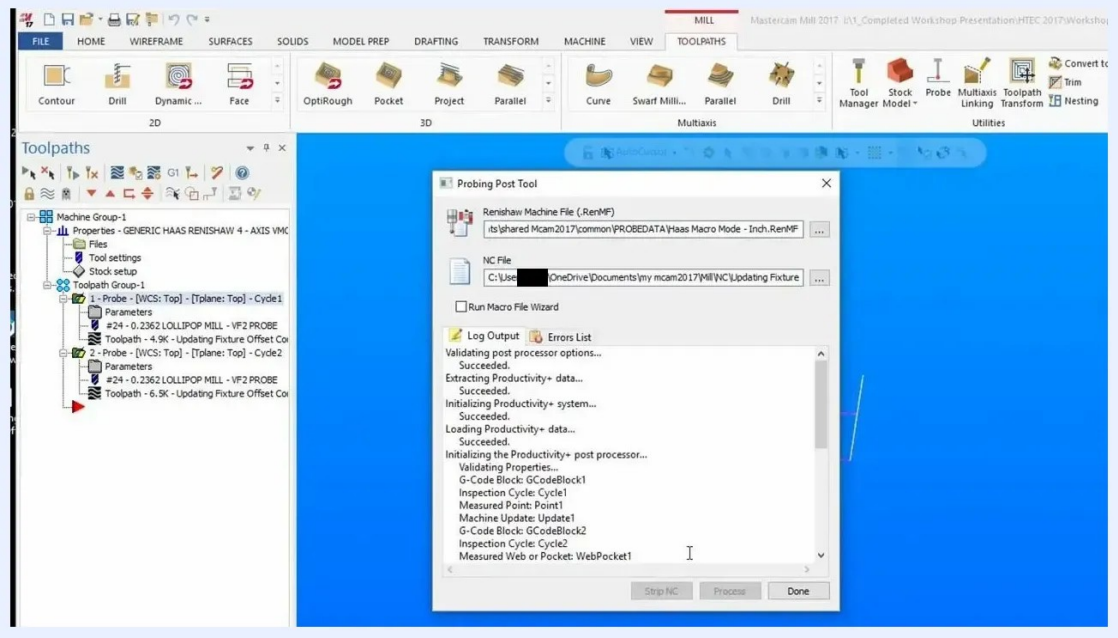


Save As dialog for the NC file named Updating Fixture Offset Completed.NC

The **Probing Post Tool** dialog then opens. It shows the Renishaw Machine File and NC File paths. The log lists the Productivity+ steps and the inspection cycles:

- Validating post processor options
- Extracting Productivity+ data
- Initializing Productivity+ system
- Loading Productivity+ data
- Initializing Productivity+ post processor
- Validating Properties
- G-Code Block: GCodeBlock1
- Inspection Cycle: Cycle1 (Measured Point: Point1, Machine Update: Update1)
- G-Code Block: GCodeBlock2
- Inspection Cycle: Cycle2 (Measured Web or Pocket: WebPocket1)

Click **Done** to close the dialog.



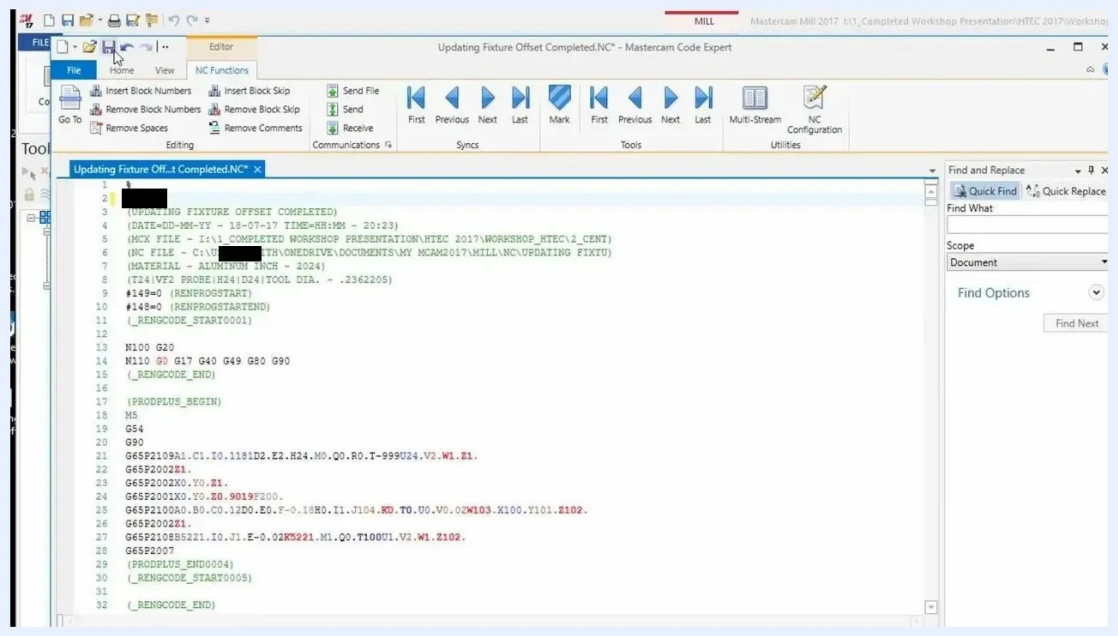
Probing Post Tool dialog with a log showing the successful post-processing steps

Step 8: Open the NC file in Code Expert

Open "Updating Fixture Offset Completed.NC" in Mastercam Code Expert. The header comments include:

- Program number: O7777
- Description: UPDATING FIXTURE OFFSET COMPLETED
- Date and time: 18-07-17, 20:23
- MCX file path and NC file path
- Material: ALUMINUM INCH - 2024
- Tool: T24 (VF2 PROBE), diameter 0.2362205

Below the header, you find initialization commands, probing cycles, and G-code blocks for the probing routine.



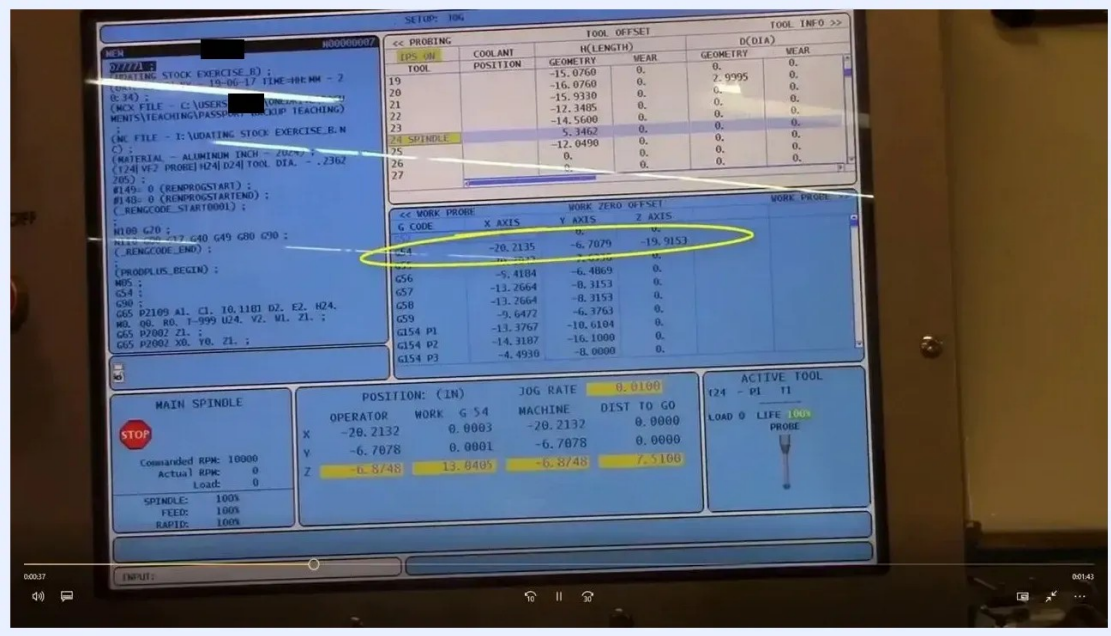
Mastercam Code Expert window displaying the generated NC file with probing and fixture offset update code

Step 9: Check the updated work offset on the CNC control

On the CNC control, find the **WORK ZERO OFFSET** section with G54 selected. After the probe routine, G54 shows these values:

- X AXIS: -20.2135
- Y AXIS: -6.7079
- Z AXIS: -19.9153

The bottom of the screen shows the main spindle status, position readouts, jog rate, and the active tool (T24 - PROBE). The highlighted values confirm the fixture offset update.



CNC control screen showing updated G54 work offset values with the probe tool active

Step 10: Insert a .015" shim under the workpiece

Place a .015" shim under the workpiece in the vise. Hold the shim with one hand and adjust the vise handle with the other. This keeps the workpiece supported and aligned.



Operator inserting a .015 inch shim under the workpiece in the vise, labeled with an arrow

Step 11: Reposition the probe over the workpiece

Position the probe above the workpiece, ready to contact the top surface. The setup matches the initial probing, so you repeat the process to verify the shim adjustment.



Probe repositioned above the workpiece after the shim adjustment

Step 12: Finish the probing cycle

Wait for the probing routine to finish and for the CNC control to display the new offset values. Follow all safety protocols before you continue to the next machining operation.



CNC machine interior with the probe contacting the workpiece after the shim adjustment

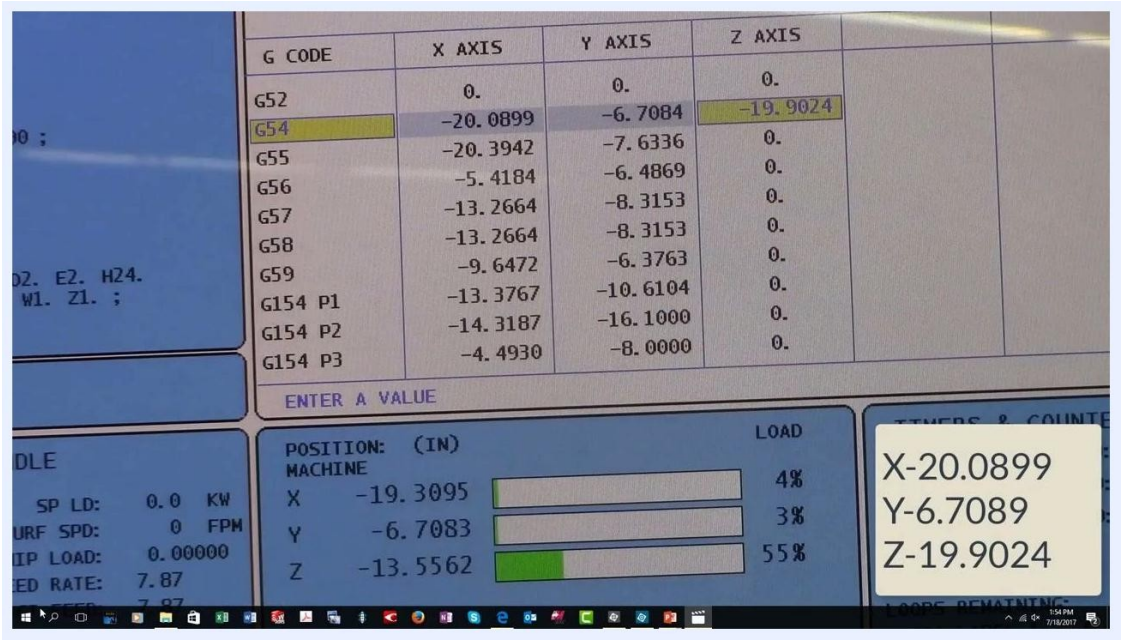
Compare work offset values

Step 13: Compare the offsets with the work probe results

On the CNC control, review the work offsets for the G codes (G52, G54, G55, and so on). G54 is highlighted with these values:

- X: -20.0899
- Y: -6.7084
- Z: -19.9024

The lower section of the screen shows the current machine position and the load percentage for each axis.



CNC control screen showing G-code offsets, machine position, and load indicators, with G54 highlighted

Next, compare these values with the **WORK PROBE** values for G54. Yellow ovals highlight both sets:

- X: -20.2135
- Y: -6.7079
- Z: -19.9153

Use this comparison to verify the probe results against the programmed offsets.

Update a tool diameter offset

Step 14: Open the Mastercam file

The CAD/CAM Programming MFGT220 slide gives the task: create a probe routine to update a tool diameter offset. A Microsoft Office security dialog asks whether to open "Update Tool Dia Offset Ready for Probe.emcam". Click **OK** to open the file.

Step 15: Select the geometry for probing

In Mastercam, zoom in on the 3D model. With the **Select** tool active, choose the circular feature to probe. The options **End Selection** and **Clear Selection** are available, and the probe toolpath is highlighted.

Step 16: Configure the probing parameters

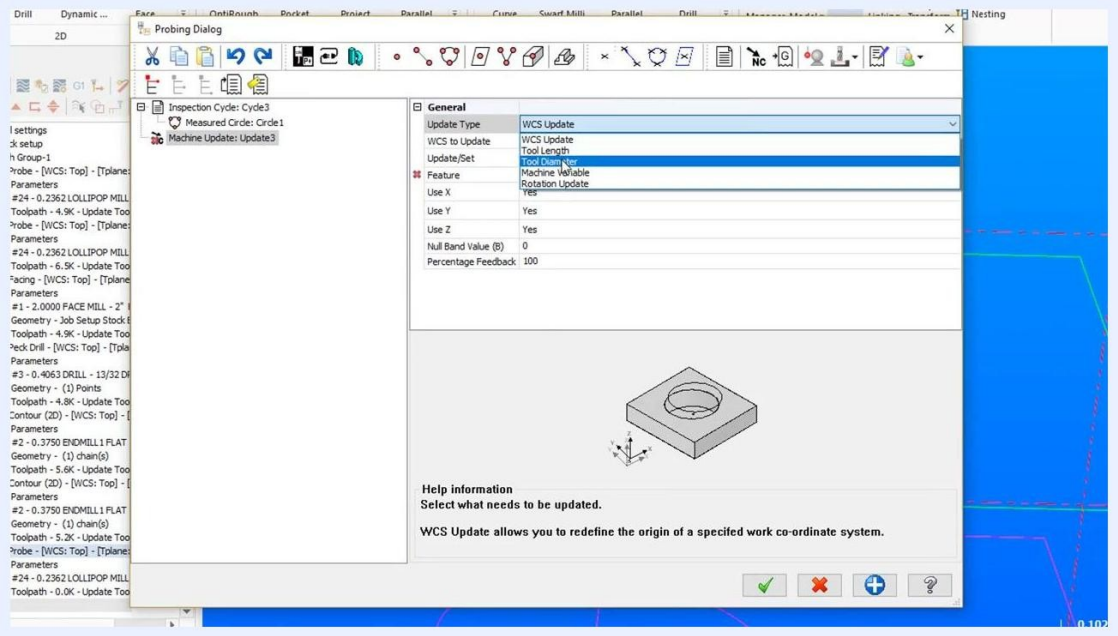
In the **Probing Dialog**, the inspection cycle lists the measured feature, for example "Measured Circle: Circle1". Under **General**, set:

- Update Type: WCS Update
- WCS to Update: WCS Update
- Update/Set: Tool Diameter (select it from the dropdown)

Under **Feature**, set:

- Feature: Machine Variable
- Use X: Yes
- Use Y: Yes
- Use Z: Yes
- Null Band Value (B): 0
- Percentage Feedback: 100

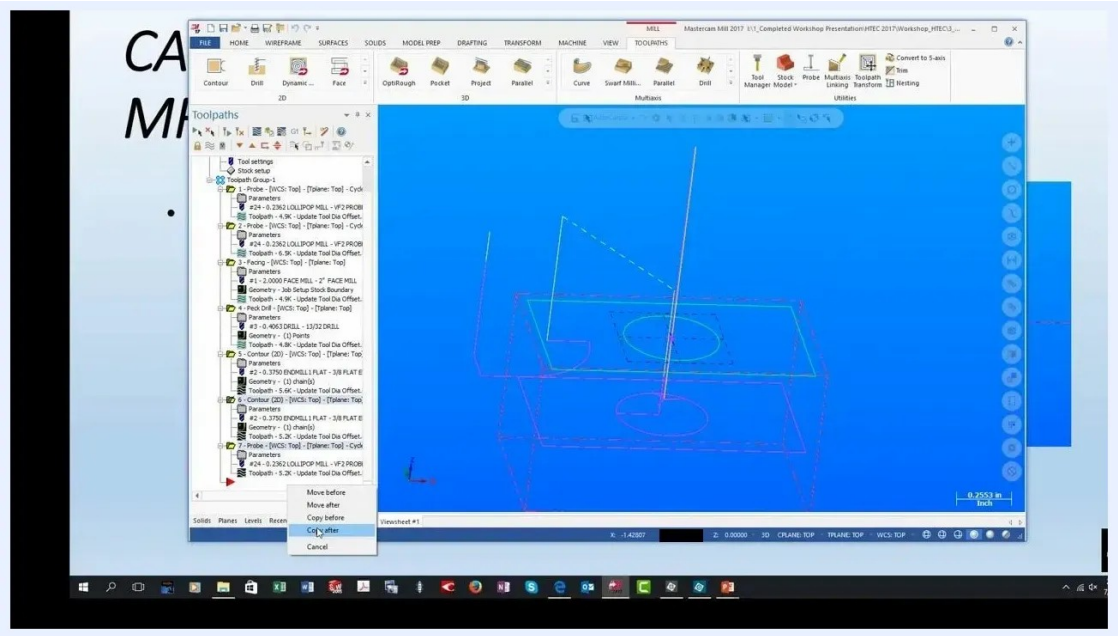
The help text explains that WCS Update lets you redefine the origin of a specified work coordinate system.



Probing Dialog in Mastercam showing the WCS Update and Tool Diameter configuration with all axes enabled

Step 17: Copy the probe operation

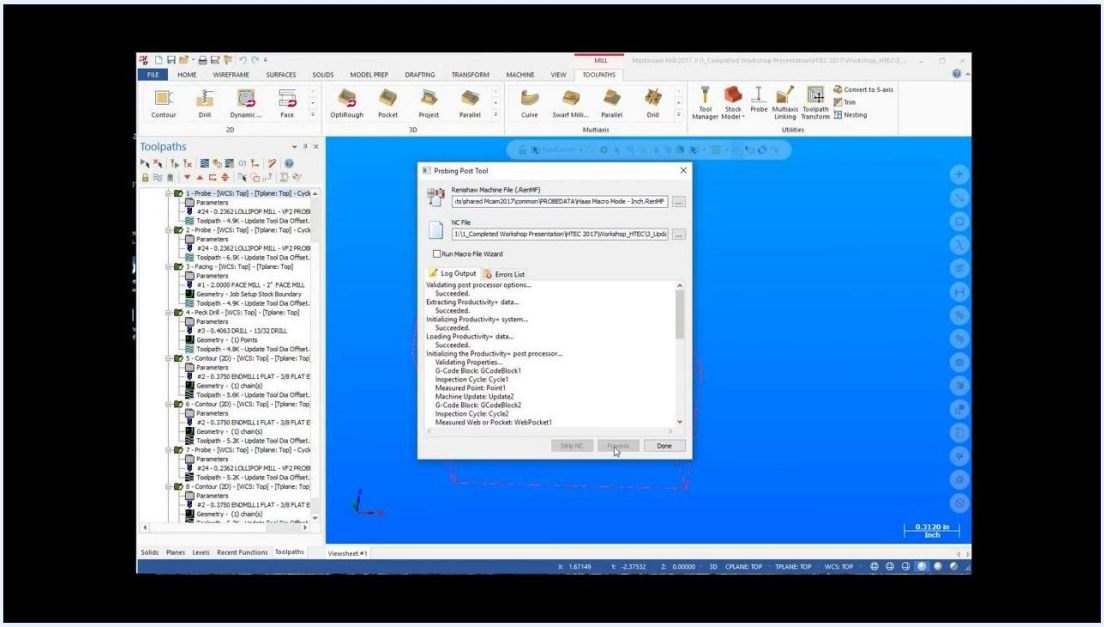
In the **Toolpaths** panel, right-click a probe operation. The context menu offers **Move before**, **Move after**, **Copy before**, **Copy after**, and **Cancel**. Select **Copy after** to duplicate the probe operation after the selected toolpath. This keeps the probe routine in the correct sequence.



Context menu in the Mastercam Toolpaths panel with Copy after highlighted for a probe operation

Step 18: Save the NC file

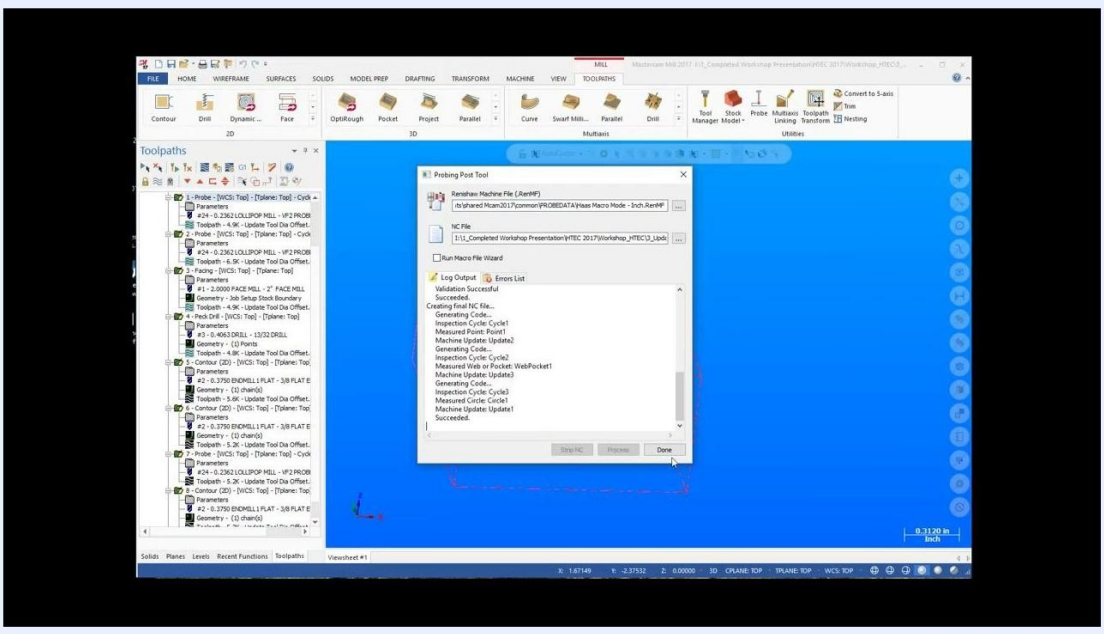
In the **Save As** dialog, name the file "Update Tool Dia Offset.NC". The example folder is "HTEC 2017\Workshop_HTEC\3_Update Tool Dia Offset". Confirm the name and location, then click **Save**.



Save As dialog in Mastercam for saving the NC file as Update Tool Dia Offset.NC

Step 19: Post process the probe routine

The **Probing Post Tool** dialog shows the selected post processor (Renishaw Machine File), the NC file path, and a log. The log lists the validation steps, code generation, and inspection cycles, such as "Measured Point: Point1", "Measured Web or Pocket: WebPocket1", and "Machine Update: Update2". Click **Done** to finish.



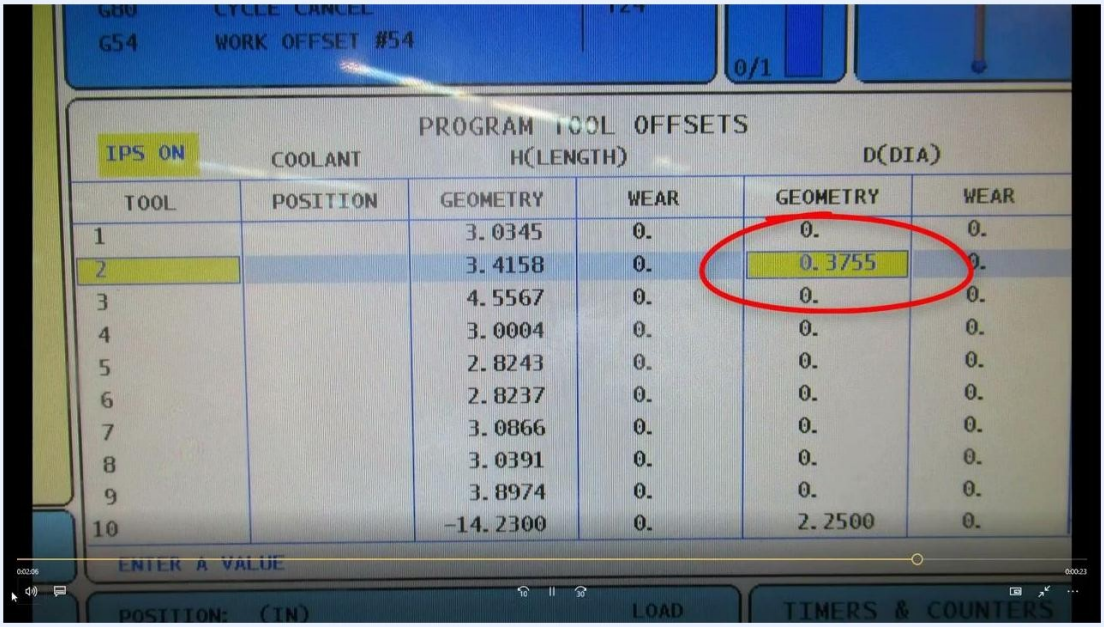
Probing Post Tool dialog in Mastercam showing the post-processing log and successful code generation

Step 20: Check the tool offset on the CNC control

On the CNC control, open the **PROGRAM TOOL OFFSETS** table. Tool #2 is highlighted with these values:

- Position: 3.4158
- Geometry: 3.4158
- Wear: 0.0
- Diameter Geometry: 0.3755 (circled in red)
- Diameter Wear: 0.0

The IPS is ON and the work offset is set to G54.



CNC control panel showing PROGRAM TOOL OFFSETS with Tool #2 highlighted and Diameter Geometry 0.3755 circled

Summary

In this guide, you built probe routines in Mastercam to update a fixture offset and a tool diameter offset. You posted each routine with the Renishaw machine file, where the log shows the Productivity+ data being extracted, initialized, and loaded. You then confirmed the updated G54 work offset and the tool diameter offset on the CNC control.

What's next

After you confirm the offsets on the control, continue to the next machining operation. Follow your shop's safety protocols before you start.