Introduction

This document describes operation and an overview of the tool and part probe canned cycles in conversational format. Probing is an option in 3000M and 5000M CNC products, and is standard in 6000M. The cycles provided perform the most common tool and part probing functions. If Probing has been added post-sale, besides Setup Utility changes, there may be Integral Programmable Intelligence (IPI) program modifications required.

The tool probe cycles are only supported on machines with automatic spindle forward/reverse and spindle speed, and homing with a permanent X, Y, and Z machine position. The method described assumes the use of negative tool-length offsets. In this method, the Tool-Length Offset (TLO) in the length column for each tool represents the distance from the tool tip at machine home to top of work piece and is a negative number. This method does not require the use of any Z work coordinate offset to be active. This procedure will find the effective tool diameter by turning the spindle on in reverse and touching two sides of the probe stylus, then storing the tool's diameter in the tool's diameter offset table.

The part probing cycles are designed to assist in part setup. Using these cycles, one or more features (edges) of a part can be measured. Using the data obtained with these measurements, calculations are made that can be used to set a given fixture offset. It is also possible to find the orientation angle of a part so as to not always have to align the part exactly.

Tool and part probing do not allow rotation, scaling, and mirroring. If they are active before calling the canned cycles, they will be deactivated. Plane will be set to XY when these cycles are complete.

Tool Probe Cycles

Before using your tool probe and tool probe cycles, you must setup the probe following the manufacture's specifications.

The tool probe will update the tool registers only. If you are going to use the tool being measured after the probing cycle, you must recall that tool for the new offsets to be active.

This section covers the following topics:

❑ Tool Probe cycle designations

❑ System Variable settings

❑ Detailed description of all tool probe cycles

15-August-03

Tool Probe Cycle Designations

The following summarizes the cycles available:

Probe Calibration (CalibTlPrb)	Tool Probe Calibration Cycle

This is used to set the Z datum for length preset, the effective probe stylus diameter for setting tool diameter registers, and establishes the center of the probe stylus.

> **NOTE:** Calibration must be done at least once before using the tool probe. Once the probe has been calibrated, calibration does not need to be done unless the probe is moved or a new part is being setup.

Length and Diameter (LenDiamMea)	Tool Length and Diameter Offset Preset

Updates length and diameter tool registers.

> **NOTE:** If the tool has a hole on the bottom so that the probe would fall between the tool teeth, do not use this cycle. Damage to the probe could result. In this case, use **Length Special** for manual length preset or **Diameter Special** for manual diameter preset.

Length Special (LenSpecMea)	Manual Tool-Length Offset Preset

Updates tool-length register. To be used for large face mill style tools or shell mill tools that has a hole in the center of the bottom of the tool.

Diameter Special (DiaSpecMea)	Manual Tool Diameter Preset

Updates tool diameter register for irregular shaped tools or tools with a hole in the center of the bottom.

Break and Wear (BrkWearDet)	Tool Breakage, Length and Diameter Wear Detection Breakage

Checks the tool and gives an alarm if not within tolerance.

Length and Diameter Wear – Check the Length and/or Diameter and updates the Length and/or Diameter wear registers up to a user-defined limit. Once the user-defined limit has been reached, the cycle will give an alarm.

System Variable Settings

Before you set the parameters for the tool probe you must:

❑ Know the diameter of the calibration standard.

❑ Know that the calibration standard is a standard that is specifically designed for calibrating the probe. The **DiamOfStd** parameter is the diameter of the part of the calibration standard that comes in contact with the probe stylus during calibration and should be an exact measurement.

15-August-03

❑ Ensure when entering the values in the parameters that you are in the same units (inch/metric) as when you are running the tool preset canned cycle. Set the units in parameters to the units in which you are going to run the cycles. See "Units" in 3000M Setup Utility. If you run programs in Inch, then the unit parameter must be set to Inch, and all probe system variables must be entered in Inch. If you want to switch to millimeter programs and use these tool-preset cycles, you must go back and change the unit parameter to metric then reset the values in these variables and recalibrate the probe stylus.

See the 3000M CNC Setup Utility Manual, P/N 70000499, "Section 2 - Builder Setup" for a detailed description of the setup parameters.

> **NOTE:** 3000M contains a "Probing" section within "Section 2 - Builder Setup" where the probing parameters are described.

1. Set 3-D probe, this should be set to Corded, Cordless, or Cordless SG for a strain gauge type probe.

2. Set **Nominal probe stylus diameter**, the overall nominal probe stylus diameter. For example, to 0.5" (12.7 mm) for the Renishaw probe, or for the Heidenhain probe use 1.575" (40 mm). This is dependent on the probe style and specifications (refer to your probe documentation).

3. Set **Maximum stroke from home for first pick**, represents the distance from machine Z home with the shortest tool or the spindle face to just below the probe stylus top as the maximum stroke for the initial probe pick.

4. Set **RPM for calibration and tool measurement**, the spindle spin RPM for tool touch. (For example, set to 800.)

5. Set **Probe orientation**, the proper probe orientation. For example, if set to –1, the probe should be installed on the right side of the table pointing toward the left in the –X direction. See **Table 1**.

Table 1, Probe Orientation Settings

Probe Orientation Setting	Direction
1	Probe is pointing to the right as you are facing the machine in +X direction
-1	Probe is pointing to the left of the machine in the -X direction.
2	Probe is pointing away from you, toward the back of the machine in the +Y direction
-2	Probe is pointing toward you, toward the front of the machine in the -Y direction

6. Set **Z first pick, FAST feedrate**, the Z fast feedrate. [For example, set to 50.0 inches per minute (in/min) (2540 mm per minute (mm/min).]

> **Warning:** When using **Length and Diameter**, the tool will travel down beyond the top of the probe after the probe is tripped. For this reason, make sure that the fast feedrate is not so high as to cause the tool to travel past the probe travel causing damage to the probe. The maximum feedrate that can be used is specific to the machine and may need to be set much lower to prevent damage to the probe.

7. Set the **Z first pick, Z first pick, MEDIUM feedrate**, the Z medium feedrate. [For example, set to 5.0 in/min (127 mm/min).]

8. Set **Z first pick, SLOW feedrate**, the Z slow feedrate. [For example, set to 1.0 in/min (25.4 mm/min).]

9. Set **Z retract amount**. [For example, set to 0.2" (5.08 mm).]

10. Set **XY retract amount**. [For example, set to 0.2" (5.08 mm).]

11. Set **Z rapid to start position from home**. Install the longest tool in the spindle and bring the Z-axis to machine home. With a tape measure, measure the distance from the tool tip to within 0.5" (12.7 mm) above the top of the probe stylus and enter that number into **Z start position**. When using **Length and Diameter**, this will cause the tool to rapid to this position in the Z-axis before starting the initial probe touch in the Z-axis. This will save time especially if the Z-feed must be set relatively slow to prevent probe over travel after the probe has been tripped.

12. Set **Diameter of tool probe gauge**. The default gauge diameter of the tool calibration standard. **Diameter of tool probe gauge** can be overwritten by the **DiamOfStd** address word in the **Probe Calibration** cycle. **Diameter of tool probe gauge** is used in the **Probe Calibration** calibration only. [For example, set to 2.0" (50.8 mm).]

❑   For tool probing or tool length presetting, Tool-Length Offset (TLO) is the distance from machine home to top of work piece or wherever you set your part "**Z**" zero.

❑   Before starting to set your tools, you must calibrate the probe. Once the probe has been calibrated, calibration does not have to be done again unless you remove the probe or replace the stylus.

❑   Recalibration may also be required if the Z location of the top of the part changes, and is not compensated for by a Z work offset.

Description of Tool Probe Cycles

This section contains detailed descriptions of the tool probe cycles:

*   Tool Probe Calibration Cycle (CalibTlPrb)
*   Tool Length and Diameter Offset Preset (LenDiamMea)
*   Manual Tool Length Measure for Special Tools (LenSpecMea)
*   Manual Tool Diameter Measure for Special Tools (DiaSpecMea)
*   Tool Breakage, Length, and Diameter Wear Detection (BrkWearDet)

Tool Probe Calibration Cycle (CalibTlPrb)

Format: CalibTlPrb DiamOfStd(n) DistDown(n)

This cycle is used to calibrate the probe. This is used to set the Z datum for length preset, establishing the center of the probe stylus, and the effective probe stylus diameter for setting tool diameter registers. Refer to **Table 2**.

Table 2, CalibTlPrb Entry Fields

Entry Fields	Description
DiamOfStd	The diameter of the part of the calibration standard that comes in contact with the probe stylus during calibration. This should be an exact measurement. (Optional override for **Diameter of tool probe gauge**)
DistDown	The distance to go down along the side of the probe stylus with the probe calibration standard when touching the side of the stylus for diameter calibration. The maximum **DistDown** value is 0.55" (13.97 mm). Without any **DistDown** value, the cycle will bring the calibration standard down past the top of the probe stylus the default 0.1" (2.54mm). If you put a number higher than 0.55" (13.97 mm), the control displays an error. (Optional) [Default: .1"]

To calibrate the tool probe:

1. Jog the calibration standard (the calibration standard should be in the spindle) to the top of your work piece, and set its tool-length offset to the top of the work piece or to wherever you would like your Z zero to be. To calibrate the tool number:

   a) Jog the tip of the calibration standard to the proper spot

   b) Press the **Calib Z** function key.

2. Manually jog the calibration standard over the probe stylus center and less then 0.2" (5.08 mm) above the probe stylus. It should be no more then 0.062" (2.0 mm) from the center of the stylus.

3. From the MDI mode, pick
   F5 (**Mill**) → **F10** (**Probe**) → **F1** (**ToolPro**) → **Probe Calibration**
   For example: **CalibTlPrb** exit by pressing **F9** twice and **F10** to exit.

4. The Z-axis will initially go down and touch the top of the probe stylus at the feedrate specified in **Z first pick, MEDIUM feedrate**. Then retouch at the slow feedrate, **Z first pick, SLOW feedrate**, establishing the zero probe stylus top.

5. Then incrementally rapid up whatever value that is in **Z retract amount**.

15-August-03

6. The spindle will come on at the RPM specified in the **RPM for calibration and tool measurement** and then the calibration standard will move over an incremental amount that is equal to (Half the value entered in the **DiamOfStd** parameter (or Setting Diameter of probe gauge) + Half the value entered in **Nominal probe stylus diameter** + The value in **XY retract amount**). The direction the probe will move over depends on what is placed in **Probe orientation**:

   –1   Go first to the left

   1   Go first to the right

   –2   Go first to the front

   2   Go first to the back

7. The Z-axis will then do a guarded Z move down 0.1" (2.54 mm) or whatever amount was placed in the **DistDown** parameter and then move over toward the probe stylus 0.3" (7.62 mm) or until it touches the probe stylus. If contact is not made with the probe or if contact is made during a guarded move, then an alarm will be generated and the canned cycle will terminate.

8. After the probe stylus is touched on the first side, the machine will then rapid up and over the stylus then down on the opposite side then over to the other two sides until it has touched the probe stylus on all four quadrants. This will establish the center of the probe stylus.

9. The spindle will then turn off and the machine will touch off on two sides of the probe with the spindle off finding the effective probe stylus diameter. Then, will rapid up above the probe stylus and over to the center.

10. Remove the calibration standard. You are now ready to start running the **Length and Diameter** cycle (or one of the other cycles for setting or checking length and diameter of the tool) to set your tool-length offsets or tool diameter registers.

Tool Length and Diameter Offset Preset (LenDiamMea)

Format: LenDiamMea Tool#(tool#) EstDiam(n) MeasType (Length, Diameter, or Both) DistDown(n) OvrFstFeed(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

Each tool must have the length set once before trying to set the diameter. Call this cycle up the first time using **Both** because it will automatically set the length first then the diameter.

❑ Calibrate the tool probe at least once before trying to automatically preset a tool. This is done initially, but if the stylus is ever changed or the probe is moved, then you must again calibrate the tool probe.

❑ This tool preset (**LenDiamMea**) can be run from within a program or from the MDI mode. Refer to **Table 3**.

Table 3, LenDiamMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required)
	With only the **Tool#** parameter present, the canned cycle will not step over half the tool's diameter but come straight down measuring the tool length and storing it in the tool register.
EstDiam	This is the rough diameter of the tool. This should be within 0.04" (1.0 mm). (Optional)
	If the **EstDiam** parameter is present, the tool will step over half of its diameter, the spindle will turn on in reverse and then the canned cycle will measure the tool's length.
	A negative **EstDiam** value is for a left-handed tool and will cause the spindle to come on forward instead of reverse.
	For on center length measurement, do not give a **EstDiam** parameter.
MeasType	The option specifies to measure length, diameter, or both and the appropriate tool registers are updated. (Optional) [Default: **Length**]
	Diameter Measure the diameter only **Length** Measure the length only **Both** Measure both length and diameter
	If **Length** is not set, the cycle will measure the tool length only.
	If **Diamete**r or **Both** are programmed, you must also have a **EstDiam** parameter or the control will display an error message.

(Continued…)

Table 3, LenDiamMea Entry Fields (Continued)

Entry Fields	Description
DistDown	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum **DistDown** value is 0.55" (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55" (13.97 mm), the control will issue an error message. If **DistDown** is not set, the cycle will use a default value of 0.1" (2.54 mm). (Optional) [Default: 0.1"] Ball nose cutters and special cutters that require a move down more than 0.55" (13.97 mm) are not supported.
OvrFstFeed	This is the override for the fast Z feedrate that was set in the User definable variable **Z first pick, FAST feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the User definable variable **Z first pick, MEDIUM feedrate**. This is used for the same reason as the **OvrFstFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the User definable variable **Z first pick, SLOW feedrate**. This is used for the same reason as the **OvrFstFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the User defined variable **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrFstFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original RPM. (Optional)

To use the tool preset probing cycle:

1. Install all the tools you wish to set, in the tool changer.

2. Input:
   LenDiamMea Tool#(tool#) EstDiam(tool rough diameter) MeasType

   If run from the inside of a program, this line needs to be repeated for every tool that you want to set.

3. Execute that line if you are in MDI mode, or run the program if you have set all the tools up in a program.

4. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, all your tools are measured and ready for use.

Shell mill style tools that have a hole in the center of the bottom will not work with this canned cycle; in this case, you must use the manual canned cycles **LenSpecMea (Length Special Manual Tool Length Measure for Special Tools)** for length and **DiaSpecMea (Diameter Special Manual Tool Diameter Measure for Special Tools)** for diameter. See Table 3, LenDiamMea Entry Fields. This cycle is only good for drills, taps, reamers, ball nosed endmills, and standard endmills with a flat bottom, the cycle updates length and diameter tool registers clearing anything in the wear registers.

Format: LenDiamMea Tool#(tool#)

With **Tool#** parameter only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** parameter and rapid directly over the center of the probe stylus.

2. The Z-axis will rapid down the distance placed in **Z rapid to start position from home**, then start feeding down toward the probe for the initial touch at the feedrate that was placed in **Z first pick, FAST feedrate**, then will back up and retouch the probe at the feedrate that is in **Z first pick, SLOW feedrate**.

3. The tool-length register for that tool is now updated, and that tool's length-wear register is set to zero.

4. Then the Z-axis will rapid up to home position.

5. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 4 until all the tools have been measured.

Format: LenDiaMea Tool#(tool#) EstDiam(tool rough diameter)

With **Tool#** and **EstDiam** parameters only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** parameter and rapid directly over the center of the probe stylus.

2. The Z-axis will rapid down the distance placed in **Z rapid to start position from home** then start feeding down toward the probe for the initial touch at the feedrate that was placed in **Z first pick, FAST feedrate** then will back up.

3. The machine will rapid over half the diameter of the cutter from the probe stylus center in the direction related to the **Probe orientation**.

4. The spindle will then come on in reverse at the RPM specified in **RPM for calibration and tool measurement** and retouch the probe at the feedrate that is in **Z first pick, SLOW feedrate**.

5. The tool-length register for that tool is now updated, and that tool's length-wear register is set to zero.

6. Then the Z-axis will rapid up to the home position.

7. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 6 until all the tools have been measured.

Format: LenDiaMea Tool#(tool#) EstDiam(tool rough diameter)
MeasType (Both)

With **Tool#**, **EstDiam**, and **MeasType** parameters set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** parameter and rapid directly over the center of the probe stylus.

2. The Z-axis will rapid down the distance placed in **Z rapid to start position from home** then start feeding down toward the probe for the initial touch at the feedrate that was placed in **Z first pick, FAST feedrate** then will back up.

3. The machine will rapid over half the diameter of the cutter from the probe stylus center in the direction related to the **Probe orientation**.

4. The spindle will then come on counter clockwise at the RPM specified in **RPM for calibration and tool measurement** and retouch the probe at the feedrate that is in **Z first pick, SLOW feedrate**.

5. The tool-length register for that tool is now updated, and any value in the length wear register will be reset to zero.

6. Then the Z-axis will rapid up above the probe stylus the distance specified in **Z retract amount** and then rapid the X & Y axes over the center of the probe and turn the spindle on in reverse.

7. The machine will move the tool's edge off to one side of the probe stylus in the direction indicated in **Probe orientation** before making a guarded move down 0.1" (2.54 mm) or whatever value has been placed in the **DistDown** Parameter.

8. The machine will then touch the tool to the probe stylus on two opposite sides at the feedrate specified in **Z first pick, MEDIUM feedrate** with the spindle running at the RPM specified in **RPM for calibration and tool measurement**, backing up 0.02" (0.508 mm) after each first touch then retouching and the feedrate specified in **Z first pick, SLOW feedrate**, calculating the diameter of the tool and placing the calculated diameter value in the diameter register for the tool being preset and any value in the diameter wear register will be reset to zero.

9. Then the Z-axis will rapid up to the home position.

10. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 9 until all the tools have been measured.

15-August-03

Manual Tool Length Measure for Special Tools (LenSpecMea)

Format: LenSpecMea Tool#(tool#) DiamOfStd(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

This cycle is used to measure the length of large face mill style tools that have a hole in the center of the bottom of the tool. Refer to **Table 4**.

Table 4, LenSpecMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) With only the **Tool#** parameter present, the spindle will turn on in reverse and the canned cycle will come straight down measuring the tool length and storing it in the tool-length register. The **Tool#** parameter must the same as the current tool in the spindle.
EstDiam	This is the rough diameter of the tool and is only used in this cycle to determine if the spindle should be turned on in reverse or forward. If you have a left-handed tool you would give a negative value to the diameter. If this parameter is left off the control will always turn on in reverse by default. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the User definable variable **Z first pick, MEDIUM feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the User definable variable **Z first pick, SLOW feedrate**. This is used for the same reason as the **OvrMedFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the User defined variable **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrMedFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original RPM. (Optional)

> **Warning:** Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for just the specific tool being checked or set.

You must have the tool positioned over the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less then 0.200" (5.08 mm).

> **NOTE:** If the spindle is locked, you may have to unlock it to manually orient the tool tooth over the probe stylus.

To measure the tool length:

1. Jog the tool to the top of the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus.

2. From the MDI mode, input:

   LenSpecMea Tool#(tool#) EstDiam(n)

   and press the **START** button. Where **Tool#** is the tool number and **EstDiam** is roughly the diameter of the special tool.
   For example: LenSpecMea Tool# 3 EstDiam 3.5

3. The spindle will turn on in reverse and the Z-axis should go down and touch the top of the probe stylus keeping the X and Y position the same then rapid up 0.02" (0.508 mm) and then retouch using the slow feedrate programmed in the machine variables. The cycle will then update the tool-length offset register clearing any value in the length wear register, turn the spindle off and return the tool to the Z height where it started.

4. The Tool Length has been set and you can now change to another tool, and repeat steps 1 through 3.

15-August-03

Manual Tool Diameter Measure for Special Tools (DiaSpecMea)

Format: DiaSpecMea Tool#(tool#) EstDiam(n) DistDown(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

This cycle is used to measure the diameter of irregularly shaped tools or tools with a hole in the center of the bottom. Refer to **Table 5**.

Table 5, DiaSpecMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required)
	The **Tool#** parameter must be the same as the current tool in the spindle.
EstDiam	This is the rough diameter of the tool. (Required)
	The diameter specified in this parameter should be larger then the actual diameter of the tool being measured but no more then 0.04" (1.0 mm) over. If you have a left-handed tool, you would give a negative value to the diameter so the spindle will turn on in forward direction.
DistDown	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum **DistDown** value is 0.55" (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55" (13.97 mm), the control will issue an error message. If **DistDown** is not set, the cycle will use a default value of 0.1" (2.54 mm). (Optional) [Default: 0.1"]
	Ball nose cutters and special cutters that require a move down more than 0.55" (13.97 mm) are not supported.
OvrMedFeed	This is the override for the medium feedrate that was set in the User definable variable **Z first pick, MEDIUM feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the User definable variable **Z first pick, SLOW feedrate**. This is used for the same reason as the **OvrMedFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)

(Continued…)

ANILAM

Table 5, DiaSpecMea Entry Fields (Continued)

Entry Fields	Description
OvrRPM	This is the override for the RPM that was set in the User defined variable **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrMedFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original RPM. (Optional)

> **Warning:** Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for just the specific tool being checked or set.

You must:

❑ Load the tool in the spindle and call up that tools offset.

❑ Know the distance from the top of the probe stylus down you will have to move so that the largest part of the tool diameter is even with the side of the probe stylus for diameter measurement. That value will be placed in **DistDown** if different then the default 0.1" (2.54 mm).

❑ Position the tool over the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less then 0.200 (5.08 mm).

To measure the tool diameter:

1. Jog the tool to the top of the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus.

2. From the MDI mode and the spindle off, input:

   DiaSpecMea Tool#(n) EstDiam(n) DistDown(n)

   and press the **START** button. Where **Tool#**(n) is the tool number, **EstDiam**(n) is roughly the diameter of the special tool (this should be larger but not more then 0.100" (2.54 mm) larger), and **DistDown**(n) is the Z-axis move down needed if different then the default 0.100" (2.54mm) so that the largest part of the tool diameter comes in contact with the edge of the probe stylus.
   For example:
   DiaSpecMea Tool# 3 EstDiam 3.5 DistDown .25
   exit and press the **START** button.

15-August-03

3. The Z-axis will feed down with the spindle on, touching the top of the probe stylus. Once the top of the probe is found, the Z-axis will rapid back up above the probe and move over to one side of the probe stylus. The tool will then move down the distance in **DistDown** or 0.1" (2.54 mm) if **DistDown** is not programmed. Then, with the spindle turning in reverse, the canned cycle will touch the side of the tool to the probe stylus twice on opposite sides establishing the tool's diameter. The new diameter will then be stored in that tool's diameter register and clear any value in the diameter wear register. The Z-axis will rapid up to machine home.

4. The Tool Diameter has now been set and you can change to another tool and repeat steps 1 through 3.

Tool Breakage, Length, and Diameter Wear Detection (BrkWearDet)

Format: BrkWearDet Tool#(tool#) EstDiam(n) MaxLenAdj(n) MaxDiaAdj(n) DistDown(n) Update(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

Refer to **Table 6**.

Table 6, BrkWearDet Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required)
	The **Tool#** parameter will be the tool number you want checked.
EstDiam	This is the rough diameter on the bottom of the tool. (Optional)
	The diameter specified in this parameter should be roughly the diameter on the bottom of the tool that you want to be over the center of the probe stylus. If you have a left-handed tool, you would give a negative value to this diameter so the spindle will turn on forward verses reverse. When stepping over for checking the diameter of the tool, this cycle will use the diameter in the tool table for the tool being checked.
MaxLenAdj	The maximum length wear value limit. The cycle will check to see if the cutter length has changed by more then this amount and will alarm, stopping the program if exceeded. If not set, the cycle will not check the tool length. (Optional)
	NOTE: At least one, **MaxLenAdj** or **MaxDiaAdj** must be set or the cycle will alarm.
MaxDiaAdj	The maximum diameter wear value limit. The cycle will check to see if the cutter diameter has changed by more then this amount and will alarm, stopping the program if exceeded. If not set, the cycle will not check the tool diameter. (Optional)
	NOTE: At least one, **MaxLenAdj** or **MaxDiaAdj** must be set or the cycle will alarm.
DistDown	The distance to go down along the side of the probe stylus when doing a diameter check. The maximum **DistDown** value is 0.55" (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55" (13.97 mm), the control will issue an error message. If **DistDown** is not set, the cycle will use a default value of 0.1" (2.54 mm). (Optional) [Default: 0.1"]
	Ball nose cutters and special cutters that require a move down more than 0.55" (13.97 mm) are not supported.

(Continued…)

Table 6, BrkWearDet Entry Fields (Continued)

Entry Fields	Description
Update	If this is undefined or set to **No**, the **Break and Wear** cycle will not update the diameter or length wear register each time it checks a tool. If set to **Yes**, the cycle will update the wear registers. In both cases the control will alarm when the maximum limit set in **MaxLenAdj** or **MaxDiaAdj** has been exceeded. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the User definable variable **Z first pick, MEDIUM feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the User definable variable **Z first pick, SLOW feedrate**. This is used for the same reason as the **OvrMedFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the User defined variable **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrMedFeed** parameter. This can only be set slower. Trying to set this higher will result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for just the specific tool being checked or set.

Warning: Running this cycle without first initially setting the length and diameter offset could result in damage to the probe and/or the machine tool. **CalTIPrb** (**Probe Calibration)** and **LenSpecMea (Length and Diameter) Automatic Tool Length and Diameter set**, or **LenSpecMea (Length Special Manual Tool Length Measure for Special Tools)** and **DiaSpecMea (Diameter Special Manual Tool Diameter Measure for Special Tools)** set must be run first before using the **BrkWearDet (Break and Wear)** cycle.

The **Break and Wear** cycle loads the tool, checks, and updates length and diameter wear registers, if specified, until a maximum value is exceeded, then it will alarm out stopping the program.

This cycle can be used in place of calling up a tool before running it.

You must know the distance from the top of the probe stylus down that you will have to move so that the largest part of the tool diameter is even with the side of the probe stylus for diameter measurement. That value will be placed in **DistDown** if different then the default 0.1" (2.54 mm).

To check the tool length and/or tool diameter for wear or breakage:

In place of the usual "**Tool#(tool#) MCode6**" command, use:

BrkWearDet Tool#(tool#) EstDiam(n) MJaxLenAdy(n) MaxDiaAdj(n) DistDown(n) Update(n)

at a tool change according to the instructions above and the control will check the tool prior to using it. To activate the new offset wear values you must call that tool with "Tool#(Tool#) MCode6" after this cycle has been run.

Part Probing Cycles

This section describes operation and an overview of the part probing cycles available in 3000M (Optional).

These cycles are designed to assist in part setup only.

Before using your part probe for part setup, you must set the probe up according to its manufacture specification so that it is set to turn on with a signal (if cordless) from the optical module sending unit and to automatically time out after approximately 120 seconds.

Rotation, Mirroring, and Scaling (RMS) subprograms are not allowed while running these cycles. If any of these cycles are in a subprogram, you cannot call them using RMS. Plane will be set to XY when these cycles are complete.

This section contains the following topics:

- ❑ Part Probe Cycle Designations
- ❑ System Variable Settings
- ❑ Part Probe Cycle Descriptions

ANILAM

Part Probe Cycle Designations

The following summarizes the cycles available:

CalibPtPrb   **Part Probe Calibration Cycle**
This is used to set the effective probe stylus diameter and set the compensation factor for any run-out of the probe stylus.

You will also need to calibrate the probe using the **CalibPtPrb** cycle.

> **NOTE:** On machines that do not have spindle orientation, the part probe stylus needs to be indicated true to the spindle centerline. In this case, the accuracy of the part probe is only as good as the stylus is running true to the spindle. Calibration must be done at least once before using the part probe. Once the probe has been calibrated, calibration does not have to be done again unless you replace the probe stylus.

EdgeFind   **Single Surface Measure/Edge Find**
This cycle will find a single surface and store that surface in a work register or fixture offset register if programmed.

CornerOut   **Outside Part Corner Find**
This cycle will find the X & Y surface on an outside corner of a part and store that location in a work register or fixture offset register if programmed.

CornerIn   **Inside Part Corner Find**
This cycle will find the X & Y surface in an inside corner of a part and store that location in a work register or fixture offset register if programmed.

InOutBoss   **Inside or Outside Hole or Boss Center Find**
This cycle will find the X & Y center of an inside hole or outside standing boss on a part and store that location in a work register or fixture offset register if programmed.

InOutWeb   **Inside or Outside Web or Slot Center Find**
This cycle will find the X or Y center of an inside or outside web or slot on a part and store that location in a work register or fixture offset register if programmed. The slot or standing web must be parallel to either the X- or Y-axes.

ProbeMove   **Protected Positioning Move**
This cycle allows for safe positioning of the probe around the part and will generate an alarm if an obstruction is encountered.

SkewComp   **Skew Error or Angle Find**
This cycle will make two touches on a surface in the X- or Y-axes and stores the angle relative to the 3 O'clock position. This cycle can also activate **SkewComp** at the same time as it is measured or in a subsequent call at another place in the program without measuring again.

System Variable Settings

Before you set the parameters for the Part probe you must:

❑ Know what the diameter of the Ring Gauge (calibration standard).

❑ Know that the Ring Gauge is a standard that is specifically designed for calibrating the probe. The **GaugeDiam** parameter is the diameter of hole that comes in contact with the probe stylus during calibration and should be an exact measurement.

❑ Ensure when entering the values in the parameters, that you are in same units (inch/metric) as when you are running the Part Probing canned cycle. Set the units in parameters to the units in which you are going to run the cycles. See "Units" in 3000M Setup Utility.

❑ If you want to switch to millimeter programs and use these part probing cycles, you must go back and change the unit parameter to metric then reset the values in these variables and recalibrate the probe stylus.

Set the following part probe cycle parameters:

1. Set **3-D probe type**, this should be set to Corded, Cordless, or Cordless SG for a strain gauge type probe.

2.–12. These parameters are for tool probe cycles.

13. Set **Probe Positioning Feedrate**, the feedrate the control will use while positioning the probe around the part. (For example, set to 200 in/min [5080 mm/min].)

14. Set **First Touch Feedrate**, the feedrate the control will use while making its initial touch finding the surface it is measuring. (For example, set to 50 in/min [1270 mm/min].)

15. Set **Nominal Probe Stylus Ball Diameter**, the diameter as measured with a micrometer. (For example, set to 0.2362" [6 mm].)

16. Set **Diameter of Spindle Probe Gauge**, the exact diameter of the ring gauge used to calibrate the spindle probe. (For example, set to 2.0" [50.80 mm].)

17. Set Invert probe logic to **No** for standard Normally Open (N.O.) probe interface and to **Yes** for a Normally Closed (N.C.) probe interface.

Part Probe Cycle Descriptions

This section contains detailed descriptions of the part probe cycles:

- Part Probe Calibration (CalibPtPrb)
- Edge Finding (EdgeFind)
- Outside Corner Finding (CornerOut)
- Inside Corner Finding (CornerIn)
- Out/Inside Boss/Hole Finding (InOutNpBoss)
- Out/Inside Web Finding (InOutWeb)
- Protected Probe Positioning (ProbeMove)
- Skew Error Find (SkewComp)

Part Probe Calibration (CalibPtPrb)

Format: CalibPtPrbn) Top(n) DistDown(n) DistBack(n) GaugeDiam(n) DistInX(n) DistInY(n)

Refer to **Table 7**.

Table 7, CalibPtPrb Entry Fields

Entry Fields	Description
Boss	Set **Boss** to **Yes** if you are calibrating to a boss verses a ring gauge; otherwise, do not set or set to **No**. Default is **No**. (Optional)
Top	If set to **Yes**, the cycle will find the top of the part before calibrating the probe. If **Boss** parameter is set to **Yes**, "**Top**" is forced to **Yes** as well; otherwise, the default is **No**. (Optional)
DistDown	The distance to go down from the top of the ring gauge or standing boss for calibration. This is only used if **Boss** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe down past the top of the ring gauge after finding the top, 0.1". (Optional)
DistBack	The **DistBack** parameter specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm) if not set. (Optional)
GaugeDiam	The diameter of the ring gauge hole the probe stylus will come in contact with. This is only to override the value in **Diameter of Spindle Probe Gauge** if needed and should be an exact measurement. (Optional)
DistInX	The distance from the starting point to move in the X-axis to find the top of the gauge. The default, if **Boss** is not set or set to **No**, is 0.1" (2.54 mm) beyond the edge of the ring gauge hole. If **Boss** is set to **Yes**, the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the gauge. The default is the current probe position. (Optional)

You must have:

1. The probe in the spindle.
2. The Ring Gauge mounted on the machine table.

To calibrate the probe:

1. Jog the probe to the approximate center of the ring gauge by eye and into the hole of the ring gauge at the depth that you wish the probe stylus to come in contact with the inside of the ring gauge hole.

2. From the MDI mode, go to the **F5** (**Mill**), then **F10** (**Probe**), then **F3** (**SpinPro**), then select **Probe Calibration**. Type the appropriate information, press **Exit**, and then press **START**.

3. The probe will touch four sides of the inside of the hole. The spindle will rotate (if the machine has spindle orientation) 180 degrees and touch the same four sides again establishing the center of the ring gauge. The spindle will then orient and touch four sides one more time calibrating the probe.

4. Remove the ring gauge from the machine and you are now ready to start part probing.

NOTE: On machines that allow the part probe to be installed in the spindle with more than one orientation, the probe stylus <u>must</u> be indicated true to the spindle centerline or the probe will not be accurate once removed and replaced into the spindle again.

Edge Finding (EdgeFind)

Format: EdgeFind SearchDir(XPlus, XMinus, YPlus, YMinus, ZPlus, or ZMinus) Offset(0–9)

❑ Calibrate the work probe at least once before trying to use this cycle.

❑ A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.

❑ A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.

❑ The EdgeFind Edge Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 8**.

Table 8, EdgeFind Entry Fields

Entry Fields	Description
SearchDir	Axis and direction to find edge. **XPlus**, **XMinus**, **YPlus**, **YMinus**, **ZPlus**, or **ZMinus** (Required)
Offset	Work Coordinate to update with edge location in X- or Y-axes. If set, work coordinate will be updated if **XPlus**, **XMinus**, **YPlus**, and **YMinus** are specified for **SearchDir**, or tool-length offset will be set for the current tool number if **SearchDir** is set to **ZPlus**, or **ZMinus**. **NOTE:** Before any tool-length offset is active, you must re-call that tool. Work coordinate register or Tool-length register will not be updated if **Offset** is not set and a warning message will tell the operator no update has taken place. [Default: **0**] Range (0–9) (Optional)

To use the Edge Finding Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Manually jog the probe stylus into proximity to the surface you wish to find and less then 0.1" (2.54 mm) away from the surface to be found.

3. Input EdgeFind SearchDir(?) Offset(n). If this is run from inside a program, this line needs to be repeated for every surface you wish to find.

> **Caution:** When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning) cycle** (see **ProbeMove** instructions later in this document).

4. Execute that line in MDI by exiting and pressing **START**.

15-August-03

Outside Corner Finding (CornerOut)

Format: CornerOut SearchQuad(XPlusYPlus, XMinusYPlus, XMinusYMinus, XPlusYMinus) Top(Yes/NO) DistDown(n) DistSide(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset (0–9))

❑ Calibrate the work probe at least once before trying to use this cycle.

❑ A preliminary tool-length offset must be set by eye for the work probe and that tool offset must be active before using this cycle. See the operations manual for setting and activating tool-length offsets.

❑ A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.

❑ The CornerOut Outside Corner Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 9**.

Table 9, CornerOut Entry Fields

Entry Fields	Description
SearchQuad	Quadrant of corner to find. **XPlusYPlus** = +,+ (upper right) **XMinusYPlus** = -,+ (upper left) **XMinusYMinus** = -,- (lower left) **XPlusYMinus** = +,- (lower right) (Required)
Top	If set to **Yes**, the cycle will find the top of the part before finding the X & Y corner coordinate. Default is **No**. If **Top** is not set or is set to **No**, the Z-axis must be at the picking depth. If **Top = Yes**, then the Z-axis must be within 0.1" (2.54 mm) above the part. The probe stylus must be positioned within 0.1" (2.54 mm) from the outside of the corner in X & Y. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistSide	The distance over from the corner to find X & Y edge. This will allow for a part corner that has a large chamfer or radius where you cannot pick the edge close to the theoretical corner or has an obstruction interfering with the default move. Default is 0.4" (10.16 mm). (Optional)

(Continued…)

Table 9, CornerOut Entry Fields (Continued)

Entry Fields	Description
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. (Optional)
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the corner. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with edge location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)

To use the Outside Corner Finding Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Manually jog the probe stylus less then 0.1" (2.54 mm) away from the outside of the corner you wish to find in X & Y. If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.

3. Input CornerOut SearchQuad(XPlusYPlus) Offset(n)
   If this is run from inside a program, this line needs to be repeated for every corner you wish to find or whose position you want to reestablish.

   > **Caution:** When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see ProbeMove instructions later in this document) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside Corner Finding (CornerIn)

Format: CornerIn SearchQuad(XPlusYPlus, XMinusYPlus, XMinusYMinus, XPlusYMinus) Top(Yes/No) DistDown(n) DistSide(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0-9)

❑ Calibrate the work probe at least once before trying to use this cycle.

❑ A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.

❑ A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.

❑ The CornerIn Inside Corner Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 10**.

Table 10, CornerIn Entry Fields

Entry Fields	Description
SearchQuad	Quadrant of corner to find. XPlusYPlus = +,+ (upper right) XMinusYPlus = -,+ (upper left) XMinusYMinus = -,- (lower left) XPlusYMinus = +,- (lower right) (Required)
Top	If set to **Yes**, the cycle will find the top of the part before finding the X & Y corner coordinate. Default is **No**. If **Top** is not set or is set to **No**, the Z-axis must be at the picking depth. If **Top = Yes**, then the Z-axis must be within 0.1" (2.54 mm) above the part. The probe stylus must be positioned within 0.1" (2.54 mm) from the outside of the corner in X & Y. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistSide	The distance over from the corner to find X & Y edge. This will allow for a part corner that has a large chamfer or radius where you cannot pick the edge close to the theoretical corner or has an obstruction interfering with the default move. Default is 0.4" (10.16 mm). (Optional)

(Continued…)

Table 10, CornerIn Entry Fields (Continued)

Entry Fields	Description
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. (Optional)
DistInX	The distance from the starting point to move in the "X" axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
DistInY	The distance from the starting point to move in the "Y" axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the corner. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with edge location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)

To use the Inside Corner Finding Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Manually jog the probe stylus less then 0.1" (2.54 mm) away from the outside of the corner you wish to find in X & Y. If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part; otherwise, the Z-axis should be at the side picking depth.

3. Input CornerIn SearchQuad(XPlusYPlus) Offset(0–9)
   If this is run from inside a program, this line needs to be repeated for every corner you wish to find or whose position you want to reestablish.

> **Caution:** When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see **ProbeMove** instructions later in this document) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Out/Inside Boss/Hole Finding (InOutBoss)

Format: InOutBoss Side(In/Out) Length(n) Width(n) Top(Yes/No)
DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0–9)
RepeatMeas(Yes/No)

❑ Calibrate the work probe at least once before trying to use this cycle.

❑ A preliminary tool-length offset must be set by eye for the work probe
and that tool offset active before using this cycle. See the operations
manual for setting and activating tool-length offsets.

❑ A preliminary work offset must be set by eye and that work coordinate
active before using this cycle. See the operations manual for setting
and activating work coordinate offsets.

❑ The InOutBoss Outside or Inside Boss/Hole Finding Cycle can be run
from within a program or from the MDI mode. Refer to **Table 11**.

Table 11, InOutBoss Entry Fields

Entry Fields	Description
Side	Inside or Outside. **In** = Inside Hole **Out** = Outside Boss (Required)
Length	Estimated length in the X-axis of boss/hole if rectangular or the Diameter if round. (Optional)
Width	Estimated width in the Y-axis of boss/hole. **Width** is only specified if boss or hole is rectangular in shape. (Optional)
Top	If set to "Yes" the cycle will find the top of the part before finding center of hole or boss. If **Side** parameter is set to **Out**, **Top** is forced to **Yes** as well; otherwise, the default is **No**. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the center. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. (Optional)

(Continued…)

Table 11, InOutBoss Entry Fields (Continued)

Entry Fields	Description
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default, if **Side** is not set or set to **In**, is 0.1" beyond the edge of the boss/hole. If **Side** is set to **Out**, the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is the current probe position. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the boss/hole center. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with the center location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)
RepeatMeas	If set to **Yes**, the cycle will do a preliminary measure in the "X" axis to get on center before measuring the Y-axis, making a total of 6 touches. If set to **No**, the cycle will only measure "X" once for a total of 4 touches. Default is **No**. (Optional)

To use the Out/Inside Boss/Hole Finding Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Manually jog the probe stylus the approximate center in X & Y within 0.1" (2.54 mm). If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.

3. Input InOutBoss Side(In/Out) Length(n) Width(n) Offset(0–9)
   If this is run from inside a program, this line needs to be repeated for every boss/hole you wish to find or whose position you want to reestablish.

   > **Caution:** When positioning the probe from within the program, you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see **ProbeMove** instructions later in this document) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Out/Inside Web Finding (InOutWeb)

Format: InOutWeb Side(In/Out) Length(n) Width(n) Top(Yes/No) DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0–9)

- An inside Web is a slot. An outside Web is a standing rib.

- Webs can only be measured in the X- or Y-axis.

- Calibrate the work probe at least once before trying to use this cycle.

- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.

- A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.

- The InOutWeb Outside or Inside Web Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 12**.

Table 12, InOutWeb Entry Fields

Entry Fields	Description
Side	Inside or Outside. **In** = Inside Hole **Out** = Outside Boss (Required)
Length	Estimated X width of Web if measuring in the X-axis. **Length** or **Width** must be specified but only one, not both.
Width	Estimated Y width of Web if measuring in the Y-axis. **Length** or **Width** must be specified but only one, not both.
Top	If set to **Yes**, the cycle will find the top of the part before finding center of Web. If **Side** parameter is set to **Out**, **Top** is forced to **Yes** as well; otherwise, the default is **No**. (Optional)
DistDown	The distance to go down from the top of part to find X or Y coordinate of the center. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm), if not set. (Optional)

(Continued…)

Table 12 ,InOutWeb Entry Fields (Continued)

Entry Fields	Description
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default, if **Side** is not set or set to **In**, is 0.1" beyond the edge of the web. If **Side** is set to **Out**, the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is the current probe position. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the web center. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with the center location in X- or Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)

To use the Out/Inside Web Finding Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Manually jog the probe stylus the approximate center in X or Y within 0.1" (2.54 mm). If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.

3. Input InOutWeb Side(In/Out) Length(n) Offset(0–9)
   If this is run from inside a program, this line needs to be repeated for every web you wish to find or whose position you want to reestablish.

> **Caution:** When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see **ProbeMove** instructions later in this document) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

15-August-03

Protected Probe Positioning (ProbeMove)

Format: ProbeMove X(n) Y(n) Z(n) Feed(n)

❑ When an X, Y, and/or Z move is programmed using the **ProbeMove** (Protected Positioning Cycle), the control will stop and alarm if the probe stylus is triggered before reaching the target set in the X, Y, and/or Z parameters.

❑ This cycle is intended to offer some degree of safety when moving the probe around the part; however, it is not fool proof and will not protect against gross bad programming where the probe body would encounter an obstruction before the probe stylus is triggered. Extreme care should be taken to avoid this condition as probe damage may result.

❑ Calibrate the work probe at least once before trying to use this cycle.

❑ A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.

❑ A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.

❑ The ProbeMove Protected Probe Positioning Cycle can be run from within a program or from the MDI mode. Refer to **Table 13**.

Table 13, ProbeMove Entry Fields

Entry Fields	Description
X	X Target position relative to current active work coordinate.
Y	Y Target position relative to current active work coordinate.
Z	Z Target position relative to current active work coordinate combined with the current active tool-length offset.
Feed	Feedrate at which to travel to target. **Feed** is only active for the current move so it must be restated every time or the default will take precedence. The default is set in the User defined variable **Probe positioning feedrate**. (Optional)

To use the ProbeMove Protected Probe Positioning Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Input
   ProbeMove X(n) Y(n) Z(n) Feed(n)
   If this is run from inside a program, this line needs to be repeated for every move you wish to make.

3. Execute that line in MDI by exiting and pressing **START**.

Skew Error Find (SkewComp)

Format: SkewComp Action(Find/FindActive/Activate) EstAngle(n) DistPicks(n) Top(Yes/No) DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n)

❑ RMS cannot be used with **SkewComp**, skew error find.

❑ Skew error is only supported for along the side edge of a part relative to the X,Y plane.

❑ Calibrate the work probe at least once before trying to use this cycle.

❑ A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.

❑ A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.

❑ The probe must be pre-positioned to the proper spot in relation to the part in accordance with the specified **EstAngle** parameter as described below or an X, Y, and/or Z should be included for pre-positioning.

❑ The SkewComp Skew Error Finding Cycle can be run from within a program or from the MDI mode. Refer to Table 14, SkewComp Entry Fields.

ANILAM

Table 14, SkewComp Entry Fields

Entry Fields	Description
Action	**Find** Finds the skew angle, but does not activate skew compensation. **FindActive** Finds the skew angle, and activates skew compensation. **Activate** Activates skew compensation with the current skew value, but will not rerun the cycle on the part. [Default: **Find**] NOTE: If **Activate** is used, all other **SkewComp** parameters are ignored. NOTE: Before using **SkewComp Activate**, you must have called **SkewComp** at least once with **Find** or **FindActive**, or the error message "Skew error has not been found!" is displayed. Skew compensation will be activated around the current active work coordinate and will only work from within the program being run. Skew compensation cannot be activated directly or indirectly using **SkewComp** from the MDI mode. The operator can run the **SkewComp** from MDI but must place **SkewComp Activate** inside the program for skew compensation to take effect. **Offset** work coordinate call will deactivate skew compensation, necessitating a re-issuance of **SkewComp Activate** to activate skew compensation. Using **FindActive** or **Activate** will default the control to Absolute. If you are in Incremental, you will need to switch back after the cycle has been run. (Optional)

EstAngle	Estimated amount of angle from 3 O'clock. Default is "0" which will cause the cycle to find the angle of the back edge of the part starting its first pick in the upper-left corner and making the second pick to the left of that, as you are facing the surface being picked. Examples:
	EstAngle=90
	      Would start in the lower-left side, picking in the X positive direction, finding the skew of the left side of the part.
	EstAngle=-90
	      Would start in the upper-right side, picking in the X negative direction, finding the skew of the right side of the part.
	EstAngle=180
	      Would start in the lower-right front edge of the part, picking in the Y positive direction, finding the skew of the front edge of the part.
	[Default: 0] (Optional)

DistPicks	The distance from the first pick to the second pick. Default is 2.0" (50.8 mm) (Optional)
Top	If set to **Yes**, the cycle will find the top of the part before finding part skew angle. Default is **No**. If **Top** is set to **Yes**, the probe stylus should be pre-positioned within 0.1" (2.54 mm) above the part. If **Top** is set to **No**, the probe stylus should be positioned at the Z-axis depth from which you want to make side picks. (Optional)

Table 14, SkewComp Entry Fields (Continued)

Entry Fields	Description
DistDown	The distance to go down from the top of part to find part skew angle. This is only used if **Top** parameter is set to **Yes**. Without any **DistSide** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. This would be used to make sure that the cycle is picking from far enough away from the edge so that it will not trigger the probe prematurely when stepping over to make the second pick. **Hint:** If the **EstAngle** parameter is relatively accurate, this parameter will not be needed because the default will be good enough. (Optional)
DistInX	The distance from the starting point to move in the "X" axis to find the top of the part. The default is 1.0" (25.4 mm) toward the part at the angle specified in the **EstAngle** parameter. (Optional)
DistInY	The distance from the starting point to move in the "Y" axis to find the top of the part. The default is 1.0" (25.4 mm) toward the part at the angle specified in the **EstAngle** parameter. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the skew angle. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)

To use the Skew Error Finding Cycle:

1. Place the probe in the spindle and make sure that its tool offset is active.

2. Manually jog the probe stylus to the appropriate start position relative to the part as specified by the **EstAngle** parameter in **Table 14**. X or Y should be within 0.1" (2.54 mm) of the part edge. If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part; otherwise, the Z-axis should be at the side picking depth. If run from within a program, probe must be pre-positioned.

3. Input SkewComp
   If this is run from inside a program, this line needs to be repeated every time you wish to find a skew angle.

   > **Caution:** When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see **ProbeMove** instructions in this document) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.