PROBING PROGRAMMING MANUAL

for
CINCINNATI MILACRON®
MACHINING CENTERS
with
ACRAMATIC® 2100E CNC CONTROL
Release 3

PUBLICATION NO. : 6-SP-98075-1
PART No. : 9703431814A

CINCINNATI MILACRON

PROBING PROGRAMMING MANUAL

for
CINCINNATI MILACRON®
MACHINING CENTERS
with
ACRAMATIC® 2100E CNC CONTROL
Release 3

PUBLICATION NO. : 6-SP-98075-1
PART No. : 9703431814A

IMPORTANT

Carefully read the instructions and safety precautions given in this manual. Do not attempt to program this machine until you have thoroughly read and understood the material contained in this manual and all other applicable manuals.

At the time of writing, this book was completely up-to-date. However, due to continual improvements in design, it is possible that descriptions contained herein may vary to a slight extent from the system delivered to you. This merely implies that the system has been improved to better fulfill your requirements. You are encouraged to contact the nearest Cincinnati Milacron representative for clarification.

CINCINNATI MILACRON

We Build Productivity

Cincinnati Milacron Company
Cincinnati, Ohio 45209-1086

Printed in U.S.A. 1/99 CM

FOREWORD

The purpose of this manual is to provide the necessary information to enable suitably experienced personnel, to program the tool probing features and spindle probing features supplied with Vickers 2100E controls.

The manual has not been prepared to enable inexperienced personnel to operate and program the machine without further training.

Information contained in this manual is not warranted and is snbject to change without notice.

The owner/user is responsible for the training of inexperienced personnel and for providing the background necessary for experienced personnel to safely operate and program these machines.

The chapter on general safety precautions should be observed at all times during machine operation and maintenance. Read this chapter before reading the remaining chapters in this manual and operating the machine.

Space is reserved at the rear of this manual for current and future addenda and supplements that pertain to the machine and/or control. In an effort to maintain the manuals as current as possible when new features are added to the system, addenda and supplement will be added to this chapter.

Table of Contents

Table of Contents

Table of Contents

Table of Contents

Table of Contents

Table of Contents

Table of Contents

Chapter 2
Spindle Probe

Spindle Probe Cycle Overview

Spindle Probe cycles provide the capability to measure one or more work piece coordinates. Each cycle can be created by using Resident Assistant Programming (RAP), Manual Data Input (MDI), or created off line and placed within the part program itself. Information generated by these cycles may be used to setup offsets or to control NC program operation.

What G Codes Create Spindle Probe Cycles

Spindle probe G codes are divided into two groups, Calibration Cycles and Measurements Cycle.

Calibration Cycles

Before work piece coordinates can be measured, the spindle probe must be calibrated. The following G codes are used for calibration cycles:

G72 Calibrates the amount of stylus offset from the spindle centerline.
G73 Calibrate more than one probe stylus offset.
G74 Calibrates probe length.

Measurement Cycles

Spindle probe measurement cycles are as follows:

G51 Is used to vector probe a surface.
G51.1 Is used to vector probe a surface and set offsets.
G51.2 Measures line slop between two points for rotary axis.
G51.3 Is used to measure the difference between two points in the X, Y plane.
G51.4 Locates multiple parts in the XY axis plane.
G51.5 Locates multiple parts in the Z axis plane using other surface measurement data.
G75 Locates X, Y, and Z dimensions of an inside corner.
G76 Locates X, Y, and Z dimensions of an outside corner.
G77 Locates the surface of a part.
G77.1 Is used to measure maximum and minimum stock amounts.
G78 Probe to locate and measure a bore.
G78 Probe to locate and measure a bore with an obstruction.
G78 Probe to locate and measure a boss.
G79 Probe to measure a web.
G79 Probe to measure a pocket.
G79 Probe to measure a pocket with an obstruction.

CAUTION

When the probe is disarmed, there is no protection against accidental contact with the part or other obstructions. Failure to follow this instruction can result in damage to the workpiece, probe, tooling, or machine.

M58 Disarm Spindle Probe

M58 causes the control to ignore probe contact signals from the probe in the spindle. This function may be used when the probe is positioned at high speed to avoid false trigger alarms caused by high acceleration.

M59 Arm Spindle Probe

M59 code arms the surface sensing probe in the spindle. The probe is armed following a tool change or by executing any of the probe cycles. When the probe is armed, the control is sensitive to any probe contact.

Surface Probe Measurement Overview

To gain an understanding of how surface probe data interacts with the control, we strongly recommend you read the overview material presented here before proceeding to an individnal cycle.

The surface measurement cycles listed on the previous page perform automatic measurement of common workpiece features. These cycles record their measurements in a common set of system registers for use by subsequent NC program statements. Typical uses of probe surface measurements and general information include:

- Determine whether there is sufficient stock to machine the part
- Determination of the location of a feature to compensate the NC program for part to part variation
- Computation of various program or tool offsets to correct for measured machining results
- Location of the part or a portion of the part
- Be sure the spindle probe is properly calibrated.
- Be sure all Tool Data Table information is entered before performing calibrations.
- Only the most recent spindle probe calibration values are retained by the control.
- Probe Part Location values are retained in System register [PRB_PART_LOC] X, Y, Z.
- Cycle Parameter Probe Gage Height PROBE_GH determines clearance or back off distance after a probe hit.
- Cycle Parameter Probe Approach PRB_APPR_FRT specifies probe approach feed rate for first probe hit.
- Cycle Parameter Probe Measurement Feed rate PRB_MEAS_FRT specifies probe approach feed rate for second measurement move.
- Spindle Probe Operations can be generated in Manual Data Input (MDI), by using Resident Assistant Programmer (RAP), from Multiple Setup, or NC program.

- Be sure that the probe stylus runs concentric with the spindle.

- Surface measurement cycles can also adjust the diameter and length offsets for the tool used to machine a surface. To do this, it is necessary to record the tool number of the tool that was used to machine the feature to be measured, since the active tool at the time of the measurement will be the probe. The T word on the surface measurement cycles is used to pass this information to the probe cycle. The T word used for probe cycles is different from the T word normally used for tool changes.

- An additional correction can be applied to tool offset updates based on feature size measurements. The L word on a surface measurement probe cycle specifies an amount to be added to the measured size before computing the system register [SIZE_ERROR] value. Clamping forces can sometimes affect the measured size values obtained by the probe cycles. The L word provides an adjustment to relate the measurements on the machine tool to a traceable such as a coordinate measurement machine.

- The L word can also be used in some cases to compensate for thermal compensations. One method for doing this is to measure some known artifact located in the machine envelop and subject to the same coolant and other environmental factors as the workpiece. The difference between the known size of the artifact and the measured size can be used to calibrate the other measurements. The L word provides the means to apply the calibration amount to the tool offset corrections.

- The surface measurement cycles have a default overtravel distance that varies for the type of cycle. The overtravel distance is the amount of motion beyond the expected (programmed) surface position that is allowed before the cycle stops with a no hit error condition. The default amount of overtravel may not always be suitable. The D word can be programmed on any probe cycle to specify the overtravel distance for that cycle, overriding the default.

What Data Interacts With Spindle Probe Measurements

The spindle probe is used to locate part coordinates on the machining surface. After this data is acquired, knowing where it resides, and how it can be used must be understood. Basically, the control interacts with spindle probe data, the Tool Data Table, Programmable Tool Offsets, Programmable Coordinate Offsets, Cycle Parameter Table, and System Registers. A brief description of each is listed below:

G51 Probe Multiple Axes

When multiple axes are used to locate a part, System Registers store the latest axis data.

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0

G51.1 Vector Probe a Surface and Set Offsets

When vector probing a surface the following System Registers store data.

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_PART_LOC] X, Y, Z	Probe Part Location	
[$X_POS_ERROR]	X axis Position Error	Probe difference between I word and measured position
[$Y_POS_ERROR]	Y axis Position Error	Probe difference between J word and measured position
[$Z_POS_ERROR]	Z axis Position Error	Probe difference between K word and measured position
[$TRUE_POS_ERR]	True Position Error	True position vector error value of X, Y, and Z axis
[$SIZE_ERROR]	Size Error	Size error value is the true position value plus the L word
[$OUT_OF_TOL]	Out Of Tolerance	If the measured [SIZE_ERROR] exceeds the V word value [OUT_OF_TOL] is set true, non-zero
[$TOL_EXCEEDED]	Tolerance Exceeded	If the measured [SIZE_ERROR] exceeds the U word value [TOL_EXCEEDED] is set true, non-zero

G51.2 Rotary Axis Measurement

When vector probing a rotary axis surface the following System Registers store data.

System Register Name	Definition	Comment
[$PRB_POS_PC] Z	Probe Position Program Coordinates	
[$PRB_POS_MC] Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_A_ANGLE]	Probe A axis Angle	Angle between two measured points in the Y axis plane, R word specifies rotary A axis
[$PRB_B_ANGLE]	Probe B axis Angle	Angle between two measured points in the X axis plane, P word specifies rotary B axis
[$ANGLE_ERROR]	Angle Error	If [PRB_A_ANGLE] or [PRB_B_ANGLE] B word tolerance is exceeded [ANGLE_ERROR] is set true

G51.3 Angle Measurement in X or Y Plane

Angled Measurement of X or Y plane data is stored in the following System Registers.

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y	Probe Position Machine Coordinates	
[$PROBE_IIIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_ANGLE]	Probe Angle	Angle of two measured points (P word for X or R word for Y) with the +X axis
[$ANGLE_ERROR]	Angle Error	If B word tolerance is exceeded [ANGLE_ERROR] is set true

G51.4 Measure Feature-to-Feature in X, Y Plane

The following System Registers are effected in this cycle.

System Register Name	Definition	Comment
[SPRB_PART_LOC]	Probe Part Location	Data used by G51.4 set by previous surface measurement cycle
[$PRB_PART_LOC(X)]	Probe Part Location X Axis	Data used by G51.4 set by previous surface measurement cycle
[$PRB_PART_LOC(Y)]	Probe Part Location Y Axis	Data used by G51.4 set by previous surface measurement cycle
[$X_POS_ERR]	X axis Position Error	Is the difference between the expected and measured distance in X axis between two features
[$Y_POS_ERR]	Y axis Position Error	Is the difference between the expected and measured distance in Y axis between two features
[$TRUE_POS_ERR]	True Position Error	True position vector error value of X, Y, axis between two features
[$PRB_ANGL_ERR]	Probe Angle Error	The error between the measured and expected angle between two features is
[$SIZE_ERROR]	Size Error	The error in the radial distance measured between two features
[$OUT_OF_TOL]	Out Of Tolerance	If second G51.4 block [TRUE_POS_ERR] exceeds V word [OUT_OF_TOL] is set true non-zero.
[$TOL_EXCEEDED]	Total Tolerance Exceeded	If second G51.4 block [TRUE_POS_ERR] exceeds U word [TOL_EXCEEDED] is set true non-zero
[$ANGLE_ERROR]	Angle Error	If second G51.4 block [PRB_ANGL_ERR] exceeds B word [ANGLE_ERROR] is set true non-zero

G51.5 Measure Feature-to-Feature in Z Plane

The following System Registers are effected in this cycle.

System Register Name	Definition	Comment
[$PRB_PART_LOC]	Probe Part Location	Data used by G51.5 set by previous surface measurement cycle
[$PRB_PART_LOC(Z)]	Probe Part Location Z Axis	Data used by G51.5 set by previous surface measurement cycle
[$Z_POS_ERR]	Z Axis Position Error	Is the expected difference between two features in Z axis
[$TRUE_POS_ERR]	True Position Error	True position vector error value of Z axis between two features
[$PRB_ANGL_ERR]	Probe Angle Error	The error between the measured and expected angle between two features
[$SIZE_ERROR]	Size Error	Size error value is the true position error value plus the L word measured between two features
[$OUT_OF_TOL]	Out Of Tolerance	If second G51.5 block [TRUE_POS_ERR] exceeds V word [OUT_OF_TOL] is set true non-zero
[$TOL_EXCEEDED]	Total Tolerance Exceeded	If second G51.5 block [TRUE_POS_ERR] exceeds U word [TOL_EXCEEDED] is set true non-zero
[$ANGLE_ERROR]	Angle Error	If second G51.5 block [PRB_ANGL_ERR] exceeds B word [ANGLE_ERROR] is set true non-zero

G72 and G73 Calibration Cycles

The following Tool Data Table Cycle Parameter Table and System Register fields are effected by these cycles.

Tool Data Table	Definition	Comment1.5
X Probe Offset	X Probe Offset	contain the latest computed spindle center line offset values for X axes after calibration is performed. **Note**: these field are not effected by G73
Y Probe Offset	Y Probe Offset	contain the latest computed spindle center line offset value Y axes after calibration is performed. **Note**: these field are not effected by G73

Cycle Parameter Table	Definition	Comment
X_POS_TIP	X Axis Position Tip Dimension	The latest computed dimensions of the stylus tip for +X, after calibration is performed
X_NEG_TIP	X Axis negative Tip Dimension	The latest computed dimensions of the stylus tip for -X after calibration is performed
Y_POS_TIP	Y Axis Position Tip Dimension	The latest computed dimensions of the stylus tip for +Y after calibration is performed
Y_NEG_TIP	Y Axis negative Tip Dimension	The latest computed dimensions of the stylus tip for -Y after calibration is performed

System Register Name	Definition	Comment
[$PROBE_CALIB]	Probe Calibration	If 360 degree calibration is selected values are stored here

G74 Set Probe Length

The following Tool Data Table is affected by this cycle.

Tool Data Table	Definition	Comment
Tool Length	Tool Length	field retains the latest calibrated probe length

G75, G76 Probe to Corner Location System Registers

When either internal or external corners are located, data is stored in the following System Registers.

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates X, Y, Z axis	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates X, Y, Z axis	
[$PRB_PART_LOC] X, Y, Z	Probe Part Location X, Y, Z axis	
[$PROBE_HIT]	Probe Hit	Probe hit set 1 surface contact, no probe hit 0
[$X_POS_ERROR]	X Axis Position Error	Probe difference between I word and measured position
[$Y_POS_ERROR]	Y Axis Position Error	Probe difference between J word and measured position
[$Z_POS_ERROR]	Z Axis Position Error	Probe difference between K word and measured position
[$TRUE_POS_ERR]	True Position Error	True position vector error value of X, Y, and Z
[$PRB_X_ANGLE]	Probe X Axis Angle	Is the computed X axis side of the corner when P word is programmed
[$PRB_Y_ANGLE]	Probe Y Axis Angle	Is the computed Y axis side of the corner when R word is programmed
[$PRB_INCL_ANG]	Probe Include Angle	Data is used by G51.4 and G51.5 for second measurement positioning
[$OUT_OF_TOL]	Out Of Tolerance	If the measured exceeds the V word valne [OUT_OF_TOL] is set true, non-zero
[$TOL_EXCEEDED]	Tolerance Exceeded	If the measnred exceeds the U word value [TOL_EXCEEDED] is set true, non-zero
[$ANGLE_ERROR]	Angle Error	If second measured X and Y axis angles exceeds B word [ANGLE_ERROR] is set true non-zero

G77 Locate Probe Surface System Registers

When a surface is located the following System Registers store the latest axis data.

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_PART_LOC] X, Y, Z	Probe Part Location	
[$X_POS_ERROR]	X axis Position Error	Probe difference between I word and measured position
[$Y_POS_ERROR]	Y axis Position Error	Probe difference between J word and measured position
[$Z_POS_ERROR]	Z axis Position Error	Probe difference between K word and measured position
[$TRUE_POS_ERR]	True Position Error	True position vector error value of X, Y, and Z axis
[$SIZE_ERROR]	Size Error	Size error value is the true position value plus the L word
[$OUT_OF_TOL]	Out Of Tolerance	If the measured [SIZE_ERROR] exceeds the V word value [OUT_OF_TOL] is set true, non-zero
[$TOL_EXCEEDED]	Tolerance Exceeded	If the measured [SIZE_ERROR] exceeds the U word value [TOL_EXCEEDED] is set true, non-zero

G77.1 Stock Allowance System Registers

The following System Registers store data when this cycle is used

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_PART_LOC] X, Y, Z	Probe Part Location	
[$MAX_STOCK]	Maximum Stock	Is data for maximum stock amount for the current set of measurements
[$MIN_STOCK]	Minimum Stock	Is data for minimum stock amount for the current set of measurements
[$VARIATION]	Variation in Stock	Is the difference between lowest and highest measured surface location
[$X_POS_ERROR]	X axis Position Error	Probe difference between I word and measured position
[$Y_POS_ERROR]	Y axis Position Error	Probe difference between J word and measured position
[$Z_POS_ERROR]	Z axis Position Error	Probe difference between K word and measured position
[$TOL_EXCEEDED]	Tolerance Exceeded	If the measured [SIZE_ERROR] exceeds the U word value [TOL_EXCEEDED] is set true, non-zero

G78 Probe a Bore System Registers

When a bore is located, the System Registers store the latest axis data, also three bore measurements are retained as follows:

System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_PART_LOC(X)]	Probe Part Location in X axis	
[$PRB_PART_LOC(Y)]	Probe Part Location in Y axis	
[$PRB_X_DIA]	Probe X Axis Diameter	
[$PRB_Y_DIA]	Probe Y Axis Diameter	
[$PRB_AVG_DIA]	Probe Average Diameter	
[$X_POS_ERROR]	X axis Position Error	Probe difference between I word and measured position
[$Y_POS_ERROR]	Y axis Position Error	Probe difference between J word and measured position
[$OUT_OF_TOL]	Out Of Tolerance	If the measured value exceeds the V word [OUT_OF_TOL] is set true, non-zero
[$TOL_EXCEEDED]	Tolerance Exceeded	If the measured value exceeds the U word value [TOL_EXCEEDED] is set true, non-zero

G79 Probe to Measure a Web or Pocket System Registers

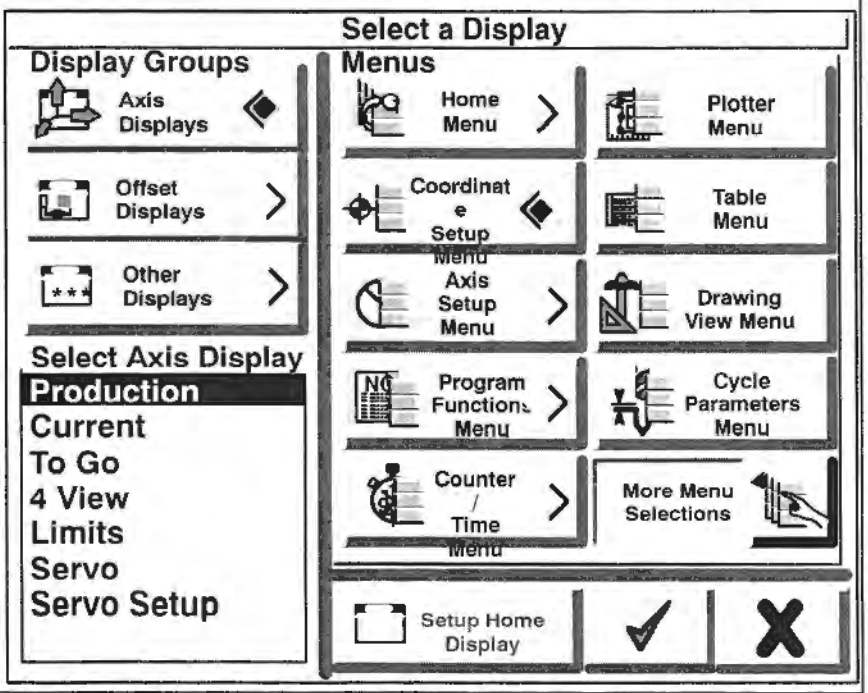
When a web or pocket is measured the following System Registers store the latest axis data. An additional field, Probe Width contains the measured web or pocket width.

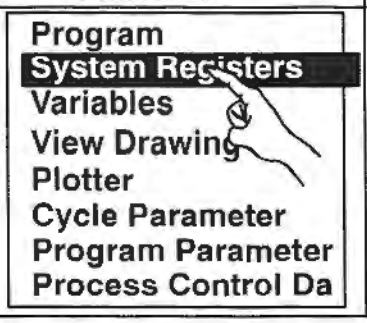
System Register Name	Definition	Comment
[$PRB_POS_PC] X, Y, Z	Probe Position Program Coordinates	
[$PRB_POS_MC] X, Y, Z	Probe Position Machine Coordinates	
[$PROBE_HIT]	Probe hit	Probe hit set 1, no probe hit 0
[$PRB_PART_LOC(X)]	Probe Part Location in X axis	
[$PRB_PART_LOC(Y)]	Probe Part Location in Y axis	
[$PRB_WIDTH]	Probe Width	Is the measured web or pocket width
[$X_POS_ERROR]	X axis Position Error	Probe difference between I word and measured position
[$Y_POS_ERROR]	Y axis Position Error	Probe difference between J word and measured position
[$TRUE_POS_ERR]	True Position Error	True position vector error value of X, and Y axis
[$OUT_OF_TOL]	Out Of Tolerance	If the measured value exceeds the V word [OUT_OF_TOL] is set true, non-zero
[$TOL_EXCEEDED]	Tolerance Exceeded	If the measured value exceeds the U word value [TOL_EXCEEDED] is set true, non-zero

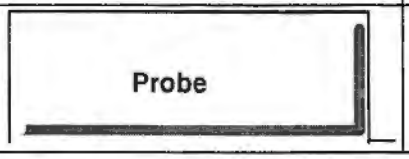
How Do I View System Register Data

Touch, press or set the following	Comments
	
	

The following display will appear:



	Touch to change
	The System Registers Table will be displayed.
	Touch to highlight

How Do I View System Register Data (continued)	
	The System Registers Table will be displayed.
	

Note: To change values on this menu the Service password must be selected.

What Tool Types are Permitted

PROBE

What Cycle Parameters Are Used With the Spindle Probe

Cycle Parameters are used to modify spindle probe operations. When the Probing Cycle Parameter Table is selected, two field values are presented: *Programmable Value*, which can be altered two ways, by the probe cycles themselves, or by the operator as needed. The *Base Value* field is used to establish a set of default values which are configured under the Setup password.

CYCLE PARAMETERS

The Cycle Parameter Table also provides the programmer with a method of changing the value of fixed cycle features; (i.e. Probe Approach Feedrate).

The following chart lists the Probe Cycle Parameters, Program Reference, Base Value, and Programmable Value of cycle parameters used by probe fixed cycles G51 - G51.5 &G72 - G79.

PROBE CYCLE PARAMETER TABLE (EXAMPLE)

PROBE CYCLE PARAMETERS	PROGRAM REFERENCE	BASE VALUE	PROGRAMMABLE VALUE
Probe Approach Feedrate	PRB_APPR_FRT	+ 2500.000	+ 2500.000
Probe Measurement Feedrate	PRB_MEAS_FRT	+ 250.000	+ 250.000
Rotating Tool Retract Distance	FIX_PRB_RRET	+ 1.000	+ 1.000
Probe Gage Height	PROBE_GH	+ 2.5400	+ 2.5400
+X Stylus Tip Dimension	X_POS_TIP	+ 0	+ 0
-X Stylus Tip Dimension	X_NEG_TIP	+ 0	+ 0
+Y Stylus Tip Dimension	Y_POS_TIP	+ 0	+ 0
-Y Stylus Tip Dimension	Y_NEG_TIP	+ 0	+ 0
Tram Surface	TRAM_SURFACE	+ 0	+ 0
Fixed Probe Tram Surface	FIX_PRB_TRAM	+ 0	+ 0
Fixed Probe Clearance Height	FIX_PRB_CLR	+ 950.0000	+ 950.0000

The cycle parameters can be changed by the NC program. An example follows:

N10 [$CYCLE_PARAMS (2) PRB_APPR_FRT] = 1250.000

Block N10 assigns the PRB_APPR_FRT cycle parameter in column two (Programmable Value) the value of 1250.000 instead of thevalue in column one (Base Value) of 2500.000.

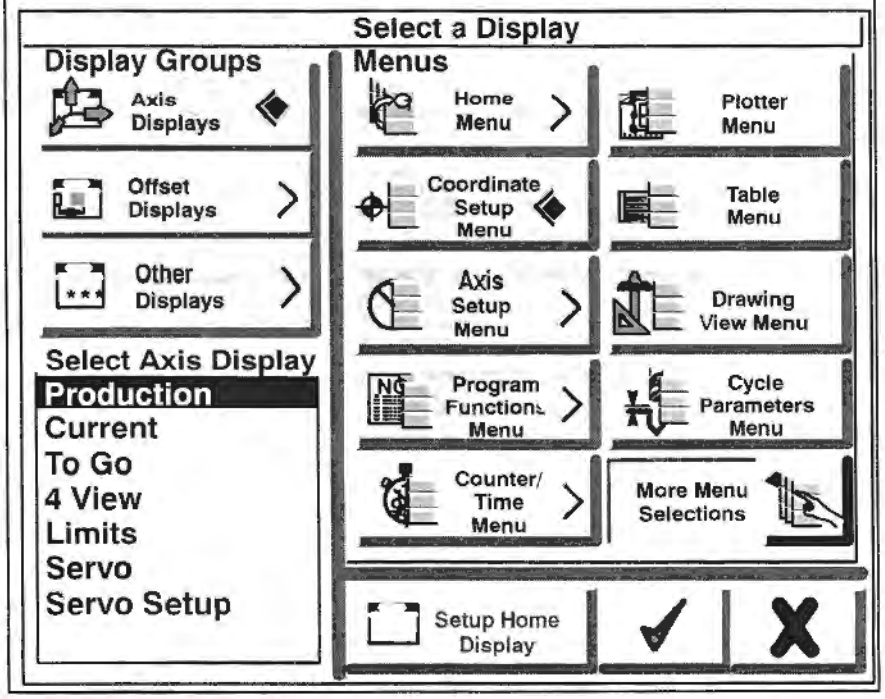
NOTE: It is extremely important that any cycle parameters changed during program execution be restored to their original table values. Restoring the original values either immediately after use or at the end of the part program will assure that programs that do not assign cycle parameter values work in a consistent manner.

Changing Probe Cycle Parameters

Changes can be made to parameters associated with each NC Cycle operations. This table is supplied with a set of configurable default values established at machine installation. Values in this table can be overridden programmatically or entered at the machine. **Only experience personnel, totally familiar with with the machine operation and programming, may be permitted to make these type of changes.** Exact information and illustration may vary somewhat from that shown depending on machine type, application and options supplied.

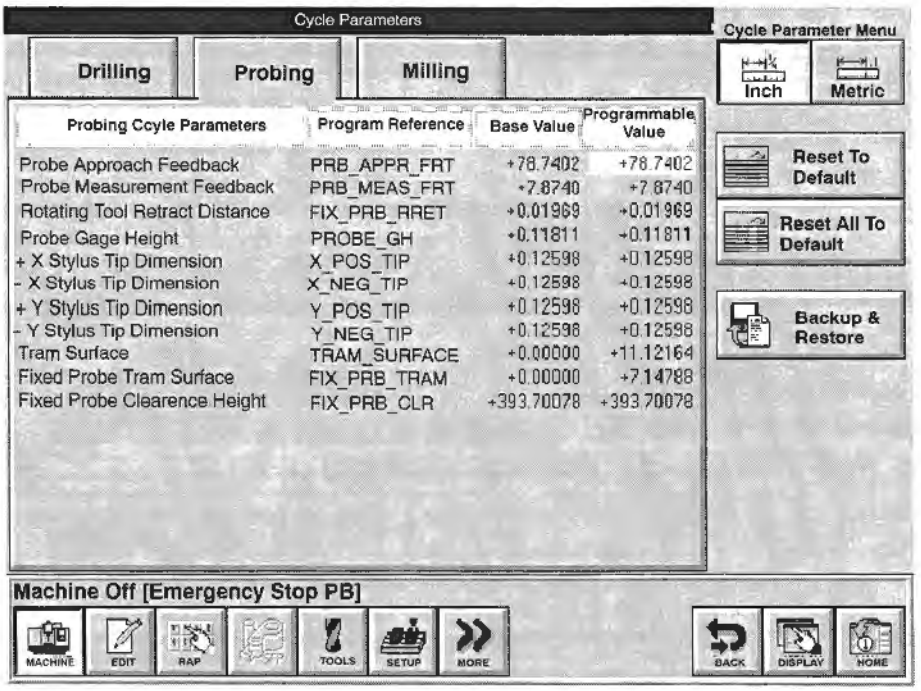
Touch, press or set the following	Comments
	
	

The following display will appear:



Changing Probe Cycle Parameters (continued)

Cycle Parameters Menu	Touch to highlight Cycle Parameters Menu
✓	
Probing	Touch to highlight Cycle Parameters Table button if table is not active.



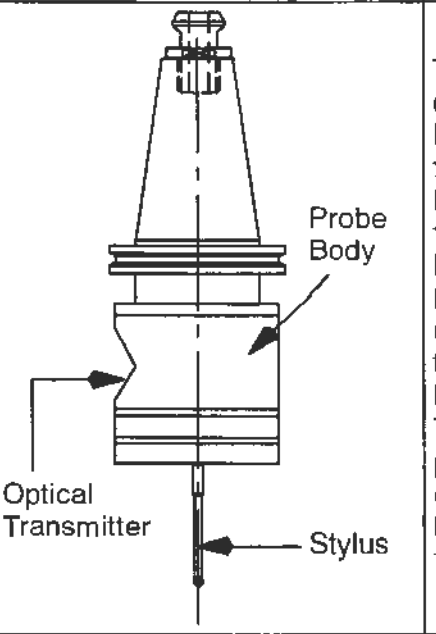
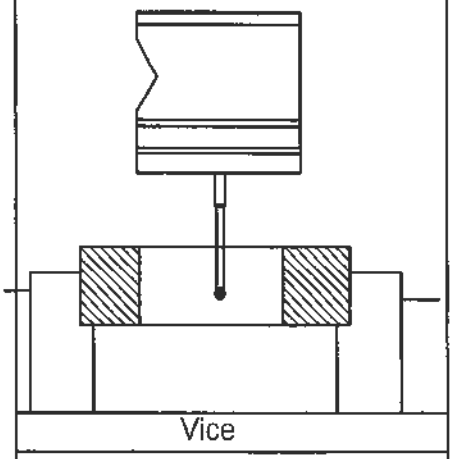
Check that Inch or Metric mode is correct. Tram Surfaces machine coordinate is displayed in table. See WARNING

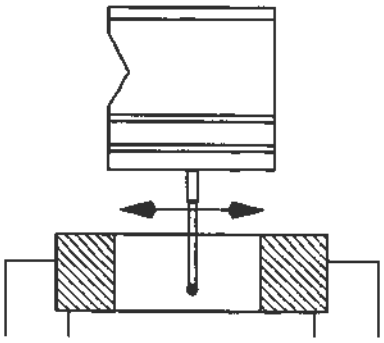
Parameter Name	Definition	Range	Usage
PRB_APPR_FRT	Probe Approach Feedrate	0 to 99.99999 inch 0 to 9999.9999 mm	Specifies probe approach feed rate for first probe contact.
PRB_MEAS_FRT	Probe Measurement Feed rate	0 to 9.99999 inch 0 to 999.9999 mm	Specifies probe approach feed rate on a second measurement move following the initial hit. A second measurement move is selected when the Q word = 0 or absent.
PROBE_GH	Probe Gage Height	0 to 9.99999 inch 0 to 9.9999 mm	This value determines clearance or back off distance after a probe hit.
X_POS_TIP X_NEG_TIP Y_POS_TIP Y_NEG_TIP	+X, -X, +Y, and -Y Stylus Tip Dimension	0 to 9.99999 inch 0 to 99.9999 mm	Defines the latest computed dimensions of the stylus tip after calibration is performed

Why Calibrate the Spindle Probe

The spindle probe must be calibrated before accurate part coordinates can be acquired. Normally, this procedure is performed at setup, but to review what is involved it will be repeated here. Also, the control retains only the most recent calibration data.

Probe Calibration	
Touch, press or set the following	**Comments**
The Probe must be calibrated before it can be used. Two Probe cycles, G72 and G74, are used to calibrate the Probe Stylus dimensions and length. Once this procedure has been preformed, the probe will locate parts along the X, Y and Z axes. Prior to calibrating the probe, ensure that the Tool Data Table contains: an approximate length and PROBE as tool type. The following example procedure is for reference purposes. Due to the numerous workpiece configurations, tooling and set-up possibilities this procedure is to be used as a guide. Refer to the Programming manual for additional probe information.	
G72 Set Stylus and Tip Dimensions	
Touch, press or set the following	**Comments**
	
	

G72 Set Stylus and Tip Dimensions (continued)	
Touch, press or set the following	**Comments**
	The exact centerline and diameter of a bored hole must be known prior to performing the following procedure. This can be accomplished by making a finished boring cut with a boring bar, by tramming an existing bore with a dial indicator, or by utilizing an existing bored hole that is at a predetermined location on the table or fixture. The stylus should be as close as possible to the spindle centerline and the reference bore should be as large as feasible.
	- Load the Probe into the spindle [TxxxM6]. Using your finger, lightly tap the stylus. The message "Unexpected Hit Probe Cycle" should appear on the CRT. If this message doesn't appear, the Probe isn't functioning properly. DO NOT use the Probe until the cause of malfunctioning is found and corrected. If the message does appear, clear the alert or press Data Reset to extinguish the alarm and rearm the Probe. - Position the spindle center line to the exact centerline of the reference bore and feed the stylus tip until it is fully within the bore.

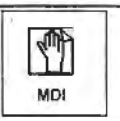
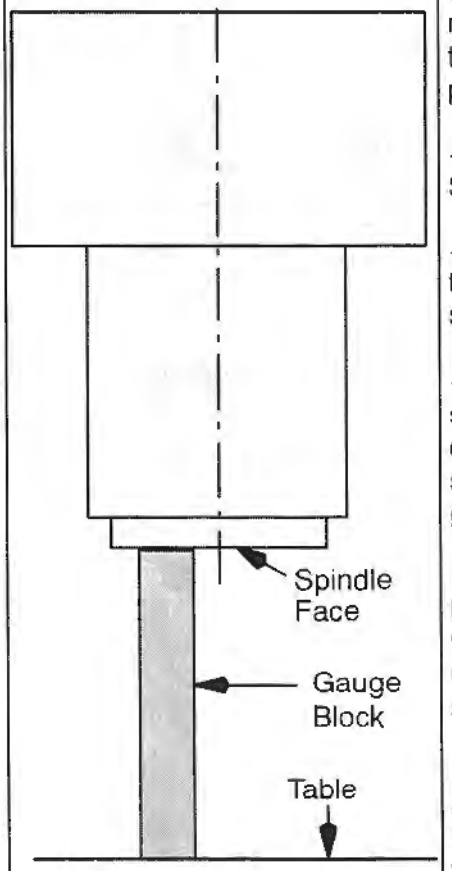
G72 Set Stylus and Tip Dimensions (continued)	
Touch, press or set the following	**Comments**
	- In MDI, enter G72Pxxxx, where Pxxx is the exact diameter of the reference bore. - Press Cycle Start, the Probe will begin the calibration cycle making a series of X and Y moves to determine the stylus size and amount that is offset from the true spindle centerline. When the G72 begins, all previous data concerning stylus size and Offset is erased, so be sure not to interrupt the cycle once it has begin, unless an emergency situation is encountered.
	The G72 will enter four pieces of information into the control that can be checked to ensure proper calibration has taken place: X+ probe tip radius, X- probe tip radius, Y+ probe tip radius, Y-probe tip radius. **NOTE** These dimensions are all within a few tenths (.0001 in.) of each other. If there seems to be a greater variance, recalibrate the Probe. Also, in the Tool Table, the X and Y Probe offset values reflect the amount that the tip of the stylus is offset from the spindle centerline. The stylus tip dimensions are stored in Cycle Parameter Table entries: +X Stylus Tip Dimension -X Stylus Tip Dimension + Y Stylus Tip Dimension -Y Stylus Tip Dimension This completes the stylus calibration in the X, Y planes.

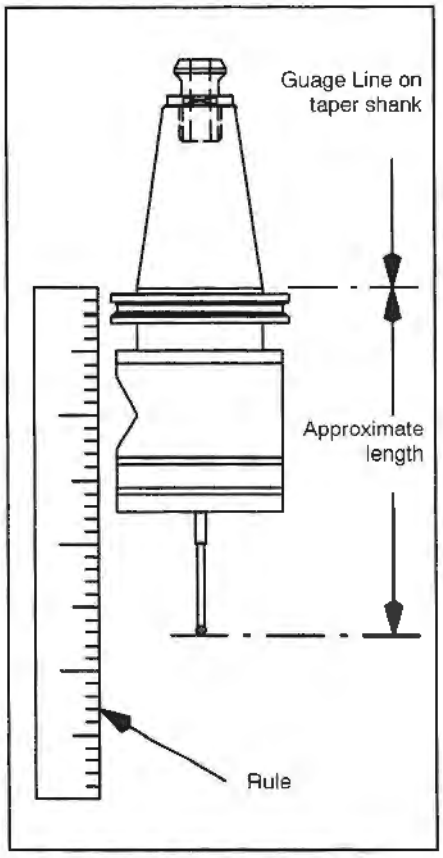
Caution 1

If T0 is not displayed, an erroneous Tool Length Set could occur resulting in subsequent damage to the workpiece, fixture and machine.

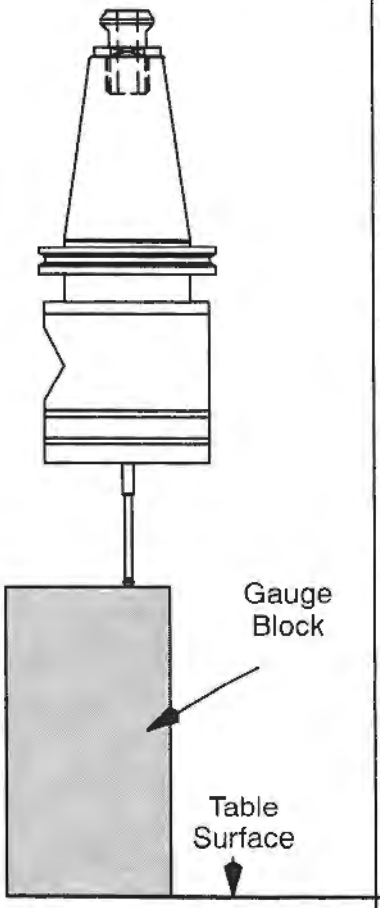
Caution 2

Do not feed the spindle face/ tool tip into the set-up gauge block. Move the Z axis a small increment and then slide the gauge block between the spindle face or tool tip and the workpiece/fixture or machine table until a proper precision slip-fit is achieved. Move the gauge block clear before moving the Z axis. Failure to follow this instruction may result in damage to machine components.

G74 Set Probe Length	
Touch, press or set the following	**Comments**
The following example procedure is for reference purposes. Due to the numerous workpiece configurations, tooling and set-up possibilities this procedure is to be used as a guide. Refer to the Programming manual for additional probe information.	
	
	
Set Z Axis Position	
	Prior to using the Probe G74, an exact Z axis position of a surface must be known. To determine the exact position of a surface, perform the following: - Remove tool from spindle. See *Unload Tool From Spindle.* - Ensure that T0 (Tool Zero) is the Active Tool displayed on the screen. See **Caution 1.** - Power feed the non rotating spindle face in X and Y axes over the surface to be used to set the probe, example 6.000 in. gauge block height. - Place 6.0000 in. height gage block on top surface of table or workpiece. Manually jog Z axis until spindle face is close to top surface of gauge block. - Using the handwheel and small increments, move Z axis until the spindle face barely touches top of gauge block - see **Caution 2**. A snug slip fit (feel) between the spindle and the top of the gauge block is required. - Record the current Z axis position displayed on the control screen.



Probe length approximation

G74 Set Probe Length (continued)	
Touch, press or set the following	**Comments**
	- Setup the Probe, refer to manufacturers documentation. - Load the Probe/holder into the Spindle, TxxM6. - Enter against the appropriate record in the tool table; - 'Probe' as the Tool Type - An approximate length of the probe within 0.1" (3mm). See figure. - Position probe in X and Y axes so the probe tip is in close proximity over the top surface of the gauge block. - In MDI, Enter G74 Zxxx, where xxx is the Z axis coordinate of the top of the gauge block. - Press Cycle Start. The Probe will move the Z axis slide to "find" the surface, and the dynamic length of the probe will be determined and automatically stored in the Tool Length column in the Tool Data table. The length calibration is complete. - The Z coordinate need not be programmed if the top surface of the gauge block also represents the specified Tram Surface value in the Cycle Parameter table.

Spindle Probe System Variables

System variables are defined and maintained by the control. They are used to make information about the control state available to the NC program. System variables used by the tool setter probe are listed below. Consult your programming manual for additional information.

Name	Definition	Program Field Name	Description
[$PRB_AVG_DIA]	Measured probe average diameter of a bore or boss. (G78 probe cycle		Range of 99999.9999mm
[$PRB_PART_LOC]	Probe Part Location is the coordinates of the measured part feature for G75-79	X,Y,Z	Range of 99999.9999mm
[$PRB_POS_MC]	Location of the most recent probe hit in machine coordinates	X,Y,Z,U,V,W,A,B,C	Range of 99999.9999mm Range of 99999.9999degrees
[$PRB_POS_PC]	Location of the most recent probe hit in program coordinates	X,Y,Z,U,V,W,A,B,C	Range of 99999.9999mm Range of 99999.9999degrees
[$PRB_WIDTH]	Measured width of a pocket or web (G79 probe cycle)		Range of 99999.9999mm
[$PRB_X_DIA]	Measured probe X axis diameter or a bore or boss. (G78 probe cycle)		Range of 99999.9999mm
[$PRB_Y_DIA]	Measured probe Y axis diameter or a bore or boss. (G78 probe cycle)		Range of 99999.9999mm
[$PROBE_HIT]	Probe Hit		FALSE = 0 TRUE = 1
[$X_POS_ERROR]	Difference between I word and measured X axis position	X	
[$Y_POS_ERROR]	Difference between J word and measured Y axis position	Y	
[$Z_POS_ERROR]	Difference between K word and measured Z axis position	Z	
[$TRUE_POS_ERR]	True vector error position of X,Y and Z axes	X,Y,Z	
[$SIZE_ERROR]	True position value plus tool offset	X,Y,Z	
[$OUT_OF_TOL]	Measured size error exceeds V word value		FALSE = 0 TRUE = 1
[$TOL_EXCEEDED]	Measured size error exceeds U word value		FALSE = 0 TRUE = 1

Name	Definition	Program Field Name	Description
[$PRB_A_ANGLE]	Angle between two measured points in the Y axis plane, R word specifies rotary A axis	R	
[$PRB_B_ANGLE]	Angle between two measured points in the X axis plane, P word specifies rotary B axis	P	
[$ANGLE_ERROR]	[$PRB_A_ANGLE] or [$PRB_B_ANGLE] angle exceeds B word tolerance	B	FALSE = 0 TRUE = 1
[$PRB_ANGL_ERR]	The error between the measured and expected angle between two features (G51.4 and G51.5)		
[$PRB_X_ANGLE]	Is the computed X axis side of the corner when P word is programmed (G75 and G76)	P	
[$PRB_Y_ANGLE]	Is the computed Y axis side of the corner when R word is programmed (G75 and G76)	R	
[$PRB_INCL_ANG]	Angle data used by G51.4 and G51.5 for second measurement positioning		
[$MAX_STOCK]	Is data for maximum stock amount for the current set of measurements (G77.1)		
[$MIN_STOCK]	Is data for minimum stock amount for the current set of measurements (G77.1)		
[$VARIATION]	Is the difference between lowest and highest measured surface location (G77.1)		

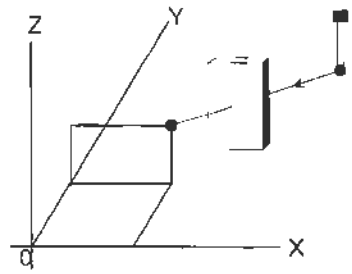
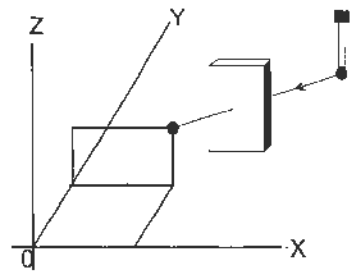
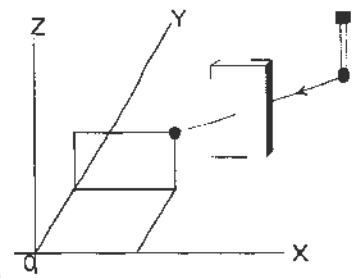
G51 Probe Multiple Axes

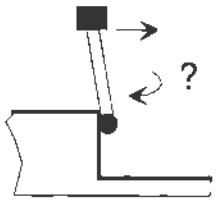
The G51.1 measures the location of a surface along an approach move made using any combination of the X, Y, and Z axes. The measured true location of the surface in the selected axis is saved in the following system registers.

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_POS_MC] X,Y,Z	axis machine coordinate data of the point of contact
[$PRB_POS_PC] X,Y,Z	axis program coordinate data of the point of contact

Note: Before performing measurements, calibrate the probe for 360 degrees using G72 A1.

G51 Probe Multiple Axes
Program Words

Word Description	Comments
Position X Word 	Specifies location to which the probe measurement span moves
Position Y Word 	Specifies location to which the probe measurement span moves
Position Z Word 	Specifies location to which the probe measurement span moves

G51 Probe Multiple Axes Program Words (continued)	
Word Description	**Comments**
Probe Process Q Word  	Q0 Double Hit Q1 Single Hit If a probe hit occurs during the approach move and the Q word specifies a single hit cycle, the coordinates of the point of contact of the probe are stored in system register [$PRB_POS_MC] and [$PRB_POS_PC]. If the Q word is zero or absent, specifying a double hit cycle, the probe retracts from the initial hit location along the approach vector by the *Probe Gage Height* distance cycle parameter. A second approach move at the *Probe Measurement Feedrate* cycle parameter is used to make the final measurement of the surface location, which is stored in system registers [$PRB_POS_MC] and [$PRB_POS_PC].
Over travel Distance D Word	Non-modal probe over travel distance allowed beyond the programmed X, Y, or Z dimension. If D is absent, no over travel beyond the programmed end point is allowed.
Controls Action E Word = 1	E0 or not programmed abort and report an alarm. E1 Set system register [$PROBE_HIT] false and continue NC program execution.

G51 Cycle Action

The probe is first positioned near the nominal location of the surface to be located.

1. Probe moves in a straight line to the programmed location plus overtravel D word distance.

2. If a probe hit occurs location of surface is measured and stored. Measurement action is determined by the Q word.

3. At cycle conclusion, the probe returns to its position prior to the G51 block execution.

G51 Programming Considerations

- Prior to executing the G51 cycle, the NC program must position the probe near the surface to be measured.

- For each axis programmed in the G51 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.

- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.

- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.

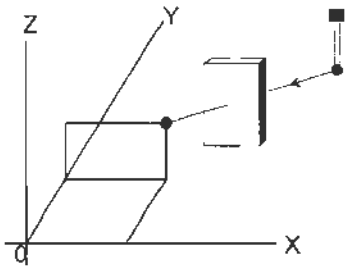
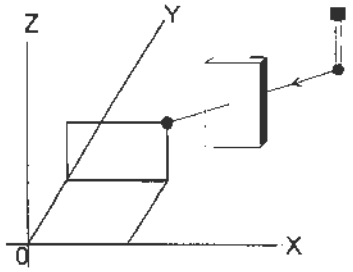
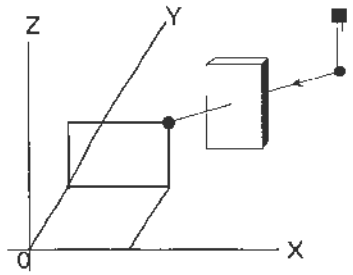
- When using 3-point measurement the probe must be calibrated for 360 degrees.

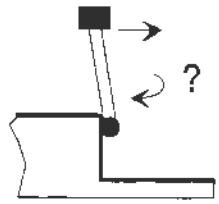
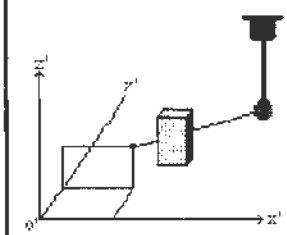
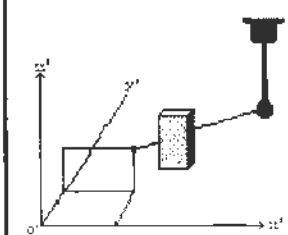
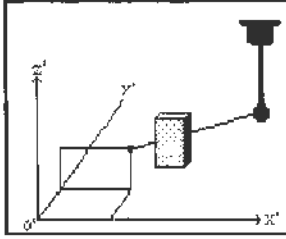
G51.1 Vector Probe Surface and Set Offsets (Optional)

The G51.1 measures the location of a surface along an approach move made using any combination of the X, Y, and Z axes. The measured true location of the surface in the selected axis is saved in the following system registers.

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_POS_MC] X,Y,Z	axis machine coordinate data of the point of contact
[$PRB_POS_PC] X,Y,Z	axis program coordinate data of the point of contact
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)] [$PRB_PART_LOC(Z)]	records part measurement coordinates

G51.1 Vector Probe Surface and Set Offsets Program Words

Word Description	Comments
Position X Word 	Specifies location to which the probe measurement span moves
Position Y Word 	Specifies location to which the probe measurement span moves
Position Z Word 	Specifies location to which the probe measurement span moves
Over travel Distance D Word	Non-modal probe over travel distance allowed beyond the programmed X, Y, or Z dimension. If D is absent, no over travel beyond the programmed end point is allowed.

G51.1 Vector Probe Surface and Set Offsets Program Words (continued)	
Word Description	**Comments**
Probe Process P Word 	Q0 Double Hit Q1 Single Hit If a probe hit occurs during the approach move and the Q word specifies a single hit cycle, the coordinates of the point of contact of the probe are stored in system register [$PRB_POS_MC] and [$PRB_POS_PC]. If the Q word is zero or absent, specifying a double hit cycle, the probe retracts from the initial hit location along the approach vector by the *Probe Gage Height* distance cycle parameter. A second approach move at the *Probe Measurement Feedrate* cycle parameter is used to make the final measurement of the surface location, which is stored in system registers [$PRB_POS_MC] and [$PRB_POS_PC].
Controls Action E Word = 1	E0 or not programmed abort and report an alarm. E1 Set system register [$PROBE_HIT] false and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages for additional information
	Non-modal parameter which specifies the expected location of the surface in X axis. The H word is used to set offsets
	Non-modal parameter which specifies the expected location of the surface in Y axis. The H word is used to set offsets
	Non-modal parameter which specifies the expected location of the surface in Z axis. The H word is used to set offsets

G51.1 Vector Probe Surface and Set Offsets Program Words (continued)	
Word Description	**Comments**
Offset Type H Word	Non-modal offset to be set based on the true position error (size error for H5) from the expected surface location specified by the I, J, and K words: 0 or not programmed Setup Offset 1 Pallet Offset 2 Fixture Offset 3 Programmable Coordinate Offset 5 Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.
Offset Number O Word	Non-modal Multiple Setup Offset number to be set Optional with H = 0 Is required with H = 2, 3, or 5
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.

G51.1 Vector Probe Surface and Set Offsets Program Words (continued)	
Word Description	**Comments**
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$PROBE_HIT] false or set [$OUT_OF_TOL] or [$TOL_EXCEEDED] Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance n the following pages for additional information.

G51.1 Cycle Action

The probe is first positioned near the nominal location of the surface to be located.

1. Probe moves in a straight line to the programmed location plus overtravel D word distance.

2. If a probe hit occurs location of surface is measured and stored. Measurement action is determined by the Q word.

3. At cycle conclusion, the probe returns to its position prior to the G51.1 block execution.

What Offsets Can Be Adjusted

Following a successful measurement, the measured true position of the bore center can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I and J words must be programmed.

G51.1 computes error between the nominal center of the bore (specified by the I and J words) and the measured center. This data is stored in the following system registers:

[$X_POS_ERROR] [$Y_POS_ERROR] [$Z_POS_ERROR]	is the difference between the I, J, and K word values and the measured position
[$TRUE_POS_ERR]	is the vector error value of X, Y, and Z axes
[$SIZE_ERROR]	is the measured size error of X, Y, and Z axes

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X, Y, and Z axis can be changed when the H word is 0 or absent, and the O word is absent. The X Y and Z axis position, and I, J and K word values are used to compute offsets. The control performs a G92.1 Position Set to the active setup offset values.

G51.1 X10 Y12 Z-10 I0 J0 K0 ; Find surface and update current active multiple setup offset

Setting Selected Setup Offsets

To apply offsets to a setup other than the active setup, the H word is 0 or absent, and the O word value is used to specify the setup record number. The X, Y and Z axis position, and I, J and K word values are used to compute offsets. The control performs a G92.1 Position Set to the O word setup selection.

G51.1 X10 Y12 Z-10 I0 J0 K0 O2 ; Find surface and update multiple setup offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values, the H word specifies the type of offset, and the O word specifies which offset record. The X, Y and Z axis position, and I, J and K word values are used to compute centerline offset. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G51.1 X10 Y12 Z-10 I0 J0 K0 H1 ; Find surface and update current active pallet values

Setting a Fixture Offset

G51.1 X10 Y12 Z-10 I0 J0 K0 H2 O3 ; Find surface and update Fixture offset 3 values

Setting a Programmable Coordinate Offset

G51.1 X10 Y12 Z-10 I0 J0 K0 H3 O5 ; Find surface and update Programmable coordinate offset 5 values

Changing Programmable Tool Offset Table values

The Programmable Tool Offset table diameter and length values can be change by programming H5 with an O word and no T word. The diameter and length tool is adjusted by data in system register [$SIZE_ERROR].

G51.1 X10 Y12 Z-5 I10.2 J12.2 K-5 H5 O6 ; change tool diameter and length record number 6 programmable tool table value.

Changing Tool Table Diameter Offset value

The Tool Table Offset diameter value can be change by programming H5 with a T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 K-5 H5 T6 ; Change tool table diameter offset and length value record number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$SIZE_ERROR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$SIZE_ERROR] exceeds the U word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

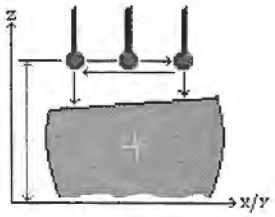
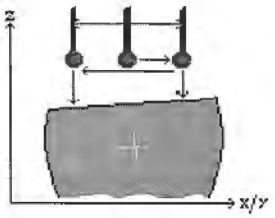
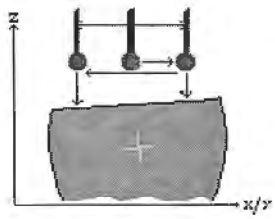
G51.1 Programming Considerations

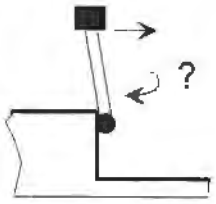
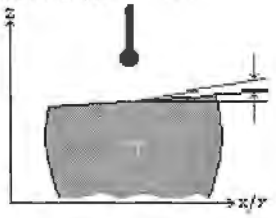
- Prior to executing the G51.1 cycle, the NC program must position the probe near the surface to be measured.
- For each axis programmed in the G51.1 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.
- When using 3-point measurement the probe must be calibrated for 360 degrees.

G51.2 Rotary Axis Measurement (Optional)

G51.2 measures the slope of a surface between two points by measuring the Z axis depth. The angle that the line between the two measured points makes with the XY plane is computed and stored in following system registers:

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_POS_MC]	Z axis machine coordinate data of the point of contact
[$PRB_POS_PC]	Z axis program coordinate data of the point of contact
[$PRB_A_ANGLE]	Is the angle that the line between two measured points makes with the XY axis plane
[$PRB_B_ANGLE]	Is the angle that the line between two measured points makes with the XY axis plane
[$ANGLE_ERROR]	Interacts with B word. If present, and the value of [$PRB_A_ANGLE] or [$PRB_B_ANGLE] exceeds the tolerance expressed by the B word [$ANGLE_ERROR] is set true.

G51.2 Rotary Axis Measurement Program Words	
Word Description	**Comments**
Position Z Word 	The expected Z axis position of the surface to be measured.
P Word 	X axis incremental distance between the two Z axis measurements. If the rotary axis is a B axis (that is, rotates about the Y axis) P must be programmed, specifying two measurements separated by an X increment.
R Word 	Y axis incremental distance between the two Z axis measurements. If the rotary axis is A axis (that is, rotates about the X axis) R must be programmed, specifying two measurements separated by an Y increment.
Over travel Distance D Word	Non-modal probe over travel distance allowed beyond the programmed X, Y, or Z dimension. If D is absent, no over travel beyond the programmed end point is allowed.

G51.2 Rotary Axis Measurement Program Words (continued)	
Word Description	**Comments**
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit If a probe hit occurs during the approach move and the Q word specifies a single hit cycle, the coordinates of the point of contact of the probe are stored in system register [$PRB_POS_MC] and [$PRB_POS_PC]. If the Q word is zero or absent, specifying a double hit cycle, the probe retracts from the initial hit location along the approach vector by the *Probe Gage Height* distance cycle parameter. A second approach move at the *Probe Measurement Feedrate* cycle parameter is used to make the final measurement of the surface location, which is stored in system registers [$PRB_POS_MC] and [$PRB_POS_PC].
Angle Tolerance B Word 	Non-modal angle tolerance for the measured angle of the surface. If the B word is present and the value of system register [$PRB_A_ANGLE] or [$PRB_B_ANGLE] exceeds the tolerance expressed by the B word, system register [$ANGLE_ERROR] is set *true*.
Offset Type H Word	Non-modal rotary axis coordinate offset to be set based on the measured angular error from the expected angle of the rotary axis. H0: Set Multiple Setup Offset for A rotary axis (if P word is programmed) or B rotary axis (if R word is programmed) to adjust the current position by the measured angle. The result of activating the offset is to make the measured surface horizontal. H absent: Do not set Setup Offset. Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.

G51.2 Rotary Axis Measurement Program Words (continued)	
Word Description	**Comments**
Offset Number O Word	Non-modal Multiple Setup Offset number to be set With H0 set Multiple Setup Offset record specified by O word. If no O word is present current Multiple Setup Offset adjusted. With H word absent, Multiple Setup Offset is not updated. However, computed values are stored in system registers [$PRB_A_ANGLE] or [$PRB_B_ANGLE] Refer to **What Offsets Can Be Adjustedon** the following pages for additional information.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$PROBE_HIT] false or set [$ANGLE_ERROR] *true* and continue NC program Execution. Refer to **What Offsets Can Be Adjusted** Offset Tolerance on the following pages for additional information.

G51.2 Cycle Action

To use G51.2, the NC program first positions the rotary axis (A or B) to place the feature to be measured in the horizontal position. Next, move the probe to a position midway between the two locations to be measured and a suitable distance above the surface.

1. Move in the + direction by P/2 in X (if P word is programmed) or R/2 in Y (if R word is programmed). Probe in Z axis.

2. If probe hit occurs, Z coordinate is stored in system registers [$PRB_POS_MC] and [$PRB_POS_PC].

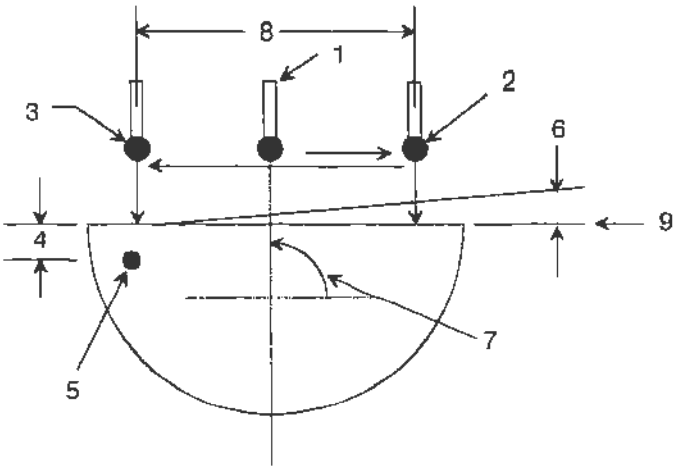
3. If Q word is zero or absent (double hit) the probe retracts from the initial hit location along the approach vector by cycle parameter *Probe Gage Height* distance. A second approach move at cycle parameter *Probe Measurement Feedrate* is used to make the final measurement of the surface location.

4. The Probe retracts to the original Z axis position, then moves by -P word or -R word and repeats measurement as before.

5. At the conclusion of the G51.2 cycle, the probe returns to its position prior to the G51.2 block execution regardless of whether or not a probe hit was detected.

Cycle Action Sample Drawing

Drawing Reference Number	Definition
1	Initial Probe Position
2	First measurement position
3	Second measurement position
4	Overtravel distance (**D word**)
5	Probe target Position
6	System register angle tolerance [$PRB_A_ANGLE] or [$PRB_B_ANGLE] (**B word**)
7	A or B axis position
8	incremental measurement distance between two points (**P or R word**)
9	expected surface location (**Z word**)



What Offsets Can Be Adjusted

G51.2 can adjust rotary axis (A or B) error by using the computed value in system register [$PRB_A_ANGLE] or [$PRB_A_ANGLE]. The H and O words are used to adjust setup offset coordinates to make the surface level.

Note: if the H word is absent, no offset data is acted on.

Setting Active Setup Offsets

G51.2 H0 ; Adjust current active setup for A (if P is programmed) or B (if R is programmed

Setting Selected Setup Offsets

G51.2 H0 O3; Adjust setup record 3 for A (if P is programmed) or B (if R is programmed

Offset Tolerances

If the second G51.2 block specifies a B word, and the value in system register [$PRB_A_ANGLE] or . [$PRB_B_ANGLE] exceeds the tolerance expressed by the B word. System register [$ANGLE_ERROR] is set true.

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test system registers [$ANGLE_ERROR] and [$PROBE_HIT] to determine whether or not the cycle completed it measurement.

G51.2 Programming Considerations

- Prior to executing the G51.2 cycle, the NC program must position the probe near the angle to be measured.
- The probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate.*
- If the rotary axis is an A axis (that is, rotates about the X axis) the R word must be programmed specifying two measurements separated by a Y increment.
- If the rotary axis is an B axis the P word must be programmed specifying two measurements separated by a X increment.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height.*
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate.* When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate.*

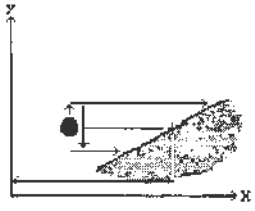
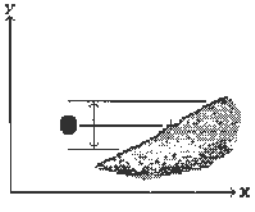
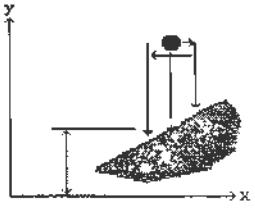
G51.3 Angle Measurement in X or Y Plane (Optional)

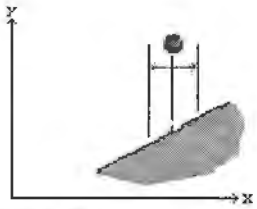
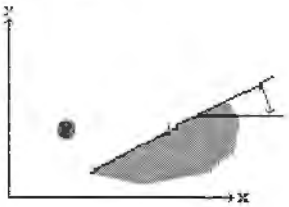
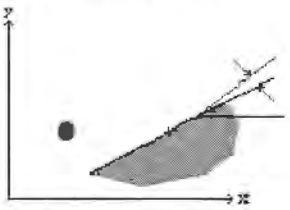
G51.3 measures the angle that a vertical surface makes with the +X axis by measuring the X or Y axis difference between two points. The angle of the line between the two measured points makes with the +X axis is computed and stored in the following system register:

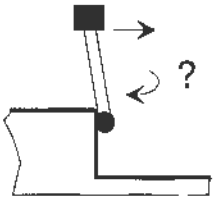
[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_POS_MC]	X or Y machine coordinate data when probe hit occurs
[$PRB_POS_PC]	X or Y program coordinate data when probe hit occurs
[$PRB_ANGLE]	Is the angle that the line between two measured points makes with the +X axis
[$PRB_ANGL_ERR]	This value is the computed angular error between the measured and the expected angle A word if provided.

Note: The measured angle is made available to the NC program, which can use it to rotate the coordinate system in the XY plane using the Type II ROT block.

G51.3 Angle Measurement in X and Y Plane Program Words

Word Description	Comments
Location X Word 	The expected X axis position of the surface to be measured midway between the two points of measurements. R word is required.
R Word Incremental Distance 	Is the distance between the two measurements in the Y axis. If the X location of the expected surface is specified, R must be programmed, specifying two measurements separated by Y axis increment.
Location Y Word 	The expected Y axis position of the surface to be measured midway between the two points of measurements. P word is required.

G51.3 Angle Measurement in X and Y Plane Program Words (continued)	
Word Description	**Comments**
P Word Incremental Distance 	Is the distance between the two measurements in the X axis. If the Y location of the expected surface is specified, R must be programmed, specifying two measurements separated by X axis increment.
Over travel Distance D Word	Non-modal probe over travel distance allowed beyond the programmed X or Y dimension. If D is absent, no over travel beyond the programmed end point is allowed.
Nominal Angle A Word 	Is the non-modal or expected angle of the measured surface with the +X axis. If the A word is not programmed, zero is assumed. The angle between the two measured points is stored in system register [$PRB_ANGLE]. If the A word is present, the error between the measured and expected angle (A word or zero degrees) is stored in system register [$PRB_ANGL_ERR]
Angle Tolerance B Word 	Is the non-modal angle tolerance for the measured angle of the surface. If the B word is present and the value of system register [$PRB_ANGL_ERR] exceeds the tolerance expressed by the B word, system register [$ANGLE_ERROR] is set *true*.

G51.3 Angle Measurement in X and Y Plane Program Words (continued)	
Word Description	**Comments**
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$PROBE_HIT] false or set [$ANGLE_ERROR] *true* and continue NC program Execution.
Probe Process Q Word  	Q0 Double Hit Q1 Single Hit If a probe hit occurs during the approach move and the Q word specifies a single hit cycle, the coordinates of the point of contact of the probe are stored in system register [$PRB_POS_MC] and [SPRB_POS_PC]. If the Q word is zero or absent, specifying a double hit cycle, the probe retracts from the initial hit location along the approach vector by the *Probe Gage Height* distance cycle parameter. A second approach move at the *Probe Measurement Feedrate* cycle parameter is used to make the final measurement of the surface location, which is stored in system registers [$PRB_POS_MC] and [$PRB_POS_PC].

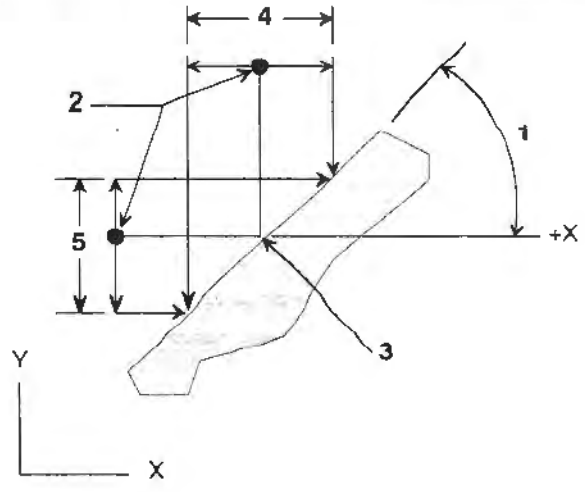
G51.3 Cycle Action

To use G51.3, the NC program first positions the probe to a position midway between the two locations to be measured and at a suitable distance away from the surface in X and Y axes, and at a suitable Z axis depth.

1. Move in the + direction by P/2 in X (if P word is programmed) or R/2 in Y (if R word is programmed).
2. If probe hit occurs, X or Y coordinate are stored in system registers [$PRB_POS_MC] and [$PRB_POS_PC].
3. If Q word is zero or absent (double hit) the probe retracts from the initial hit location along the approach vector by cycle parameter *Probe Gage Height* distance. A second approach move at cycle parameter *Probe Measurement Feedrate* is used to make the final measurement of the surface location.
4. The Probe retracts to the original X or Y position, then moves by -P word or -R word and repeats measurement as before.
5. At the conclusion of the G51.2 cycle, the probe returns to its position prior to the G51.2 block execution regardless of whether or not a probe hit was detected.

Cycle Action Sample Drawing

Drawing Referenece Number	Definition
1	Non-modal nominal or expected angle of the measured surface with the +X axis. If (A word) is not programmed, zero is assumed. (**A word**)
2	Initial probe location
3	Programmed X or Y axis location
4	Y axis location of the expected surface (**P word**)
5	X axis location of the expected surface (**R word**)



G51.3 Programming Considerations

- Prior to executing the G51.3 cycle, the NC program must position the probe near the angle to be measured.
- The probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- If X location surface is specified the R word must be programmed. If Y location surface is specified the P word must be programmed
- If X and R words are specified, the two points to be measured is the R word distance apart in the Y axis, and the measurements are make by moving the probe parallel to the X axis.
- If Y and P words are specified, the two points to be measured is the P word distance apart in the X axis, and the measurements are make by moving the probe parallel to the Y axis.
- Angle measurement is made with respect to the +X axis, and is between +90 and
  -90 degrees.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.

G51.4 Measure Feature-to-Feature XY Plane (Optional)

G51.4 obtains the measurement distance (in the XY axis plane) between two features. Basically, G51.4 obtains previous surface measurement (G51.1, G75, G76, G77, G78 or G79) data to compute distance between parts.

The data from the first measurement cycle (which can be any cycle that sets system register [$PRB_PART_LOC(X Y or Z)) is captured by programming G51.4 following the probe cycle. The second G51.4 computes the distance between the first and second feature and can optionally test the distance against the expected values.

Note: Distance measurements to another feature or Part are always referenced from the first feature or Part 1 you define. The first feature or Part 1 is defined by programming G51.4 with no program words.

Example:

G76 Xx Yx ; locate external corner of first part
G51.4; Use data obtained by G76 as reference point to Part 2 or Part 3.

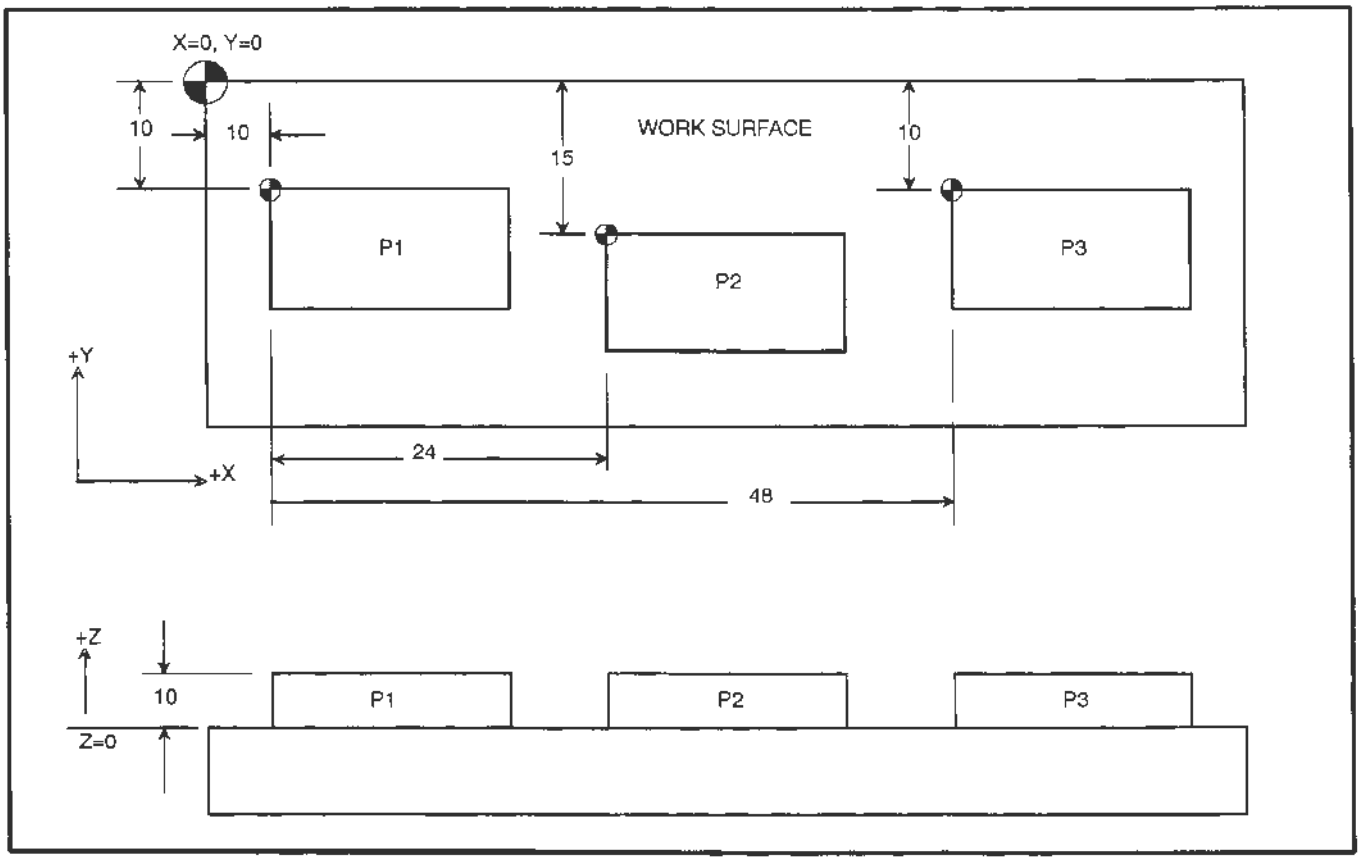
G51.4 Measure Feature-to-Feature in XY Plane Program Words	
Word Description	**Comments**
X Word	Non-modal expected X axis signed incremental distances from Part 1 to Part 2. Either X and Y or A and P words must be programmed.
Y Word	Non-modal expected Y axis signed incremental distances from Part 1 to Part 2. Either X and Y or A and P words must be programmed.
A Word	Is the non-modal expected angle from the +X axis from Part 1 to Part 2.. Either X and Y or A and P words must be programmed.
P Word	Is the non-modal expected distance from the +X axis from Part 1 to Part 2. Either X and Y or A and P words must be programmed..
B Word	Is the non-modal angle tolerance for error between the measured angle the line makes from P1 to P2 and the expected angle specified by the A word or by the X and Y words.
Offset Type H Word	Non-modal type of offset to be set based on the true position error (H0 - H3) or size error H5: 0 or not programmed Setup Offset 1 Pallet Offset 2 Fixture Offset 3 Programmable Coordinate Offset 5 Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** at the end of this chapter for additional information.

G51.4 Measure Feature-to-Feature in XY Plane Program Words (continued)	
Word Description	**Comments**
Offset Number O Word	Non-modal Multiple Setup Offset number to be set Optional with H = 0 Is required with H = 2, 3, or 5
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
R Word	Non-modal cycle command used with T word: 0 or not programmed: outside measurement, system register [SSIZE_ERROR] is **added** to the tool table diameter offset 1: inside measurement, system register [$SIZE_ERROR] is **subtracted** from the tool table diameter offset.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.

G51.4 Measure Feature-to-Feature in XY Plane Program Words (continued)	
Word Description	**Comments**
L Word	Non-modal experience correction for setting tool offsets. The L word valne on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages for additional information.

G51.4 Program example

The following program segment illustrates the basic concept of the G51.4 cycle.



G51.4 Programming Example

```
G0  X9.75 Y-9.75 ; Move X and Y axes to the corner of P1
    Z9.5 ; Position probe in Z axis .5 inch below the top of P1
G76 X10 Y-10 ; Find the corner of P1
G51.4 ; Use P1 as the reference distance to P2 and P3
G0  Z10.5 ; Retract probe above P1
    X33.75 Y-14.75 ; Move X and Y axes to the corner of P2
    Z9.5 ; Position probe in Z axis .5 inch below the top of P2
G76 X34 Y-5 ; Find the corner of P2
G51.4 X24 Y-5 H2 O2 ; Check distance between expected and measured distance and replace fixture offset 2
G0  Z10.5 ; Retract probe above P2
    X57.75 Y-9.75 ; Move X and Y axes to the corner of P3
    Z9.5 ; Position probe in Z axis .5 inch below the top of P3
G76 X58 Y-10 ; Find the corner of P3
G51.4 X48 Y0 H2 O3 ; Check distance between expected and measured distance and replace fixture offset 3
G0  Z10.5 ; Retract probe above P3
```

Multiple Setup Coordinate Offset

To set a setup coordinate system offset, G51.4 uses H0 and the O word to specify the Multiple Setup table value. If no O word is included, the Multiple Setup coordinate of the active setup is changed.

G51.4 H0 O2 ; Sets multiple setup table record 2 values

X and Y coordinate offset of the Multiple Setup are changed by adding system register [$X_POS_ERROR] and [$Y_POS_ERROR] to the existing setup X and Y offset respectively. Also, the Multiple Setup coordinates are relative to P1 and are not simply a set of program coordinates of P2.

Setting A Pallet Offset

For Pallet, Fixture and Programmable Offsets [$X_POS_ERROR] and [$Y_POS_ERROR] replace the previous offsets.

G51.4 H1 ; Set the active pallet table values

Setting Fixture Offset

To set a Fixture Offset, G51.4 uses H2 and the O word to specify the Fixture Offset table value.

G51.4 H2 O2 ; Set fixture offset table record 2 values

For Pallet, Fixture and Programmable Offsets [$X_POS_ERROR] and [$Y_POS_ERROR] replace the previous offsets.

Setting NC Programmable Offset

To set an NC Programmable Coordinate Offset, G51.4 uses H5 and the O word to specify the NC Programmable Offset table value.

G51.4 H5 O2 ; Set NC Program Offsets table record 2 values

For Pallet, Fixture and Programmable Offsets [$X_POS_ERROR] and [$Y_POS_ERROR] replace the previous offsets.

What Tool Offsets Can I apply

G51.4 can adjust tool diameter by adding system register data [$SIZE_ERROR] to the tool used to machine the features. Tool diameter offsets can be adjusted in two areas, the Programmable Tool Offset table and Tool Data Table.

The G51.4 block must contain H5, the Tool Reference Number or Tool Record Number of the tool used to machine the features, and the O word to reference the Programmable Tool Offset table value or R word to reference the Tool Data Table value.

Changing Programmable Tool Offset Table Diameter value

Since this cycle can measure either the outside or inside of a part, the R word controls how system value [$SIZE_ERROR] is used to update the Programmable Tool Offset Table Diameter value.

R Word:

0 or not programmed: outside measurement, system register [$SIZE_ERROR] is **added** to the tool table diameter offset

1: inside measurement, system register [$SIZE_ERROR] is **subtracted** from the tool table diameter offset.

G51.4 H5 O6 R0 ; Adjust programmable tool offset table record 6 diameter by system register [$SIZE_ERROR]

Since the R word is 0, Record number 6 (tool diameter) is changed by adding system register value [$SIZE_ERROR].

Changing Tool Table Diameter Offset value

Since this cycle can measure either the outside or inside of a part, the R word controls how system value [$SIZE_ERROR] is used to update the Tool Data Table Offset Diameter value.

R Word:

0 or not programmed: outside measurement, system register [$SIZE_ERROR] is **added** to the tool table diameter offset

1: inside measurement, system register [$SIZE_ERROR] is **subtracted** from the tool table diameter offset.

G51.4 H5 T6 R1 ; Subtract [$SIZE_ERROR] value from tool table diameter offset value for tool number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the second G51.4 block contains a V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the second G51.4 block contains a U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

If the second G51.4 block specifies a B word, it is the tolerance on the angle between the line from P1 to P2 and the +X axis. If system register [$PRB_ANGL_ERR] exceeds the B word value, system register [$ANGLE_ERROR] is set true.

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test system registers [$OUT_OF_TOL], [$TOL_EXCEEDED], and [$ANGLE_ERROR] to determine whether an error occurred.

G51.4 Program Considerations

- The first feature or Part 1 is defined by programming G51.4 with no program words. All reference to additional features or parts are taken from the first part (P1) you define.
- The distance to Part 2 must be specified on the second G51.4 block.
- Tool diameter offset can not be set if the measurements were made using G78 or G79.
- X and Y or A and P signed incremental distance must be programmed
- To set the tool diameter offset in the Tool Table, the T word in the G51.4 block must contain the Tool Reference Number or the Tool Record Number of the tool used to machine the part.
- To set a Programmable Tool Table Offset value the G51.4 block must contain H5 and the O word must contain the number of the Programmable Tool Offset to be adjusted.

G51.5 Measure Feature-to-Feature Z Plane (Optional)

G51.5 obtains the measurement distance (in the Z axis plane) between two features and the angle formed by the line between the two measurements and the XY plane. Basically, G51.5 obtains previous surface measurement (G51.1, G75, G76, G77, G78 or G79) data to compute distance between parts.

The data from the first measurement cycle (which can be any cycle that sets system register [$PRB_PART_LOC(Z)) is captured by programming G51.5 following the probe cycle. The second G51.5 computes the distance between the first and second feature and can optionally test the distance against the expected values.

Note: Distance measurements to another feature or Part are always referenced from the first feature or Part 1 you define. The first feature or Part 1 is defined by programming G51.5 with no program words.

Example:

G77 Zx ; locate surface of first part
G51.5 ; Use data obtained by G77 as reference point to Part 2 or Part 3.

G51.5 Measure Feature-to-Feature Z Plane Program Words

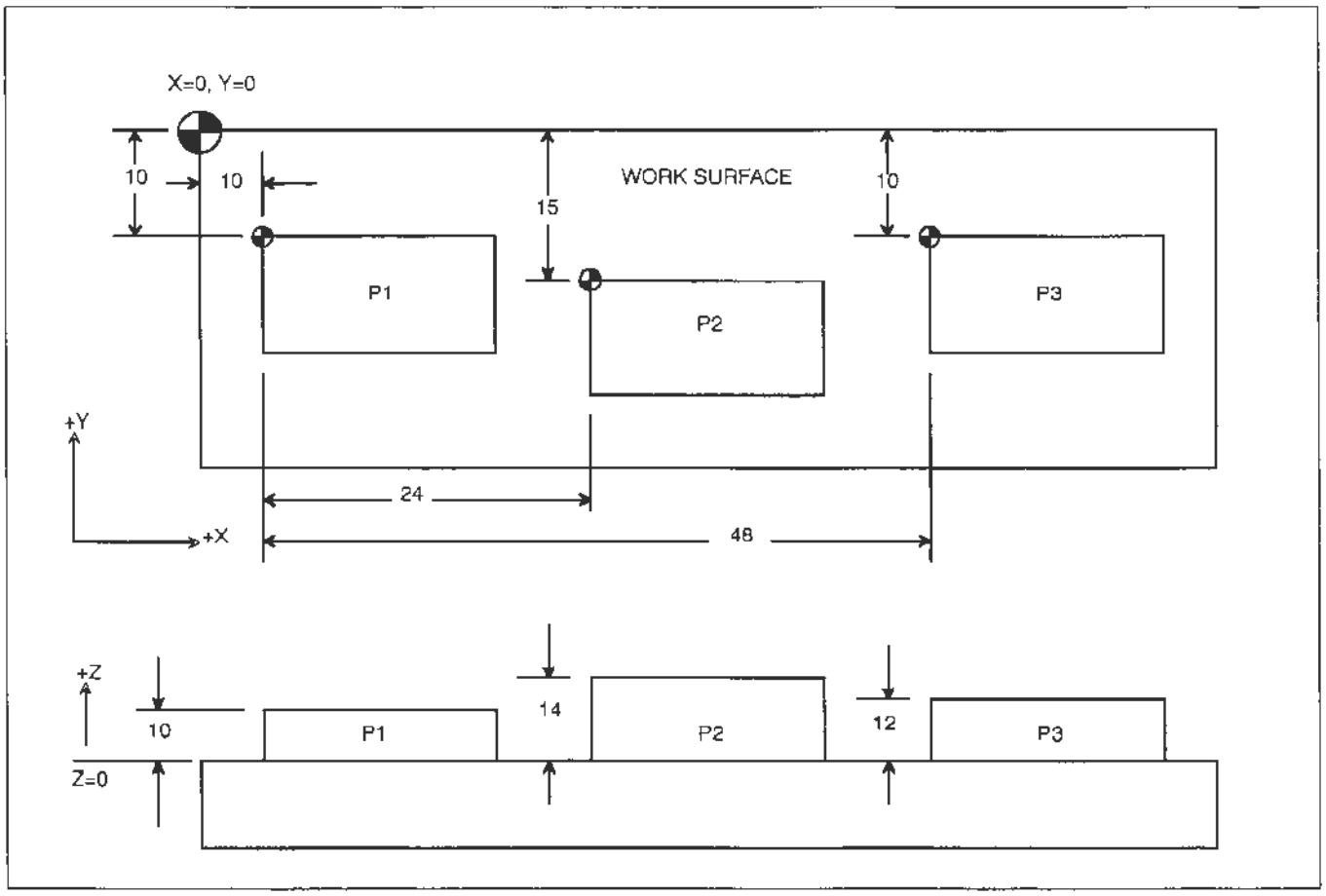
Word Description	Comments
Z Word	Non-modal expected Z axis signed incremental distances from Part 1 to Part 2. The Z word is required.
A Word	Is the optional non-modal expected angle from the XY plane to the line from Part 1 to Part 2.
P Word	Is the optional non-modal expected distance between Part 1 to Part 2 in the XY plane.
B Word	Is the non-modal angle tolerance for error between the measured angle the line makes from P1 to P2 and the expected angle specified by the A word or by the Z and P words. If the B word is programmed, either the A word or P word must be programmed.
Offset Type H Word	Non-modal type of offset to be set based on the true position error (H0 - H3) or size error H5: 0 or not programmed Setup Offset 1 Pallet Offset 2 Fixtnre Offset 3 Programmable Coordinate Offset 5 Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.

G51.5 Measure Feature-to-Feature Z Plane Program Words (continued)	
Word Description	**Comments**
Offset Number O Word	Non-modal Multiple Setup Offset number to be set Optional with H = 0 Is required with H = 2, 3, or 5
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.

G51.5 Measure Feature-to-Feature Z Plane Program Words (continued)	
Word Description	**Comments**
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages for additional information.

G51.5 Program example

The following program segment illustrates the basic concept of the G51.5 cycle.




```
G0  X12.0 Y-12.0 ; Move X and Y axes to the top of P1 surface
    Z10.25 ; Position probe in Z axis .25 inch above the top of P1
G77 Z10 D.1 ; Find the top surface of P1
G51.5 ; Use P1 as the reference distance to P2 and P3
G0  Z15.5 ; Retract probe above P2 for clearance
    X35.0 Y-16.0 ; Move X and Y axes above surface of P2
    Z14.25 ; Possition probe .25 above P2
G77 Z14 D.1 ; Find the surface of P2
G51.5 Z14 H5 T2 ; Check distance and adjust tool table length for T2
G0  Z15.5 ; Retract probe
    X59 Y-12.0 ; Move X and Y axes above surface of P3
    Z12.25 Position probe .25 above P3
G77 Z12 D.1 ; Find surface of P3
G51.5 Z12 H5 T2 O2 ; Check distance and adjust programmable tool offset table record 2 length
G0  Z12.5 ; Retract probe
```

What Offsets Can I Adjust

Multiple Setup Coordinate Offset

To set a setup coordinate system offset, G51.5 uses H0 and the O word to specify the Multiple Setup table value. If no O word is included, the Multiple Setup coordinate of the active setup is changed.

G51.5 H0 O2 ; Sets Z axis multiple setup table record 2 value

Z axis coordinate offset of the Multiple Setup are changed by adding system register [$Z_POS_ERROR] to the existing setup Z offset respectively. Also, the Multiple Setup coordinates are relative to P1 and are not simply a set of program coordinates of P2.

Setting A Pallet Offset

For Pallet, Fixture and Programmable Offsets [$X_POS_ERROR] and [$Y_POS_ERROR] replace the previous offsets.

G51.5 H1 ; Set the active pallet table values

Setting Fixture Offset

To set a Fixture Offset, G51.5 uses H2 and the O word to specify the Fixture Offset table value.

G51.5 H2 O2 ; Set Z axis fixture offset table record 2 value

For Pallet, Fixture and Programmable Offsets [$X_POS_ERROR] and [$Y_POS_ERROR] replace the previous offsets.

Setting NC Programmable Offset

To set an NC Programmable Coordinate Offset, G51.5 uses H5 and the O word to specify the NC Programmable Offset table value.

G51.5 H5 O2 ; Set Z axis NC Program Offsets table record 2 value

For Pallet, Fixture and Programmable Offsets [$X_POS_ERROR] and [$Y_POS_ERROR] replace the previous offsets.

What Tool Offsets Can I apply

G51.5 can adjust tool length by adding system register data [$SIZE_ERROR] to the tool used to machine the features. Tool length offsets can be adjusted in two areas, the Programmable Tool Offset table and Tool Data Table.

The G51.5 block must contain H5, the Tool Reference Number or Tool Record Number of the tool used to machine the features, and the O word to reference the Programmable Tool Offset table value or R word to reference the Tool Data Table value.

Changing Programmable Tool Offset Table Length value

G51.4 H5 O6 ; Adjust programmable tool offset table record 6 length by system register [$SIZE_ERROR]

Record number 6 (tool length) is changed by adding system register value [$SIZE_ERROR].

Changing Tool Table Length Offset value

G51.4 H5 T6 R1 ; Adjust programmable tool table length value for tool number 6 by system register [$SIZE_ERROR]

Tool Table Record number 6 (tool length) is changed by adding system register value [$SIZE_ERROR].

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the second G51.5 block contains a V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the second G51.5 block contains a U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

If the second G51.5 block specifies a B word, it is the tolerance on the angle between the line from P1 to P2 and the +X axis. If system register [$PRB_ANGL_ERR] exceeds the B word value, system register [$ANGLE_ERROR] is set true.

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test system registers [$OUT_OF_TOL], [$TOL_EXCEEDED], and [$ANGLE_ERROR] to determine whether an error occurred.

G51.5 Program Considerations

- The first feature or Part 1 is defined by programming G51.5 with no program words. All reference to additional features or parts are taken from the first part (P1) you define.
- The distance to Part 2 must be specified on the second G51.5 block.
- Z word must be programmed.
- When the B word is programmed, either A and P must be programmed.
- To set the tool length offset in the Tool Table, the T word in the G51.5 block must contain the Tool Reference Number or the Tool Record Number of the tool used to machine the part.
- To set a Programmable Tool Table Length Offset value the G51.5 block must contain H5 and the O word must contain the number of the Programmable Tool Offset to be adjusted.

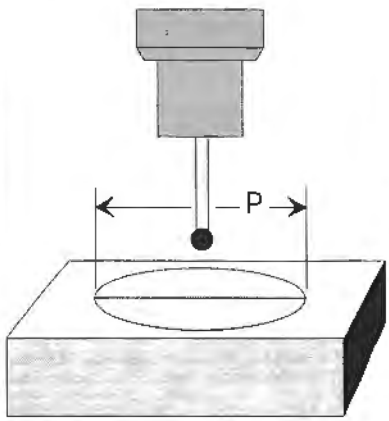
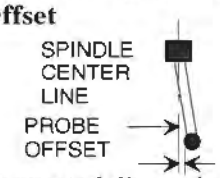
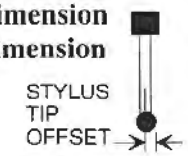
G72 Calibrate Probe Dimension

G72 performs a complete calibration of the probe. Calibration measures two separate quantities: the amount by which the center of the stylus is offset from the spindle centerline and the effective stylus size (measured from the centerline to the contact point) in the +X, -X, +Y and -Y directions (for XY calibration no A word) or at 30 degree intervals (for 360 degree calibration when A word is 1).

Before G72 can be executed, the probe must be positioned such that the spindle centerline is concentric with the centerline of a reference hole of known dimension, and the tip is located just below the hole surface.

Note: Ensure the probe tip Nominal Diameter value is entered in the Tool Data Table. A typical value might be .236 inch.

G72 Calibrate Probe Dimension Program Words

Word Description	Comments
Probe Diameter P Word 	The P word is the unsigned non-modal diameter of the reference hole being used for calibration.Computed Spindle Centerline calibration Offset values are stored in tool data table fields **X Probe Offset and Y Probe Offset**  Computed dimensions of the stylus tip are stored in Probe Cycle Parameter Table fields: **+X Stylus Tip Dimension** **-X Stylus Tip Dimension** **+Y Stylus Tip Dimension** **-Y Stylus Tip Dimension**  At the conclusion of calibration, the probe positions at the cycle start location or not programmed XY axis calibration selected
A Word	A0 or not programmed XY axis calibration selected A1 360 degree calibration is selected. Calibration values are stored in system register [$PROBE_CALIB] If the probe type is capable of operation at any spindle angle, an A word of 1 will cause the spindle to enter C axis mode. The spindle will rotate 30 degrees for each of 12 hits at 30 degree increments.

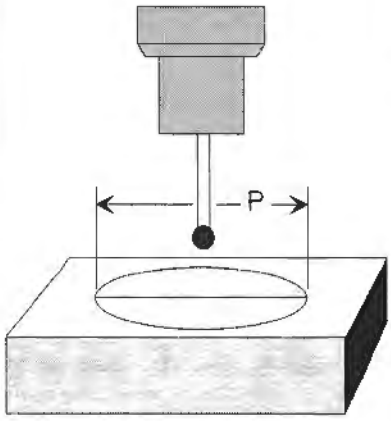
G73 Set Probe Stylus Tip Dimensions

G73 performs the same calibration procedures as G72. The difference between these G codes is G73 does not update Tool Data Table fields **X Probe Offset** and **Y Probe Offset**. G73 will use Tool Data Table fields **X Probe Offset** and **Y Probe Offset** values during the measurement process, but will only update Cycle Parameter Stylus Tip Dimensions +X, -X, +Y, -Y. Basically, this gives you the flexibility if more than one probe is used, or if the tip dimensions must be recalibrated without recomputing the Probe Stylus Offset. Also, keep in mind only the last set of calibrated Stylus Tip Dimension values are retained by the control.

Before G72 can be executed, the probe must be positioned such that the spindle centerline is concentric with the centerline of a reference hole of known dimension, and the tip is located just below the hole surface.

Note: Ensure the probe tip Nominal Diameter value is entered in the Tool Data Table. A typical value might be .236 inch.

G73 Set Probe Stylus TipDimensions Program Words

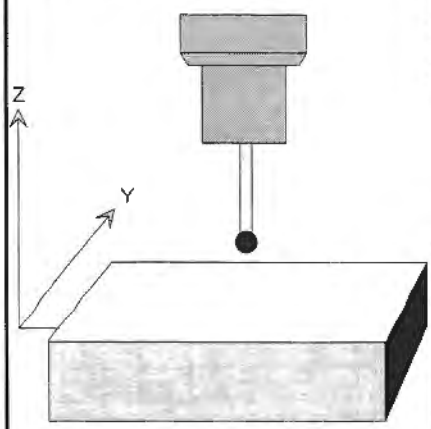
Word Description	Comments
Probe Diameter P Word 	The P word is the unsigned non-modal diameter of the reference hole being used for calibration. G73 will use Tool Table Values **X Probe Offset** and **Y Probe Offset** Computed dimensions of the stylus tip are stored in Probe Cycle Parameter Table fields: **+ X Stylus Tip Dimension** **-X Stylus Tip Dimension** **+Y Stylus Tip Dimension** **-Y Stylus Tip Dimension** At the conclusion of calibration, the probe returns to the Cycle Start Position
A Word	A0 or not programmed XY axis calibration selected A1 360 degree calibration is selected. Calibration values are stored in system register [$PROBE_CALIB]

G74 Set Probe Length

The G74 Set Probe Length cycle measures the dynamic probe length and saves the value in the Tool Data Table field **Length** for the probe. The dynamic length is the distance from the tool reference surface (spindle nose) to the tip of the probe at the instant the probe registers a hit.

Prior to performing the G74 cycle, the approximate length of the probe stylus must be entered into the Tool Data Table. This approximate length must be accurate to ± the *Probe Gage Height* cycle parameter value. The probe is then loaded into the spindle and positioned above (along the Z axis) a reference surface of known position.

G73 Set Probe Stylus Tip Dimensions Program Words

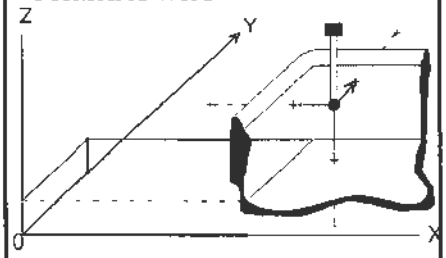
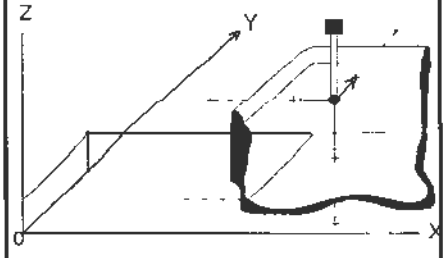
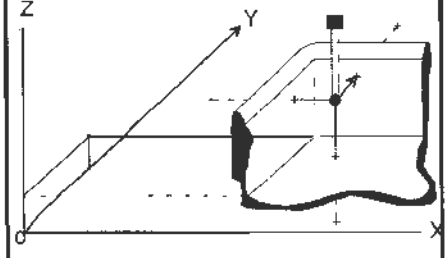
Word Description	Comments
Reference Surface Z Word 	Specifies the Optional absolute Z coordinate of the reference surface in program coordinates Executing G74 causes the probe to move in the minus Z direction until a hit is detected. After the probe hit, the probe retracts to *Probe Gage Height* above the surface detected. The probe's Tool Length field iu the tool table is updated with the dynamic probe length. If the Z word is not present in the G74 block, the *Tram Surface* value is used. **Note:** To protect the probe and machine from damage, the maximum depth of the Z axis move is limited to *Probe Gage Height* below the reference surface dimension, using the approximate probe tool length present in the tool table when the G74 cycle starts.

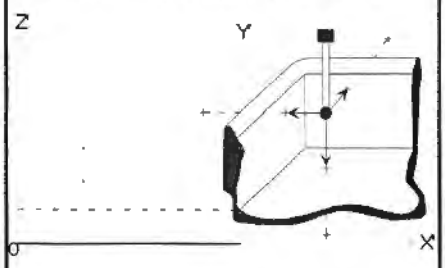
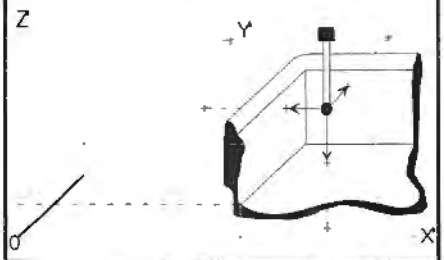
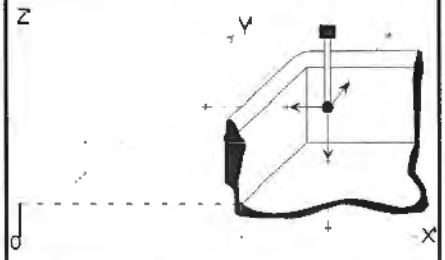
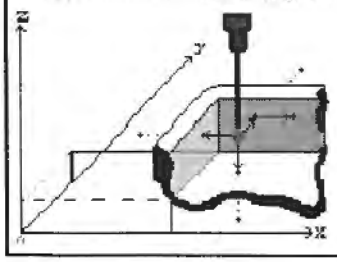
G75 Probe to Locate Internal Corner

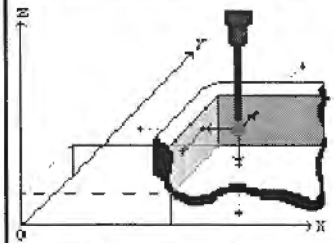
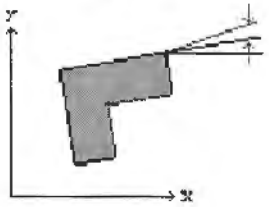
This cycle is used to locate the X, Y, and Z dimensions of an inside corner whose sides are parallel or nearly parallel to the X, Y, and Z axes. For true corner location G75 provides a B word for angle measurement of the X and Y axis surface. Once measurement is made, the following system register store the data.

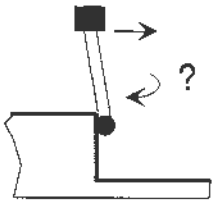
[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)] [$PRB_PART_LOC(Z)]	records the true corner location of the axis used to take measurement
[$PRB_POS_MC(X,Y, Z)] [$PRB_POS_PC(X,Y, Z)]	records X, Y, and Z machine coordinates, and program coordinate surface locations of the last probe hit when detected
[$PRB_X_ANGLE]	Angle of the X axis surface located
[$PRB_Y_ANGLE]	Angle of the Y axis surface located
[$PRB_INCL_ANG]	If both X and Y are included in the G75 block, and the optional incremental move (G51.3, G51.4, or G51.5) to a second measurement position is specified, the included angle value is used.

G75 Set Probe to Locate Internal Corner Program Words

Word Description	Comments
Position X Word 	Specifies nominal inside location of the corner in X axis.
Position Y Word 	Specifies nominal inside location of the corner in Y axis.
Position Z Word 	Specifies nominal inside location of the corner in Z axis.

G75 Set Probe to Locate Internal Corner Program Words (continued)	
Word Description	**Comments**
Optional I Word Reset Coordinate 	Optional non-modal X axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinates on the following pages for additional information.
Optional J Word Reset Coordinate 	Optional non-modal Y axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinates on the following pages for additional information.
Optional K Word Reset Coordinate 	Optional non-modal Z axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinates on the following pages for additional information.
D Word Overtravel Distance	Non-modal probe over travel distance allowed beyond the programmed X or Y dimension. If D is absent, no over travel beyond the programmed end point is allowed.
P Word Incremental Distance (Opt.) 	Non-modal signed X axis incremental distance to the second measurement location. If P is present, X and Y must also be programmed. When used, the computed angle or the corner is stored in system register [$PRB_X_ANGLE] If the P word is absent only a single measurement is made in the Y axis.

G75 Set Probe to Locate Internal Corner Program Words (continued)	
Word Description	**Comments**
R Word Incremental Distance (Opt.) 	Non-modal signed Y axis incremental distance to the second measurement location. If R is present, X and Y must also be programmed. When used, the computed angle or the corner is stored in system register [$PRB_X_ANGLE] If the R word is absent only a single measurement is made in the X axis.
B Word Angle Tolerance (Opt.) 	Is the non-modal angle tolerance for the measured angle of the X and Y surface. If either system registers [$PRB_X_ANGLE] or [$PRB_Y_ANGLE] is outside to ± the B word, system register [$ANGLE_ERROR] is set true. Note: the B word value is equal to half of the total angle tolerance.
O Word defines offset number to be set.	Optional with H = 0 Required with H = 2 or 3 ignored otherwise
H Word Type of offset to be set based on I, J, K words. Actions of this word are effected by the O word.	0 or not programmed: Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.
E Word	E0 or not programmed abort and report an alarm. E1 Set system register[$PROBE_HIT] false and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tolerance on the following pages for additional information.

G75 Set Probe to Locate Internal Corner Program Words (continued)	
Word Description	**Comments**
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit If the Q word value is 1, specifying a single hit cycle, the measurements are made during the initial move toward the part at the *Probe Approach Feedrate* cycle parameter. If the Q word is absent or zero, specifying a double hit cycle, the initial move toward the part is made at the *Probe Approach Feedrate* cycle parameter. When the probe hits the part, ~~the~~ it retracts by the *Probe Gage Height* cycle parameter, then executes a second approach span limited to twice the *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter for the actual data acquisition.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.

G75 External Corner Cycle Action

Measure X Coordinate Of The Corner

1. Advance along the X axis toward the programmed nominal value and record the probe hit position.
2. Return to the start position.
3. Report hit status based on the E word.
4. If R word is programmed, advance along the Y axis by the signed R increment, then make a second measurement in X axis by moving toward the programmed X value and record the probe hit position.
5. Return to the position.
6. Report hit status based on the E word.
7. Perform any required offsets

Measure Y Coordinate Of The Corner

1. Advance along the Y axis toward the programmed nominal value and record the probe hit position.
2. Return to the start position.
3. Report hit status based on the E word.
4. If P word is programmed, advance along the X axis by the signed P increment, then make a second measurement in Y axis by moving toward the programmed Y value and record the probe hit position.
5. Return to the start position.
6. Report hit status based on the E word.
7. Perform any required offsets

Measure Z Coordinate Of The Corner

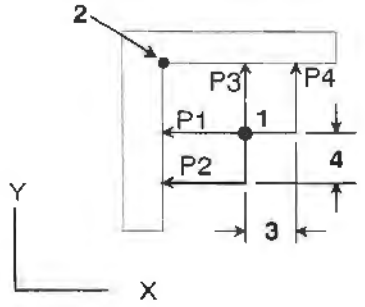
1. Advance along the Z axis toward the programmed nominal value and record the probe hit position.
2. Return to the start position.
3. Report hit status based on the E word.
4. Perform any required offsets

Cycle Action Sample Drawing

G75 With Sides Parallel To X and Y axis

Drawing Reference Number	Definition
1	Probe Start Point
2	Nominal corner location (programmed X, Y, Z)
3	Incremental distance to the second measurement location (**P word**)
4	Incremental distance to the second measurement location (**R word**)

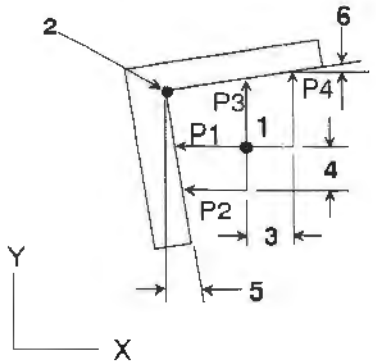
First measurement is made in X direction position 1 (P1)
Second measurement is made in X direction if R word is present position 2 (P2)
Third measurement is made in Y direction position 3 (P3)
Fourth measurement is made in Y direction if P word is present position 4 (P4)



G75 With Sides Not Parallel To X and Y axis

Drawing Reference Number	Definition
1	Initial Probe Position
2	Nominal Corner location (programmed X, Y, Z)
3	Incremental distance to the second measurement location (**P word**)
4	Incremental distance to the second measurement location (**R word**)
5	Represents Y axis probe angle [$PRB_Y_ANGLE]
6	Represents X axis probe angle [$PRB_X_ANGLE]

First measurement is made in X direction position 1 (P1)
Second measurement is made in X direction if R word is present position 2 (P2)
Third measurement is made in Y direction position 3 (P3)
Fourth measurement is made in Y direction if P word is present position 4 (P4)



What Offsets Can Be Adjusted

Following a successful corner measurement, the measured position of the corner can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, at least one of the I, J, or K words corresponding to the axis or axes (X, Y, or Z that was measured) must be specified. The following system registers are used to compute offset data:

[$X_POS_ERROR] [$Y_POS_ERROR] [$Z_POS_ERROR]	The difference between the I, J, and K word value and the measured position if both X and I are programmed, if both Y and J are programmed, or if both Z and K are programmed
[$TRUE_POS_ERR]	is the true position error vector value of the corner located

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X, Y, or Z axis can be changed when the H word is 0 or absent, and the O word is absent. The X, Y, or Z axis position, and I, J, and K word values are used to compute the corner offset. The control performs a G92.1 Position Set to the active setup offset values.

G75 X10 Y12 Z-10 I10.2 J12.2 K-10.2 ; Find corner and update current active multiple setup offset

Setting Selected Setup Offsets

To set the coordinates for a setup other than the active setup, the G77 block must contain the I, J, and K words and specify the number of the setup by using the O word. The X, Y, or Z axis coordinate of the setup specified by the O word is changed by setting the measured corner location to the I, J, or K value respectively.

G75 X10 Y12 Z-10 I10 J12 K-10 O2; Find corner and update multiple setup offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set the X, Y, or Z axis component of one of these offsets, the H word specifies the type of offset, and the O word specifies which offset, and the offset is computed from the measured corner position and the I, J, and K word value. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G75 X10 Y12 Z-10 I10 J12 K-10 H1 ; Set the current active pallet coordinate offset values

Setting a Fixture Offset

G75 X10 Y12 Z-10 I10 J12 K-10 H2 O3 ; Set Fixture offset 3 coordinate offset values

Setting a Programmable Coordinate Offset

G75 X10 Y12 Z-10 I10 J12 K-10 H3 O5 ; Set Programmable coordinate offset 5 values

Tolerances

The U and V word specify the upper tolerance limit and the tolerance respectively.

If a tolerance error occurs because the position error exceeds the V word value, system register [$OUT_OF_TOL] is set true (non-zero).

If the error exceeds the U word value, system register [$TOL_EXCEEDED] is set true

If the measured X and Y angles exceed the B word value system register [$ANGLE_ERROR] is set true.

The E word determines what action is taken as follows:

If the E word is zero or absent, an alarm is reported and the NC program halts.

If the E word is 1, no alarm is reported and NC program execution continues with the next block. When the E word value is 1, system register [$PROBE_HIT] is tested to determine whether or not the cycle completed its measurement. Also, the NC program test [$OUT_OF_TOL], [$TOL_EXCEEDED], and [$ANGLE_ERROR] to determine whether an error occurred.

G75 Programming Considerations

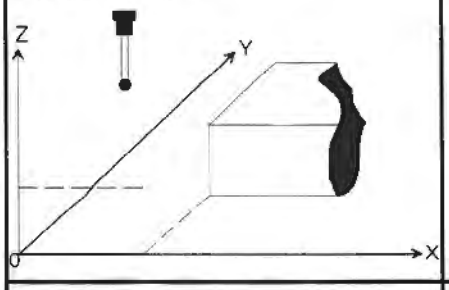
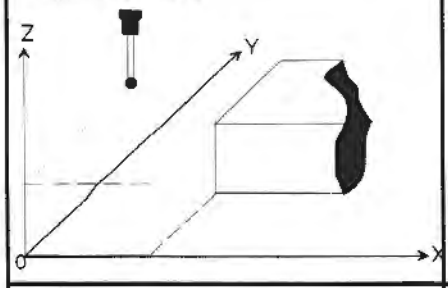
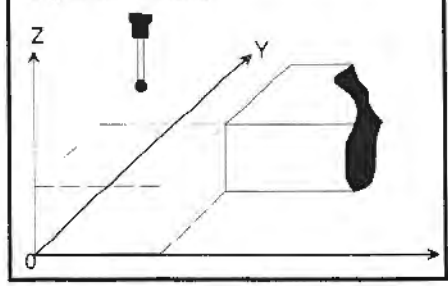
- Prior to executing the G75 cycle, the NC program must position the probe near the inside corner to be located.
- For each axis programmed in the G75 block the probe moves from the current position toward the programmed endpoint in X, Y, or Z at the cycle parameter *Probe Approach Feedrate.*
- If any axis is omitted X, Y, or Z, no motion occurs along that axis.
- If one axis is omitted, the corner is measured in the other two axes. In this case, the omitted axis must be positioned so that the probe stylus will hit the surfaces making up the corner to be measured.
- If the D word is present, it specifies the allowable overtravel beyond the programmed X, Y, or Z dimension. If the D word is absent, the allowable overtravel is the same as the distance from the initial position to the programmed X, Y, or Z dimension.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height.*
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate.* When the probe hits the part, it retracts by cycle parameter *Probe Gage Height,* then executes a second approach span limited to twice the *Probe Gage Height* at cycle parameter *Probe Measurement Feedrate* for the actual data acquisition.

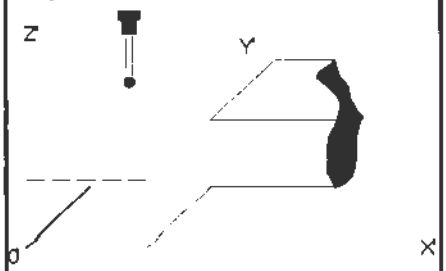
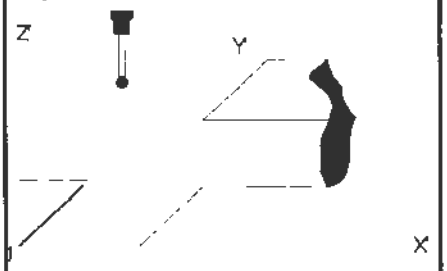
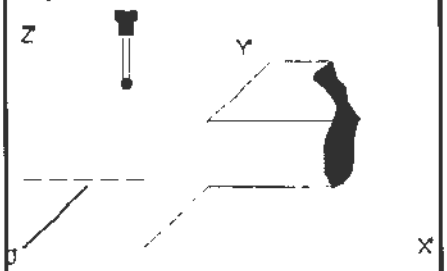
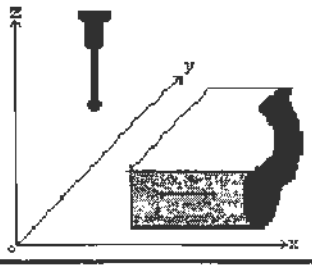
G76 Probe to Locate External Corner

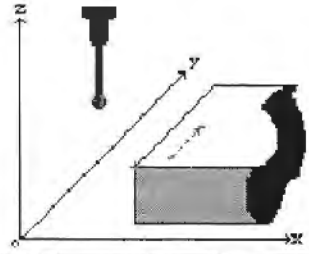
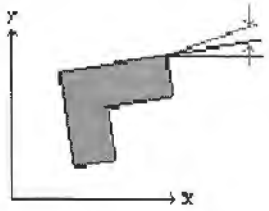
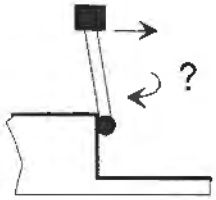
This cycle is used to locate the X, Y, and Z dimensions of an outside corner whose sides are nearly parallel to the X, Y, and Z axes. For true corner location G76 provides a B word for angle measurement of the X and Y axis surface. Once measurement is made, the following system register store the data.

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)] [$PRB_PART_LOC(Z)]	records the true corner location of the axis used to take measurement
[$PRB_POS_MC(X,Y, Z)] [$PRB_POS_PC(X,Y, Z)]	records X, Y, and Z machine coordinates, and program coordinate surface locations of the last probe hit when detected
[$PRB_X_ANGLE]	Angle of the X axis surface located
[$PRB_Y_ANGLE]	Angle of the Y axis surface located
[$PRB_INCL_ANG]	If both X and Y are included in the G76 block, and the optional iucremental move (G51.3, G51.4, or G51.5) to a second measurement position is specified, the included angle value is used.

G76 Probe to Locate External Corner Program Words

Word Description	Comments
Position X Word 	Specifies nominal external location of the corner in X axis.
Position Y Word 	Specifies nominal external location of the corner in Y axis.
Position Z Word 	Specifies nominal external location of the corner in Z axis.

G76 Probe to Locate External Corner Program Words (continued)	
Word Description	**Comments**
Optional I Word Reset Coordinate 	Optional non-modal X axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinates on the following pages
Optional J Word Reset Coordinate 	Optional non-modal Y axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinateson the following pages
Optional K Word Reset Coordinate 	Optional non-modal Z axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinates on the following pages
D Word Overtravel Distance	Non-modal probe over travel distance allowed beyond the programmed X or Y dimension. If D is absent, no over travel beyond the programmed end point is allowed.
P Word Incremental Distance (Opt.) 	Non-modal signed X axis incremental distance to the second measurement location. If P is present, X and Y axis must also be programmed. When used, the computed angle or the corner is stored in system register [$PRB_X_ANGLE] If the P word is absent only a single measurement is made in the Y axis.

G76 Probe to Locate External Corner Program Words (continued)	
Word Description	**Comments**
R Word Incremental Distance (Opt.) 	Non-modal signed Y axis incremental distance to the second measurement location. If R is present, X and Y axis mnst also be programmed. When used, the computed angle or the corner is stored in system register [$PRB_X_ANGLE] If the R word is absent only a single measurement is made in the X axis.
B Word Angle Tolerance (Opt.) 	Is the non-modal angle tolerance for the measured angle of the X and Y surface. If either system registers [$PRB_X_ANGLE] or [$PRB_Y_ANGLE] is outside to ± the B word, system register [$ANGLE_ERROR] is set true. Note: the B word value is equal to half of the total angle tolerance.
O Word defines offset number to be set.	Optional with H = 0 Required with H = 2 or 3 ignored otherwise
H Word Type of offset to be set based on I, J, K words. Actions of this word are effected by the O word.	0 or not programmed: Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset Refer to **What Offsets Can Be Adjusted** on the following pages
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit If the Q word value is 1, specifying a single hit cycle, the measurements are made during the initial move toward the part at the *Probe Approach Feedrate* cycle parameter. If the Q word is absent or zero, specifying a double hit cycle, the initial move toward the part is made at the *Probe Approach Feedrate* cycle parameter. When the probe hits the part, ~~the~~ it retracts by the *Probe Gage Height* cycle parameter, then executes a second approach span limited to twice the *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter for the actual data acquisition.

G76 Probe to Locate External Corner Program Words (continued)	
Word Description	**Comments**
E Word	E0 or not programmed abort and report an alarm. E1 Set system register[$PROBE_HIT] false and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tolerance on the following pages
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measnred error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.

G76 External Corner Cycle Action

Note: Standoff distance is the distance between the initial probe stylus position and the nominal corner location X, Y, and Z axis programmed dimensions.

Measure X Coordinate Of The Corner

1. Move Y and Z axis simultaneously toward the corner by twice the YZ standoff distance.
2. Move X axis toward the corner and measure the surface location using one or two hits specified by the Q word.
3. Report hit status based on the E word.
4. If R word is programmed, advance along the Y axis by the signed R increment, then make a second measurement in X axis by moving toward the programmed X value and record the probe hit position.
5. Return X axis, then Y and Z axis together to the initial position.
6. Report hit status based on the E word.
7. Perform any required offsets

Measure Y Coordinate Of The Corner

1. Move X and Z axis simultaneously toward the corner by twice the XZ standoff distance.
2. Move Y axis toward the corner and measure the surface location using one or two hits specified by the Q word.
3. Report hit status based on the E word.
4. If P word is programmed, advance along the X axis by the signed P increment, then make a second measurement in Y axis by moving toward the programmed Y value and record the probe hit position.
5. Return Y axis, then X and Z axis together to the initial position.
6. Report hit status based on the E word.
7. Perform any required offsets

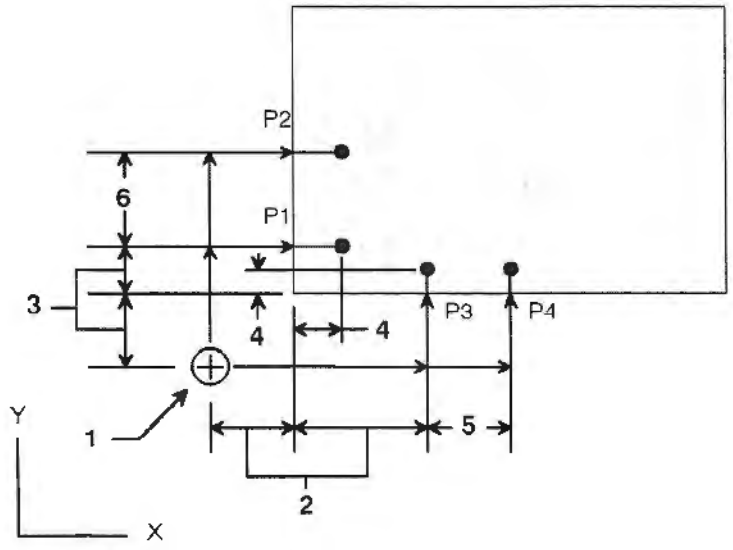
Measure Z Coordinate Of The Corner

1. Move X and Y axis simultaneously toward the corner by twice the XY standoff distance.
2. Move Z axis toward the corner and measure the surface location using one or two hits specified by the Q word.
3. Return Z axis, then X and Y axis together to the initial position.
4. Report hit status based on the E word.
5. Perform any required offsets

G76 With Sides Parallel To X and Y axis

Drawing Reference Number	Definition
1	Probe Start Point
2	X axis stand off distance
3	Y axis stand off distance
4	Overtravel distance (**D word**)
5	Incremental distance to the second measurement location (**P word**)
6	Incremental distance to the second measurement location (**R word**)

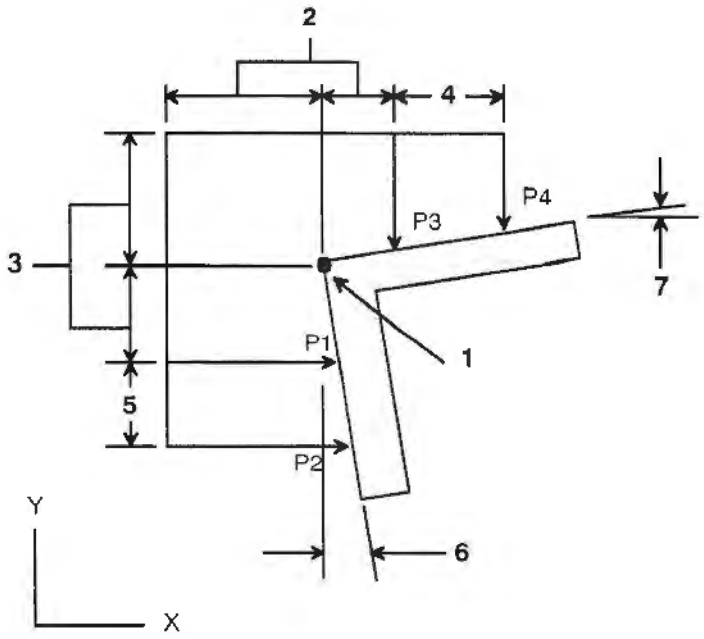
First measurement is made in X direction position 1 (P1)
Second measurement is made in X direction if R word is present position 2 (P2)
Third measurement is made in Y direction position 3 (P3)
Fourth measurement is made in Y direction if P word is present position 4 (P4)



G76 With Sides Not Parallel To X and Y axis

Drawing Reference Number	Definition
1	Nominal Corner location (programmed X, Y, Z)
2	X axis stand off distance
3	Y axis stand off distance
4	Incremental distance to the second measurement location (**P word**)
5	Incremental distance to the second measurement location (**R word**)
6	Represents Y axis probe angle [$PRB_Y_ANGLE]
7	Represents X axis probe angle [$PRB_X_ANGLE]

First measurement is made in X direction position 1 (P1)
Second measurement is made in X direction if R word is present position 2 (P2)
Third measurement is made in Y direction position 3 (P3)
Fourth measurement is made in Y direction if P word is present position 4 (P4)



What Offsets Can Be Adjusted

Following a successful corner measurement, the measured position of the corner can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, at least one of the I, J, or K words corresponding to the axis or axes (X, Y, or Z that was measured) must be specified. The following system registers are used to compute offset data:

[$X_POS_ERROR] [$Y_POS_ERROR] [$Z_POS_ERROR]	The difference between the I, J, and K word value and the measured position if both X and I are programmed, if both Y and J are programmed, or if both Z and K are programmed
[$TRUE_POS_ERR]	is the true position error vector value of the corner located

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X, Y, or Z axis can be changed when the H word is 0 or absent, and the O word is absent. The X, Y, or Z axis position, and I, J, and K word values are used to compute the corner offset. The control performs a G92.1 Position Set to the active setup offset values.

G76 X10 Y12 Z-10 I10.2 J12.2 K-10.2 ; Find corner and update cnrrent active multiple setup offset

Setting Selected Setup Offsets

To set the coordinates for a setup other than the active setup, the G77 block must contain the I, J, and K words and specify the number of the setup by using the O word. The X, Y, or Z axis coordinate of the setup specified by the O word is changed by setting the measured corner location to the I, J, or K value respectively.

G76 X10 Y12 Z-10 I10 J12 K-10 O2; Find corner and update multiple setup 2 offsets

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set the X, Y, or Z axis component of one of these offsets, the H word specifies the type of offset, and the O word specifies which offset, and the offset is computed from the measured corner position and the I, J, and K word value. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G76 X10 Y12 Z-10 I10 J12 K-10 H1 ; Set the current active pallet coordinate values

Setting a Fixture Offset

G76 X10 Y12 Z-10 I10 J12 K-10 H2 O3 ; Set Fixture offset 3 coordinate values

Setting a Programmable Coordinate Offset

G76 X10 Y12 Z-10 I10 J12 K-10 H3 O5 ; Set Programmable coordinate offset 5 values

Tolerances

The U and V word specify the upper tolerance limit and the tolerance respectively.

If a tolerance error occurs because the position error exceeds the V word value, system register [$OUT_OF_TOL] is set true (non-zero).

If the error exceeds the U word value, system register [$TOL_EXCEEDED] is set true

If the measured X and Y angles exceed the B word value system register [$ANGLE_ERROR] is set true.

The E word determines what action is taken as follows:

If the E word is zero or absent, an alarm is reported and the NC program halts.

If the E word is 1, no alarm is reported and NC program execution continues with the next block. When the E word value is 1, system register [$PROBE_HIT] is tested to determine whether or not the cycle completed its measurement. Also, the NC program test [$OUT_OF_TOL], [$TOL_EXCEEDED], and [$ANGLE_ERROR] to determine whether an error occurred.

G76 Programming Considerations

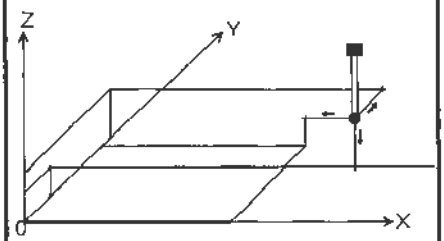
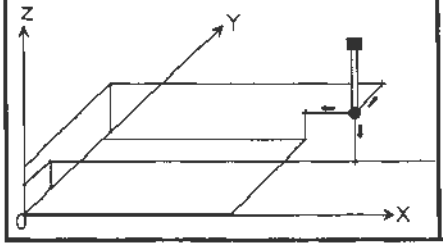
- Prior to executing the G76 cycle, the NC program must position the probe near the outside corner to be located.
- For each axis programmed in the G76 block the probe moves from the current position toward the programmed endpoint in X, Y, or Z at the cycle parameter *Probe Approach Feedrate*.
- If any axis is omitted X, Y, or Z, no motion occurs along that axis.
- If one axis is omitted, the corner is measured in the other two axes. In this case, the omitted axis must be positioned so that the probe stylus will hit the surfaces making up the corner to be measured.
- If the D word is present, it specifies the allowable overtravel beyond the programmed X, Y, or Z dimension. If the D word is absent, the allowable overtravel is the same as the distance from the initial position to the programmed X, Y, or Z dimension.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second approach span limited to twice the *Probe Gage Height* at cycle parameter *Probe Measurement Feedrate* for the actual data acquisition.

G77 Probe to Locate Surface

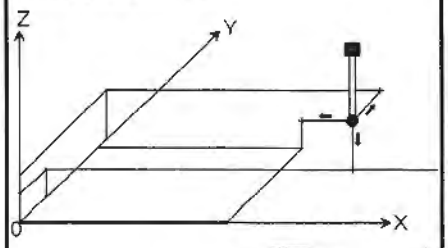
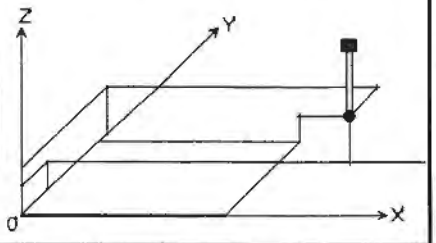
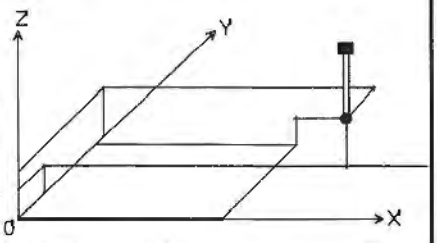
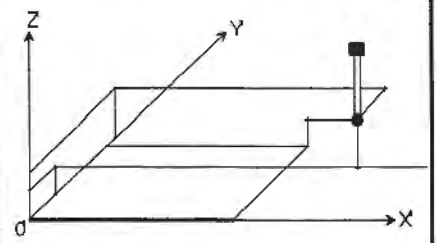
G77 Locate Surface cycle probes for a surface in a single machine axis. The measured true location of the surface in the selected axis is saved, The location can be treated as a reference surface position and used to set the setup offset for that axis by having the G77 cycle perform an automatic G92.1 Position Set, or a coordinate offset can be computed from the measured location. Alternatively, the measured position can be treated as a size, and used to adjust the length or diameter offset for the tool used to machine the surface. Once measurement is made, data is collected and stored in the following system registers.

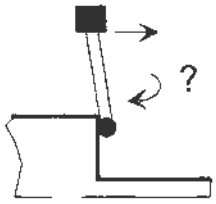
[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_POS_MC(X,Y, Z)] [$PRB_POS_PC(X,Y, Z)]	records X, Y, and Z machine coordinates, and program coordinate surface locations when a hit is detected
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)] [$PRB_PART_LOC(Z)]	If the X axis is programmed, the X axis value in the probe location variables is updated, if Y is programmed, the Y axis value is updated, and if Z is programmed, the Z value is updated. The other axis values are not changed.
[$X_POS_ERROR] [$Y_POS_ERROR] [$Z_POS_ERROR]	The difference between the I, J, or K word value and the measured position is stored in system registers [$X_POS_ERROR] if both X and I are programmed, [$Y_POS_ERROR] if both Y and J are programmed or [$Z_POS_ERROR] if both Z and K are programmed.
[$TRUE_POS_ERR]	Is the recorded true position error value
[$SIZE_ERROR]	Is the recorded size error value

G77 Probe to Locate Surface Program Words

Word Description	Comments
Position X Word 	Specifies nominal surface location in the X axis.
Position Y Word 	Specifies nominal surface location in the Y axis.

G77 Probe to Locate Surface
Program Words (continued)

Word Description	Comments
Position Z Word 	Specifies nominal surface location in the Z axis.
Optional I Word Reset Coordinate 	Optional non-modal X axis program coordinates to assign to the measured corner location or to be nsed for computing offsets See Changing Coordinates on the following pages for additional information
Optional J Word Reset Coordinate 	Optional non-modal Y axis program coordinates to assign to the measured corner location or to be used for compnting offsets See Changing Coordinates on the following pages for additional information
Optional K Word Reset Coordinate 	Optional non-modal X axis program coordinates to assign to the measured corner location or to be used for computing offsets See Changing Coordinates on the following pages for additional information
D Word Overtravel Distance	Non-modal probe overtravel distance allowed beyond the nominal snrface location specified by the programmed X, Y, or Z axis dimension. If the D word is absent, no overtravel beyond the programmed dimension is permitted.
H Word Type of offset to be set based on I, J, K words. Actions of this word are effected by the O word.	0 or not programmed: Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset 5 = Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.

G77 Probe to Locate Surface Program Words (continued)	
Word Description	**Comments**
O Word defines offset number to be set.	Optional with H = 0 Required with H = 2 or 3 ignored otherwise
Probe Process Q Word  	Q0 Double Hit Q1 Single Hit If a probe hit occurs during the approach move, system register [$PROBE_HIT]is set *true* and the location of the surface is measured and stored in the appropriate elements of system registers [$PRB_POS_MC], [$PRB_POS_PC], and [$PRB_PART_LOC]. The measurement method is specified by the Q word value. If the Q word value is 1, specifying a single hit cycle, the measurements are made during the initial move toward the part at the *Probe Approach Feedrate* cycle parameter. If the Q word is absent or zero, specifying a double hit cycle, the initial move toward the part is made at the *Probe Approach Feedrate* cycle parameter. When the probe hits the part, it retracts by the *Probe Gage Height* cycle parameter, then executes a second approach span limited to twice the *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameters for the actual data acquisition. Following the measurement, the probe retracts to *Probe Gage Height* above the measured surface.
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.

G77 Probe to Locate Surface Program Words (continued)	
Word Description	**Comments**
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$PROBE_HIT] false or set [$OUT_OF_TOL] or [$TOL_EXCEEDED] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages for additional information

G77 Cycle Action

G77 moves the probe from the current position toward the programmed endpoint in X, Y, or Z axis at probe cycle parameter *Probe Approach Feedrate*.

The approach move continues until either the probe contacts a surface or reaches the programmed dimension. If the D word is present, it specifies the allowable overtravel beyond the programmed X, Y, or Z dimension.

What Offsets Can I Adjust

Following the probe hit, the measured surface location can be used to set either a coordinate offset or the length or diameter offset of the tool that was used to machine the surface. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I, J, and K word corresponding to the axis word must be specified.

The computed error is stored in the following system registers error data.

[$X_POS_ERROR] [$Y_POS_ERROR] [$Z_POS_ERROR]	The difference between the I, J, and K word value and the measured position if both X and I are programmed, if both Y and J are programmed, or if both Z and K are programmed
[$TRUE_POS_ERR]	is the true position error of the surface
[$SIZE_ERROR]	this value is the error between the nominal size and measured size used to adjust tool offsets

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. To set the active setup coordinate system offset, the G77 block must contain the I, J, or K word. The X, Y, or Z axis coordinate offset of the active setup is changed when the H word is 0 or absent and the O word is absent. The control performs a G92.1 Position Set to set the coordinates of the measured corner location to the coordinate specified by the I, J, and K values.

G77 X10 Y12 Z-10 I10.2 J12.2 K-10.2 ; Find surface and update current active multiple setup offset

Setting Selected Setup Offsets

To set the coordinates for a setup other than the active setup, the G77 block must contain the I, J, and K words and specify the number of the setup by using the O word. The X, Y, or Z axis coordinate of the setup specified by the O word is changed by setting the measured corner location to the I, J, or K value respectively.

G77 X10 Y12 Z-10 I10 J12 K-10 O2; Find surface and update multiple setup offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values the H word specifies the type of offset, and the O word specifies which offset record. The X, Y, or Z axis position, and I, J or K word values are used to compute offsets. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G77 X10 Y12 Z-10 I10 J12 K-10 H1 ; Set the current active pallet coordinate offset values

Setting a Fixture Offset

G77 X10 Y12 Z-10 I10 J12 K-10 H2 O3 ; Set Fixture offset 3 coordinate values

G77 X10 Y12 Z-10 I10 J12 K-10 H3 O5 ; Set Programmable coordinate offset 5 values

Changing Programmable Tool Offset Table Diameter value

G77 X10 Y12 I10.2 J12.2 H5 O6

Since X and Y axes are used for measurement and the O word is present with no T word the programmable tool offset table record 6 diameter is adjusted by system register [$SIZE_ERROR]

Record number 6 (tool diameter) is changed by adding system register value [$SIZE_ERROR].

Changing Tool Table Diameter Offset value

G77 X10 Y12 I10.2 J12.2 H5 T6

Since X and Y axes are used for measurement and no O word is present, the T word defines tool offset table record 6 diameter offset is adjusted by system register [$SIZE_ERROR]

Record number 6 (tool diameter) is changed by system register value [$SIZE_ERROR].

Changing Programmable Tool Offset Table Length value

G77 Z10.2 K10 H5 O6

Since the Z axis is used for measurement and the O word is present with no T word, the programmable tool offset table record 6 length is adjusted by system register [$SIZE_ERROR]

Record number 6 (tool length) is changed by system register value [$SIZE_ERROR].

Changing Tool Table Length Offset value

G77 Z10 K10 H5 T6

Since Z axis is used for measurement and no O word is present, the T word defines tool offset table record 6 length is adjusted by system register [$SIZE_ERROR]

Record number 6 (tool diameter) is changed by adding system register value [$SIZE_ERROR].

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

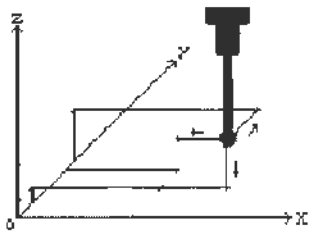
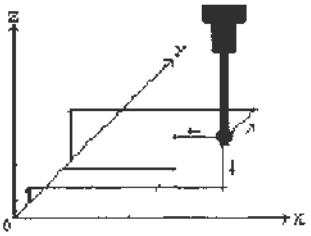
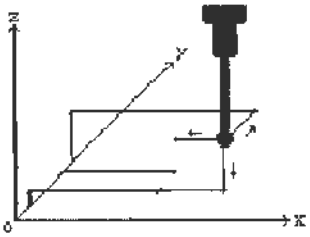
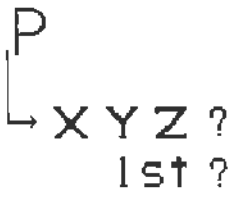
If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

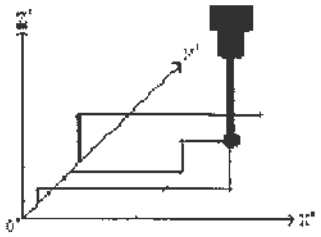
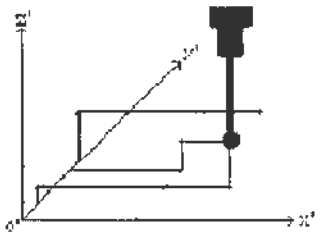
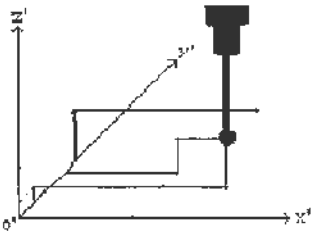
G77 Programming Considerations

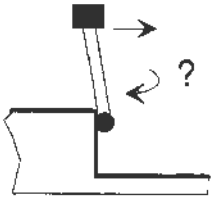
- Prior to executing the G77 cycle, the NC program must position the probe near the surface to be located.
- For each axis programmed in the G77 block the probe moves from the current position toward the programmed endpoint in X, Y, or Z at the cycle parameter *Probe Approach Feedrate.*
- If the D word is present, it specifies the allowable overtravel beyond the programmed X, Y, or Z dimension. If the D word is absent, the allowable overtravel is the same as the distance from the initial position to the programmed X, Y, or Z dimension.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height.*
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate.* When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second approach span limited to twice the *Probe Gage Height* at cycle parameter *Probe Measurement Feedrate* for the actual data acquisition.

G77.1 Stock Allowance (Optional)

G77.1 Stock Allowance cycle probes for a surface in a single machine axis (X, Y, or Z). Multiple executions of G77.1 blocks (based on the P word) can be used to sample and record stock variation at several part locations. Also, this cycle can set a tool length offset or coordinate system value above the surface of the part.

G77.1 Stock Allowance Program Words	
Word Description	**Comments**
X Word Position 	Modal axis command point in the X axis. The axis in which probe motion occurs is determined by the P word value.
Y Word Position 	Modal axis command point in the Y axis. The axis in which probe motion occnrs is determined by the P word value.
Z Word Position 	Modal axis command point in the Z axis. The axis in which probe motion occurs is determined by the P word value.
P Word Stock Cycle Command 	Non-modal cycle command for probe motion: 1: Probe in X axis 2: Probe in Y axis 3: Probe in Z axis 4: Probe in X axis; first operation on this surface 5: Probe in Y axis; first operation on this surface 6: Probe in Z axis; first operation on this surface See **Surface Measurement Overview** on the following pages for additional information

G77.1 Stock Allowance Program Words (continued)	
Word Description	**Comments**
D Word Overtravel Distance	Non-modal probe overtravel distance allowed beyond the nominal surface location specified by the programmed X, Y, or Z axis dimension. If the D word is absent, no overtravel beyond the programmed dimension is permitted.
I Word Probe Location (Optional) 	A single non-modal axis parameter which specifies the expected location of the axis being probed. The I word must agree with the P word command, that is, I goes with P1. The I word should only be specified on the last of a series of G77.1 cycles for a given surface. See **What Offsets Can Be Adjusted** on the following pages for additional information
J Word Probe Location (Optional) 	A single non-modal axis parameter which specifies the expected location of the axis being probed. The J word must agree with the P word command, that is, J goes with P2. The J word should only be specified on the last of a series of G77.1 cycles for a given surface. See **What Offsets Can Be Adjusted** on the following pages for additional information
K Word Probe Location (Optional) 	A single non-modal axis parameter which specifies the expected location of the axis being probed. The K word must agree with the P word command, that is, K goes with P3. The K word should only be specified on the last of a series of G77.1 cycles for a given surface. See **What Offsets Can Be Adjusted** on the following pages for additional information
H Word Type of offset to be set based on I, J, K words. Actions of this word are affected by the O word.	0 or not programmed: Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information
O Word defines offset number to be set.	Optional with H = 0 Required with H = 2 3 or 5 ignored otherwise

G77.1 Stock Allowance Program Words (continued)	
Word Description	**Comments**
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit If a probe hit occurs during the approach move, system register [$PROBE_HIT]is set *true* and the location of the surface is measured and stored in the appropriate elements of system registers [$PRB_POS_MC], [$PRB_POS_PC], and [$PRB_PART_LOC]. If the Q word value is 1, specifying a single hit cycle, the measurements are made during the initial move toward the part at the *Probe Approach Feedrate* cycle parameter. If the Q word is absent or zero, specifying a double hit cycle, the initial move toward the part is made at the *Probe Approach Feedrate* cycle parameter. When the probe hits the part, it retracts by the *Probe Gage Height* cycle parameter, then executes a second approach span limited to twice the *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameters for the actual data acquisition. Following the measurement, the probe retracts to *Probe Gage Height* above the measured surface.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured surface value is more than the U word tolerance above the nominal surface: 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tolerance on the following pages for additional information

Surface Measurement Overview

The P word is used to define the measurement axis. The G77.1 block moves the probe from the current position to the programmed endpoint in two axes perpendicular to the measurement axis. P word selections are as follows:

Positioning Axis	Measurement Axis
P1 or P4 move YZ axes	measure in X axis
P2 or P5 move XZ axes	measure in Y axis
P3 or P6 move XY axes	measure in Z axis

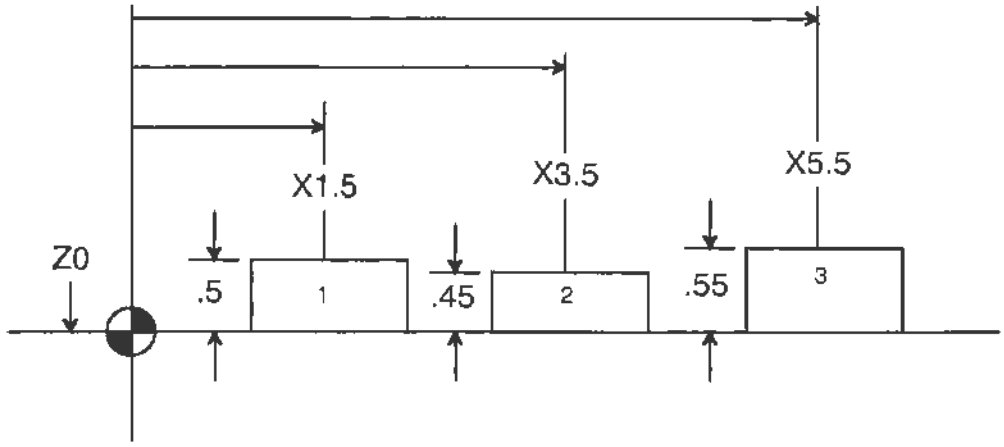
Once measurement is made, the following system register interact with the P word selection

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_POS_MC]	The probe machine position measured surface location for all axes
[$PRB_POS_PC]	The probe program coordinates position measured surface location for all axes
[$PRB_PART_LOC(X)]	This value is the measured axis part location when P1 or P4 word is used.
[$PRB_PART_LOC(Y)]	This value is the measured axis part location when P2 or P5 word is used.
[$PRB_PART_LOC(Z)]	This value is the measured axis part location when P3 or P6 word is used.
[$MIN_STOCK]	Sets minimum stock for first series measurements using P4, P5 or P6 word
[$MAX_STOCK]	Sets maximum stock for first series measurements using P4, P5 or P6 word
[$VARIATION]	Is the difference between [$MIN_STOCK] and [$MAX_STOCK], the lowest and highest measured surface.

G77.1 Surface Measurement Sample Program

The following program segment is intended to illustrate multiple G77.1 surface measurements in Z axis. The table below illustrates system register interaction during the measurements process.

N100 G77.1 X1.5 Y12 Z0 D.05 P6 : Move to position 1 measure in Z axis set minimum and maximum values
N110 Z1
N120 G77.1 X3.5 Z0 D.05 P3 ; Move to position 2 set variation value
N130 Z1
N140 G77.1 X5.5 Z0 D.05 P3 ; Move to position 3 set variation value



	[$MAX_STOCK]	[$MIN_STOCK]	[$VARIATION]	[$PRB_POS_MC]	[$PRB_POS_PC]	[$PRB_PART_LOC(Z)]
1	.500	.500	.000	Set all axes	Set all axes	10
2	.500	.450	.05	Set all axes	Set all axes	9
3	.550	.450	.100	Set all axes	Set all axes	8

What Offsets Can Be Adjusted

Following a successful measurement, the measured surface location can be used to set offsets based on the H word programmed. The I, J, and K words determine the measurement axis. To update an offset, I, J, or K words must be programmed in the G77.1 block. The following system register contain the data:

[$X_POS_ERROR]	If both X and I are programmed
[$Y_POS_ERROR]	If both Y and J are programmed
[$Z_POS_ERROR]	If both Z and K are programmed
[$TRUE_POS_ERR]	is the error distance from the nominal value to the true surface

Setting Active Setup Offsets

To set the coordinate system offset, the last G77.1 block in the series must contain the I, J, or K word. The X, Y, or Z axis coordinate offset of the active setup is changed by the control performing a G92.1 Position Set at the minimum measured coordinate specified by the I, J, and K value. The active setup is changed when the H word is 0 or absent, and the O word is absent.

G77.1 P3 X10 Y12 Z-10 I102 J12.2 K-10.2 ; Find surface and update current active multiple setup offset

Setting Selected Setup Offsets

To set coordinates for a setup other than the active setup, the G77.1 block must contain the I, J, or K word and specify setup in the O word. The X, Y, or Z axis coordinate of the setup is specified by the O word is changed by setting the minimum measured position to the I, J, or K value respectively.

G77.1 P3 X10 Y12 Z-10 K-10 O2; Measure surface and update multiple setup offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set the X, Y, or Z axis component of one of these offsets, the H word specifies the type of offset, and the O word specifies which offset. The offset is computed from the minimum measured position and the I, J, and K word value.

Setting a Pallet Offset

G77.1 P3 X10 Y12 Z-10 K-10 H1 ; Set Z axis value for the current active pallet coordinates

Setting a Fixture Offset

G77.1 P3 X10 Y12 Z-10 I10 J12 K-10 H2 O3 ; Set X, Y, and Z axes values for Fixture offset 3

Setting a Programmable Coordinate Offset

G77.1 P3 X10 Y12 Z-10 I10 J12 K-10 H3 O5 ; Set X, Y, and Z axes values for Programmable coordinate offset record 5

Tolerances

The U word specifies the upper tolerance limit for the amount of stock present. If the I, J, and K word is not present (that is, if an offset is not being changed), the U word represents the maximum allowable value for the measured stock amount above the Programmed X, Y, or Z dimension. For example:

N010 G77.1 P3 X10 Y5 Z-10 K-10 D.1 U.2

This example specifies that a probe measurement is to be made at X = 10 and Y = 5, with an expected surface location of Z10.0. The surface can be as low as 9.9 (Z10 minus the allowable overtravel of D0.1), or as high as 10.2 (Z1 plus the upper tolerance limit of U0.2).

If I, J, or K word is present, U is the tolerance on the value of system variable [$VARIATION].

If the error exceeds the U word value, system register [$TOL_EXCEEDED] is set true

The E word determines what action is taken as follows:

If the E word is zero or absent, an alarm is reported and the NC program halts.

If the E word is 1, no alarm is reported and NC program execution continues with the next block. When the E word value is 1, system register [$PROBE_HIT] is tested to determine whether or not the cycle completed its measurement. Also, the NC program test [$TOL_EXCEEDED] to determine whether an error occurred.

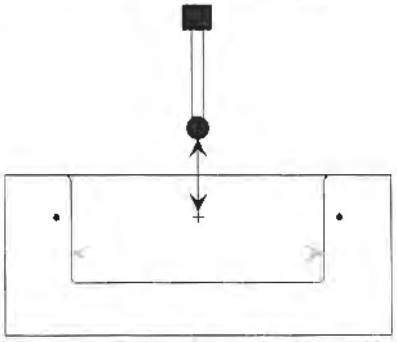
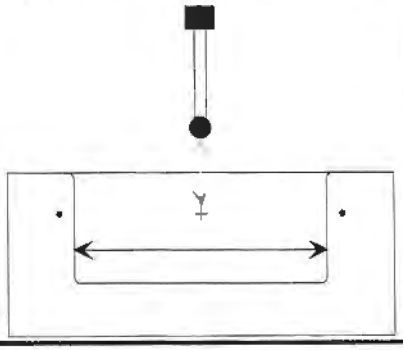
G77.1 Programming Considerations

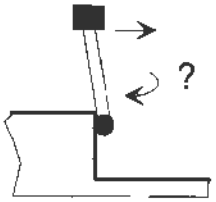
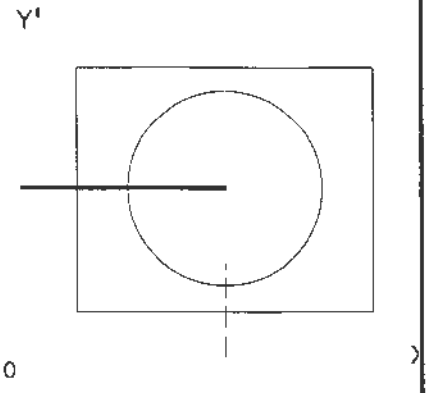
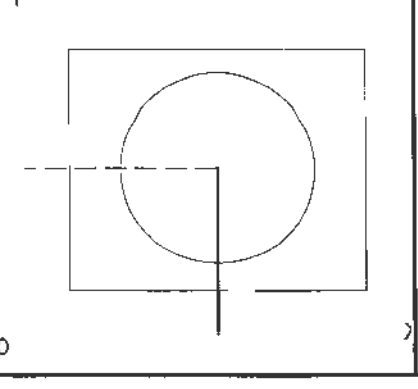
- Prior to executing the G77.1 cycle, the NC program must position the probe near the surface to be located.
- For each axis programmed in the G77.1 block the probe moves from the current position toward the programmed endpoint in X, Y, or Z at the cycle parameter *Probe Approach Feedrate*.
- If the D word is present, it specifies the allowable overtravel beyond the programmed X, Y, or Z dimension. If the D word is absent, the allowable overtravel is the same as the distance from the initial position to the programmed X, Y, or Z dimension.
- The U word value represents the maximum allowable value for the measured stock amount above the Programmed X, Y, or Z dimension.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second approach span limited to twice the *Probe Gage Height* at cycle parameter *Probe Measurement Feedrate* for the actual data acquisition.

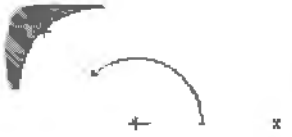
G78 Probe a Bore

To select bore measurement the G78 program block R word **must** equal 0 or be absent The cycle measurement that takes place is made along the *measurement line*, which is a line through the starting probe stylus position to the programmed XY endpoint. G78 can measure a bore either by using the X and Y axis process or by a 3-point measurement method, which requires a 360 degree calibrated probe. In either case, once measurement is made, data is collected and stored in the following system registers:

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)]	records center measurement coordinates
[$PRB_PART_LOC(Z)]	used to set the Z axis dimension at which the measurements were made.
[$PRB_X_DIA] [$PRB_Y_DIA]	XY axis diameter measurements
[$PRB_AVG_DIA]	measured diameter

G78 Probe a Bore Program Words	
Word Description	**Comments**
Incremental Distance Z Word 	Specifies nominal signed incremental distance to move from the initial location to the Z axis location at which the measurements are to be made. Z is normally negative since the probe normally is positioned above the workpiece.
Diameter P Word 	Specifies non-modal unsigned nominal diameter of the bore to be measured. All measurements moves are limited to half of the nominal bore diameter plus the overtravel distance (P/2 + D word) in length if the D word is specified, otherwise the moves are limited to the nominal diameter (P word) in length.

G78 Probe a Bore Program Words (continued)	
Word Description	**Comments**
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit The first move for a double hit cycle (the only move for single hit measurements) the probe retracts by *Probe Gage Height* cycle parameter and makes a second measurement move of twice *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter. The final position of the probe is on the measured centerline of the bore at the initial Z axis position. The values of system registers [$PROBE_HIT], [$PRB_POS_MC], and [$PRB_POS_PC] reflect the location of the last probe move completed in the cycle. If any probe move fails to hit the part, the cycle aborts, the probe returns to the start point, and system register [$PROBE_HIT] is set false.
R Word defines the cycle type	R word = 0 or absent = bore measurement where the center of the bore is free of obstructions.
Optional I Word Reset Coordinate to 	Specifies type of offset (based on H word) to assign to the measured X axis bore location or to be used for computing offsets.
Optional J Word Reset Coordinate to 	Specifies type of offset (based on H word) to assign to the measured Y axis bore location or to be used for computing offsets.

G78 Probe a Bore Program Words (continued)	
Word Description	**Comments**
A Word Angle  B Word Angle  C Word Angle 	These cycle modal words are sign angles measured counter clockwise from the +X axis at which the three measurements are made. Specifying the A, B, and C words cause the 3-point measurement to be made instead of the XY axis measurement. All three word angles must be programmed if any angle word is present. **Note:** Before 3-point measurement can be used the probe must be calibrated for 360 degrees. For additional information refer to **3-point measurement** on the following pages for additional informatioo
H Word Type of offset to be set based on I and J words. Actions of this word are effected by the O word.	0 or not programmed = Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset 5 = Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.
D Word Overtravel Distance	Specifies cycle modal probe overtravel distance allowed beyond the programmed motion (half the nominal diameter from the center). If the D word is absent, the full programmed diameter (P word) is allowed before a "no hit" condition is detected.
O Word defines offset number to be set	Optional with H = 0 Required with H = 2, 3 or 5 ignored otherwise
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The nou-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.

G78 Probe a Bore Program Words (continued)	
Word Description	**Comments**
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages for additional information.

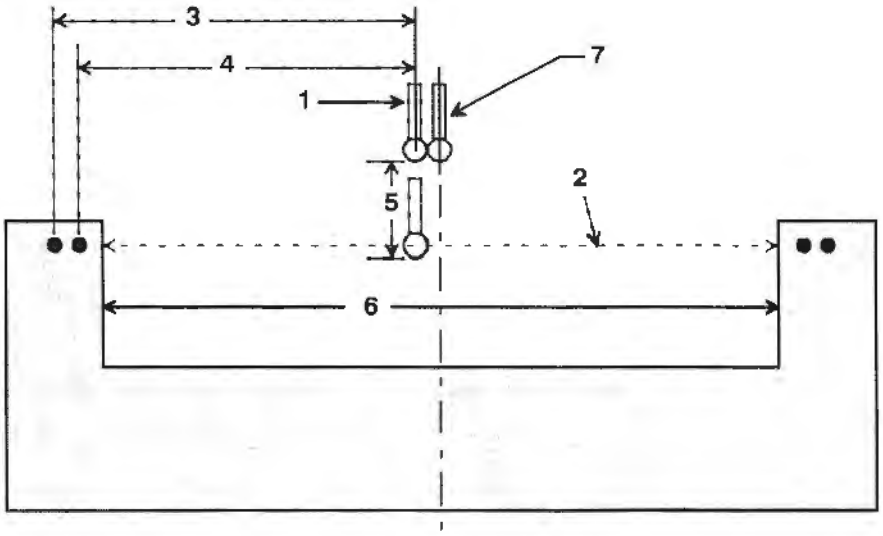
XY Axes Bore Measurement Cycle Action

Clear center bore measurements are determined by the R word equaling 0 or absent. To measure an open bore the following steps will occur:

1. Position the probe near the nominal center of the bore.
2. The probe advances along the Z axis by the incremental value specified in the Z word.
3. Probe in the + X direction for P/2 + D word (or P word if D word is absent).
4. Return to the start position.
5. Probe in the - X direction for P/2 + D word (or P word if D word is absent).
6. Move to a position midway between the locations found in steps 3 and 5.
7. Probe in the + Y direction for P/2 + D word (or P word if D word is absent).
8. Return to the starting Y axis position.
9. Probe in the -Y direction for P/2 + D word (or P word if D word is absent).
10. Move to a position midway between the locations found in steps 7 and 9.
11. Probe again in the + X direction for P/2 + D word (or P word if D word is absent).
12. Return to the X position from step 6.
13. Probe again in the - X direction for P/2 + D word (or P word if D word is absent).
14. Position to the computed center of the bore.
15. Retract Z axis to the starting Z axis position.
16. Perform any required coordinate offset or tool length offset adjustments.

XY Axes Bore Sample Drawing

Drawing Reference Number	Definition
1	Probe Start Point
2	Measurement Line
3	Probe Target Position with (D word) Absent
4	Probe Target Position with (D word) Present
5	Z Incremental (**Z word**)
6	Nominal Width (**P Word**)
7	Measured Center



3-Point Measurement

The 3-point measurement capability allows partial measurement of bores with cutouts located on the X or Y axis centerline. The A, B, and C angle words programmed in the G78 block command the probe to make three moves from the center of the bore toward the periphery of the bore. All measurements moves are limited to half of the nominal bore diameter plus the overtravel distance (P/2 + D word) in length if the D word is specified, otherwise the moves are limited to the nominal diameter (P word) in length.

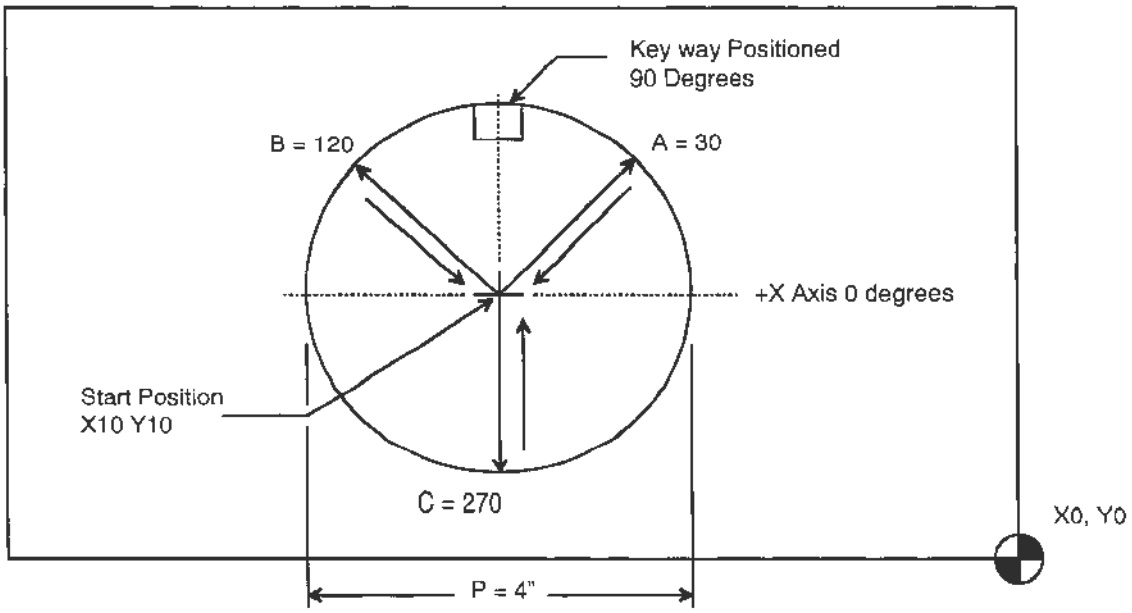
Note: Before 3-point measurements can be used the you must have a probe calibrated for 360 degree.

Basic Movements of 3-Point Measurement

1. Advance the probe along the Z axis by the incremental value specified in the Z word. Note that if the probe
2. Probe at A degree from the + X direction for P/2 + D (or P word if D word is absent).
3. Return to the start position.
4. Probe at B degree from the + X direction for P/2 + D (or P word if D word is absent).
5. Return to the start position.
6. Probe at C degree from the + X direction for P/2 + D (or P word if D word is absent).
7. Return to the start position.
8. Position the computed center of the bore.
9. Retract Z axis to the starting Position
10. Any coordinate offset or tool length offset adjustments are perform.

3-Point Measurement Sample Program

G78 X10 Y10 Z-12 A30 B120 C270 H2 O2 ; Take angled measurements update Fixture Offsets record 2



What Offsets Can Be Adjusted

Following a successful bore measurement, the measured true position of the bore center can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I and J words or 3-point measurement A, B and C words must be programmed.

G78 computes error between the nominal center of the bore (specified by the I and J words) and the measured center. This data is stored in the following system registers:

[$X_POS_ERROR] [$Y_POS_ERROR]	is the error in X and Y axis
[$TRUE_POS_ERR]	is the error distance from the nominal value to the true center
[$SIZE_ERROR]	if the optional P word nominal size is specified, this value is the error between the nominal size and measured size

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X and Y axis can be changed when the H word is 0 or absent, and the O word is absent. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10.2 J12.2 P4 ; Find center of 4 inch bore and update the current active multiple setup offset

Setting Selected Setup Offsets

To apply offsets to a setup other than the active setup, the H word is 0 or absent, and the O word value is used to specify the setup record number. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10 J12 O2 P2 ; Find center of 2 inch bore and update multiple setup record offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values, the H word specifies the type of offset, and the O word specifies which offset record. The X and Y axis position, and I and J word values or A, B, and C words values are used to compute centerline offset. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G78 X10 Y12 Z-10 I10 J12 H1 P3 ; Find center of 3 inch bore and update current active pallet values

Setting a Fixture Offset

G78 X10 Y12 Z-10 I10 J12 H2 O3 P2 ; Find center of 2 inch bore and update Fixture offset 3 values

Setting a Programmable Coordinate Offset

G78 X10 Y12 Z-10 I10 J12 H3 O5 P3 ; Find center of 3 inch bore and update Programmable coordinate offset 5 values

Changing Programmable Tool Offset Table Diameter value

Since X and Y axes are used for measurement, the Programmable Tool Offset table diameter value can be change by programming H5 with an O word and no T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G78 X10 Y12 Z-5 I10.2 J12.2 H5 O6 P4 ; change tool diameter record number 6 programmable tool table value.

Changing Tool Table Diameter Offset value

Since X and Y axes are used for measurement, the Tool Table Offset diameter value can be change by programming H5 with a T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 T6 P3 ; Change tool table diameter offset value record number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

G78 Bore Programming Considerations

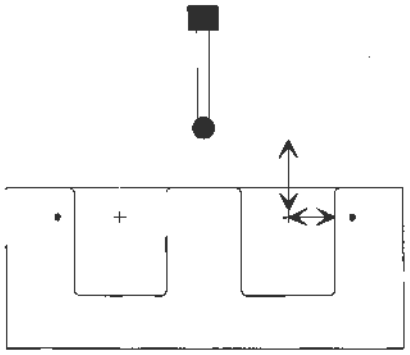
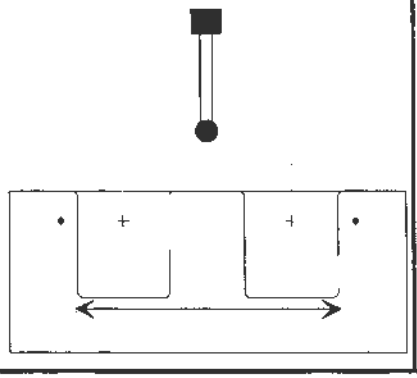
- Prior to executing the G78 cycle, the NC program must position the probe near the center of the bore to be measured.
- To measure a bore with no obstructions the R word **must** be 0 or absent.
- For each axis programmed in the G78 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- All of the measurement moves are limited to half of the nominal bore diameter plus the specified overtravel
(P/2 + D) in length if the D word is specified, otherwise the moves are limited to the nominal diameter (P word) in length.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.
- When using 3-point measurement the probe must be calibrated for 360 degrees.
- When using 3-point measurement the A, B, and C words must all be programmed.

G78 Probe Bore with Obstacle

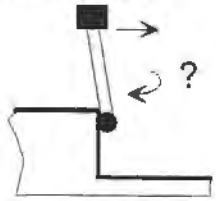
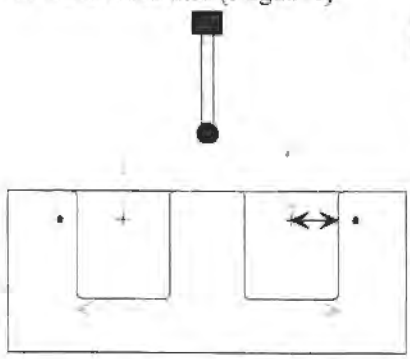
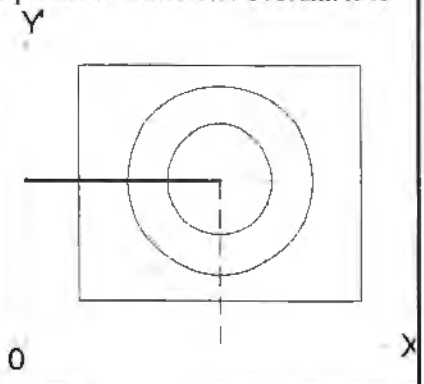
To use this measurement the G78 program block **must** contain an R word with a negative value R < 0. The cycle measurement that takes place is made along the *measurement line*, which is a line through the starting probe stylus position to the programmed XY endpoint. G78 can measure a bore either by using the X and Y axis process or by a 3-point measurement method, which requires a 360 degree calibrated probe. In either case, once measurement is made, data is collected and stored in the following system registers:

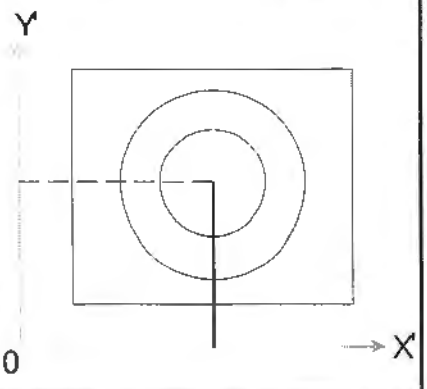
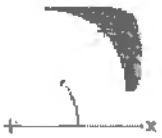
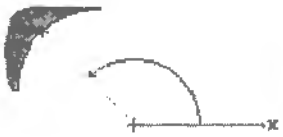
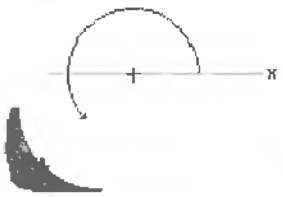
[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)]	records center measurement coordinates
[$PRB_PART_LOC(Z)]	used to set the Z axis dimension at which the measurements were made.
[$PRB_X_DIA] [$PRB_Y_DIA]	XY axis diameter measurements
[$PRB_AVG_DIA]	measured diameter

G78 Probe a Bore with Obstacle Program Words

Word Description	Comments
Incremental Distance Z Word 	Specifies nominal signed incremental distance to move from the initial location to the Z axis location at which the measurements are to be made. Z is normally negative since the probe normally is positioned above the workpiece.
Diameter P Word 	Specifies unsigned nominal diameter of the bores or bosses to be measured All of the measnrements moves are limited to half of the nominal bore diameter plus the radial clearance R word distance (P/2 + R word).

G78 Probe a Bore with Obstacle Program Words (continued)

Word Description	Comments
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit The first move for a double hit cycle (the only move for single hit measurements) the probe retracts by *Probe Gage Height* cycle parameter and makes a second measurement move of twice *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter. The final position of the probe is on the measured centerline of the bore or boss at the initial Z axis position. The values of system registers [$PROBE_HIT], [$PRB_POS_MC], and [$PRB_POS_PC] reflect the location of the last probe move completed in the cycle. If any probe move fails to hit the part, the cycle aborts, the probe returns to the start point, and system register [$PROBE_HIT] is set false.
R Word Clearance (Negative) 	R < 0 : bore measurement with obstructed center. The R word is the clearance distance inside of the nominal diameter (P word) of the bore available for probe motion. The R word **must** be a negative value.
Optional I Word Reset Coordinate to 	Specifies type of offset (based on H word) to assign to the measured X axis bore location or to be used for computing offsets.

G78 Probe a Bore with Obstacle Program Words (continued)	
Word Description	**Comments**
Optional J Word Reset Coordinate to 	Specifies type of offset (based on H word) to assign to the measured Y axis bore location or to be used for computing offsets.
A Word Angle  B Word Angle  C Word Angle 	These cycle modal words sign angles measured counter clockwise from the +X axis at which the three measurements are made. Specifying the A, B, and C words cause the 3-point measurement to be made instead of the XY axis measurement. All three word angles must be programmed if any angle word is present. **Note:** Before 3-point measurement can be used the probe must be calibrated for 360 degrees. For additional information refer to **3-point measurement** at the end of this chapter
H Word Type of offset to be set based on I and J words. Actions of this word are effected by the O word.	0 or not programmed = Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset 5 = Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages
D Word Overtravel Distance	Specifies cycle modal probe overtravel distauce allowed beyond the programmed motion (half the nominal diameter from the center). If the D word is absent, the full programmed diameter (P word) is allowed before a "no hit" condition is detected.
O Word defines offset number to bc set	Optional with H = 0 Required with H = 2, 3 or 5 ignored otherwise

G78 Probe a Bore with Obstacle Program Words (continued)	
Word Description	**Comments**
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The non-modal U word specifies the npper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.

G78 Probe a Bore with Obstacle Program Words (continued)	
Word Description	**Comments**
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages

XY Axes Bore Measurement With an Obstructed Center Cycle Action

Obstructed center bore measurements are determined by programming an R word with a minus value R < 0. The following steps will occur:

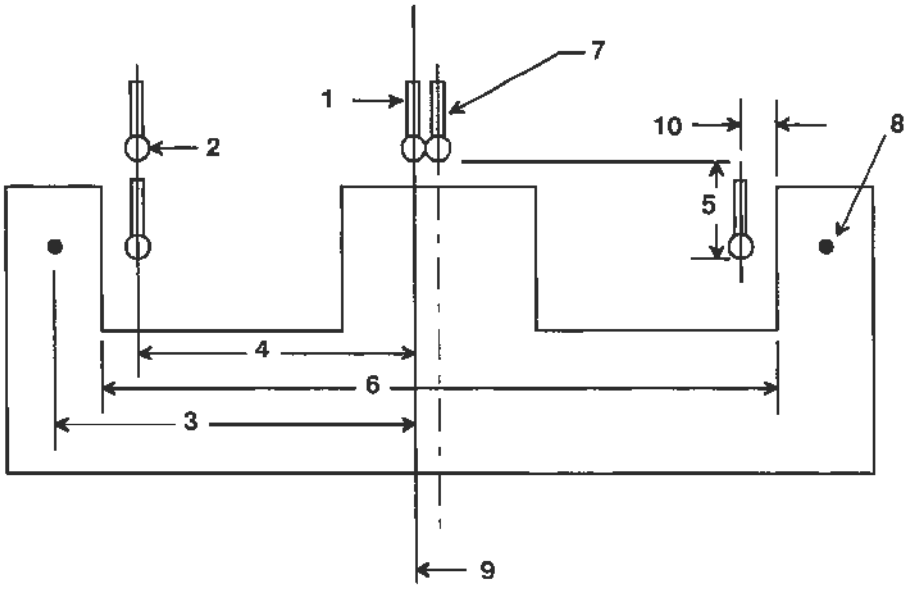
1. Position the probe near the nominal center of the bore.
2. X axis will move in the + X direction to the radical clearance position (P/2 + R word) (note the R word is negative).
3. Advance the probe along the Z axis to the measurement depth.
4. Probe in the + X direction, limiting the move to P/2 + D word if D is programmed, P/2 + R word if D is not programmed.
5. Retract in X axis by cycle parameter *Probe Gage Height*.
6. Retract Z axis to the start position.
7. Move in the - X direction to the radial clearance position.
8. Advance the probe along the Z axis to the measurement depth.
9. Probe in the - X direction, limiting the move to P/2 + D word if D is programmed, P/2 + R word if D is not programmed.
10. Retract in X axis by cycle parameter *Probe Gage Height*.
11. Retract Z axis to the start position.
12. Move to a position midway between the locations found in steps 4 and 9.
13. Y axis will move in the + Y direction to the radical clearance position (P/2 + R word).
14. Advance the probe along the Z axis to the measurement depth.
15. Probe in the + Y direction, limiting the move to P/2 + D word if D is programmed, P/2 + R word if D is not programmed.
16. Retract Y axis by cycle parameter *Probe Gage Height*.
17. Retract Z axis to the start position.
18. Move in the - Y direction to the radial clearance position.
19. Advance the probe along the Z axis to the measurement depth.
20. Probe in the - Y direction, limiting the move to P/2 + D word if D is programmed, P/2 + R word if D is not programmed.
21. Retract Y axis by cycle parameter *Probe Gage Height*.
22. Retract Z axis to the start position.
23. Move to a position midway between the locations found in steps 15 and 20.
24. Repeat steps 2 through 12 at the new Y axis position.

25. Position to the computed center of the bore.

26. Perform any require coordinate offset or tool length offset adjustments

XY Axes Bore Sample Drawing With Obstruction

Drawing Reference Number	Definition
1	Probe Start Point
2	Radial Clearance Position
3	Limit the move to P/2 + D if (D word) is programmed
4	Limit the move to P/2 + R if (D word) is not programmed
5	Z Incremental (**Z word**)
6	Nominal Width (**P Word**)
7	Measured Center
8	Probe Target Position
9	Estimated Centerline
10	Inside clearance distance (**R word**)



3-Point Measurement Over Obstruction

The 3-point measurement capability allows partial measurement of bores with cutouts located on the X or Y axis centerline. The A, B, and C angle words programmed in the G78 block command the probe to make three moves from the R word clearance toward the periphery of the bore. Measurement moves over an obstruction are defined by an R word with a minus value R < 0.

Note: Before 3-point measurements can be used the you must have a probe calibrated for 360 degree.

Basic Movements of 3-Point Measurement with Obstruction

1. Move axes to nominal center of bore start point
2. Move at A degrees from the + X direction to the radial clearance position (P/2 + R) (note that R is negative).
3. Advance the probe along the Z axis to the measurement depth
4. Probe A degrees from the + X direction limiting the move to P/2 + D if D word is programmed, P/2 + R if D word is not programmed.
5. Retract by *Probe Gage Height*.
6. Retract in Z to the start position.
7. Move at B degrees from the + X direction to the radial clearance position (P/2 + R) (note that R is negative).
8. Advance the probe along the Z axis to the measured depth.
9. Probe B degrees from the + X direction limiting the move to P/2 + D if D word is programmed, P/2 + R if D word is not programmed.
10. Retract by Probe Gage Height.
11. Retract in Z to the start position.
12. Move at C degrees from the + X direction to the radial clearance position (P/2 + R) (note that R is negative).
13. Advance the probe along the Z axis to the measured depth.
14. Probe C degrees from the + X direction limiting the move to P/2 + D if D word is programmed, P/2 + R if D word is not programmed.
15. Retract by Probe Gage Height.
16. Retract in Z to the start position.
17. Position to the computed center of the bore.
18. Perform any required coordinate offset or tool length offset adjustments.

What Offsets Can Be Adjusted

Following a successful measurement, the measured true position of the bore center can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I and J words or 3-point measurement A, B and C words must be programmed.

G78 computes error between the nominal center of the bore (specified by the I and J words) and the measured center. This data is stored in the following system registers:

[$X_POS_ERROR] [$Y_POS_ERROR]	is the error in X and Y axis
[$TRUE_POS_ERR]	is the error distance from the nominal value to the true center
[$SIZE_ERROR]	if the optional P word nominal size is specified, this value is the error between the nominal size and measured size

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X and Y axis can be changed when the H word is 0 or absent, and the O word is absent. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G78 X10 Y12 Z-10 I10.2 J12.2 R-.025 P4 ; Find center of 4 inch bore and update current active multiple setup offset

Setting Selected Setup Offsets

To apply offsets to a setup other than the active setup, the H word is 0 or absent, and the O word value is used to specify the setup record number. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the O word selection.

G78 X10 Y12 Z-10 I10 J12 O2 R-.025 P2; Find center of 2 inch bore and update multiple setup offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values, the H word specifies the type of offset, and the O word specifies which offset record. The X and Y axis position, and I and J word values or A, B, and C words values are used to compute centerline offset. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G78 X10 Y12 Z-10 I10 J12 H1 R-.025 P3 ; Find center of 3 inch bore and update current active pallet values

Setting a Fixture Offset

G78 X10 Y12 Z-10 I10 J12 H2 O3 R-.025 P2 ; Find center of 2 inch bore and update Fixture offset 3 values

Setting a Programmable Coordinate Offset

G78 X10 Y12 Z-10 I10 J12 H3 O5 R-.025 P3 ; Find center of 3 inch bore and update Programmable coordiuate offset 5 values

Changing Programmable Tool Offset Table Diameter value

Since X and Y axes are used for measurement, the Programmable Tool Offset table diameter value can be change by programming H5 with an O word and no T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G78 X10 Y12 Z-5 I10.2 J12.2 H5 O6 R-.025 P4 ; change tool diameter record number 6 programmable tool table value.

Changing Tool Table Diameter Offset value

Since X and Y axes are used for measurement, the Tool Table Offset diameter value can be change by programming H5 with a T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 T6 R-.025 P3 ; Change tool table diameter offset value record number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$TRUE_POS_ERR] exceeds the U word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

G78 Programming Considerations

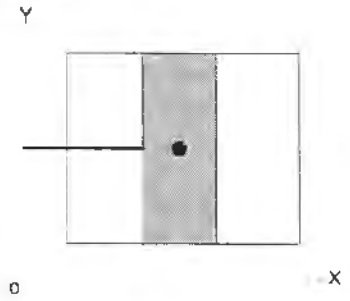
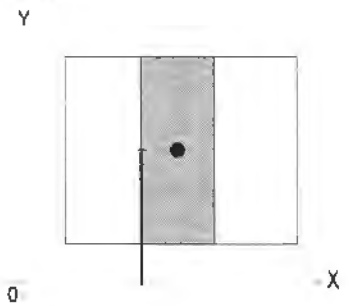
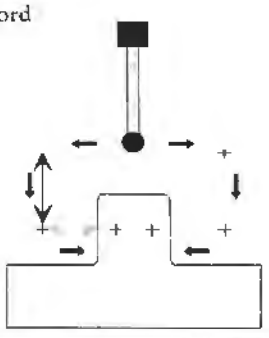
- Prior to executing the G78 cycle, the NC program must position the probe near the center of the bore to be measured.
- For each axis programmed in the G78 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- The R word must be a minus number R < 0.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.
- When using 3-point measurement the probe must be calibrated for 360 degrees.
- When using 3-point measurement the A, B, and C words must all be programmed.

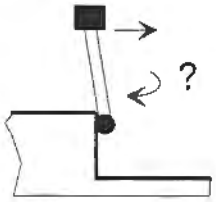
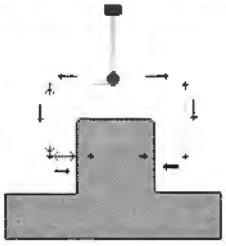
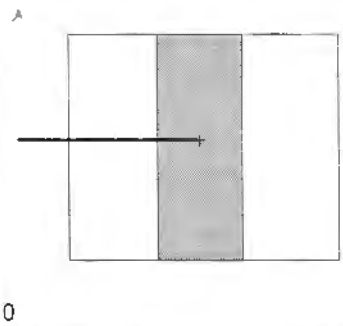
G79 Probe to Measure a Web

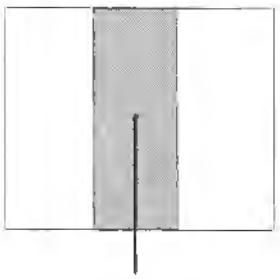
To select web measurement the G79 program block **must** contain an R word with a positive value R > 0. The cycle measurement that takes place is made along the *measurement line*, which is a line through the starting probe stylus position to the programmed XY endpoint. Once measurement is made, data is collected and stored in the following system registers:

[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_WIDTH]	is the measured width
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)]	records center measurement coordinates
[$PRB_PART_LOC(Z)]	used to set the Z axis dimension at which the measurements were made.

Note: If the web measurement is made along a vector that lies at an angle to the X and Y axis, you should be using a calibrated 360 degree probe for greater measurement accuracy.

G79 Probe to Measure a Web Program Words	
Word Description	**Comments**
X Word 	Specifies nominal coordinates of one end of the measurement line.
Y Word 	Specifies nominal coordinates of one end of the measurement line.
Z Word 	Specifies nominal signed incremental distance to move from the start point to the Z axis location at which the measurements are to be made.

G79 Probe to Measure a Web Program Words (continued)	
Word Description	**Comments**
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit The first move for a double hit cycle (the only move for single hit measurements) the probe retracts by *Probe Gage Height* cycle parameter and makes a second measurement move of twice *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter. The final position of the probe is on the measured centerline of the web at the initial Z axis position. The values of system registers [$PRB_POS_MC], and [$PRB_POS_PC] reflect the location of the last probe move completed in the cycle. If any probe move fails to hit the part, the cycle aborts, the probe returns to the start point, and system register [$PROBE_HIT] is set false.
R+ Word Clearance Distance (positive) 	Specifies non-metal signed radial measurement clearance distance outside of the wall of the web (the programmed XY coordinate) available for probe motion. The R word **must** be positive for web measurement R > 0.
Optional P Word Width	Specifies non-modal unsigned nominal width of the web to be measured.
Optional I Word Reset Coordinate to 	Specifies non-modal type of offset (based on H word) to assign to the measured X axis coordinates of the center of the measured web.

G79 Probe to Measure a Web Program Words (continued)	
Word Description	**Comments**
Optional J Word Reset Coordinate to Y'  0 X'	Specifies non-modal type of offset (based on H word) to assign to the measured Y axis coordinates of the center of the measured web.
H Word Type of offset to be set based on I and J words. Actions of this word are effected by the O word.	Non-modal offset number used to set: 0 or not programmed = Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset 5 = Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages additional information.
D Word Overtravel Distance	Non-modal probe overtravel distance allowed beyond the programmed motion (the XY coordinate programmed). If the D word is absent, the value of cycle parameter *Probe Gage Height* is used.
O Word defines offset number to be set	Optional with H = 0 Required with H = 2, 3 or 5 ignored otherwise
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.

G79 Probe to Measure a Web Program Words (continued)	
Word Description	**Comments**
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less thau the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages additional information.

Web Cycle Action

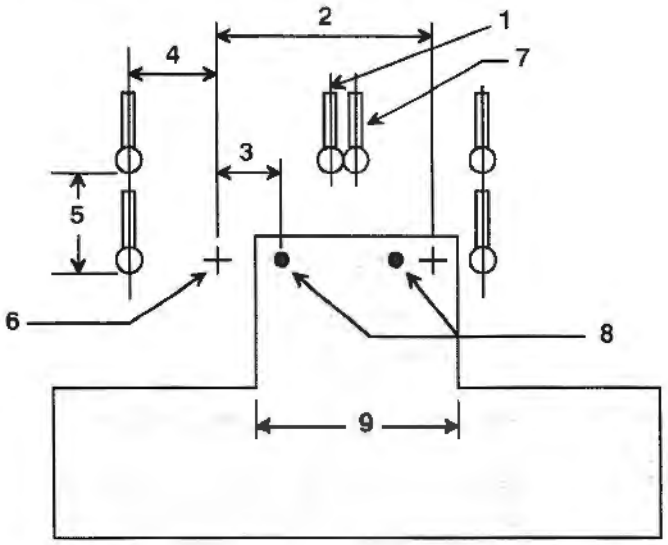
To measure a web the G79 block **must** contain an R word with a positive value R > 0.

Before the G79 block is executed, the NC program must position the probe near the center of the web to be measured at a Z axis position that is clear of any obstruction. When G79 is executed the following steps will occur:

1. X and Y axis move to the programmed location (the end of the measurement line, which is the nominal part surface location) plus the clearance amount specified by the R word.
2. Advance Z axis to depth.
3. Probe toward center of web along measurement line. This move is the R word clearance plus D word overtravel allowance.
4. Retract from the probe hit position by cycle parameter *Probe Gage Height*.
5. Retract in Z axis to the starting Z dimension.
6. Move X and Y to the opposite end of the measurement line plus the clearance specified by the R word..
7. Advance the probe along the Z axis by the Z word increment.
8. Probe toward center of web along measurement line. This move is the R word clearance plus D word overtravel allowance.
9. Retract from the probe hit position by cycle parameter *Probe Gage Height*.
10. Retract in Z axis to the starting Z dimension.
11. Position in XY to the measured center of the web

Cycle Action Sample Drawing

Drawing Reference Number	Definition
1	Probe Start Point
2	Measurement Line
3	Overtravel Allowance (**D word**)
4	Outside clearance distance (**R word**)
5	Z Incremental (**Z word**)
6	Programmed XY Location
7	Measured Center
8	Probe Target
9	Nominal Width (**P word**)



What Offsets Can Be Adjusted

Following a successful web measurement, the measured true position of the web center can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I and J words must be programmed.

G79 computes error between the nominal center of the web (specified by the I and J words) and the measured center. This data is stored in the following system registers:

[$X_POS_ERROR] [$Y_POS_ERROR]	is the error in X and Y axis
[$TRUE_POS_ERR]	is the error distance from the nominal value to the true center
[$SIZE_ERROR]	if the optional P word nominal size is specified, this value is the error between the nominal size and measured size

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X and Y axis can be changed when the H word is 0 or absent, and the O word is absent. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10.2 J12.2 R.025 P4 ; Find center of 4 inch web and update the current active multiple setup offset

Setting Selected Setup Offsets

To apply offsets to a setup other than the active setup, the H word is 0 or absent, and the O word value is used to specify the setup record number. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10 J12 O2 R.025 P2 ; Find center of 2 inch web and update multiple setup record offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values, the H word specifies the type of offset, and the O word specifies which offset record. The X and Y axis position, and I and J word values are used to compute centerline offsets. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G79 X10 Y12 Z-10 I10 J12 H1 R.025 P3 ; Find center of 3 inch web and update current active pallet values

Setting a Fixture Offset

G79 X10 Y12 Z-10 I10 J12 H2 O3 R.025 P2 ; Find center of 2 inch web and update Fixture offset 3 values

Setting a Programmable Coordinate Offset

G79 X10 Y12 Z-10 I10 J12 H3 O5 R.025 P3 ; Find center of 3 inch web and update Programmable coordinate offset 5 values

Changing Programmable Tool Offset Table Diameter value

Since X and Y axes are used for measurement, the Programmable Tool Offset table diameter value can be change by programming H5 with an O word and no T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 O6 R.025 P4 ; change tool diameter record number 6 programmable tool table value.

Changing Tool Table Diameter Offset value

Since X and Y axes are used for measurement, the Tool Table Offset diameter value can be change by programming H5 with a T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 T6 R.025 P3 ; Change tool table diameter offset value record number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$TRUE_POS_ERR] exceeds the V word value. [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

G79 Programming Considerations

- Prior to executing the G79 cycle, the NC program must position the probe near the center to be measured.
- For each axis programmed in the G79 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- The R word must be a plus number R > 0 for web measurements.
- If no probe hit is detected, system register [SPROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.
- If the web measurement is made along a vector that lies at an angle to the X and Y axis, you should use be using a calibrated 360 degree probe for greater measurement accuracy.

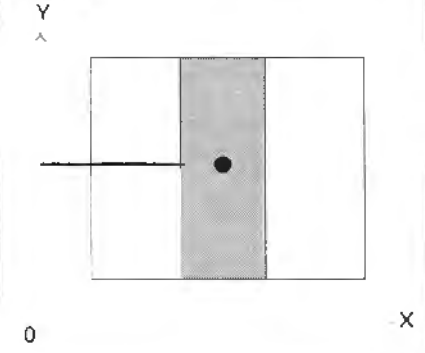
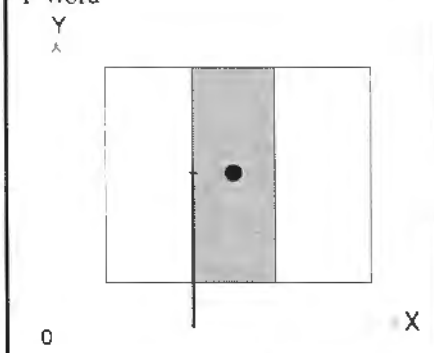
G79 Probe to Measure a Pocket

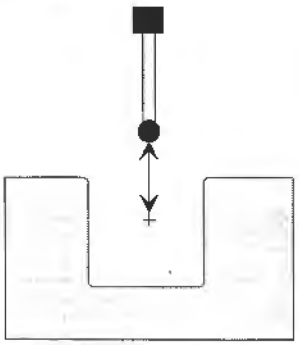
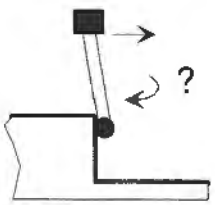
To select pocket measurement the G79 program block R word **must** equal 0 or be absent. The cycle measurement that takes place is made along the *measurement line*, which is a line through the starting probe stylus position to the programmed XY endpoint. Once measurement is made, data is collected and stored in the following system registers:

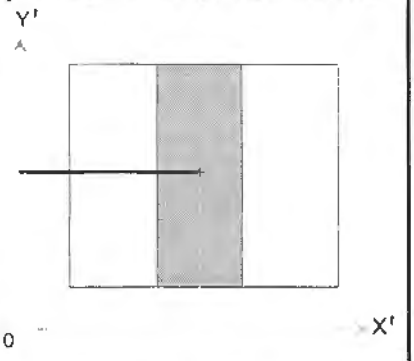
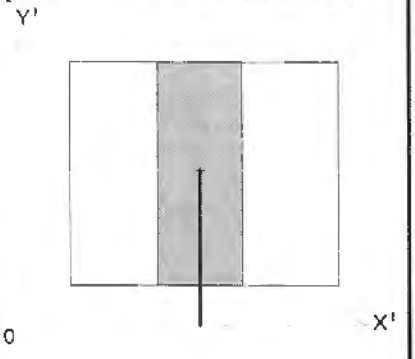
[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_WIDTH]	is the measured width
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)]	records center measurement coordinates
[$PRB_PART_LOC(Z)]	used to set the Z axis dimension at which the measurements were made.

Note: If the pocket measurement is made along a vector that lies at an angle to the X and Y axis, you should use a be calibrated 360 degree probe for greater measurement accuracy.

G79 Probe to Measure a Pocket Program Words

Word Description	Comments
X Word 	Specifies nominal coordinates of one end of the measurement line.
Y Word 	Specifies nominal coordinates of one end of the measurement line.

G79 Probe to Measure a Pocket Program Words (continued)	
Word Description	**Comments**
Z Word 	Specifies nominal signed incremental distance to move from the start point to the Z axis location at which the measurements are to be made.
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit The first move for a double hit cycle (the only move for single hit measurements) the probe retracts by *Probe Gage Height* cycle parameter and makes a second measurement move of twice *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter. The final position of the probe is on the measured centerline of the web at the initial Z axis position. The values of system registers [$PRB_POS_MC], and [$PRB_POS_PC] reflect the location of the last probe move completed in the cycle. If any probe move fails to hit the part, the cycle aborts, the probe returns to the start point, and system register [$PROBE_HIT] is set false.
R Word Clearance Distance (R = 0 or absent)	Specifies pocket measurement where the center of the pocket is free of obstructions. The R word **must** equal 0 or absent for pocket measurement with no obstruction. R = 0 or absent.
Optional P Word Width	Specifies non-modal unsigned nominal width of the pocket to be measured.

G79 Probe to Measure a Pocket Program Words (continued)	
Word Description	**Comments**
Optional I Word Reset Coordinate to 	Specifies non-modal type of offset (based on H word) to assign to the measured X axis coordinates of the center of the measured pocket.
Optional J Word Reset Coordinate to 	Specifies non-modal type of offset (based on H word) to assign to the measured Y axis coordinates of the center of the measured pocket.
H Word Type of offset to be set based on I and J words. Actions of this word are affected by the O word.	Non-modal offset number used to set: 0 or not programmed = Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset 5 = Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pages for additional information.
D Word Overtravel Distance	Non-modal probe overtravel distauce allowed beyond the programmed motion (the XY coordinate programmed). If the D word is absent, the value of cycle parameter *Probe Gage Height* is used.
O Word defines offset number to be set	Optional with H = 0 Required with H = 2, 3 or 5 ignored otherwise
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.

G79 Probe to Measure a Pocket Program Words (continued)	
Word Description	**Comments**
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.
W Word	The non-modal W word specifies some variation in the measnred valnes to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measnrement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error actiou taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pages for additional information.

Pocket Cycle Action

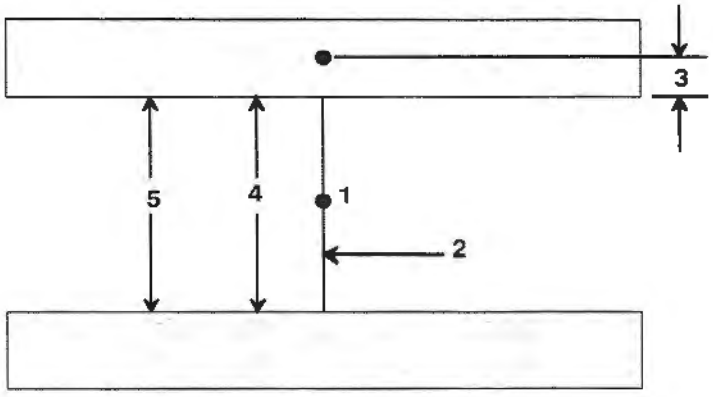
The actions to measure a pocket center is determined by programming an R word equaling 0 or absent. The following steps will occur:

Position the probe near the nominal center of the pocket.

1. Advance the probe along the Z axis by the incremental value specified in the Z word.
2. Probe from the start point along the measurement line to the programmed XY axis location, continue beyond XY axis by the overtravel allowance specified by the D word if necessary.
3. Probe in the opposite direction along the measurement line until the surface is found. This move continues for twice the distance from the initial point to the programmed XY location plus the overtravel allowance specified by the D word.
4. Position to the computed midpoint of the measurement line.
5. Retract the probe to the Z axis start point.
6. Next, any required coordinate offset or tool offset adjustments are performed.

Cycle Action Sample Drawing

Drawing Reference Number	Definition
1	Probe Start Point
2	Measurement Line
3	Overtravel Allowance (**D word**)
4	Measured Pocket Width, System Register [$PRB_WIDTH]
5	Nominal Width (**P word**)



Changing Coordinates

Following a successful pocket measurement, the measured true position of the pocket center can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I and J words must be programmed.

G79 computes error between the nominal center of the pocket (specified by the I and J words) and the measured center. This data is stored in the following system registers:

[$X_POS_ERROR] [$Y_POS_ERROR]	is the error in X and Y axis
[$TRUE_POS_ERR]	is the error distance from the nominal value to the true center
[$SIZE_ERROR]	if the optional P word nominal size is specified, this value is the error between the nominal size and measured size

Setting Offset of the Active Setup

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X and Y axis can be changed when the H word is 0 or absent, and the O word is absent. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10.2 J12.2 P4 ; Find center of 4 inch web and update the current active multiple setup offset

Setting Offset of a Selected Setup

To apply offsets to a setup other than the active setup, the H word is 0 or absent, and the O word value is used to specify the setup record number. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10 J12 O2 P2 ; Find center of 2 inch web and update multiple setup record offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values, the H word specifies the type of offset, and the O word specifies which offset record. The X and Y axis position, and I and J word values are used to compute centerline offsets. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G79 X10 Y12 Z-10 I10 J12 H1 P3 ; Find center of 3 inch pocket and update current active pallet values

Setting a Fixture Offset

G79 X10 Y12 Z-10 I10 J12 H2 O3 P2 ; Find center of 2 inch pocket and update Fixture offset 3 values

Setting a Programmable Coordinate Offset

G79 X10 Y12 Z-10 I10 J12 H3 O5 P3 ; Find center of 3 inch pocket and update Programmable coordinate offset 5 values

Changing Programmable Tool Offset Table Diameter value

Since X and Y axes are used for measurement, the Programmable Tool Offset table diameter value can be change by programming H5 with an O word and no T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 O6 P4 ; change tool diameter record number 6 programmable tool table value.

Changing Tool Table Diameter Offset value

Since X and Y axes are used for measurement, the Tool Table Offset diameter value can be change by programming H5 with a T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 T6 P3 ; Change tool table diameter offset value record number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

G79 Programming Considerations

- Prior to executing the G79 cycle, the NC program must position the probe near the center of the pocket to be measured.
- For each axis programmed in the G79 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- The R word must be a plus number R > 0.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.
- If the pocket measurement is made along a vector that lies at an angle to the X and Y axis, you should use a be calibrated 360 degree probe for greater measurement accuracy.

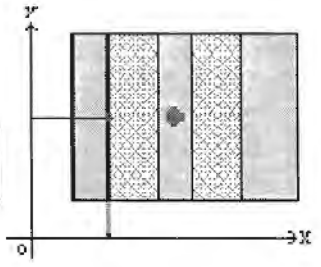
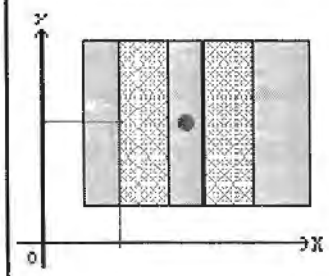
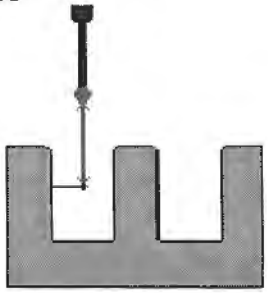
G79 Probe Pocket with Obstacle

To select this measurement the G79 program block **must** contain an R word with a negative value R < 0. The cycle measurement that takes place is made along the *measurement line*, which is a line through the starting probe stylus position to the programmed XY endpoint. Once measurement is made, data is collected and stored in the following system registers:

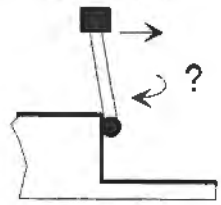
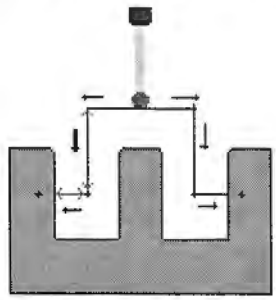
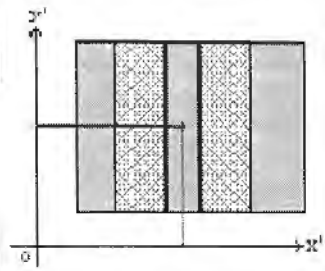
[$PROBE_HIT]	Set true non-zero with probe hit
[$PRB_WIDTH]	is the measured width
[$PRB_PART_LOC(X)] [$PRB_PART_LOC(Y)]	records center measurement coordinates
[$PRB_PART_LOC(Z)]	used to set the Z axis dimension at which the measurements were made.

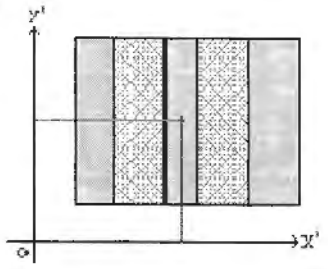
Note: If the web measurement is made along a vector that lies at an angle to the X and Y axis, you should be using a calibrated 360 degree probe for greater measurement accuracy.

G79 Probe Pocket with Obstacle Program Words

Word Description	Comments
X Word 	Specifies nominal coordinates of one end of the measurement line.
Y Word 	Specifies nominal coordinates of one end of the measurement line.
Z Word 	Specifies nominal signed incremental distance to move from the start point to the Z axis location at which the measurements are to be made.

G79 Probe Pocket with Obstacle Program Words (continued)

Word Description	Comments
Probe Process Q Word 	Q0 Double Hit Q1 Single Hit The first move for a double hit cycle (the only move for single hit measurements) the probe retracts by *Probe Gage Height* cycle parameter and makes a second measurement move of twice *Probe Gage Height* at the *Probe Measurement Feedrate* cycle parameter. The final position of the probe is on the measured centerline of the pocket at the initial Z axis position. The values of system registers [$PRQBE_HIT], [$PRB_POS_MC], and [$PRB_POS_PC] reflect the location of the last probe move completed in the cycle. If any probe move fails to hit the part, the cycle aborts, the probe returns to the start point, and system register [$PROBE_HIT] is set false.
R Word Clearance Distance (negative) 	Specifies the clearance distance inside of the pocket edge (the programmed XY coordinate) for probe motion. The R word **must** be negative R < 0.
Optional P Word Width	Specifies non-modal unsigned nominal width of the pocket to be measured.
Optional I Word Reset Coordinate to 	Specifies non-modal type of offset (based on H word) to assign to the measured X axis coordinates of the center of the measured pocket.

G79 Probe Pocket with Obstacle Program Words (continued)	
Word Description	**Comments**
Optional J Word Reset Coordinate to 	Specifies non-modal type of offset (based on H word) to assign to the measured Y axis coordinates of the center of the measured pocket.
H Word Type of offset to be set based on I and J words. Actions of this word are effected by the O word.	Non-modal offset number used to set: 0 or not programmed = Setup Offset 1 = Pallet Offset 2 = Fixture Offset 3 = Programmed Coordinate Offset 5 = Programmable Tool Offset Refer to **What Offsets Can Be Adjusted** on the following pagesfor additional information.
D Word Overtravel Distance	Non-modal probe overtravel distance allowed beyond the programmed motion (the XY coordinate programmed). If the D word is absent, the value of cycle parameter *Probe Gage Height* is used.
O Word defines offset number to be set	Optional with H = 0 Required with H = 2, 3 or 5 ignored otherwise
T Word	Non-modal tool reference or record number of the tool to be updated. Used with H5.
U Word	The non-modal U word specifies the upper tolerance limit above which no offset update is performed and tolerance limit exceeded error is reported.
V Word	The non-modal V word specifies a tolerance below which the measured error is applied to the specified tool offset and above which the offset is updated but an out of tolerance error is reported.

G79 Probe Pocket with Obstacle Program Words (continued)	
Word Description	**Comments**
W Word	The non-modal W word specifies some variation in the measured values to occur without changing tool offset. The W word is only valid when programmed with a T word or with H5. **Note:** the null band should be large enough to prevent changing the tool offsets for measurement errors that are less than the measurement accuracy.
F Word	Non-modal feedback which controls the percent of feedback applied. Range values are from 0 to 100, representing the percentage of the measured error to be applied to the tool length offset.
L Word	Non-modal experience correction for setting tool offsets. The L word value on a surface measurement probe cycle specifies an amount to be added to the measured size before computing system register data [$SIZE_ERROR] The L word provides an adjustment to relate the measurements on the machine tool to a traceable standard such as a coordinate measurement machine.
E Word	Non-modal error action taken if the probe does not contact a surface or the measured error exceeds a specified tolerance limit 0 or not programmed: abort and report an alarm 1: set system register [$OUT_OF_TOL] or [$TOL_EXCEEDED] or [$ANGLE_ERROR] *true* and continue NC program execution. Refer to **What Offsets Can Be Adjusted** Tool Offset Tolerance on the following pagesfor additional information.

Obstructed Pocket Cycle Action

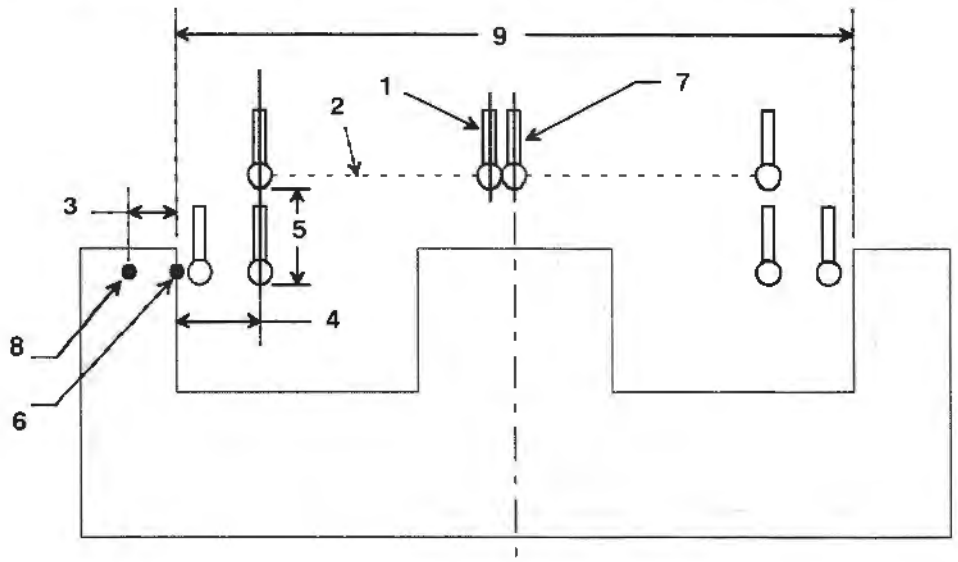
To measure an obstructed pocket the G79 block **must** contain an R word with a negative value R < 0.

Before the G79 block is executed, the NC program must position the probe near the center of the web to be measured at a Z axis position that is clear of any obstruction. When G79 is executed the following steps will occur:

1. X and Y axis move to the programmed location (the end of the measurement line, which is the nominal part surface location) plus the clearance amount specified by the R word.
2. Advance Z axis to depth.
3. Probe toward outside of pocket along measurement line. This move is the R word clearance plus D word overtravel allowance.
4. Retract from the probe hit position by cycle parameter *Probe Gage Height*.
5. Retract in Z axis to the starting Z dimension.
6. Move X and Y to the opposite end of the measurement line plus the clearance specified by the R word..
7. Advance the probe along the Z axis by the Z word increment.
8. Probe toward outside of pocket along measurement line. This move is the R word clearance plus D word overtravel allowance.
9. Retract from the probe hit position by cycle parameter *Probe Gage Height*.
10. Retract in Z axis to the starting Z dimension.
11. Position in XY to the measured center of the pocket

Cycle Action Sample Drawing

Drawing Reference Number	Definition
1	Probe Start Point
2	Measurement Line
3	Overtravel Allowance (**D word**)
4	Clearance (**R word**)
5	Z Incremental (**Z word**)
6	Programmed XY Location
7	Measured Center
8	Probe Target
9	Nominal Width (**P word**)



What Offsets Can Be Adjusted

Following a successful web measurement, the measured true position of the web center can be used to set a coordinate offset. The selection of which offset to adjust is controlled by the optional parameters specified. In order to set any offset, the I and J words must be programmed.

G79 computes error between the nominal center of the web (specified by the I and J words) and the measured center. This data is stored in the following system registers:

[$X_POS_ERROR] [$Y_POS_ERROR]	is the error in X and Y axis
[$TRUE_POS_ERR]	is the error distance from the nominal value to the true center
[$SIZE_ERROR]	if the optional P word nominal size is specified, this value is the error between the nominal size and measured size

Setting Active Setup Offsets

No matter what your machine configuration is, you will always have one active setup. The active setup offset values for X and Y axis can be changed when the H word is 0 or absent, and the O word is absent. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10.2 J12.2 R-.025 P4 ; Find center of 4 inch pocket and update the current active multiple setup offset

Setting Selected Setup Offsets

To apply offsets to a setup other than the active setup, the H word is 0 or absent, and the O word value is used to specify the setup record number. The X and Y axis position, and I and J word values are used to compute centerline offsets. The control performs a G92.1 Position Set to the active setup offset values.

G79 X10 Y12 Z-10 I10 J12 O2 R-.025 P2; Find center of 2 inch pocket and update multiple setup record offset 2

Setting a Pallet, Fixture, or Programmable Coordinate Offset

To set these offset values, the H word specifies the type of offset, and the O word specifies which offset record. The X and Y axis position, and I and J word values are used to compute centerline offsets. The following examples will write offset values to their respected tables.

Setting a Pallet Offset

G79 X10 Y12 Z-10 I10 J12 H1 R-.025 P3 ; Find center of 3 inch pocket and update current active pallet values

Setting a Fixture Offset

G79 X10 Y12 Z-10 I10 J12 H2 O3 R-.025 P2 ; Find center of 2 inch pocket and update Fixture offset 3 values

Setting a Programmable Coordinate Offset

G79 X10 Y12 Z-10 I10 J12 H3 O5 R-.025 P3 ; Find center of 3 inch pocket and update Programmable coordinate offset 5 values

Changing Programmable Tool Offset Table Diameter value

Since X and Y axes are used for measurement, the Programmable Tool Offset table diameter value can be change by programming H5 with an O word and no T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 O6 R-.025 P4 ; change tool diameter record number 6 programmable tool table value.

Changing Tool Table Diameter Offset value

Since X and Y axes are used for measurement, the Tool Table Offset diameter value can be change by programming H5 with a T word. The tool is adjusted by data in system register [$SIZE_ERROR].

G79 X10 Y12 Z-5 I10.2 J12.2 H5 T6 R-.025 P3 ; Change tool table diameter offset value record number 6

Tool Offset Tolerances

Tool offset adjustment can be influenced by the tolerances specified by the U word (upper tolerance limit), the V word (tolerance) and the W word (null band).

If the V word and system register [$TRUE_POS_ERR] exceeds the V word value, [$OUT_OF TOL] is set true (non-zero).

If the U word and system register [$TRUE_POS_ERR] exceeds the V word value, [$TOL_EXCEEDED] is set true (non-zero).

Tolerance reporting is determined by the E word.

If the E word is absent, an alarm is reported and the NC program halts.

If the E word = 1, no alarm is reported and NC program continues with the next block. When the E word is 1 the NC program test [$PROBE_HIT] to determine whether or not the cycle completed its measurements. Also, system registers [$OUT_OF_TOL], and [$TOL_EXCEEDED] are tested to determine whether an error occurred.

G79 Programming Considerations

- Prior to executing the G79 cycle, the NC program must position the probe near the center of the pocket to be measured.
- For each axis programmed in the G79 block the probe moves from the current position toward the programmed endpoint at the cycle parameter *Probe Approach Feedrate*.
- The R word must be a negative number R < 0.
- If no probe hit is detected, system register [$PROBE_HIT] set to false (zero) and the probe retracts to the programmed position minus cycle parameter *Probe Gage Height*.
- If the Q word value is absent or zero, specifying a double hit cycle, the initial move toward the part is made at cycle parameter *Probe Approach Feedrate*. When the probe hits the part, it retracts by cycle parameter *Probe Gage Height*, then executes a second measurement move of *Probe Gage Height* plus the D word value (or *Probe Gage Height* if D is absent) at cycle parameter *Probe Measurement Feedrate*.
- If the pocket measurement is made along a vector that lies at an angle to the X and Y axis, you should be using a calibrated 360 degree probe for greater measurement accuracy.

Chapter 3
Tool Probe Option

General Information

The tool sensor, or fixed probe is positioned within the machine travel envelope so that tools can touch the probe stylus. To measure and check tools, two probe cycles are provided, G68 Set Tool Size, and G69 Check Tool Size. G68 is used to measure tool length and diameter, after which, data can selectively be recorded in either the *Tool Data Table* or *Programmable Tool Offset Table*. G69 is used to check, update, and report tool status.

G68 or G69 can be programmed using Resident Assistant Programming (RAP), Manual Data Input (MDI), or within the part program itself. For sample purposes, RAP illustrations are presented in this document.

G68 Tool Setter Probe General Information

Before any probe calibration is performed, do the Set Fixed Probe Tram Surface to define top position of the fixed probe stylus.

Calibrate the Tool Setter Probe (using G68 P5, P6, or P7) first before any tool measurements.

Diameter measurements using Y axis can be selected at the Machine Application TOOL PROBE menu, check box **Use Y Axis to Measure Tool Diameter**. Note, the Service password level must be active to change axis selection.

Set Cycle Parameter *Fixed Probe Clearance Height* with the longest tool you will be using. This value will ensure adequate clearance above the Tool Setter Probe.

G68 is a Non-modal G code; that is, it is active only in the block in which it is programmed and need not be replaced with a G1 or G0 to move the tool following a Set Tool Size cycle.

Generally, length for tools smaller in diameter than the tool probe are measured with the tool not rotating

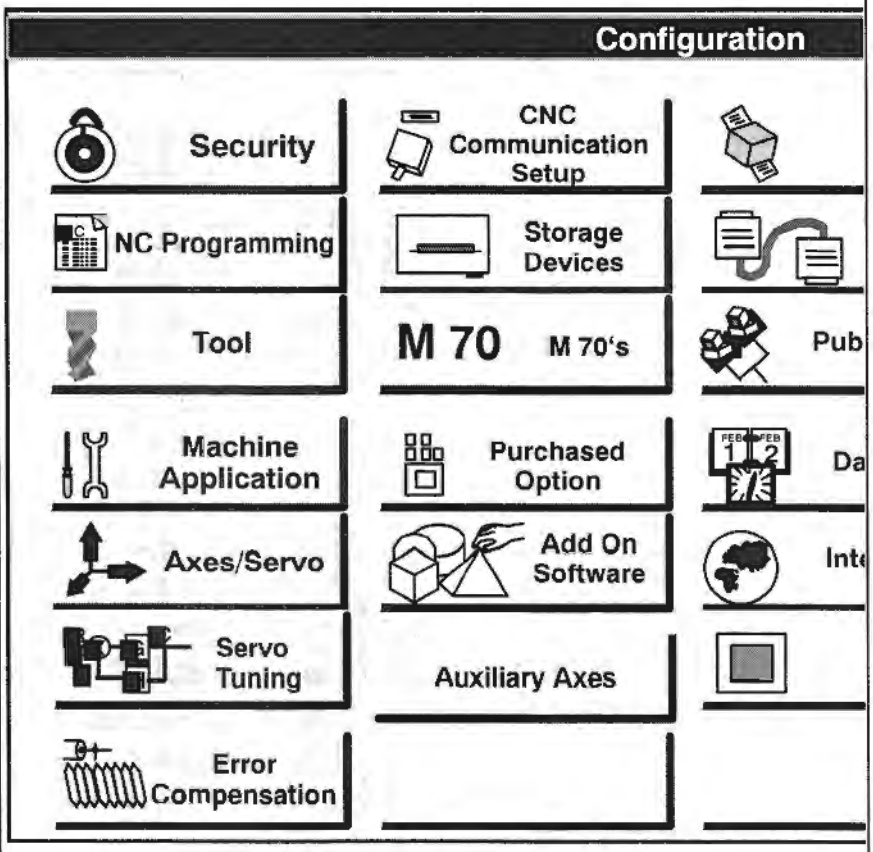
Tool diameter measurements are always made with the tool rotating. Also, two probe hits are used to determine the cutter high edge.

When using G68 L1 to set tool lengths, **always** position the tool over the probe before executing.

G68 Tool Probe Operations can be generated in MDI, by using Resident Assistant Programmer (RAP), or by NC programming.

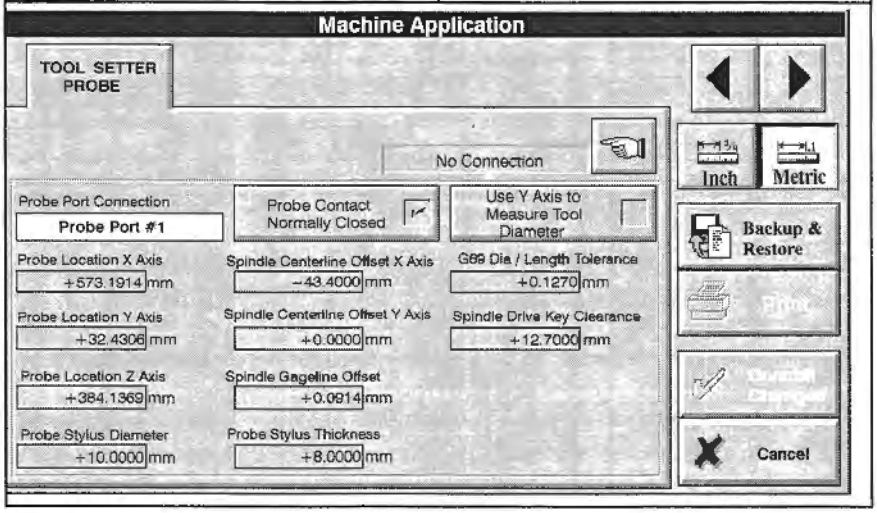
View Tool Setter Probe Coordinates	
Touch, press or set the following	**Comments**
	
	

View Tool Setter Probe Coordinates (continued)	
Touch, press or set the following	**Comments**
The following display will appear:" (partial view)	



Machine Application	
(Use arrows to locate **Tool Setter Probe**)	
TOOL SETTER PROBE	
The following display will appear: (Touch or Press **Tool Setter Probe**)	

View Tool Setter Probe Coordinates (continued)	
Touch, press or set the following	**Comments**
TOOL SETTER PROBE	



Probe Location X, Y, Z Axes, identifies the probe location in machine coordinates.

Example A: Machine zero is located at the extreme +Z axis travel position, which is usually the Z axis align position. The Probe Location Z Axis field value will be a negative number.

NOTE: In example A, Machine zero is referenced at extreme +Z axis position.

Example B: Machine zero is located at the table top or center of index, depending on machine type (vertical or horizontal). The Probe Location Z Axis field value is the distance from the table top or center of index, depending on machine type (vertical or horizontal) to the top of the fixed probe stylus. The Probe Location Z Axis field value will be a positive number.

NOTE: In example B, Machine zero referenced from the table top or center of index, depending on machine type (vertical or horizontal).

Spindle Centerline Offsets X and Y Axes values are machine coordinates which offset the spindle face for Fixed Tram Surface Measurement. When the spindle face hit occurs, the Cycle Parameter "Fixed Probe Tram Surface" FIX_PRB_TRAM value is set.

Spindle Gageline Offset: For V flange tooling, the butt of the tool's flange does not actually seat against the spindle nose. This distance is the Spindle Gageline Offset. This value is normally derived from an off line tool measurement fixture.

G69 Diameter/Length Tolerance: This default tolerance is used when the G69 block contains no tolerance values. Normally, if this value is exceeded an Alert is posted, and the tool table field "Tool Status" is changed to mark the tool worn or broken. Program execution is determined by the E word programmed in the G69 block.

Probe Related M-codes

M68 - Advance Tool Probe (If Applicable)

M68 causes the control to advance the Tool Probe into the Work Zone and prepares it for use.

M69 - Retract Tool Probe (If Applicable)

M59 causes the Tool Probe to be retracted to its parked position, usually out of the Work Zone.

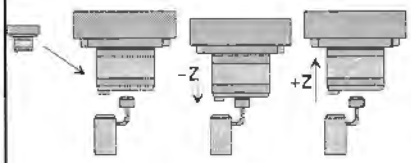
What Cycle Parameters Are Used With the Tool Setter Probe

Tool Setter Probe Cycle Parameters provide the operator with a means of entering and modifying parameters associated with tool setter probe operations. Most parameters can be overridden programmatically on an individual cycle basis. The table contains two sets of values: *Programmable Value*, the set of active values, which the operator can alter as needed, and *Base Value* a set of default values, which can be configured under the Setup password. The active value can be reset to the default values by the probe cycles.

To view or Change these cycle parameters, refer to the procedure: "Change Cycle Parameters"

Spindle Probe and Tool Setter Probe cycle parameters will be both be displayed. Cycle parameters that are used by the Tool Setter Probe are listed below. Ranges and definitions may be found in Appendix A.

Cycle	Program Reference
Probe Approach	PRB_APPR_FRT
Dimensions Probe Measurement Feed rate	PRB_MEAS_FRT
Rotating Tool Retract Distance	FIX_PRB_RRET
Probe Gage Height	PROBE_GH
Fixed Probe Tram Surface	FIX_PRB_TRAM
Fixed Probe Clearance Height	FIX_PRB_CLR

Setting the Fixed Probe Tram Surface	
Before any probe calibration is performed, the top position of the fixed prob stylus must be defined. This procedure will use the **Spindle Centerline Offsets X and Y Axis** values to position the spindle face over the probe stylus. This procedure is normally only required when the probe physical location has been changed or a new probe has been installed.	
Touch, press or set the following	**Comments**
	
 M6 T0; (Removes tool) G68 Q1 (Specifies Single Hit)	Display must be either Production, Current, To Go or Program.
	
Action Sequence 	Usually spindle orientation takes place with the keys positioned parallel with X axis. If you need to change orientation program a J word. Setup values **Spindle Centerline Offsets X and Y Axis** determine spindle centerline offset. Usually Y axis is offset. Cycle Parameter *Fixed Probe Tram Surface* FIX_PRB_TRAM is updated.

What Calibrating the Probe Does

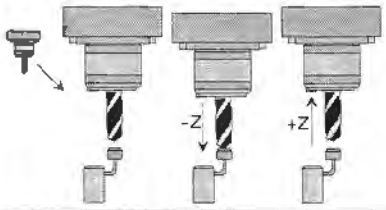
Probe calibration data is the base value from which tool length and diameter measurements are calculated. So it is important to calibrate the probe before taking any tool measurements. After calibration, probe data is stored and used for calculating tool measurements. Also, it should be noted that only the most recent calibration data is retained by the control. Probe calibration is performed by programming G68 with P5, P6 or P7. Additional word descriptions for these cycles are listed later in this document.

Calibrating the Probe Length with Known Length Tool

To use this procedure a tool of precisely known length must be placed in the spindle. Therefore, the tool used for calibration should be a drill or other tool with a single point or smaller in diameter than the probe head.

Ensure precision tool values are entered in the tool table. Place zero values in tool table fields **X Probe Offset** and **Y Probe Offset** for the precision tool.

Note: If Tool Data is manually changed while the tool is in the spindle, you must call the tool again for new tool data to register. For example, if T6 is in the spindle and data is changed, in MDI simply enter T6 M6 and press Cycle Start to register new tool data.

Touch, press or set the following	Comments
	
 M6 Txx G68 P5 Q1	Display must be either Production, Current, To Go or Program. Txx is the precision tool used for calibration loaded into the spindle
	
Action Sequence 	Axis position to the probe center line. Tool length calibration is performed with the spindle stopped and with the spindle centerline on the probe centerline.

Calibrating the Probe Diameter with Known Diameter Tool

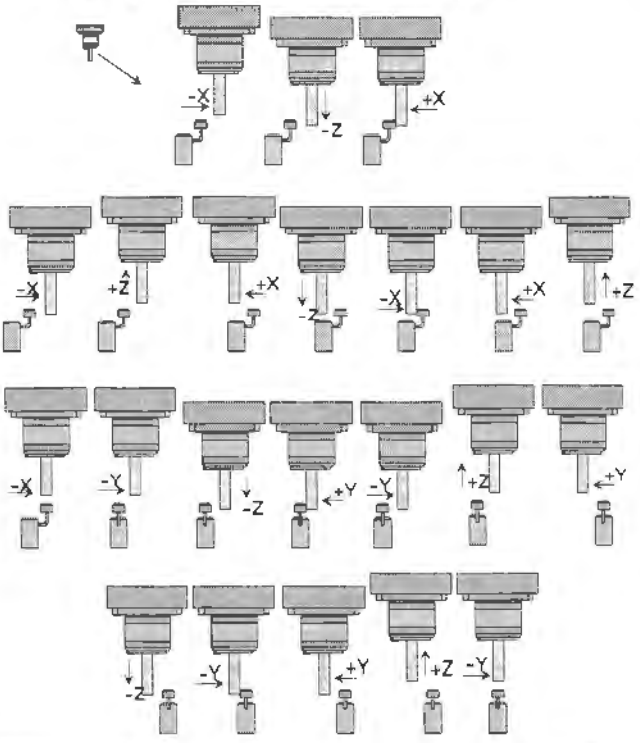
The tool probe is calibrated for diameter measurements by programming G68 P6 with a tool of precisely known diameter in the spindle. The tool used for calibration must be a smooth diameter since calibration is done with spindle stopped.

Ensure precision tool data is entered in the tool table. Also, enter zero values in tool table fields **X Probe Offset** and **Y Probe Offset** to ensure spindle center line positioning over the probe.

Note: Since the tool will position on all four sides of the probe for a full calibration, the tool diameter should be relatively small.

Touch, press or set the following	Comments
	
 M6 Txx G68 P6 Q1 A .1 R.1	Display must be either Production, Current, To Go or Program. Txx is the arbor used for calibration loaded into the spindle
	

Calibration Sequence with Arbor Tool



Calibrating the Probe Length and Diameter with Known Diameter Tool

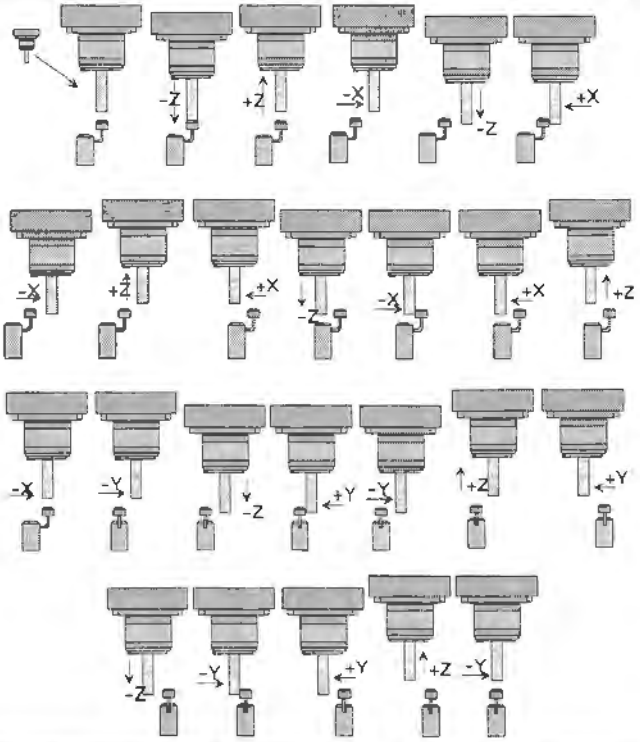
The tool probe can be calibrated for length and diameter measurements at the same time by programming G68 P7 with a tool of precisely known length and diameter in the spindle. The tool used for calibration must be a smooth diameter since calibration is done with spindle stopped.

Ensure precision tool data is entered in the tool table. Also, enter zero values in tool table fields **X Probe Offset** and **Y Probe Offset** to ensure spindle center line positioning over the probe.

Note: Since the tool will position on all four sides of the probe for a full calibration, the tool diameter should be relatively small.

Touch, press or set the following	Comments
	
 M6 Txx G68 P7 Q1 A .1 R.1	Display must be either Production, Current, To Go or Program. Txx is the arbor used for calibration loaded into the spindle
	

Calibration Sequence with Arbor Tool



How Offsets Are Updated with G68

G68 computes tool length by subtracting the tool probe location (found during the most recent calibration cycle) from the measured tool tip location. The tool probe location value is stored in the Cycle Parameter Table as field *fixed probe tram surface* FIX_PRB_TRAM. G68 determines Tool diameter by executing two probe hits on the side to the stylus, then the computed difference between two tool edge locations is subtracted from the probe diameter determined by the most recent probe calibration.

The tool data update performed by G68 can effect values in either the *Tool Data Table* or *Programmable Tool Offsets Table.* Specific actions of G68 are defined by the P word, which tables are updated is determined by H and O words. For example:

H0 or absent =

Tool length

Tool Data Table field Length is updated with the new measured tool length.

Tool Diameter

Tool Data Table field Diameter Offset will be updated. This value is calculated from the difference between measured tool diameter and the current Tool Data Table Nominal Diameter field value.

H5 programmed with O word

Tool length

The *Programmable Tool Offsets Table* column Length (record number is selected by the O word) will be updated. The value placed in this field is calculated from the difference between measured tool length and the *Tool Data Table* Length field value.

Tool Diameter

The *Programmable Tool Offsets Table* column Diameter (record number is selected by the O word) will be updated. The value placed in this field is calculated from the difference between measured diameter and the *Tool Data Table* Nominal Diameter field value.

How G68 Positions The Spindle Over The Tool Probe

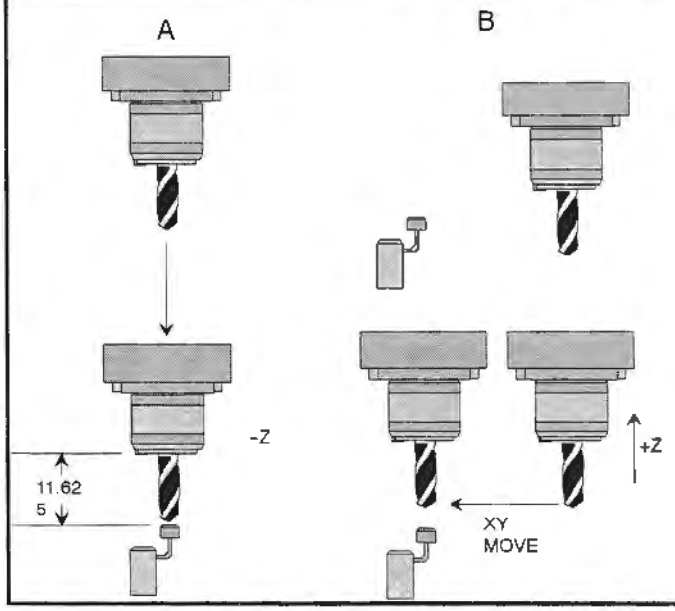
When G68 is executed, X and Y axes position in rapid traverse to the configured tool probe location. If there are no X and Y tool table probe offsets the spindle centerline is placed over the center of the probe. If there are X and Y tool table probe offsets, the spindle centerline is positioned away from the centerline of the tool probe by the these offsets.

Since the length of the tool in the spindle is to be measured, the tool tip location is uncertain before measurement. To protect the tool probe from being hit during XY moves, cycle parameter *Fixed Probe Clearance Height* FIX_PRB_CLR is provided. This Z axis retract is acted on when the probe radius, plus the tool radius, and the spindle position are less then the *Fixed Probe Clearance Height* above the top of the fixed probe. In this case, Z axis retracts to the *Fixed Probe Clearance Height* before X and Y axis move. If the spindle is already positioned such that the tool is over the probe head, Z axis retract is not acted on.

The *Fixed Probe Clearance Height* should be set to the longest tool length that is to be used. This value is the distance from the tool tip to the spindle face. For example, if the longest tool anticipated is 292.1mm (11.5 inches) in length, enter a slightly larger value 295.275mm (11.625) inches. Then, G68 and G69 will rapid the spindle nose this distance plus cycle parameter *Probe Gage Height* above the fixed probe, thus ensuring adequate clearance.

Note: Fixed Probe Clearance Height is measured from the spindle nose and must contain a value. This value is usually established during tool setup. However, the following syntax could be used to write to Cycle Parameter FIX_PRB_CLR.

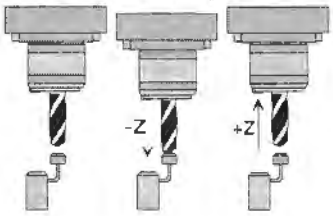
[$CYCLE_PARAMS(2)FIX_PRB_CLR]=11.625

Setting Fixed Probe Clearance Height	Comments
	Fixed Probe Clearance Height is a value. In example A, since the spindle is already positioned over the probe, the spindle rapids to *Probe Gage Height*. In example B, the tool is below the probe. Z axis retracts, then X and Y axis position over the probe.

Setting the Tool Length Offset using the Current Spindle Position

By using G68 with L1 you can update a Tool Length Offset from the current spindle position. To use this feature the tool **MUST** be positioned over the probe.

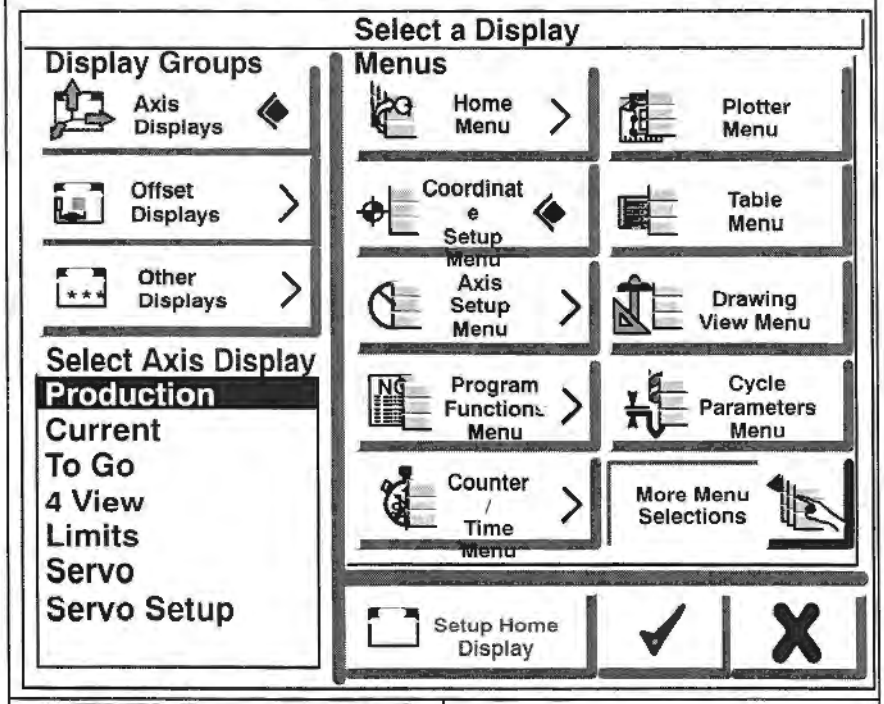
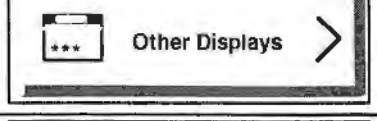
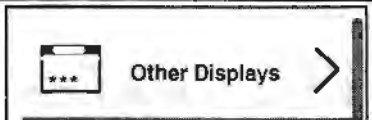
Note: When L1 is specified, X and Y words may not be programmed. Also, the L word is ignored for diameter only measurements (G68 P2)

Touch, press or set the following	Comments
	
 M6 Txx G68 L1	Display must be either Production, Current, To Go or Program. Txx is the arbor used
	
Action Sequence 	Z axis feeds to contact the probe. Tool length measurement is performed with the spindle stopped. Two hits of the probe take place. Tool Data Table **Length** field is updated.

How Does G69 Checks Tool Tolerance

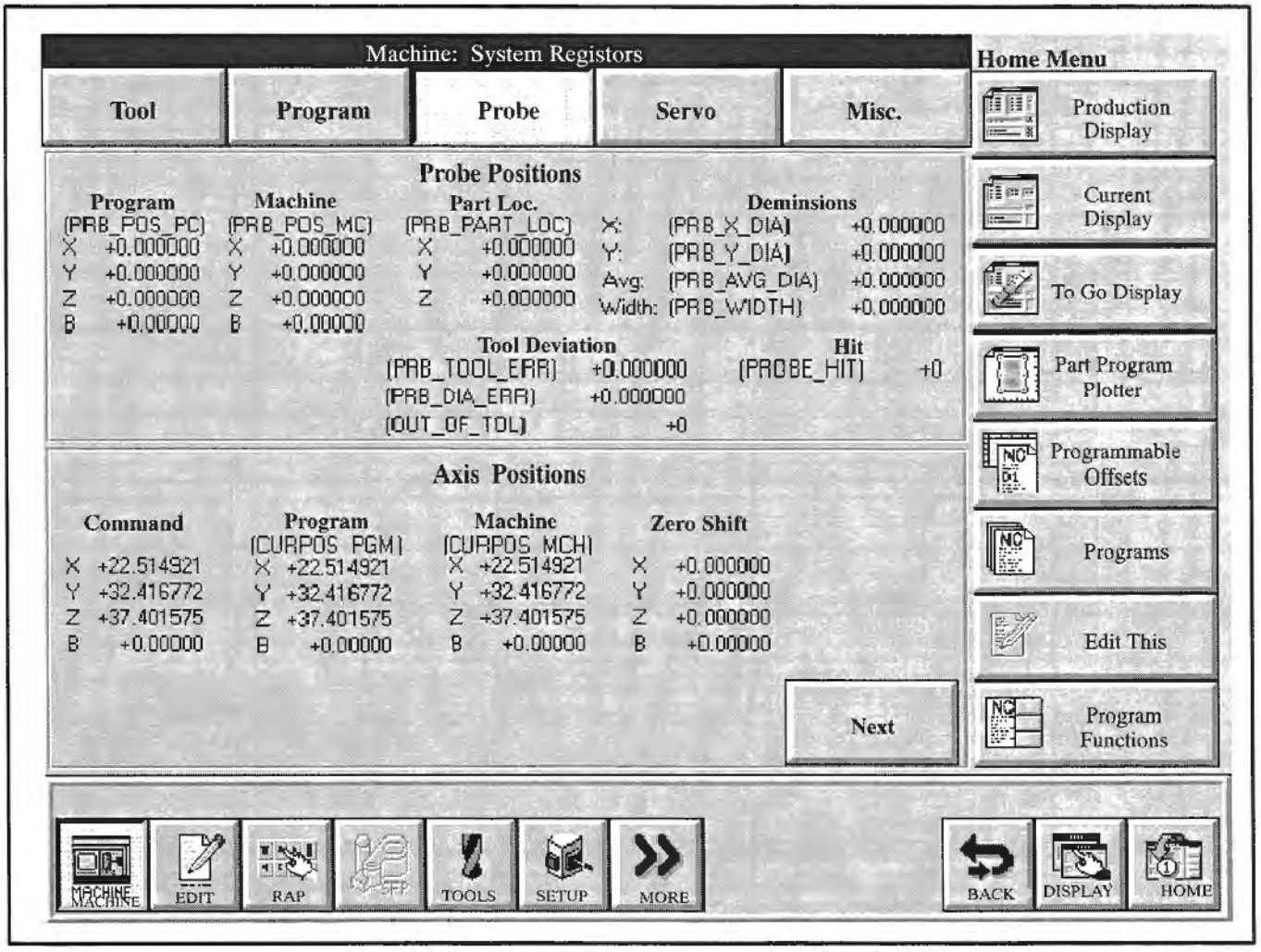
When G69 measurements are completed, length is checked against the tolerance specified by K and Z words, and diameter is checked against the tolerance specified by U and V words. If any words are omitted in the G69 block, default setting **G69 Dia/Length Tolerance** is used.

When G69 completes a measurement cycle, two fields [$PRB_TOOL_ERR] (for tool length error), and [$PRB_DIA_ERR] (for tool diameter error) will contain the error data. You can view this data by selecting the System Registers Probe field as follows:

<table>
<tr><th colspan="2">View Tool Tolerance Data</th></tr>
<tr><th>Touch, press or set the following</th><th>Comments</th></tr>
<tr><td> 
</td><td></td></tr>
<tr><td> 
</td><td></td></tr>
<tr><td colspan="2">The following display will appear:

</td></tr>
<tr><td> 
</td><td></td></tr>
<tr><td> 
</td><td>Touch to change</td></tr>
</table>

View Tool Tolerance Data (continued)	
Program System Registers Variables View Drawing Plotter Cycle Parameter Program Parameter Process Control Da	Touch to highlight

Touch **Probe**.



Under **Tool Deviation** the system registers used are as follows:

[$PRB_TOOL_ERR] This value is the error computed between tool length measurement, and the Tool Data Table **Length** field value.

[$PRB_DIA_ERR] This value is the error computed between tool diameter measurement and the Tool Data Table **Nominal Diameter** field value.

[$OUT_OF_TOL] Is set to 1 when the measured error exceeds the tolerance boundary. When set (1), the E word programmed in the G69 block is acted on.

[$PROBE_HIT] Is set to 1 when a probe hit occurs.

How G69 Determines What Action To Take

Just like G68, the type of measurement G69 will use is specified by the P word. After measurement, the action that is taken is determined by the E word selection. The table below is a quick reference for each E word selection. These selections are repeated in each G69 write-up later in this document.

P0, P1, P2, P3, P4	System Register	E0 or Absent	E1	E2
Measured Length Is Within Tolerance	Set [$PRB_TOOL_ERR] to measured error	Update Tool Length in Tool Table continue program execution	No Tool Table update continue program execution	No Tool Table update continue program execution
Measured Length Is Out Of Tolerance Too Short	Set [$PRB_TOOL_ERR] to measured error. Set [$OUT_OF_TOL] =1	Report an Alarm stop program execution Tool Table is updated. Tool Status is marked Worn	Jump to label set by most recent ATR block	Report an Alarm, stop program execution Tool Table is Not updated. Tool Status is marked Worn
Measured Length Is Out Of Tolerance Too Long	Set [$PRB_TOOL_ERR] to measured error. Set [$OUT_OF_TOL] =1	Report an Alarm stop program execution Tool Table is Not updated	Jump to label set by most recent ATR block	Report an Alarm, stop program execution Tool Table is Not updated. Tool Status is marked Worn
Tool Did Not Hit Probe	Set [$OUT_OF_TOL] =1	Report an Alarm stop program execution Tool Table is updated. Tool Status is marked Broken	Jump to label set by most recent ATR block.	Report an Alarm, stop program execution Tool Table is Not updated. Tool Status is marked Broken
Measured Diameter Is Within Tolerance	Set [$PRB_DIA_ERR] to measured deviation	Update Diameter Offset in Tool Table, continue program execution	No update continue program execution	No update continue program execution
Measured Diameter Is Out Of Tolerance Too Small	Set [$PRB_DIA_ERR] to measured deviation. Set [$OUT_OF_TOL] =1	Report an Alarm stop program execution Tool Table is Not updated	Jump to label set by most recent ATR block	Report an Alarm, stop program execution Tool Table is Not updated. Tool Status is marked Worn
Measured Diameter Is Out Of Tolerance Too Large	Set [$PRB_DIA_ERR] to measured deviation. Set [$OUT_OF_TOL] =1	Report an Alarm stop program execution Tool Table is Not updated	Jump to label set by most recent ATR block	Report an Alarm, stop program execution Tool Table is Not updated. Tool Status is marked Worn

Automatic Tool Recovery (ATR) (Option)

The Automatic Tool Recovery (or ATR type II block) provides a means to specify a section of the NC program designated to handle an exception condition such as a broken tool detected by a G69 probe cycle.

The effect of the ATR block is to define the label specified in the ATR block as the active exception handler. If any subsequent exception is reported (such as a broken tool) the NC program execution transfers to the ATR-specified label. The exception handler typically determines whether there is any feasible recovery and either attempts the recovery or aborts the program. Recovery strategies typically include loading an alternate tool and re-machining the portion of the part that was machined with the defective tool.

When an exception handler receives control, system register [$EXCEPTION] will contain a number that specifies the cause of the exception. Exceptions are shown in the table below.

[$EXCEPTION] Value	Meaning
0	No Exception Present
1	Worn Tool
2	Broken Tool
3	Oversize Tool
4	Wear Limit Exceeded

You can veiw the exception value as follows:

From the Home menu touch **DISPLAY.**

Touch **Other Displays.**

Touch **System Registers** to highlight. The touch the **Green** check button.

Touch **Program.**

Do I Have the ATR Option

To find out if you have the ATR option proceed as follows:

From the **HOME** menu touch **MORE.**

Touch **System Configuration.**

Touch **Purchased Options**

Use down arrow buttons to select **Automatic Tool Recovery** option. The Authorization field should display **Permanent**, and the Enabled field should display **Yes.**

ATR Simple Program

The following simple program briefly shows the function of an ATR block. For in depth information refer to your programming manual (ATR) Automatic Tool Recovery.

```
: T1 M6 ; Selects tool 1 for measurement
(ATR, L [MEASURE_TOOL]); Defines ATR block label
G91 Z10 ; Set incremental move to Z axis location
X-5 Y-6 ; move X and Y axis
G69 P0 E1 ; check tool length
[#OPERATION]=1
[OP1]
G91 Z10 ; move Z axis out of part
X-3 Y-3 ; move X and Y axis
M2 ; end program
[MEASURE_TOOL] ; ATR jump label
(IF [$EXCEPTION]=1 THEN) ; if exception handler sets tool worn jump
to OP1 and execute program
(GOTO[OP1])
(ENDIF)
```

G69 Tool Setter Probe General Information

Measured tool length tolerance can be viewed on the System Register menu [$PRB_TOOL_ERR].

Measured tool diameter tolerance can be viewed on the System Register menu [$PRB_DIA_ERR].

Tool diameter measurements are always made with the tool rotating.

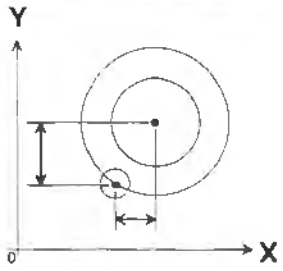
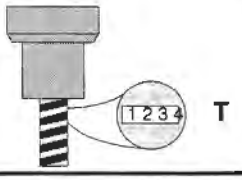
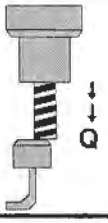
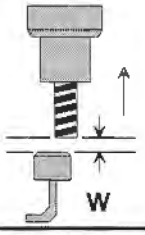
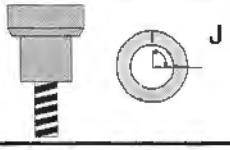
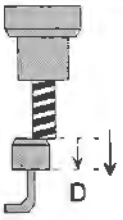
G69 cycles are Non-modal G codes, that is, they are active only in the block in which they are programmed and need not be replaced with a G1 or G0 to move execution.

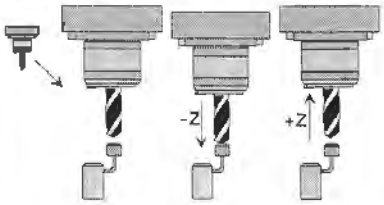
If either length or diameter tolerance words are omitted in the G69 block, the **G69 Dia/Length Tolerance** value in the Machine Application TOOL PROBE menu is used as the default tolerance.

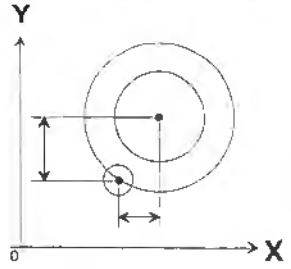
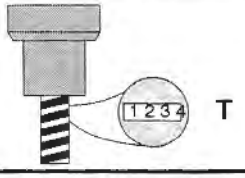
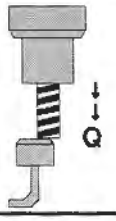
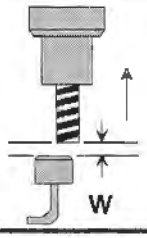
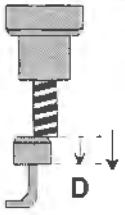
The E word defines what program action is taken.

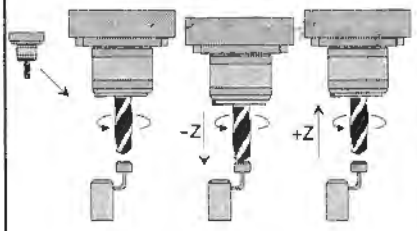
G69 Tool Probe Operations can be generated (in MDI), by using Resident Assistant Programmer (RAP), or by NC program.

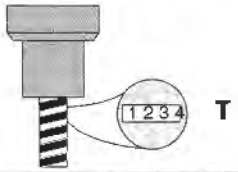
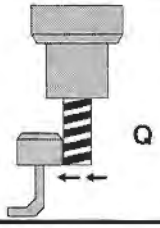
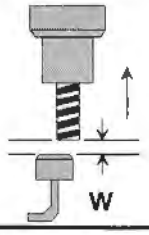
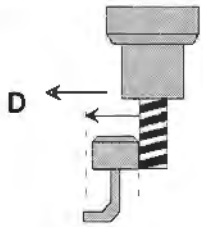
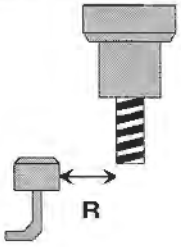
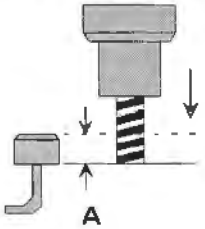
G68 - Set Tool Size Program Information

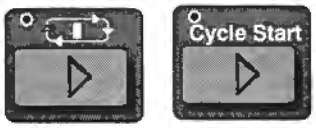
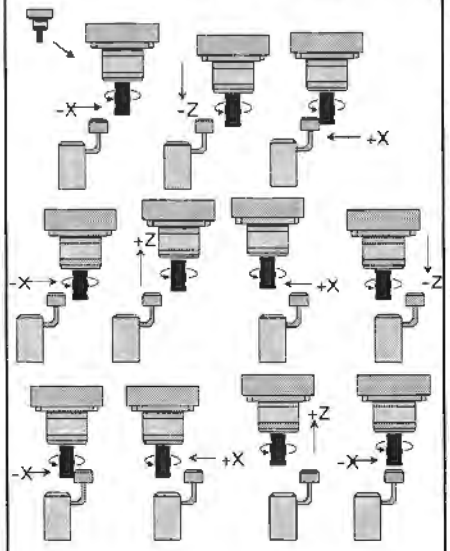
G68 Set Length P0 Spindle Stopped Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**J** is the optional nonmodal orient angle used to specify the angle at which the tool is to be measured; the default angle is zero.
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

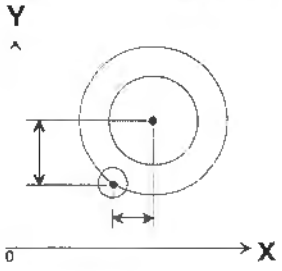
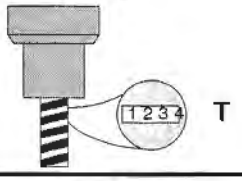
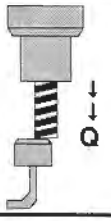
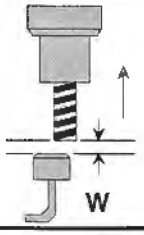
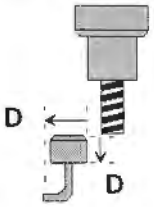
G68 P0 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle.	
Touch, press or set the following	**Comments**
	
 G68 P0 Q1 (Example)	Display must be either Production, Current, To Go or Program.
	
Action Sequence 	Z axis positions over the probe. Since no X Y offset is programmed, the X Y Probe Offset values from the Tool Table determine positioning over the probe. The *Tool Data Table* **Length** field will be updated because no H word is programmed. Z feeds to contact the probe. Q1 defines single hit. Z axis returns to the probe clearance position.

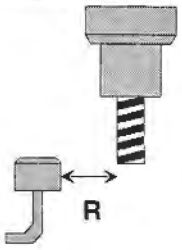
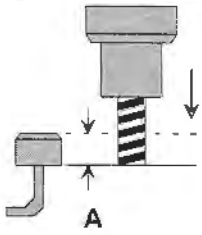
G68 Set Length P1 Spindle Rotating Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

G68 P1 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle.	
Touch, press or set the following	**Comments**
	
 G68 P1 Q0 (Example)	Display must be either Production, Current, To Go or Program.
	
Action Sequence 	G68 P1 rotates in the opposite direction from normal cutting. The spindle direction recorded in the Tool Data Table is used to determine the direction to rotate the spindle. Z axis positions over the probe. Since no X Y offset is programmed, the X Y Probe Offset values from the Tool Table determine positioning over the probe. The *Tool Data Table* **Length** field will be updated because no H word is programmed. Z feeds to contact the probe. Q0 defines two hits. Z axis returns to the probe clearance position.

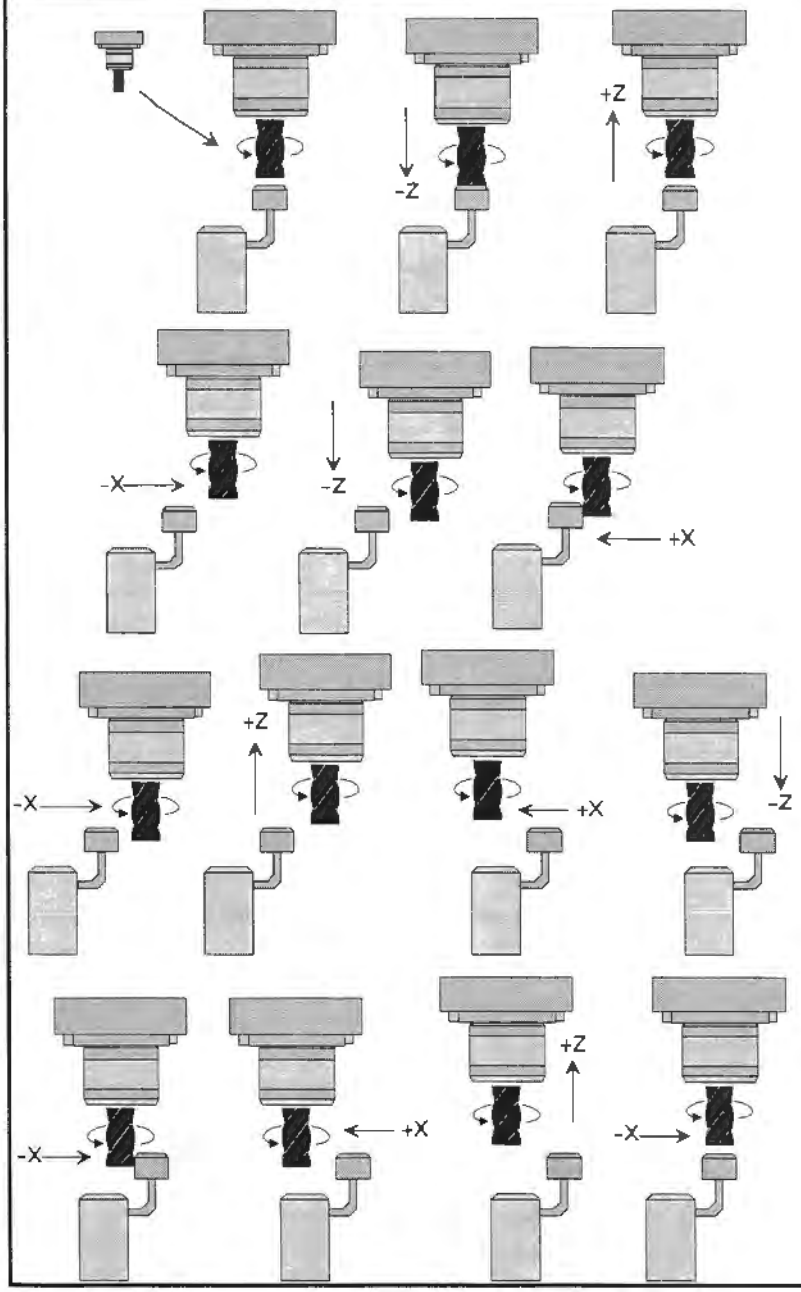
G68 Set Tool Diameter P2 Spindle Rotating Program Words	
Word Description	**Comments**
	T is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal distance from the top of the probe to the position used to measure the tool diameter. The G68 cycle moves the tool from the clearance position by A, plus cycle parameter *Probe Gage Height* to make the diameter measurements. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

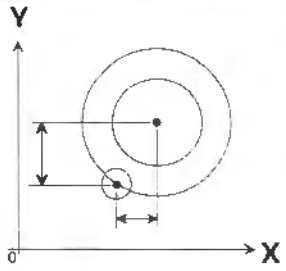
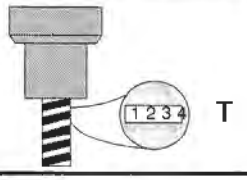
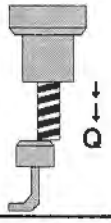
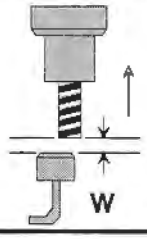
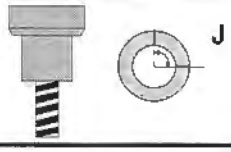
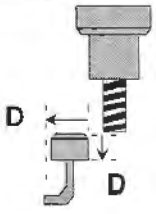
G68 P2 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle. Also, X axis measure tool diameter is active.	
Touch, press or set the following	**Comments**
	
 G68 P2 Q0 R.1 A.1 (Example)	Display must be either Production, Current, To Go or Program.
	
Action Sequence 	G68 P2 rotates in the opposite direction from normal cutting. Move X and Y axis to clearance distance next to the probe. This distance is determined by the Tool Radius + Probe Radius + R word. Z axis moves in the minus direction. This depth is determined by the A word + *Probe Gage Height*. The *Tool Data Table* **Diameter Offset** field will be updated because no H word is programmed. Z axis returns to the probe clearance position. Two hits are used to contact the probe. Z axis returns to the probe clearance position.

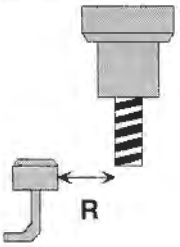
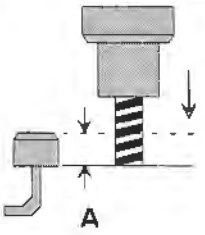
G68 Set Tool Length & Diameter P3 Spindle Rotating Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. The tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). For diameter measurement, this is the distance the edge of the tool (determined from the tool table nominal diameter) is permitted to move into the calibrated tool probe radius. If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

G68 Set Tool Length & Diameter P3 Spindle Rotating Program Words (continued)	
Word Description	**Comments**
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal distance from the top of the probe to the position used to measure the tool diameter. The G68 cycle moves the tool from the clearance position by A, plus cycle parameter *Probe Gage Height* to make the diameter measurements. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

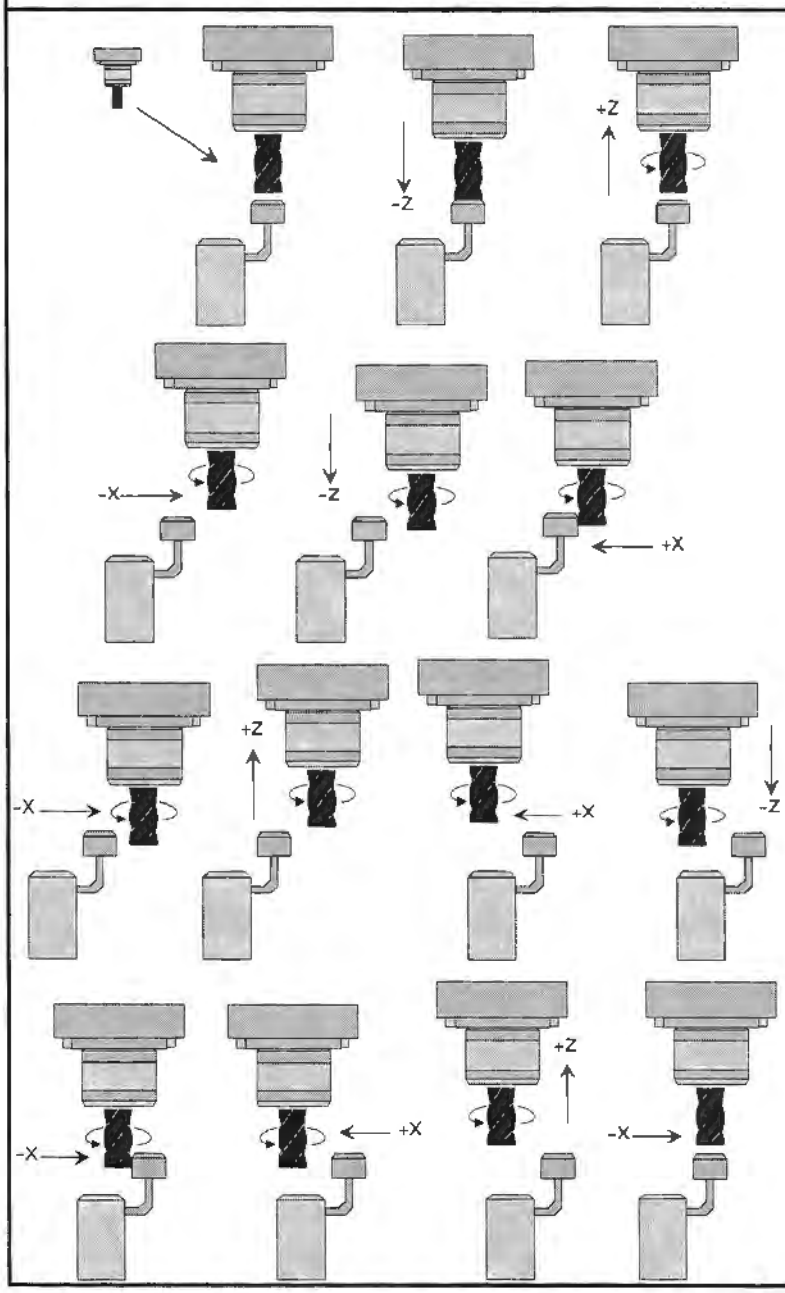
G68 P3 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle. Also, X axis measure tool diameter is active.	
Touch, press or set the following	**Comments**
	
 G68 P3 H5 O1 Q0 R.1 A.1 (Example)	Display must be either Production, Current, To Go or Program.
	

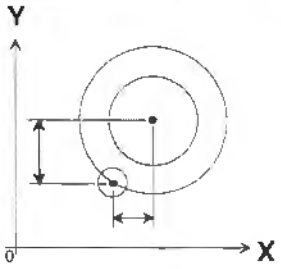
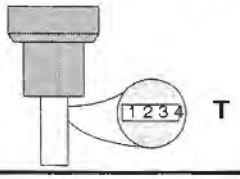
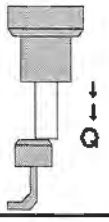
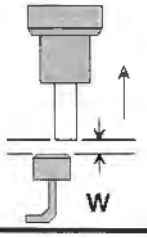
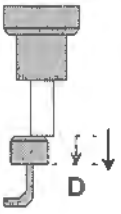
Action Sequence	Comments
	G68 P3 rotates in the opposite direction from normal cutting. Z axis positions over the probe. Since no X Y offset is programmed, the X Y Probe Offset values from the Tool Table determine positioning over the probe. Tool length measurement is performed first then diameter measurements. Move X and Y axis to clearance distance next to the probe. This distance is determined by the Tool Radius + Probe Radius + R word. Z axis moves in the minus direction. This depth is determined by the A word + *Probe Gage Height*. The *Programmable Tool Offsets Table* record O1 **Length** and **Diameter** field will be updated. Two hits are used to contact the probe. Z axis returns to the probe clearance position.

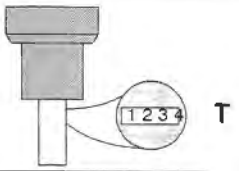
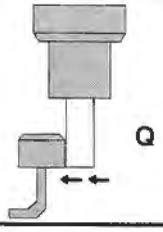
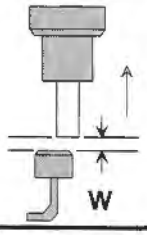
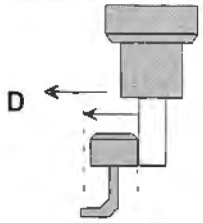
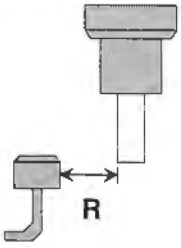
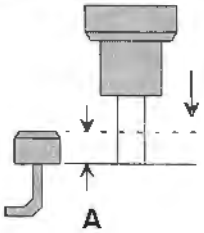
G68 Set Tool Length Spindle Stopped & Diameter P4 Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**J** is the optional nonmodal orient angle used to specify the angle at which the tool is to be measured; the default angle is zero.
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. The tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). For diameter measurement, this is the distance the edge of the tool (determined from the tool table nominal diameter) is permitted to move into the calibrated tool probe radius. If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

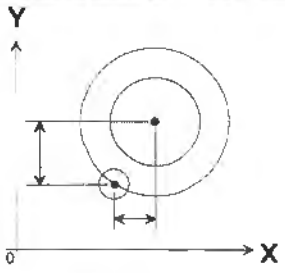
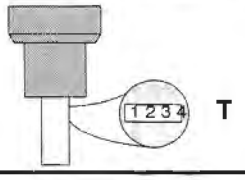
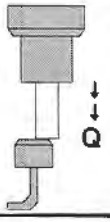
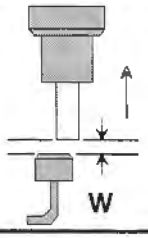
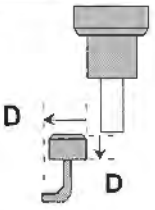
G68 Set Tool Length Spindle Stopped & Diameter P4 Program Words (continued)	
Word Description	**Comments**
	R is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	A is the nonmodal distance from the top of the probe to the position used to measure the tool diameter. The G68 cycle moves the tool from the clearance position by A, plus cycle parameter *Probe Gage Height* to make the diameter measurements. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

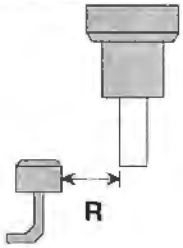
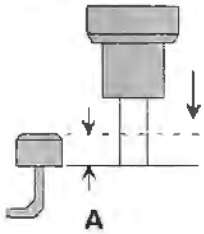
G68 P3 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle. Also, X axis measure tool diameter is active.	
Touch, press or set the following	**Comments**
	
 G68 P4 H5 O1 Q0 R.1 A.1 (Example)	Display must be either Production, Current, To Go or Program.
	

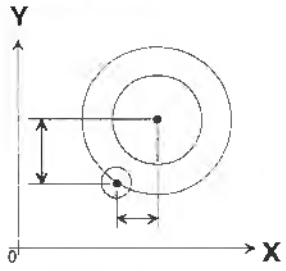
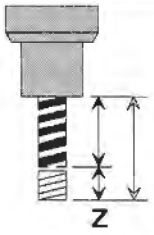
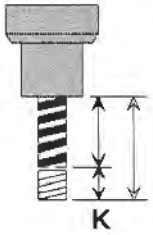
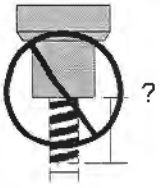
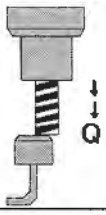
Action Sequence	Comments
	For diameter measurements G68 P4 rotates in the opposite direction from normal cutting. Z axis positions over the probe. Since no X Y offset is programmed, the X Y Probe Offset values from the Tool Table determine positioning over the probe. Tool length measurement is performed first then diameter measurements. Move X and Y axis to clearance distance next to the probe. This distance is determined by the Tool Radius + Probe Radius + R word. Z axis moves in the minus direction. This depth is determined by the A word + *Probe Gage Height*. The *Programmable Tool Offsets Table* record O1 **Length** and **Diameter** field will be updated. Two hits are used to contact the probe. Z axis returns to the probe clearance position.

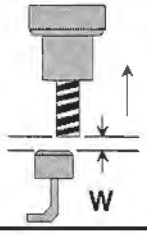
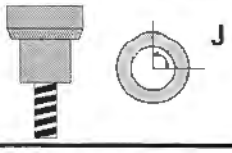
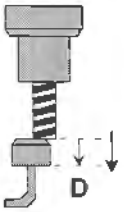
G68 Calibrate Probe Length With Tool In Spindle P5 Spindle Stopped Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For calibration, this is the distance the tool tip is permitted to move below the probe location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

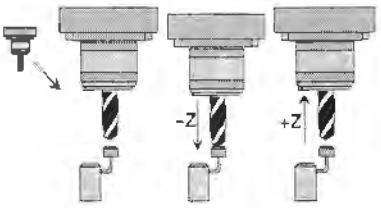
G68 Calibrate Probe Diameter With Tool in Spindle P6 Spindle Stopped Program Words	
Word Description	**Comments**
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal distance from the top of the probe to the position used to measure the tool diameter. The G68 cycle moves the tool from the clearance position by A, plus cycle parameter *Probe Gage Height* to make the diameter measurements. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

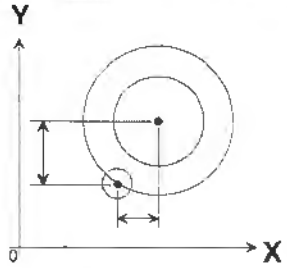
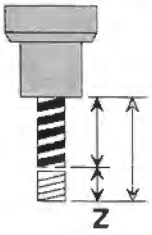
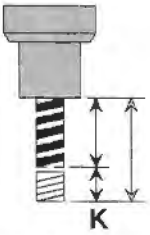
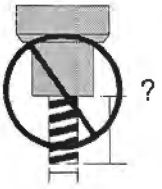
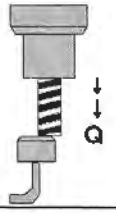
G68 Calibrate Probe Length and Diameter With Tool in Spindle P7 Spindle Stopped Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**T** is the optional tool identifier specifying the next tool to be measured.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. The tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). For diameter calibration, this is the distance the edge of the tool (determined from the tool table nominal diameter) is permitted to move into the calibrated tool probe radius. If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

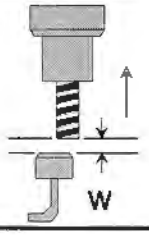
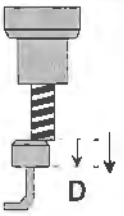
G68 Calibrate Probe Length and Diameter With Tool in Spindle P7 Spindle Stopped Program Words (continued)	
Word Description	**Comments**
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal distance from the top of the probe to the position used to measure the tool diameter. The G68 cycle moves the tool from the clearance position by A, plus cycle parameter *Probe Gage Height* to make the diameter measurements. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

G69 Check Length P0 Spindle Stopped Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**Z** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *shorter* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**K** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *longer* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**E0** = Update Length Offset value and set Tool Worn if the tool is too short. An alarm is displayed and cycle halts if the tool length exceeds the tolerance specified by Z and K word values or **G69Dia/Length Tolerance** valne. **E1** = Proceed *without updating* the Tool Length and Jump to the program label specified by the most recent ATR block if one was programmed. **E2** = Proceed *without updating* the Tool Length and Set Tool Worn if the tool is too short, or set Tool Broken if no probe hit occurs. An alarm is displayed and cycle halts if the tool length is outside of the tolerance specified by Z and K words or **G69Dia/Length Tolerance** value.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.

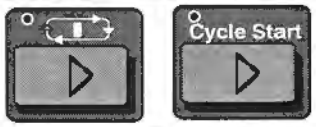
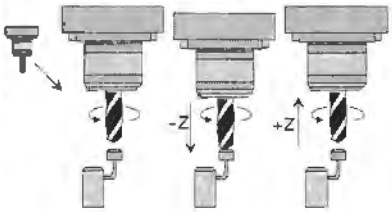
G69 Check Length P0 Spindle Stopped Program Words (continued)	
Word Description	**Comments**
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**J** is the optional nonmodal orient angle used to specify the angle at which the tool is to be measured; the default angle is zero.
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

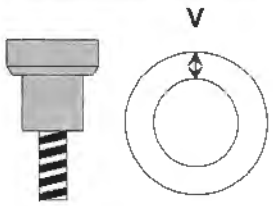
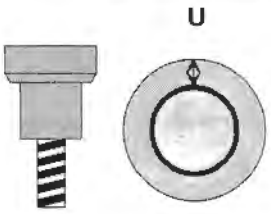
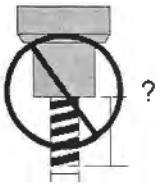
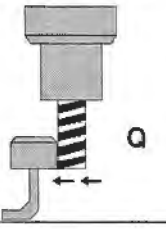
G69 P0 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle.	
Touch, press or set the following	**Comments**
	
 G69 P0 Q1 Z .002 K .001 E0 (Example)	Display must be either Production, Current, To Go or Program.
	
Action Sequence 	Z axis positions over the probe. Since no X Y offset is programmed, the X Y Probe Offset values from the Tool Table determine positioning over the probe. Z feeds to contact the probe. Q1 defines single hit. Z axis returns to the probe clearance position. The *Tool Data Table* **Length** field will be updated because no H word is programmed. The measured tolerance is displayed in System Register field [$PRB_TOOL_ERR]. E0 defines the following action: If measurement is within tolerance, update Tool Data Table, Tool Length If out of tolerance, System Register [$OUT_OF_TOL] is set to 1. An Alarm is reported and program execution is stopped.

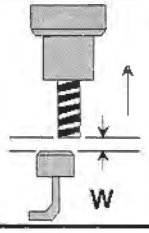
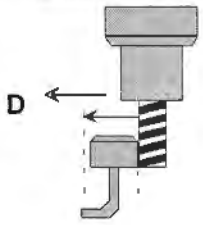
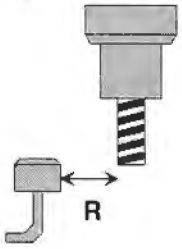
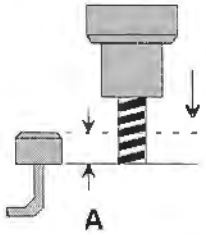
G69 Check Length P1 Spindle Rotating Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**Z** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *shorter* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**K** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *longer* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**E0** = Update Length Offset value and set Tool Worn if the tool is too short. An alarm is displayed and cycle halts if the tool length exceeds the tolerance specified by Z and K word values or **G69Dia/Length Tolerance** value. **E1** = Proceed *without updating* the Tool Length and Jump to the program label specified by the most recent ATR block if one was programmed. **E2** = Proceed *without updating* the Tool Length and Set Tool Worn if the tool is too short, or set Tool Broken if no probe hit occurs. An alarm is displayed and cycle halts if the tool length is outside of the tolerance specified by Z and K words or **G69Dia/Length Tolerance** value.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.

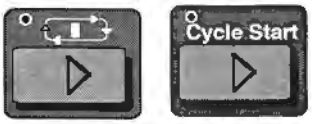
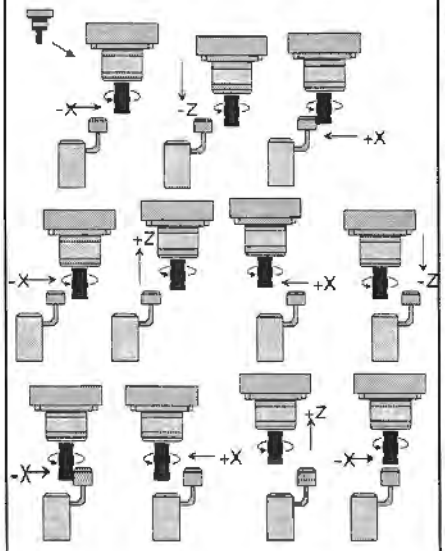
G69 Check Length P1 Spindle Rotating Program Words (continued)	
Word Description	**Comments**
	W is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	D is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. For tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

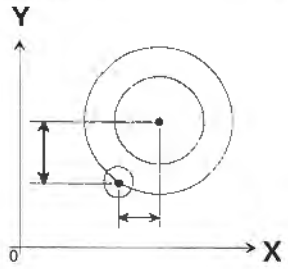
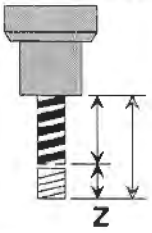
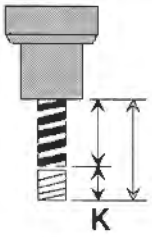
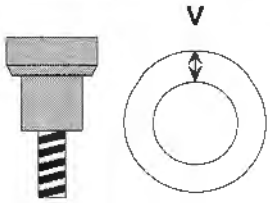
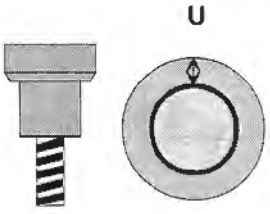
G69 P1 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle.	
Touch, press or set the following	**Comments**
	
 G69 P1 Q1 Z .002 K .001 E0 (Example)	Display must be either Production, Current, To Go or Program.

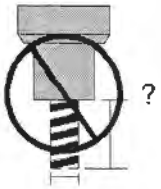
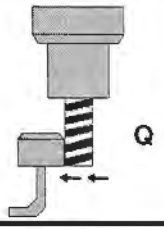
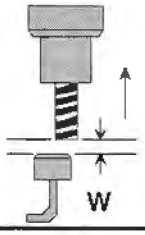
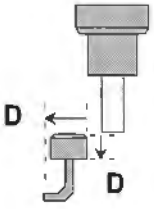
G69 P1 Cycle Action (continued)	
Touch, press or set the following	**Comments**
	
Action Sequence 	Z axis positions over the probe. Since no X Y offset is programmed, the X Y Probe Offset values from the Tool Table determine positioning over the probe. Z feeds to contact the probe. Q1 defines single hit. Z axis returns to the probe clearance position. The *Tool Data Table* **Length** field will be updated because no H word is programmed. The measured tolerance is displayed in System Register field [$PRB_TOOL_ERR]. E0 defines the following action: If measurement is within tolerance, update Tool Data Table, Tool Length If measured tolerance is too short, System Register [$OUT_OF_TOL] is set to 1. An Alarm is reported and program execution is stopped. If measured tolerance is too long, System Register [$OUT_OF_TOL] is set to 1. An Alarm is reported and program execution is stopped.

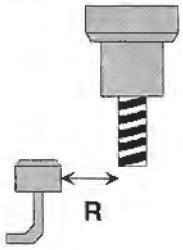
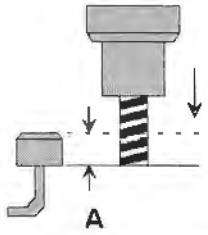
G69 Check Diameter P2 Spindle Rotating Program Words	
Word Description	**Comments**
	**V** is the optional unsigned cycle modal tolerance value by which the G69 diameter measurement is permitted to be *smaller* than the Nominal Diameter in the Tool Table. That is the amount the tool is permitted to be undersize. If this word is not programmed. Machine Application **G69Dia/Length Tolerance** value is used.
	**U** is the optional unsigned cycle modal tolerance value by which the G69 diameter measurement is permitted to be *smaller* than the Nominal Diameter in the Tool Table. That is the amount the tool is permitted to be undersize. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**E0** = Update Diameter Offset value and set Tool Worn if the tool is too short. An alarm is displayed and cycle halts if the checked tool exceeds the tolerance specified by U, and V words or **G69Dia/Length Tolerance** value. **E1** = Proceed *without updating* the Tool Diameter Offset and Jump to the program label specified by the most recent ATR block if one was programmed. **E2** = Proceed *without updating* the Tool Diameter Offset and Set Tool Worn if the tool is undersize, or set Tool Broken if no probe hit occurs. An alarm is displayed and cycle halts if the measured tool is outside of the tolerance specified by U, and V words **G69Dia/Length Tolerance** value.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.

G69 Check Diameter P2 Spindle Rotating Program Words (continued)	
Word Description	**Comments**
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. This is the distance the edge of the tool (determined from the tool table nominal diameter) is permitted to move into the calibrated tool probe radius. If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal incremental depth is the distance from the Z plane clear of the tool to the position used to check the tool diameter. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

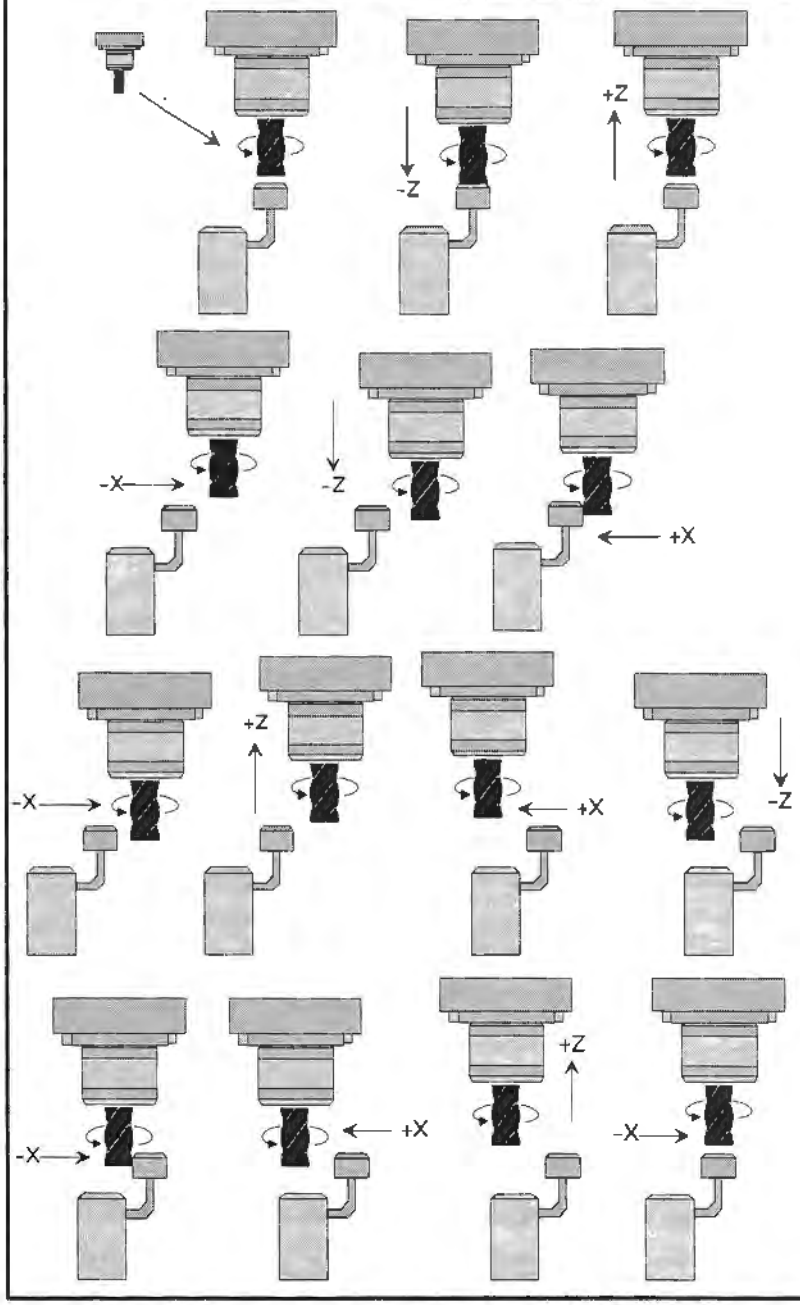
G69 P2 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle. Also, X axis measure tool diameter is active.	
Touch, press or set the following	**Comments**
	
 G69 P2 Q0 V .0001 U .0001 E0 R.1 A.1 (Example)	Display must be either Production, Current, To Go or Program.
	
Action Sequence 	G68 P2 rotates in the opposite direction from normal cutting. Move X and Y axis to clearance distance next to the probe. This distance is determined by the Tool Radius + Probe Radius + R word. Z axis moves in the minus direction. This depth is determined by the A word + *Probe Gage Height*. Two hits are used to contact the probe. Z axis returns to the probe clearance position. The checked tolerance is displayed in System Register field [$PRB_DIA_ERR]. E0 defines the following action: If measurement is within tolerance, update Tool Data Table, **Diameter Offset.** If out of tolerance, System Register [$OUT_OF_TOL] is set to 1. An Alarm is reported and program execution is stopped.

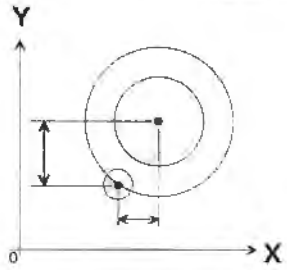
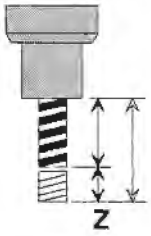
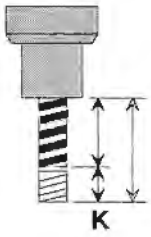
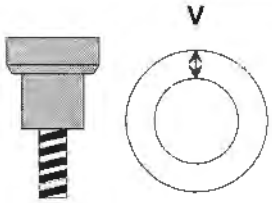
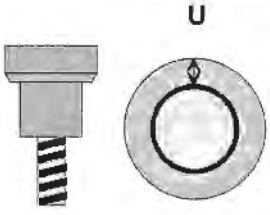
G69 Check Tool Length & Diameter P3 Spindle Rotating Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**Z** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *shorter* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**K** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *longer* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**V** is the optional unsigned cycle modal tolerance value by which the G69 diameter measurement is permitted to be *smaller* than the Nominal Diameter in the Tool Table. That is the amount the tool is permitted to be undersize. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**U** is the optional unsigned cycle modal tolerance value by which the G69 diameter measurement is permitted to be *smaller* than the Nominal Diameter in the Tool Table. That is the amount the tool is permitted to be undersize. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.

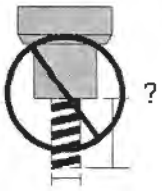
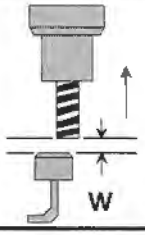
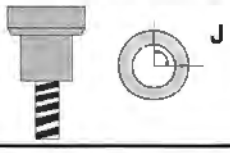
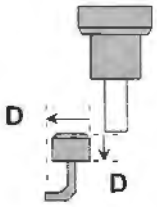
G69 Check Tool Length & Diameter P3 Spindle Rotating Program Words (continued)	
Word Description	**Comments**
	**E0** = Update Length Offset value and Diameter Offset value. An alarm is displayed and cycle halts if the measurement exceeds the tolerance specified by Z, K, V, and U words or **G69Dia/Length Tolerance** value. **E1** = Proceed *without updating* and Jump to the program label specified by the most recent ATR block if one was programmed. **E2** = Proceed *without updating* the Tool Length and Diameter offset value and Set Tool Worn if out of tolerance, or set Tool Broken if no probe hit occurs. An alarm is displayed and cycle halts if the tool length is outside of the tolerance specified by Z, K, V, and U words or **G69Dia/Length Tolerance** value.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. The tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). For diameter measurement, this is the distance the edge of the tool (determined from the tool table nominal diameter) is permitted to move into the calibrated tool probe radius. If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

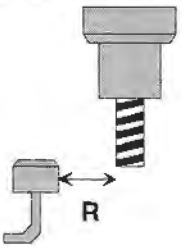
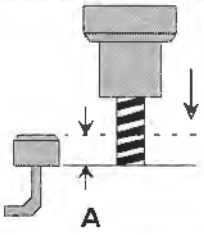
G69 Check Tool Length & Diameter P3 Spindle Rotating Program Words (continued)	
Word Description	**Comments**
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal incremental depth is the distance from the Z plane clear of the tool to the position used to check the tool diameter. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

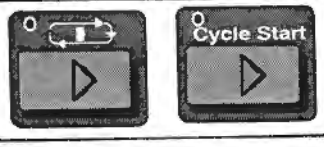
G69 P3 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle. Also, X axis measure tool diameter is active.	
Touch, press or set the following	**Comments**
	
 G68 P3 Q0 Z .001 K .001 V .0001 U .0001 R.1 A.1 E0 (Example)	Display must be either Production, Current, To Go or Program.
	

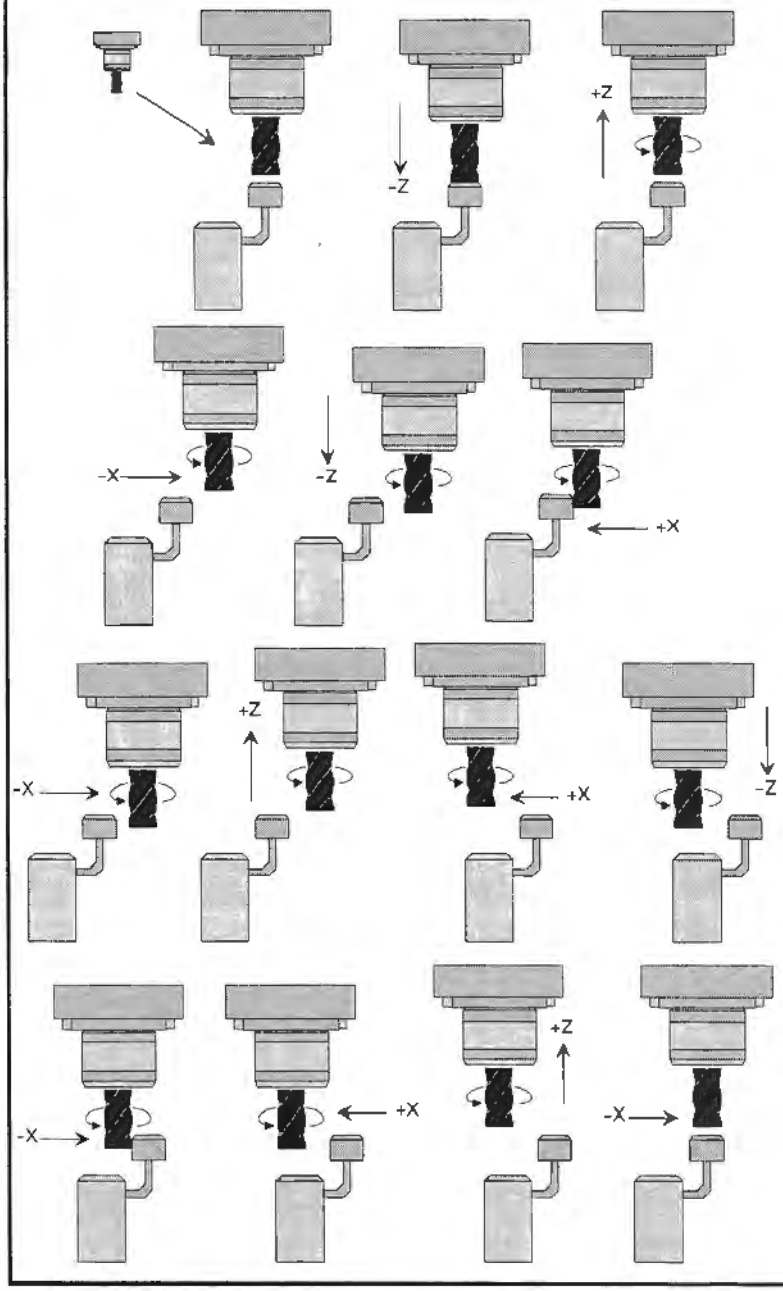
Action Sequence	Comments
	G69 P3 rotates in the opposite direction from normal cutting. Tool length measurement is performed first then diameter measurements. Z axis positions over the probe. Two hits are used to contact the probe. Z axis returns to the probe clearance position. Move X and Y axis to clearance distance next to the probe. This distance is determined by the Tool Radius + Probe Radius + R word. Move Z axis down by the A word amount. The length tolerance is displayed in System Register field [$PRB_TOOL_ERR]. The diameter tolerance is displayed in System Register field [$PRB_DIA_ERR]. E0 defines the following action: If length is within tolerance, update Tool Data Table, Tool **Length** If diameter is within tolerance, update Tool Data Table, **Diameter Offset.** If length or diameter is out of tolerance, System Register [$OUT_OF_TOL] is set to 1. An Alarm is reported and program execution is stopped.

G68 Set Tool Length Spindle Stopped & Diameter P4 Program Words	
Word Description	**Comments**
	**X** and **Y** are the optional incremental offsets from the spindle centerline to the point on the tool to be measured. If either of these values are not present, **X and/or Y Probe Offsets** from the Tool Table is used If an Offset is programmed, the **X and/or Y Probe Offset** value in the Tool Table is updated with the value in the G68 block.
	**Z** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *shorter* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**K** is the optional unsigned cycle modal tolerance value by which the G69 length measurement is permitted to be *longer* than the Tool Length in the Tool Table. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**V** is the optional unsigned cycle modal tolerance value by which the G69 diameter measurement is permitted to be *smaller* than the Nominal Diameter in the Tool Table. That is the amount the tool is permitted to be undersize. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.
	**U** is the optional unsigned cycle modal tolerance value by which the G69 diameter measurement is permitted to be *smaller* than the Nominal Diameter in the Tool Table. That is the amount the tool is permitted to be undersize. If this word is not programmed, Machine Application **G69Dia/Length Tolerance** value is used.

G68 Set Tool Length Spindle Stopped & Diameter P4 Program Words (continued)	
Word Description	**Comments**
	**E0** = Update Length Offset value and Diameter Offset value. An alarm is displayed and cycle halts if the measurement exceeds the tolerance specified by Z, K, V, and U words or **G69Dia/Length Tolerance** value. **E1** = Proceed *without updating* and Jump to the program label specified by the most recent ATR block if one was programmed. **E2** = Proceed *without updating* the Tool Length and Diameter offset value and Set Tool Worn if out of tolerance, or set Tool Broken if no probe hit occurs. An alarm is displayed and cycle halts if the tool length is outside of the tolerance specified by Z, K, V, and U words or **G69Dia/Length Tolerance** value.
	**Q0** or absent Double Hit **Q1** Single Hit Note: Rotating tool measurements are always done with two hits.
	**W** is the optional Nonmodal final retract distance (overrides the default final Z axis position).
	**J** is the optional nonmodal orient angle used to specify the angle at which the tool is to be measured; the default angle is zero.
	**D** is the optional nonmodal probe overtravel distance allowed beyond the programmed motion. The tool length measurement, this is the distance the tool tip is permitted to move below the probe Z location (using the tool length present in the tool table before the measurement). For diameter measurement, this is the distance the edge of the tool (determined from the tool table nominal diameter) is permitted to move into the calibrated tool probe radius. If D is absent, the value of the *Probe Gage Height* in the cycle parameter table is used.

G68 Set Tool Length Spindle Stopped & Diameter P4 Program Words (continued)	
Word Description	**Comments**
	**R** is the radial clearance distance used to position the tool away from the probe stylus. The total clearance distance is determined by the Tool Radius + Probe Radius + R word. Note: the R word is required for all diameter measurements and is ignored for cycles that measure only length.
	**A** is the nonmodal incremental depth is the distance from the Z plane clear of the tool to the position used to check the tool diameter. Note: the A word is required for all diameter measurements and is ignored for cycles that measure only length.

G69 P4 Cycle Action	
it is assumed the machine is aligned, all tool table information is entered, and the tool to be measured is loaded in the spindle. Also, X axis measure tool diameter is active.	
Touch, press or set the following	**Comments**
	
 G68 P4 Q0 Z .001 K .001 V .0001 U .0001 R.1 A.1 E0 (Example)	Display must be either Production, Current, To Go or Program.
	

Action Sequence	Comments
	For diameter measurements G69 P4 rotates in the opposite direction from normal cutting. Tool length measurement is performed first then diameter measurements. Z axis positions over the probe. Two hits are used to contact the probe. Z axis returns to the probe clearance position. Start spindle rotation and move X and Y axis to clearance distance next to the probe. This distance is determined by the Tool Radius + Probe Radius + R word. Move Z axis down by the A word amount. The measured length tolerance is displayed in System Register field [$PRB_TOOL_ERR]. The measured diameter tolerance is displayed in System Register field [$PRB_DIA_ERR]. E0 defines the following action: If measurement length is within tolerance, update Tool Data Table, Tool **Length** If measurement diameter is within tolerance, update Tool Data Table, **Diameter Offset.** If length or diameter is out of tolerance, System Register [$OUT_OF_TOL] is set to 1. An Alarm is reported and program execution is stopped.

Index

Index