Section 15: Touch Probes

Touch Probes

Touch Probe – Length Offset

The tool setting probe is used with the UTILITY command to establish the length offset. It can also be used for tool breakage detection and setting tool diameter offsets. The UTILITY command is used to set diameter and locate the length offset. Enter the command UT and the following menu appears:
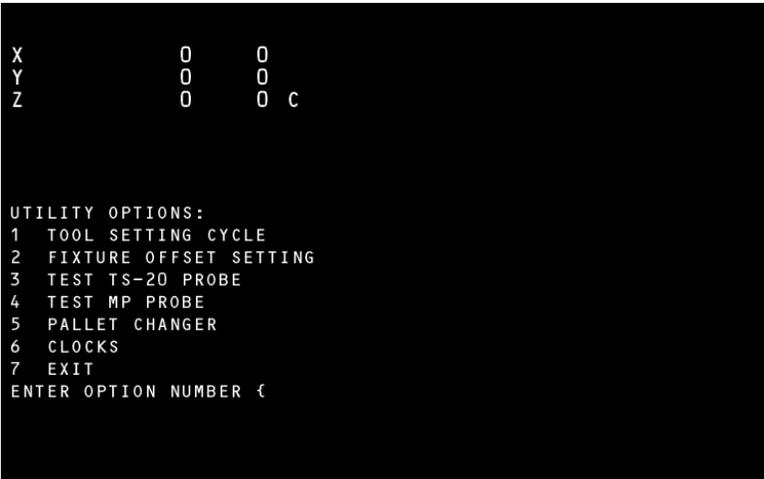
Figure 15-1 Utility Options Menu

Tool Setting Cycle

The utility command provides two basic functions to aid in the setup procedure. The first function steps the operator through the process of retrieving tools from the turret, entering the diameters and locating the length offsets.

After selecting this cycle the CNC displays the following menu:

Figure 15-2 Tool Setting Cycle Menu

The information regarding the current tool in the spindle is displayed and the CNC prompts the user for the starting tool number. After entering the first tool number, the CNC responds with the message:

Figure 15-3 Starting Tool Number Prompt

Respond by entering the last tool number to be set.

The following menu appears after the starting and ending tool numbers are defined:

```
X          0    0
Y          0    0
Z          0    0 C




TOOL SETTING OPTIONS:
1  JOG TO POSITION
2  PROBE LEFT MOUNT
3  PROBE RIGHT MOUNT
4  EXIT
PRESS OPTION NUMBER
```

Figure 15-4 Tool Setting Options Menu

Option 1=Jog to Position
This option is used for locating the tool length offset by using the JOG function of the CNC. The CNC prompts the operator to press the JOG key and manually move to the gauge point instead of the automatic process using the probe.

Option 2 & 3 - Probe Mount
The CNC allows for two probe mounting positions. The left or right orientation is defined as standing in front of the machine, looking Y+ direction.
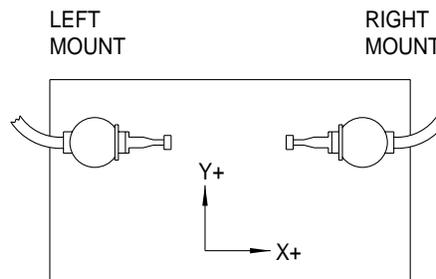

Figure 15-5 Probe Mounting Positions

Selecting left or right mount determines the X axis shift direction when locating the tool length of an end mill. The left mount probe shifts the X axis in the positive direction before moving Z down. The right mount shifts the X axis in the negative direction before moving Z down.

Probe Fixture Offset Number (0, 1-48) Once the probe is installed, a fixture offset can be dedicated to tool setting. The X and Y fixture offset value is the location of the center of the tool setter stylus relative to the HOME position.
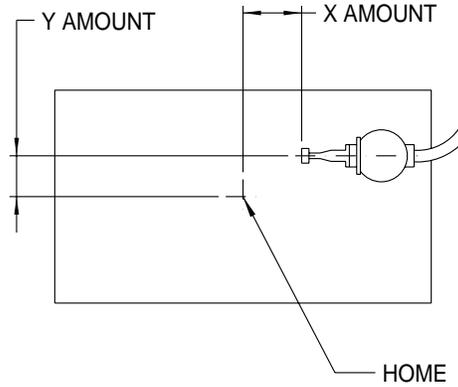

Figure 15-6 Probe Fixture Offset Number

Respond with 0 to ignore this feature or enter the fixture offset number.

**Probe Offsets**
Z Fixture Offsets
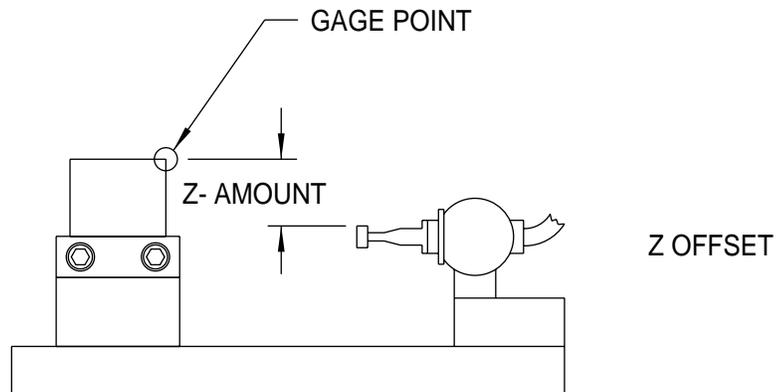
Figure 15-7 Z Fixture Offsets

After selecting a fixture offset the CNC displays the current X, Y, and Z values and prompts the user with the following message:

```
X             0    0
Y             0    0
Z             0    0 C

FIXTURE OFFSET = 3
  X= 0                    B= 0
  Y= 0                    A= 0
  Z= 0

THE Z FIXTURE OFFSET IS USED IN CALCULATING THE LENGTH OFFSET
DO YOU WANT TO SET THE (GUAGE POINT TO PROBE) Z OFFSET?
PRESS THE Y OR N KEY
```

Figure 15-8 Z Fixture Offset Message

Respond by pressing the Y key and begin the process of establishing the Z fixture offset. Then press the N key and the CNC continues the setup procedure using the current Z value.

When a fixture offset is selected, the Z fixture offset value is subtracted from the current touch position when determining the length offset. A positive value increases and a negative value shortens the length offset.

After selecting a fixture offset, the CNC moves X and Y axes to the fixture offset, loads the first tool, and begins the tool setting process by displaying the following menu:

```
X             0    0
Y             0    0
Z             0    0 C




TOOL NUMBER = 3
DIAMETER OFFSET =   0.3750
LENGTH OFFSET = 0

TOOL SETTING OPTIONS:
1 ENTER TOOL DIAMETER
2 LOCATE LENGTH
3 EXIT
PRESS OPTION NUMBER
```

Figure 15-9 Tool Setting Options Menu

If the program requires the diameter or the tool is an end mill, the operator selects function 1 and enters the diameter. Select function 2 to begin locating the length offset.

Locating Length Using the JOG Function

The CNC displays the message:

PRESS JOG AND MOVE TO HEIGHT BLOCK OR PRESS MANUAL TO EXIT

The operator then presses the JOG key and moves the tool to the desired offset position. When the tool is at the desired location press the MANUAL key.

Locating Length Using the Probe

Selecting function 2 causes the CNC to move the tool down in one of two ways:

1) If a diameter is entered, the X axis shifts the radius amount, spins the tool backwards, then locates the length.

2) Specifying a zero tool diameter causes the CNC to move the Z down without shifting the X axis or spinning the tool.

The CNC performs the following actions for locating the length of an end mill:

    a. Spindle ON reverse at 100 RPM
    b. X axis is shifted the radius of the tool
    c. Z axis down until initial touch
    d. Z axis is raised .010"
    e. Spindle speed is increased to 500 RPM
    f. Z axis down until final touch
    g. Spindle off, length offset is stored
    h. Z axis returned to COLD START
    i. X axis shift position returned

Fadal

Figure 15-11 Clearance Position & Touch Probe Selection

N1 moves X and Y to the clearance position (Left Figure Above).

N2 moves the tool .100 below the top of the probe (Right Figure Above) and selects the Touch probe.

N3 moves the tool to the center of the probe. Because the block contains the G31 code, the CNC stops moving as soon as the tool touches the probe. The P word used in combination with the G31 causes the CNC to store the touch location.

N4 verifies the previous point (P1) has been interrupted by the probe. A broken tool causes the move to go to completion (without a touch) and abort the operation. See PROBE L9101 FUNCTIONS in this section.

Touch Probe – Tool Diameter Offset

The tool diameter offset is achieved by a two step process as follows:

1) The CNC is programmed to touch the probe at two points.

2) The L9101 R1+8. code is used to perform the calculation.

EXAMPLE: After the length offset has been located, the following program is used to establish the diameter of a 1/2" end mill.

N1 G0 G90 S500 M4 E24 X0 Y-.5 (.200+.25+.05
N2 H1 Z-.1 M65
N3 G1 G31 Y0 F5. P1
N4 G0 Z.1
N5 Y.5
N6 Z-.1
N7 G1 G31 Y0 P2
N8 L9101 R1+8. R2+.4 D1

N1: The E24 shifts the XY zero to the center and the Z zero to the top of the stylus. The X0 moves to the center of the stylus. The Y-.5 moves to a clearance position, calculated as follows:

1/2 the width of the stylus: .200

1/2 the approximate tool diameter: .250

Clearance: .050

N2: moves the tip of the tool .100" below the top of the stylus while spinning the tool backwards at 500 RPM.

N3: moves to touch point 1.

N4: moves Z .100 above the stylus.

N5: moves to a clearance position in preparation for the next touch.

N6: moves Z below the top of the stylus.

N7: moves to touch point 2.

N8: performs the diameter calculation.

The stylus width is specified by R2. The D word specifies the diameter is to be stored as offset 1 in the tool table.

General Rules to Follow: MP Series Probe

1) Start the program by selecting the probe. M64 selects the MP Series probe, M65 selects the TS Series probe.

2) A move with the G31 must be a linear (G1) move.

3) No other codes are allowed with the G31 except G1, P# (Point Number), and feed rates.

4) The Probe functions may only use three points for each calculation, P1, P2, and P3.

5) The probing is to be in the absolute mode (G90).

6) CRC, Mirror Image, Rotation and Drill Cycles are not allowed during the execution of the G31 code.

Locating the Points

There are two procedures available to locate and store the points:

1) Using the G31 P# codes.

2) Using function 1 of the L9101 fixed subroutine.

G31 Probe Touch Function

The G31 is only used in conjunction with a probe. This code causes the machine to stop motion when the probe is touched and then execution continues at the next line in the program. The G31 can be used with table or spindle probes (See also G31.1).

The motion can be defined in absolute or incremental terms. The positions can be stored with a P word, a macro V variable, and output through the RS232 port. All G31 moves must be G1 linear moves. No G0, G2, or G3 moves are allowed.

- Rotation can be in effect when the G31 is used.
- CRC should not be in effect when G31 is used.
- Mirrored axes should be canceled before using the G31 code.
- Fixed cycles need to be canceled before using the probe.

Note: Program a move that would normally be excessive. For example, if a one inch move is required to get the probe up to a wall, use a two inch move in the program. The probe will stop the motion, and whatever motion is remaining, for that line, will be discarded and the control will continue execution of the program at the next line.

Expect some over travel if the feed rate used with the move is high, and also if the stylus in the probe is long. A method for accuracy would be to use the G31 and the G31.1 codes together. Use the G31 at a high feed rate to get up to the wall. With the high feed rate, the stylus is angled and over the edge because of the time required to read the probe and stop the motion. Then reverse the motion to move away from the wall with the G31.1 code in the line. Do this at a slow feed rate. At F1. the motion is slow enough that it will usually stop within one tenth. The G31.1 will stop motion when the probe is not touching. This means that the stylus will be perpendicular to the table and directly at the edge of the wall when the probe is not touching.

If the stylus is not running true, or a chip is in the spindle, the probe will not give a true position reading. For consistency use an M19 to orient and lock the spindle at the same position each time the probe inserted in the spindle. If an operator is to place the probe in the spindle by hand, orient the spindle prior to inserting it in the spindle. Sometimes the stylus will work itself loose, confirm that it is tightly screwed in before using the probe.

Storing Probed Positions

Saving positions through the RS-232 port:

1) Any software designed to save data from the port will be sufficient to retain the data.

2) When a touch is made, the motion will stop and the current position wlll be outputted through the port.

G1 G31 X1. F50. This line sends just the X axis location to the port
G1 G31 X2. Y5. F50. This line sends the X and Y locations to the port
G1 G31 X3. Y-4. Z-2. F50. This line sends the X, Y, and Z locations to the port

3) Macro SPRINT statements can be used just before the probe line to identify the information being saved

#SPRINT "PROBE TOUCH #1:" G1 X1. Y1. G31

EXAMPLE: **Saving Positions to P Words:**

1) P1, P2, and P3 are used to save the touch positions when the fixed probe subroutines are going to be used in the program.

G1 X3. Y-6. G31 P1   The first touch position is saved to P1
X0 Y0 G5
G1 X0 Y6. G31 P2   The second touch position is saved to P2
X0 Y0 G5
G1 X-3. Y-6. G31 P3   The third touch position is saved to P3
L9101 R1+2.   Use probe fixed subroutine function #2 to find center

2) P1, P2, and P3 can be used with the macro PX1-3, PY1-3, PZ1-3, PA1-3, and PB1-3 variables. When a probe touch (G31) or probe no-touch (G31.1) is used on a line with a P1, P2, or P3 each axis position is stored regardless of the axis that moved to get the touch point.

G90 G0 X3. Y-6.
Z1. H21
G1 F30. G31 Z-2.
F1. Z0 G31.1 P1   P1 has stored the XYZAB position at this line
#V1=PZ1
#PRINT "THE TOUCH POINT IS: X",PX1,", Y",PY1,", AND Z",PZ1

EXAMPLE: Saving the Position as a V Variable:

1) The current position can be saved to a V variable by using a macro AX, AY, AZ, AA, or AB command.
G90 G1 X4. Y4. G31 F50.
X0 Y0 G31.1 F1.
#V1=AX This saves the X position to V1#V2=AY This saves the Y position to V2

G31.1 Probe No Touch Function This code causes the machine to stop motion when the probe is not touching and then execution continues at the next line in the program. The G31.1 can be used with table or spindle probes. This code functions exactly like the G31 code. See also G31.

Using G31 During the probing function, the CNC is programmed as normal. Programming a G31 with an axis move causes the CNC to monitor the probe interface and stop all motion in the event of a touch.

The CNC stores three touch points in memory. Coding a P1, P2, or P3 in the same block with the G31 designates the touch point to stored. This code also sends the touch or no touch data to the RS-232 serial port.

EXAMPLE: *G1 G31 F25. X10. P1*
Moves the X axis until the probe touches or the move is completed

EXAMPLE: **If the Probe Touches:**

The axis motion stops, the X, Y, Z, A, B locations are stored as point 1 and the program continues.

EXAMPLE: **If the Probe Doesn't Touch:**

The CNC stores a value of .100E9 (to signify no touch) as the X location of point #1 and continues the program.

The following example shows how to locate three points inside a 3" diameter hole. The approximate center is at X0 and Y0. The Z0 is at the top of the diameter.
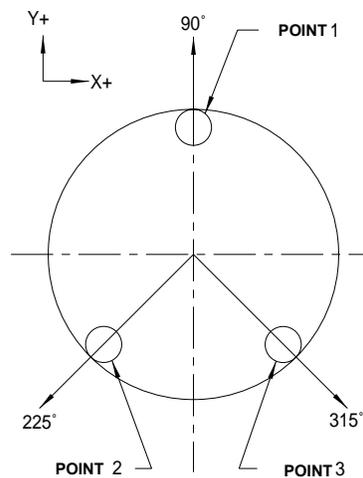
Figure 15-12 Locating Points

N1 M6 T1   (LOAD THE PROBE
N2 G0 G90 X0 Y0   (POSITION TO THE APPROX. CENTER
N3 Z-.25 H1 M64   (MOVE TIP .25 BELOW TOP OF PART
N4 G1 G31 P1 F25. Y1.8   (FIRST POINT
N5 F150. Y0   (MOVE OFF PART AND RETURN
N6 G31 P2 F25. X-1.273 Y-1.273   (APPROACH AT 225 DEGREES
N7 F150. X0 Y0   (MOVE OFF PART AND RETURN
N8 G31 P3 F25. X+1.273 Y-1.273   (APPROACH AT 315 DEGREES
N9 F150. X0 Y0   (MOVE OFF PART AND RETURN

Note: After performing a G31 touch, before another G31 is programmed, the probe must be moved off the part as in N5 and N7 above.

L9101 Probe Functions The L9101 fixed subroutine has 10 probe functions available:

1) LOCATE TOUCH POINT

2) CENTER LOCATION AND RADIUS

3) PART ORIENTATION

4) MID-POINT AND ANGLE

5) Z DATUM LOCATION

6) TOOL BREAKAGE DETECTION

7) TOUCH/POSITION CHECK

8) COMPUTE DIAMETER

9) SET PROBE CALIBRATION

10) SET TOUCH POINT

The code L9101 is used to call a probe function, the R word R1 selects the specific function. For example: L9101 R1+2. Selects function #2 - CENTER LOCATION AND RADIUS.

Upon completion of the L9101, the R words R1-R3 contain the results. The R words can then be used as indirect references throughout the remainder of the program.

The touch points are retained in memory until power is removed, thus making it possible to do a mid-program start after the points have been located. As in circular motion, the G17, G18, and G19 modes determine the output of the L9101 subroutine. G17= XY, G18= ZX, and G19= YZ.

For example, use function 2 to compute the center location of 3 points. The logical X is returned in R1. When G18 is in effect, R1 contains the physical Z center location.

The probing process is designed to be used as a three step process:

1) Locate the points.

2) Use fixed subroutine L9101 to perform the calculations.

3) Continue the program using the R words returned by step 2.

Locate Touch Point   **Function #1:** Locates Touch Point

Coding: L9101 R1+1. X, Y or Z moves, Approach/Return Feed
P1, P2 or P3 define point number
Result: Stores Location Specified By P word
R1= Logical X Touch, R2= Logical Y Touch

Moves to locate a touch point, apply the probe correction and return to the starting position. This function uses a two touch process. The first touch locates the initial point, the second touch is performed at a slow feed rate for best accuracy.

EXAMPLE: **Example Coding:**

L9101 R1+1. X1. Y1. F50. P1
Generates the following incremental motion:
X1. Y1. F50. G31 G1   (INITIAL TOUCH
X.0035 Y.0035 F10.   (OVER TRAVEL PROBE SWITCH
X-.0707 Y-.0707 F10. G31.1   (MOVES UNTIL NO TOUCH
X-.0035 Y-.0035 F10.   (MOVES OFF .005 MORE
X.0707 Y.0707 F.5 G31 P1   (FINAL TOUCH

The following program uses function 1 to locate the 3 points of a circle, instead of G31 as in the previous example.
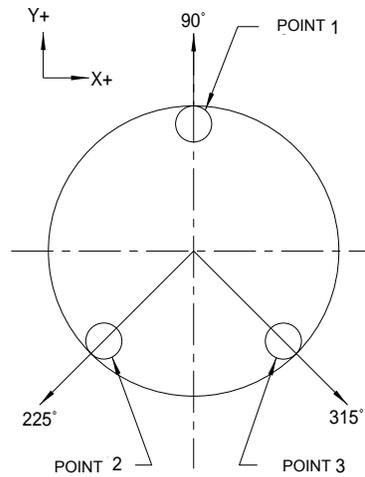

Figure 15-13 Point Location with Function 1

N1 M6 T1   (LOAD THE PROBE
N2 G0 G90 X0 Y0   (POSITION TO APPROX. CENTER
X-1. Y-1. F50. G1   (RETURN TO START POSITION
N3 Z-.25 H1 M64   (MOVE .25 BELOW TOP OF PART
N4 L9101 R1+1. P1 F25. Y1.8   (FIRST POINT
N5 L9101 R1+1. P2 F25. X-1.273 Y-1.273   (APPROACH AT 225 DEGREES
N6 L9101 R1+1. P3 F25. X+1.273 Y-1.273   (APPROACH AT 315 DEGREES

Compute Center and Radius

Function #2: Compute Center Location and Radius

Coding: L9101 R1+2.
Points Used: P1, P2, P3
Results: R1= Logical X, R2= Logical Y, R3= Radius

This function computes the center location and radius relative to the three touch points. The X center position is returned in R1, the Y center position returned in R2. and the radius returned in R3. This function may be used with the ID or OD of a circle. The following is a sample program to locate a center

and perform a drilling operation at the center of the circle, using the ID of the circle:
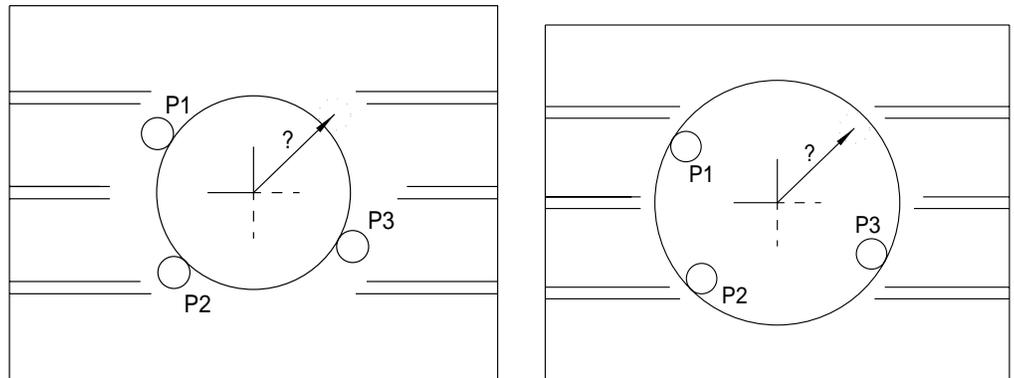
Figure 15-14 Circle Center Location

N1 M6 T1   (LOAD PROBE
N2 G0 G90 X6. Y0.   (POSITION TO CENTER
N3 Z-.25 H1 M64   (POSITION Z
N4 L9101 R1+1. X5. Y1. F25. P1   (LOCATE POINT 1
N5 L9101 R1+1. X5. Y-1. F25. P2   (LOCATE POINT 2
N6 L9101 R1+1. X7. Y-1. F25. P3   (LOCATE POINT 3
N7 L9101 R1+2.   (COMPUTE CENTER
N8 M6 T2   (LOAD DRILL
N9 G0 X+R1 Y+R2   (POSITION TO CENTER
N10 H2 Z.1   (BEGIN DRILLING OPERATION
N11 G1 F10 Z-2.0

Part Orientation   **Function #3:** Part Orientation

Coding: L9101, R1+3., R2= Expected Angle from P2 to P3
Points Used: P1, P2, P3
Results: R1= Logical X, R2= Logical Y, R3= Angular Error
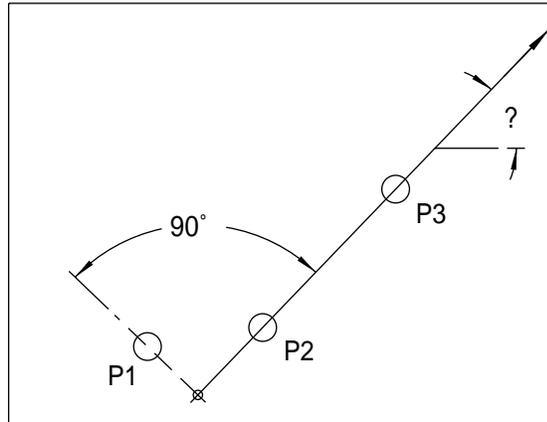
Figure 15-15 Intersection Point & Angular Correction

This function computes the location of the intersection point and angular correction needed for program rotation. The computed intersection point assumes P1 has a 90 degree relationship to the line created from P2 to P3.
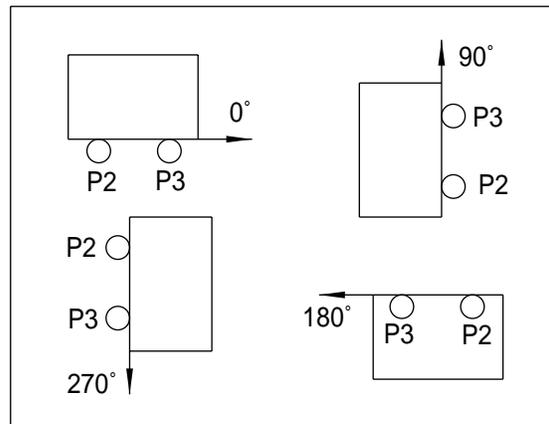
Figure 15-16 Expected Angle Changes

The expected angle is related to the touch surface. The examples above illustrate how the expected angle changes, depending upon how P2 and P3 touch the part.

The angle is coded in decimal degrees, 0 degrees starts at the X+ direction and increases in the counterclockwise direction.
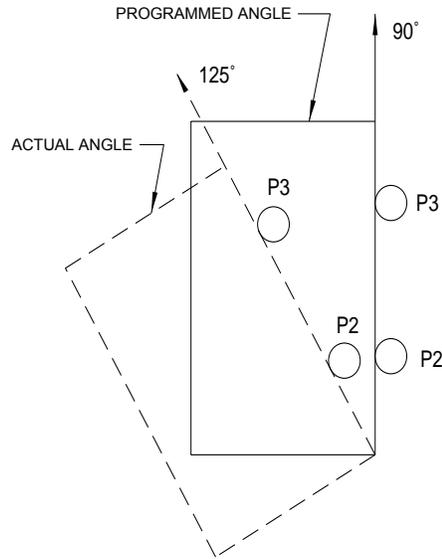
Figure 15-17 Programmed Angle P2 to P3

The angular error returned in R3 is the result of subtracting the expected angle from the probed angle (P2 to P3). This angle (R3) becomes the angular correction needed for program coordinate rotation (G68). The sample above shows the programmed angle from P2 to P3 to be 90 degrees. The angle necessary for program rotation would then be 35 degrees if the actual angle is 125 degrees.
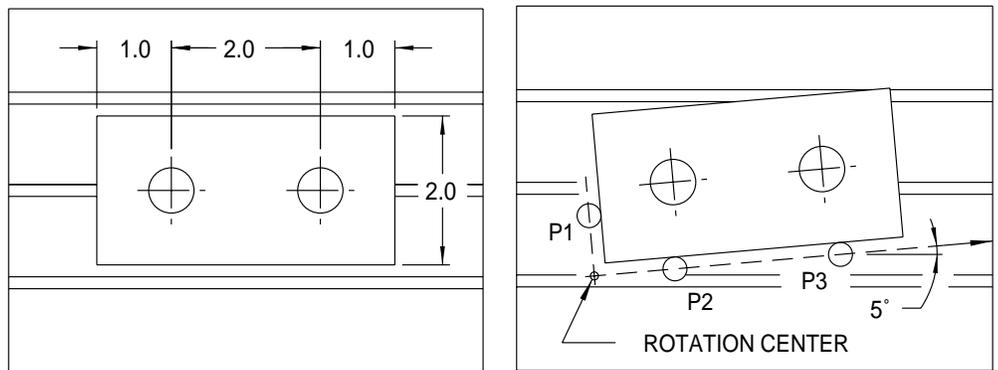
Figure 15-18 Sample Program to Drill Two Holes in a Part

The example above is a sample program to drill two holes in a part. The program XY zero position is approximately the lower left corner of part.

The probe stylus is a .25" diameter.

N1 M6 T1(LOAD PROBE
N2 G0 X-.5 Y1. (CLEARANCE POSITION FOR POINT 1
N3 Z-1.0 H1 M64
N4 L9101 R1+1. X.5 F25. P1 (MOVE TO TOUCH POINT 1
N5 Z0 G0
N6 X1.0 Y-.5 (CLEARANCE POSITION FOR POINT 2
N7 Z-1.0
N8 L9101 R1+1. Y.5 F25. P2 (MOVE TO TOUCH POINT 2
N9 G0 Z-1.0
*N10 X5.0 Y-.5 (CLEARANCE POSITION FOR POINT 3*
N11 Z-1.0
N12 L9101 R1+1. Y.5 F25. P3 (MOVE TO TOUCH POINT 3
N13 L9101 R1+3. R2+0 (CALCULATE PART ORIENTATION
N14 G90 G0 H0 Z0
N15 M6 T2 (LOAD DRILL
N16 G90 G0 X+R1 Y+R2 S10000 M3 (MOVE TO INTERSECTION POINT
N17 G92 X0 Y0 (SET ABSOLUTE LOCATION
N18 G68 R0+R3 X0 Y0 (SET ROTATION
N19 Z.1 H1 M8
N20 G81 G99 Z-1.0 R0+.05 F50. X1.125 Y1.125 (DRILL CYCLE
N21 X3.125 (DRILL SECOND HOLE
N22 G69 G80
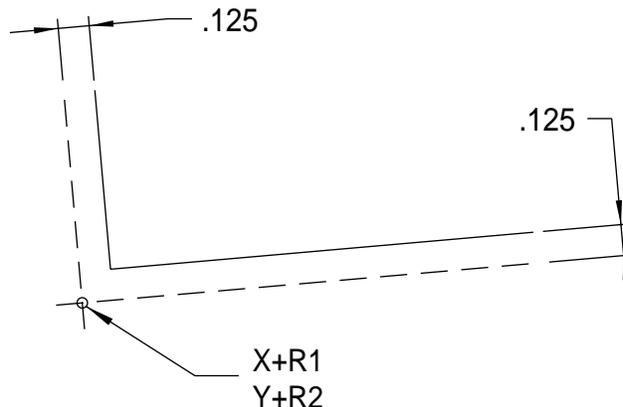
Figure 15-19 CNC Movement to Corner

After probing the part and establishing the rotation, programming X.125 Y.125 causes the CNC to move the center of the tool to the corner of the part.

Mid-Point and Angle   **Function #4:** Mid-Point and Angle

Coding: L9101, R1+4., R2= Expected Angle
Points Used: P1, P2
Results: R1= Logical X, R2= Logical Y, R3= Angular Error
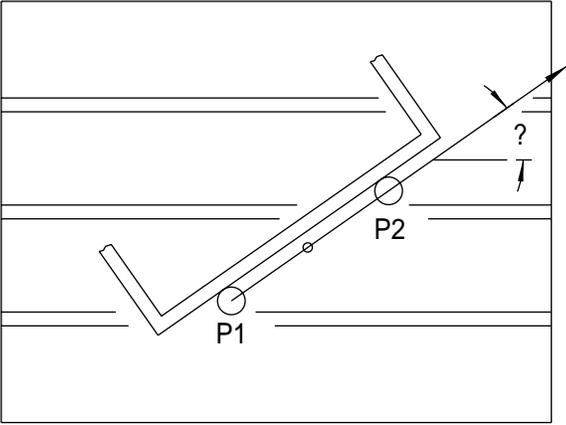
Figure 15-20 Mid-Point and Angle

This function computes the location of the point between P1 and P2 and the angular correction needed for program rotation. The angular error is determined the same as function 3 - PART ORIENTATION previously described.

The following are two examples using this function:
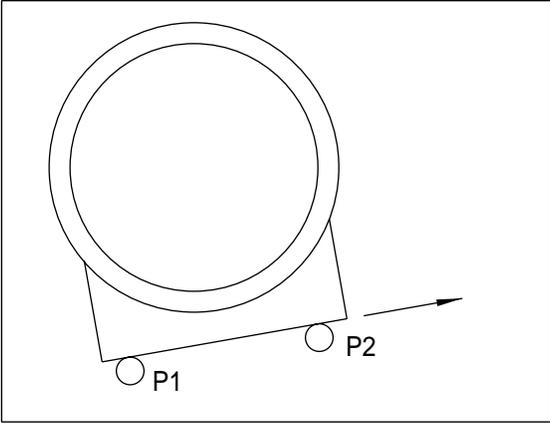
Figure 15-21 Part with Unknown Rotation

The example above shows a part having an unknown rotation. After using this function, the R3 contains the angular correction needed for rotation.
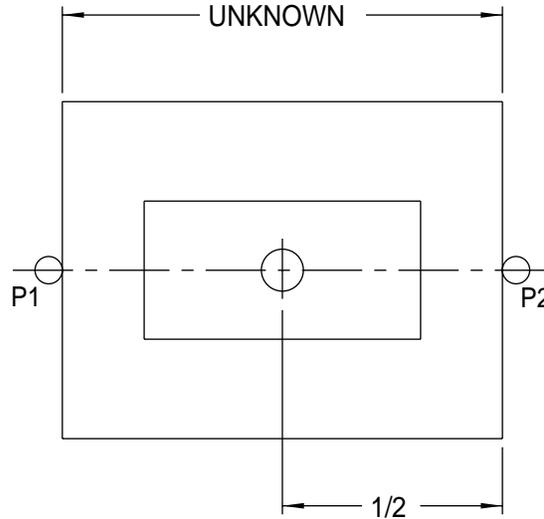
Figure 15-22 Part with Unknown Width

The example above shows a part having an unknown width. After using this function, the R1 contains the location of the midpoint.

Z Datum Location   **Function #5***:* Z Datum Location

Coding: L9101, R1+5., Optional Z Modifier
Points Used: P1
Results: R1= Distance From Length Offset Position To Z Datum
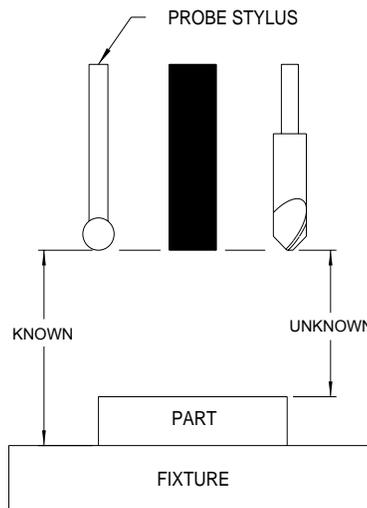
Figure 15-23 Z Datum Location

This function is used to calculate the distance between an unknown surface and the gauge point (Tool Length Offset Position).

The procedure is as follows:

1) Setup: A length offset is set for the probe as a normal tool.

2) Programming:

   a. The probe is moved to the gauge point using the H word.
   b. The Z touch for P1 is found using G31 or function 1 of L9101.
   c. Function 5 is used to calculate the distance.
   d. The R1 is referenced as the distance to the surface.

The following example shows how to locate the top surface and remove .010":

N1 M6 T1   (LOAD PROBE
N2 H1 M64   (MOVE TO OFFSET, SELECT MP8 PROBE
N3 G1 G31 F25. Z-30. P1   (LOCATE P1
N4 L9101 R1+5. Z-.010   (CALCULATE, ADD -.01
N5 M6 T2   (LOAD END MILL
N6 M3 S10000   (SPINDLE ON
N7 Z+R1 H2   (POSITION Z
N8 G1 F100. X10.   (MACHINE .010 OFF TOP

Block 7 above positions the Z axis .01 below the touch position. Another method available is to use the G92 preset code as follows:

N6 M3 S10000 (SPINDLE ON
N7 H2 (POSITION Z
N8 G92 Z-R1 (PRESET ABS. Z LOCATION
N9 Z0
N10 G1 F100. X10. (MACHINE .010 OFF TOP

Using the G92 code allows the absolute Z location to be relative to the touch position for the remainder of the program.

When machining multiple parts, the Z datum can be stored as a fixture offset using G10 L02 Z-R1 P#.

**Tool Breakage
Detection**

Function #6: Tool Breakage Detection

Coding: L9101, R1+6.
Points Used: P1
Results: A No Touch Causes The CNC Program To Stop

CONTACT   NO CONTACT
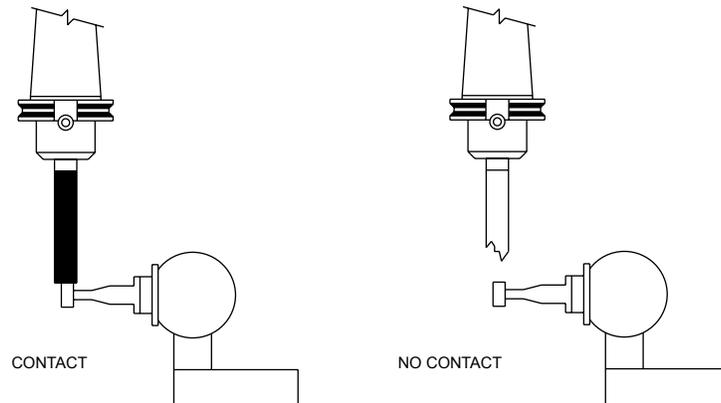
Figure 15-24 Tool Breakage Detection

This function is used to check if P1 had a successful Y or Z touch position stored. A block containing a G31, P1 with a move that doesn't touch the tool setting probe causes the CNC to store a value to signify a touch was not made.

See "USING THE TOUCH PROBE - TOOL BREAKAGE DETECTION" previously described.

**Touch Check,
Position Check**

Function #7: Touch Check, Position Check

Coding:
Touch Check: L9101, R1+7., R2= False #, R3= True #
Position Check: L9101, R1+7., R2= False #, R3= True #
R4= Approach Direction, X, Y, Z, A or B # Points Used: P1
Results: R2= True # or False #

The following diagram describes this function:
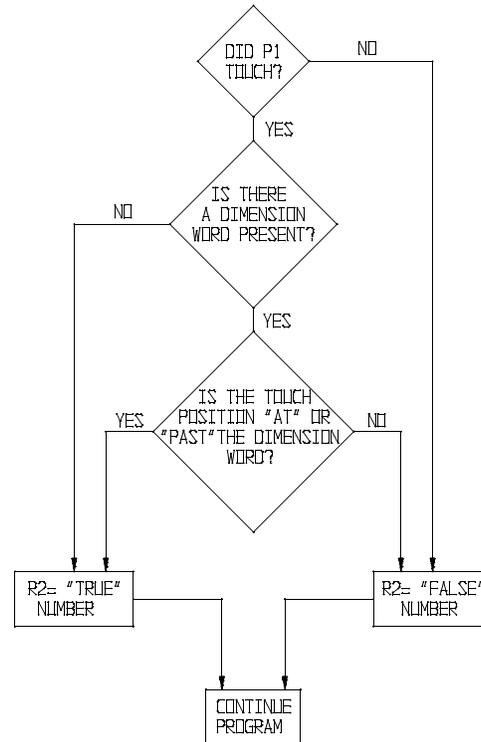
Figure 15-25 Touch Check, Position Check

An X, Y, Z, A, or B dimension word included with the L9101 block is used for the position check. Motion will not occur; the CNC uses the axis word only for the position check. The R4 indicates approach direction; R4+1.= Positive, R4-1.= Negative.

EXAMPLE: L9101 R1+7. R2+10. R3+20. R4-1. Y-2.5

1) Approaching negative with a touch at Y-3.0 returns R2 with a value of 20.

2) Approaching negative with a touch at Y+1.0 returns R2 with a value of 10.

True or False Comparison

Function #7: Continued

Coding:
Touch Check: L9101, R1+7., R2= False #, R3= True #
Position Check: L9101, R1+7., R2= False #, R3= True #
R4= Approach Direction, X, Y, Z, A or B # Points Used: P1
Results: R2=True # or False #

Using the True/False response to redirect the program allows for numerous capabilities. The following is an example how the program operation changes according to the touch position:
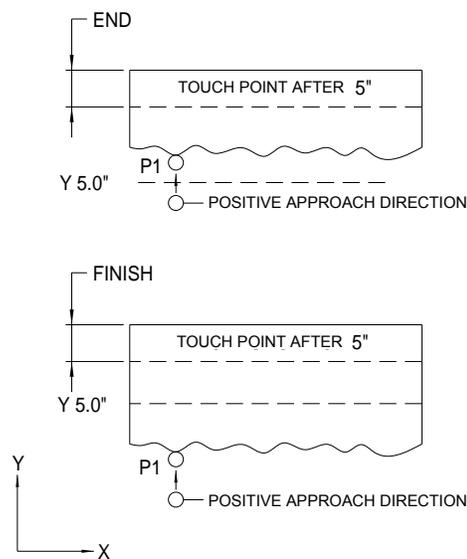
Figure 15-26 True/False Comparison

N1 M6 T1   (LOAD PROBE
N2 G0 G90 X0 Y0   (POSITION XY
N3 Z-1.0 H2 M64   (POSITION Z, SELECT MP8
N4 G1 G31 Y10. F25. P1   (MOVE TO TOUCH
N5 L9101 Y5.0 R1+7. R2+8. R3+11. R4+1.   (COMPARE POSITION
N6 M6 T2   (LOAD TOOL
N7 M99 P+R2   ("AT" or "PAST"= N11,BEFORE =
N8 G10 L12 P2 R0+.55   (SET DIAMETER FOR ROUGHING
N9 F25.0   (SET ROUGHING FEED RATE
N10 M98 P1   (CALL SUB. TO MACHINE PART
N11 G10 L12 P2 R0+.5   (SET DIAMETER FOR FINISHING
N12 F50.0   (SET FINISH FEED RATE
N13 M98 P1   (CALL SUB. TO MACHINE PART

This example finds a touch point and begins the finishing operation at N11 when the Y touch position is "AT" or "PAST" the Y+5", otherwise the roughing operation is performed.

Calculate Diameter   **Function #8:** Calculate Diameter

Coding: L9101, R1+8., R2= Stylus Width
*   D#= Offset Number (Optional)*
Points Used: "Y" of P1 and "Y" of P2
Results: R1= Tool Diameter/Radius
*   Logical X of P1= Tool Diameter/Radius*

This function performs the diameter/radius calculation. The value returned depends on the selection made using the SETP command. When the machine is in the DIAMETER mode the result is diameter, otherwise radius is returned. Specifying a D word in the block with L9101 causes the CNC to store the value in the tool offset table.

The X of P1 contains the result of the calculation. This enables function 7 to perform a Position Check. See USING THE TOUCH PROBE - TOOL DIAMETER OFFSET previously described.

Set Calibration   **Function #9:** Set Probe Calibration

Coding: L9101, R1+9., R2= Radial Over travel
*   X, Y or Z= Shift Amount*
Points Used: None
Results: Used With Function #1

This function establishes the probe compensation. The L9101 Function 1 uses these values when present for error correction. In most cases this function is not needed because most tolerances are greater than the probe errors.

XY Shift Error   The shift error is the difference between the center of the spindle and the center of the probe. The probe has an adjustment to align the centers. The CNC XY shift capability allows another method to compensate for the error.

Radial Over Travel   Before the probe indicates a touch, the stylus must touch the surface and open the contacts inside the probe. This causes a slight over travel error. This probe error varies with the length of the stylus. For example, an over travel error of approximately .0012" occurs with a 2" stylus length. The error increases to .0043" with a 7" stylus length. Applying over travel correction improves the 2" stylus error to +-.0004" and the 7" stylus error to +-.0012.

When locating the center of a circle, the over travel of the probe doesn't change the computed center location. The radius of the circle appears larger when inside a hole and smaller when outside a boss.

MP8 Probe Calibration

1) Mount an inspection ring on the table.

2) Set the XY home position at the center of the ring.

3) Install the probe and run the following program:

N1 M64 G1 G90 Z-??(MOVE Z INSIDE RING
N2 L9101 R1+1. Y5.0 F25. P1
N3 L9101 R1+1. X-4.33 Y-2.5 F25. P2(TOUCH 120 DEG. APART
N4 L9101 R1+1. X+4.33 Y-2.5 F25. P3
N5 L9101 R1+2.(COMPUTE CENTER

4) Enter the command; SUM

Function #9:   Continued

Coding: L9101, R1+9., R2= Radial Over Travel
              X, Y or Z= Shift Amount
Points Used: None
Results: Used With Function #1

After entering the SUM command the CNC displays the point data, R words, points used and the probe correction setting as follows:

P1=0,0,0,0,0,0,0,0,0
P2=0,0,0,0,0,0,0,0,0
P3=0,0,0,0,0,0,0,0,0
R WORDS R0-R9
0,0,0,0,0,0,0,0,0,0
POINTS USED   X   Y

The point values are displayed in the axis order of P1=X, Y, Z, U, V, W, A, B, C.

The R values are displayed in ascending numerical as R0, R1, R2, R3, R4, R5, R6, R7, R8, R9.

Entering XY Shift Values   The negative values of R1 and R2 are used for the shift values. R1 becomes the X shift and R2 becomes the Y shift.

EXAMPLE: The probe indicates the center location to be at X.001 and Y.002 after performing the calibration. The compensation is entered as L9101 R1+9.R2+ X-.001 Y-.002

Entering Radial Over Travel After the ring gauge test, R2 contains the effective radius. The amount entered for compensation is computed by subtracting the effective radius from the apparent radius.

EXAMPLE: *Ring Gauge Radius= 1.500"   Stylus Radius= .125"*
Apparent Radius= 1.375"   Effective Radius= 1.378
1.375 - 1.378 = -.003 (Correction Needed)
The XY shift and radial correction is entered as:
L9101 R1+9. R2-.003 X-.001 Y-.002

The correction is entered in the program before L9101 function 1 is used.

Set Counter **Function #10:** Set Counter

Coding: L9101, R1+10., P1= Increments the counter
*   by 1 X = Sets the counter starting number*
Points Used: None
Results: Used With Function #7 to create IF - Then statement

This function is used to create an IF - Then statement for program redirection. The examples below use this function to perform G52 program shifts for multiple part programming.

%
N1O101 (SUB FOR PART MACHINING
N2G1G90F250.
N4X1.
N5Y1.
N6X0
N7Y0
N8M99
N1O9110 (IF THEN PROGRAM REDIRECTION EXAMPLE
N2L100 (SUB TO SHIFT THEN MOVE
N3G52X+R8Y+R7 (LOCAL COORDINATE SYSTEM
N4M98P101 (INPUT SUB PROGRAM NUMBER
N5M17
N6M30
N7R9+0R8+0R7+0 (ASSIGN R VARIABLES
N8L101

N9G91G10L109P1 (INCREMENT X COUNT BY +1
N10G90
N11G91G10L108P2 (INCREMENT X STEP BY +2
N12G90
N13L9101R1+10.X+R9P1 (SETS THE X VALUE TO COUNT
N14L9101R1+7.X3.R2+24.R3+16.R4+1. (IF COUNT IS X OR HIGHER
N15M99P+R2 (GOTO R3 ELSE GOTO R2
N16G91G10L107P-2 (INCREMENT Y STEP BY -2
N17G90
N18L9101R1+10.Y+R7P1 (SETS THE Y VALUE TO COUNT
N19L9101R1+7.Y-5.R2+21.R3+25.R4-1. (IF COUNT IS Y OR HIGHER
N20M99P+R2 (GOTO R2 ELSE GOTO R3
N21R8+0 (ASSIGN VARIABLES
N22G52X0Y+R7 (MOVE TO X0 Y(R7 VALUE)
N23R9+0 (ASSIGN R VARIABLE
N24M99P8 (GOTO LINE #8
N25M5M9
N26M2

Set Touch Point

Function #10: Set Touch Point

Coding: L9101, R1+10.,X, Y, Z, P
*   X, Y, AND Z = Location of point*
Points Used: None
Results: The P word identified by the X, Y, and Z
*   location is used with other L9101 functions.*

This function may be used to identify the centers of two bores and calculate the center and angle between them. The example below locates the center of two bores and calculates the center and angle between them.

N1 M6 T1 (LOAD PROBE
N2 G0 G90 X6. Y0. (POSITION TO CENTER OF THE FIRST BORE
N3 Z-.25 H1 M64 (POSITION Z
N4 L9101 R1+1. X6. Y1. F25. P1 (LOCATE POINT 1
N5 L9101 R1+1. X4. Y-1. F25. P2 (LOCATE POINT 2
N6 L9101 R1+1. X8. Y-1. F25. P3 (LOCATE POINT 3
N7 L9101 R1+2. (COMPUTE CENTER OF THE FIRST BORE
N8 R9+R1 R8+R2 (RENAME THE LOCATION
N9 G0 G90 X10. Y0 (POSITION TO CENTER OF THE SECOND BORE
N10 L9101 R1+1. X10. Y1. F25. P1 (LOCATE POINT 1

N11 L9101 R1+1. X8. Y-1. F25. P2 (LOCATE POINT 2
N12 L9101 R1+1. X12. Y-1.F25. P3 (LOCATE POINT 3
N13 L9101 R1+2. (COMPUTE CENTER OF THE SECOND BORE
N14 R7+R1 R6+R2 (RENAME THE LOCATION
N15 L9101 R1+10. X+R7 Y+R6 P2 (SET THE CENTER OF THE SECOND BORE
TO POINT 2
N16 L9101 R1+10. X+R9 Y+R8 P1 (SET THE CENTER OF THE FIRST BORE TO
POINT 1
N17 L9101 R1+4.R2+0 (COMPUTE CENTER AND ANGLE OF THE TWO BORES
N18 G90 G10 L2 P1 X+R1 Y+R2 (SET THE CENTER OF THE BORES AS
FIXTURE OFFSET 1
N19 G0 G90 E1 X0 Y0 (MOVE TO FIXTURE 1 ZERO
N20 G68 X0 Y0 R0+R3 (SET ROTATION WITH THE ANGLE OF THE BORES

Probe Tutorial
Examples in Format 2

Always test the probe to determine if it is functioning properly by typing the M64 M66 code in MDI. Then go into jog, touch the stylus, and look for the touch/no touch message.

The stylus should run true when the probe is rotated by hand. Place an indicator in a magnetic base and put the indicator tip on the end of the stylus. Rotate the probe head by hand and observe the run out. Use the adjustment screws to get the stylus to run true.

1) This example will demonstrate how the G31 code stops motion and allows the control to move to the next line in the program. The move on the line with the G31 or the G31.1 is usually a move that is beyond the desired touch point.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3. G31
M0 Look at the X axis position at this point
X1. G0
M99 P1

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of the stylus should be below the top of the solid object.

2) This example will demonstrate the effect of feed rate on over travel after the probe indicates to the control that a touch has been made.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3. G31
M0
X1. G0
M99 P1

Start by jogging the probe to approximately one inch to the right side of a solid object. The position display will present the current X axis location. Press the START button to run this routine each time the position is displayed. Notice that the location changes over several times it is run. Next, change the feed rate on line one. Make it faster, then try it at slower feed rates. The position should repeat more consistently at the slower feed rates. The faster the feed rate, the further the machine will over travel at the touch point.

3) This example will demonstrate a two-touch method of finding a point on an edge.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3. G31
X.05
F1.
X-3. G31
M0
X1.G0
M99 P1

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of the stylus should be below the top of the solid object.

The first touch is at a high feed rate and is only used to get the stylus in the general area of the edge. Then the stylus moves away and the second approach is used to get a "good" point. From the second example it was demonstrated that the slower feed rate results in a more consistent touch position.

4) This example will demonstrate the use of the G31.1 code.

The G31 code will stop motion when the probe switch is opened. The G31.1 code will stop motion when the probe switch is closed. The G31.1 code is used just after a G31 code is used. The probe switch is opened when the stylus touches an edge and the motion continues to cause the stylus to open the switch. At this point, the stylus should be at an angle. It should be mentioned

that the longer the stylus, the more over travel will be required to open the switch. Motion in the opposite direction can now be stopped with a G31.1 code. When the stylus is vertical, the probe switch is now closed and the motion will stop because of the G31.1 code.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3. G31
F10.
X1. G31.1
M0
X1. G0
M99 P1

Try to vary the second feed rate. Again observe the X axis position. The slower feed rate will result in a more consistent final position.

5)   This is a variation on the fourth example.

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of the stylus should be below the top of the solid object.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3. G31
X1. G31.1
F.5
X-3. G31
M0
X1. G0
M99 P1

Notice the amount of time required to pick up a point with each of these methods. Compare the time from examples 4 and 5. Also compare the positions picked up using each method. Consistency and time should be issues to be aware of.

Using the Probe with Macro Statements
Examples in Format 2

Always test the probe to determine if it is functioning properly by typing the M64 M66 code in MDI. Then go into jog, touch the stylus, and look for the touch/no touch message.

The stylus should run true when the probe is rotated by hand. Place an indicator in a magnetic base and put the indicator tip on the end of the stylus. Rotate the probe head by hand and observe the run out. Use the adjustment screws to get the stylus to run true.

Macro statements can be used to make determinations from the positions picked up with the probe.

1) Use the AX, AY, AZ, AA, and AB macro statement to collect and use the current axis position.

Start by jogging the probe to approximately one inch to the right side of a solid object. The end of a stylus should be below the top of the solid object.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3.G31
X1. G31.1
F.5
X-3.G31
#V1=AX
M0
X1. G0
M99 P1

The macro statement states, "Make the V1 variable equal to the current X axis location (AX). To see this value in memory, use the DV command from the command mode. The current location will be stored in the V1 variable location on the screen. These examples can all be used for the other axes, and they could all be used in absolute as well.

2) This example will demonstrate how to use a macro statement to determine the center point of the part.

Start by placing a 1-2-3 block in a vice. hold the block with the three inch sides in the jaws. Jog the probe to approximately one inch to the right side of the block. The end of the stylus should be approximately .25 below the top of the block.

EXAMPLE: *G91 G1 F50.*
M64 M66
X-3. G31
X1. G31.1
F.5

X-3. G31
#V1=AX
Z1. G0
X-4.
Z-1.
F50.
X3. G31
X-3. G31.1
F.5
X3. G31
#V2=AX
#V3=(V1+V2)/2
Z1. G0
X4.
Z-1.
M0
M99 P1

The V1 variable represents the right side touch; the V2 variable represents the left side touch point. The V3 variable represents the mid-point of the block. This mid-point will be relative to the SETX position. View the V3 value in the variable table by using the DV command in the command mode.

3) This example will demonstrate the ability to alter or establish a fixture offset using the probe and a macro statement.

Start by placing a 1-2-3 block in a vice. Hold the block with the three inch sides in the jaws. Jog the probe to approximately one inch to the right side of the block. The end of the stylus should be approximately .25 below the top of the block.

EXAMPLE:
G91 G1 F50.
M64 M66
X-3. G31
X1. G31.1
F.5
X-3. G31
#V1=AX
Z1. G0
X-4.
Z-1.
F50.
X3. G31

#V2=AX
#V3=(V1+V2)/2
#FX1=V3
Z1. G0
X4.
Z-1.
M0
M99 P1

The V1 variable represents the right side touch; the V2 variable represents the left side touch point. The V3 variable represents the mid-point of the block. This mid-point will be relative to the SETX position. The FX1 macro statement is used to enter the value of V3, which is the mid-point position, into the X value of fixture offset 1. Use the DF command and the DV command to compare the values in the fixture table and the variable table.

Fixed Subroutines Examples in Format 2

Always test the probe to determine if it is functioning properly by typing the M64 M66 code in MDI. Then go into jog, touch the stylus, and look for the touch/no touch message. The stylus should run true when the probe is rotated by hand. Place an indicator in a magnetic base and put the indicator tip on the end of the stylus. Rotate the probe head by hand and observe the run out. Use the adjustment screws to get the stylus to run true.The control has fixed subroutines specially designed to work with the probe. This section will discuss the use of these routines.

1) Each subroutine requires that the positions used in the calculations are stored in P variables. There are two methods to pick up and store P values:

EXAMPLE: **Method 1**

The P1, P2, or P3 variables will store the values of the moving axes in the line with the G31 or G31.1 codes.

G91 G1 F50.
M64 M66
X-3. G31 P1
M0
X1. G0
M99 P1
The P1 on the line with the G31 will store the X value of the touch point

G91 G1 F50.
M64 M66
X-3. G31.1 P1
M0
X1. G0
M99 P1
The P1 on the line with the G31.1 will store the X value of the touch point

G91 G1 F50.
M64 M66
Z-5. G31 P1
M0
X1.G0
M99 P1
The P1 on the line with the G31 will store the Z value of the touch point

G91 G1 F50.
M64 M66
X-3. Y-3. G31 P1
M0
X1. G0
M99 P1
The P1 on the line with the G31 will store the X and Y axis values

Method 2

Use the L9101 subroutine function 1 to move and store the point. Note that the program must be written in absolute terms. When using the example use the SETX command to set the X axis home approximately one inch to the right of the object to touch.

EXAMPLE: *G90 G1*
M64 M66
L9101 R0+1. X-3 F25. P1
M0
X1. G0
M99 P1

The function for the L9101 subroutine is selected with the R0+1. selects function 1 of the L9101 subroutine. The function requires four items:

1) The R0 variable to select the function

2) The move to the point

3) The approach feed rate

4) The desired P variable

In the example above only the X axis will be stored because it is the only axis move in the L9101 line.

Compare this method of picking up and storing a touch point to the methods discussed previously in this section. Each method will store the points needed; selecting one method over the other is a matter of programmer's preference. Note that one disadvantage of using the L9101 fixed subroutine is that it must be written in absolute terms.

EXAMPLE: *G90 G1*
M64 M66
L9101 R0+1. X-3. Y-3. F25. P1
M0
X1. G0
M99 P1

In the example above, the X and Y axis positions will be stored because they are the axes in motion in the L9101 line.

This page intentionally left blank.