Programs 8190_en.h

The program is used for the measurement of hole positions by means of a 3-D touch probe and for subsequent rework at the determined exact positions.

Requirement:
A hole pattern provided as an NC program with X-Y positions and M99 must be precisely reworked at all positions.
Since the positions cannot be exactly reproduced from one machine to another, all points should be newly measured with a touch probe.
The points from the previous drilling program should be used as the measuring positions.
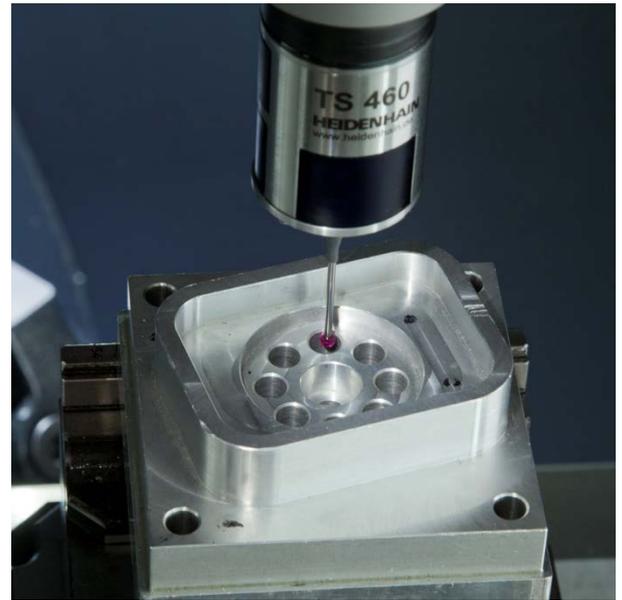
Problem:
A measuring cycle cannot be called several times with M99 because the measuring position is defined in the cycle.

Solution:
Use a separate measuring program which you define in Cycle 12 and then call at the individual machining positions.

Preparation:
A .PNT point table must be created. The number of rows in the table must equal the number of positions to be measured and reworked.

> **NOTE:**
> If the table contains more rows than there are positions, machining operations will be performed at X/Y 0 when the table is processed.

Program 8190_en.h:
A tool call for the touch probe is programmed first, followed by a reset of the counter Q0. The counter will be needed later in order to determine the row to be written to in the table.
Then Cycle 12 is programmed for the program 81902_en.h (measuring program). The file path needs to be adapted here, if necessary. The program in which the original positions are defined with a cycle call is called next; in the example, this is the program 81901_en.i. Adapt the file path here, too, if necessary.

Important! This program must not contain M30.

With each cycle call, the control now calls and executes the program 81902_en.h.
When the measuring program has been executed at all positions, the control returns to the main program where a tool change to the reworking tool is programmed.
Next, the point table—named 81903_en.pnt in the example—is defined as a machining pattern, followed by a cycle with the machining operation to be used for reworking. The example here uses Cycle 12 again, which then calls the defined program at the machining positions.
Then CYCL CALL PAT is programmed to call the machining cycle at the positions defined in the pattern.
At the end, the control retracts the tool.

```
0   BEGIN PGM 8190_EN MM
1   ;NC-program for determining the exact
2   ;positions of several holes, writing them to
3   ;a point table and then executing another
4   ;machining operation at these positions.
5   ;The measuring cycle is not to be called at
6   ;the programmed positions, for this purpose,
7   ;it is defined in a separate program that is
8   ;called with Cycle 12. The control writes the
9   ;results of the measuring cycle to a point
10  ;table. The program uses this point table
11  ;as a pattern definition for reworking.
12  ;
13  BLK FORM 0.1 Z X-50 Y-70 Z-40
14  BLK FORM 0.2  X+50  Y+70  Z+0
15  ;
16  ;Tool call for touch probe
17  TOOL CALL 500 Z
18  ;
19  Q0 = 0 ;Reset counter
20  ;
21  ;Definition of program 81902_en.h for
22  ;measuring the hole as a cycle
23  CYCL DEF 12.0 PGM CALL
24  CYCL DEF 12.1 PGM 81902_en.h
25  ;
26  ;Call program 81901_en.i with positions
27  ;and cycle call
28  CALL PGM 81901_en.i
29  ;
30  ;Tool call for rework
31  TOOL CALL 5 Z S1234 F500
32  ;
33  ;Definition of point table 81903_en.pnt
34  ;as point pattern
35  SEL PATTERN "81903_en.pnt"
36  ;
37  ;Definition of program 81904_en.h for
38  ;reworking the hole as cycle
39  CYCL DEF 12.0 PGM CALL
40  CYCL DEF 12.1 PGM 81904_en.h
41  ;
42  ;Cycle call at the positions defined in the
43  ;point pattern
44  CYCL CALL PAT FMAX M3
45  ;
46  ;Retract
47  L  Z+100 R0 FMAX
48  ;
49  ;End of program
50  M30
51  END PGM 8190_EN MM
```

Program 81901_en.i:

This is a simple ISO program in which the positions at which the measuring cycle is to be executed are approached with a G0 command and then a cycle call is performed with M99.
Important! This program must not contain M30.

```
%81901_EN G71 *
;NC program with the positions at which the
;machining of the holes and the measurement
;are to be preformed.
N2 G00 G90 X+0 Y+0 Z+10*
N3 G00 G90 X-14 Y-32 M99*
N4 G00 G90 X-14 Y+0 M99*
N5 G00 G90 X-14 Y+32 M99*
N6 G00 G90 X+13.71 Y+48 M99*
N7 G00 G90 X+13.71 Y+16 M99*
N8 G00 G90 X+13.71 Y-16 M99*
N9 G00 G90 X+13.71 Y-48 M99*
N10 G00 G90 X+0 Y+0*
N99999999 %81901_EN G71 *
```

Program 81902_en.h:

In this program, the measuring cycle is programmed and the measured results are written to the table.
An FN20 function with which the NC and PLC are synchronized is programmed at the beginning of the program. This function is important in order to ensure that the following program blocks are not executed until the approached position has been reached.
The control then reads out the current position of the X axis with an FN18 function and stores the value in parameter Q1. It then reads out the value of the Y axis and stores it in parameter Q2.
The measuring cycle 421 is defined next. Here, the previously defined parameters Q1 and Q2 are used as the center point in the first and second axes. You then need to adapt other data, such as the nominal diameter and the measuring height, to the hole you want to measure.
The table to be written to (81903_en.pnt in this example) is defined after the cycle execution.
Adapt the file path here, if necessary.
The control then writes the measured results Q151 for X and Q152 for Y to the X and Y columns of the table, in the row that is defined by the counter Q0. These results contain the measured actual center point coordinates of the hole for the respective axis.
After the table has been written to, the counter Q0 for the row is incremented by one so that the next time the program is run, the control will write to the next row of the table.

```
0  BEGIN PGM 81902_EN MM
1  ;NC program with which a measuring cycle is
2  ;executed at the current tool position. This
3  ;may be necessary if the cycle is to be
4  ;executed with a cycle call at a specific
5  ;positions. For this purpose, you could
6  ;define this program in Cycle 12 and then
7  ;call it with CYCL CALL or M99.
8  ;
9  ;Wait for synchronization of NC and PLC to
10 ;ensure that the program run is not continued
11 ;until the position has been reached.
12 FN 20: WAIT FOR  SYNC
13 ;
14 ;Read out the current position of the X axis
15 ;and save the value in Q1
16 FN 18: SYSREAD Q1 = ID270 NR1 IDX1
17 ;
18 ;Read out the current position of the Y axis
19 ;and save the value in Q2
20 FN 18: SYSREAD Q2 = ID270 NR1 IDX2
21 ;
22 ;Definition of the measuring cycle at the
23 ;current position
24 TCH PROBE 421 MEASURE HOLE ~
   Q273=+Q1   ;CENTER IN 1ST AXIS ~
   Q274=+Q2   ;CENTER IN 2ND AXIS ~
   Q262=+25.2 ;NOMINAL DIAMETER ~
   Q325=+0    ;STARTING ANGLE ~
   Q247=+90   ;STEPPING ANGLE ~
   Q261=-5    ;MEASURING HEIGHT ~
   Q320=+0    ;SET-UP CLEARANCE ~
   Q260=+20   ;CLEARANCE HEIGHT ~
   Q301=+0    ;MOVE TO CLEARANCE ~
   Q275=+0    ;MAXIMUM LIMIT ~
   Q276=+0    ;MINIMUM LIMIT ~
   Q279=+0    ;TOLERANCE 1ST CENTER ~
   Q280=+0    ;TOLERANCE 2ND CENTER ~
   Q281=+1    ;MEASURING LOG ~
   Q309=+0    ;PGM STOP TOLERANCE ~
   Q330=+0    ;TOOL ~
   Q423=+4    ;NO. OF PROBE POINTS ~
   Q365=+1    ;TYPE OF TRAVERSE
25 ;
26 ;Open the table to which the results are
27 ;to be written
28 FN 26: TABOPEN 81903_en.pnt
29 ;
30 ;Write the measured results to the table
31 FN 27: TABWRITE Q0 /"X" = Q151
32 FN 27: TABWRITE Q0 /"Y" = Q152
33 ;
34 ;Increment the counter
35 Q0 = Q0 + 1
36 END PGM 81902_EN MM
```

Program 81903_en.pnt
Point table in which the center point coordinates are saved.

```
BEGIN PNT81903_EN.PNT MM
NR    X         Y        Z        FADE CLEARANCE
0    -14       -32      +0       N
1    -14       +0       +0       N
2    -14       +32      +0       N
3    +13.71    +48      +0       N
4    +13.71    +16      +0       N
5    +13.71    -16      +0       N
6    +13.71    -48      +0       N
[END]
```

Program 81904_en.h
This program is used for reworking the holes. Some tool paths programmed for machining a recess have been defined here.

```
0  BEGIN PGM 81904_EN MM
1  ;NC program for machining a recess. This is
2  ;done by movingon a circular path in two
3  ;infeeds.
4  CC IX+0 IY+0
5  L  Z-22 R0 FMAX M3
6  LBL 1
7  LP  PR+13.1  PA+0 RL F100 M8
8  RND R2
9  CP IPA+360 DR+
10 RND R2
11 LP  PR+0  PA+0 R0
12 LBL 0
13 L IZ-8 R0 FMAX
14 CALL LBL 1
15 L  Z+10 R0 FMAX M9
16 END PGM 81904_EN MM
```