DDCSV Probing Modes V2

Version Notes

V0 Original did not have a version number!!

V1 Added note re. param #69

Added section on Tool Changing

V2 Added note re: #223

Saveed as pdf as well

Introduction to Probing

In order for probing to work parameter #223 must be set to 1

In the latest release I am using [Install_2018-07-30-98 NOR] there are 4 modes supported by the controller.

The mode is selected by parameter #68 [“Tool Setting function mode"]

0 = Disable

1 = Mode 1

2 = Mode 2

3 =Mode 3

Disable

In this mode pressing 2nd/Probe/Probe will have no effect. The display says “Probe” so use the Mode key to select the chosen mode.

Mode 1

This mode uses 2 probe sequences to first establish the height of the tool sensor and then probers the tool onto the sensor. In so doing it updates the value held in #69 ["Thickness of tool sensor"].

It uses the following parameters:

69 ["Thickness of tool sensor"] It updates this parameter with the measured tool height.

Note that this parameter can have a negative value:

It measures the "height" of the tool probe comparing the stock z-zero (defined during the zeroing sequence) and z coordinate during the tool contact with the sensor. So depending on the work setup (tool probe surface is higher or lower then the stock/code z-zero) - it could be both positive and negative, as it's not a real probe height - it's a difference between part/stock zero and surface of the probe.

#71 ["Initial tool's position"]

0 = "current position" Where the probe is in the current reference frame. All examples relate to G54

1 = "fixed position" The position of the tool sensor in the MACH machine space

If “fixed” it uses:

These 3 locate the tool sensor in the Machine co-ordinate system (MACH)

#72 ["Initial probe position on X axis"]

#73 [“Initial probe position on Y axis"]

#74 ["Initial probe position on Z axis"]

#75 [“back distance after probe"] How far to retract in Z after the contact

First the user has to manually zero the z axis onto the tool table.

This also sets #69 = 0 [only in Mode 1.

Then either manually move the tool over the tool sensor [#71 = 0] or set #71 = 1 and have the correct values in #72..#74

Mode 2

This mode is the simplest. It uses the height of the tool sensor and the retract distance after the probe to set the Z height of the tool

It uses the following parameters:

#69 ["Thickness of tool sensor"] How high is the tool sensor in the Z axis.

#71 ["Initial tool's position"]

0 = "current position" Where the probe is in the current reference frame. All examples relate to G54

1 = "fixed position" The position of the tool sensor in the MACH machine space

If “fixed” it uses:

These 3 locate the tool sensor in the Machine co-ordinate system (MACH)

#72 ["Initial probe position on X axis"]

#73 [“Initial probe position on Y axis"]

#74 ["Initial probe position on Z axis"]

#75 [“back distance after probe"] How far to retract in Z after the contact

Mode 3

This mode is the most complex and useful as it probes all three axes. In order for this the tool sensor must be a cube or similar regular rectangular block. The mode will first probe for Z, then probe the side of the block [left or right] for X and finally it will probe the face or the back of the block for Y. It then retracts back as dictated by parameter #75 and moves to the X & Y zero position as defined by #2001 and #2002. Note if you want to locate onto the corner you just probed set #2001 and #2002 to 0.

Note:

#69 is NOT used

#71 is NOT used in this mode

#75 is NOT used in this mode

It uses the following parameters:

#2000 ["Cutter diameter"] The diameter of the probe tip

#2001 ["Tool plate thick for X"]

#2002 ["Tool plate thick for Y"]

#2003 [“Tool plate thick for Z"]

#2004 ["shift of X axis before probed"] If Positive it will probe the right hand side of the block.

#2005 [“shift of Y axis before probed"} If negative it will probe the front face of the block

#2006 [“Z position before X(Y)-axis probed"] The distance to descend before moving the X or Y axes to establish probe contact

#2007 ["Back distance when the tool touches the X-axis edge"] After a n axis is touched off these 3 parameters

#2008 ["Back distance when the tool touches the Y-axis edge"] define how much to pull back before the next

#2009 ["Back distance when the tool touches the Z-axis edge"] movement

#2010 ["center of tool plate"] If you want to set the X & Y zero to the corner of the test block you would set #2001 & #2002 to 0. Refer to figure 8, and if this is the case when #2010 = 0 the tip of the tool has to be exactly on the corner. If you set #2010 to a value, the tool is offset (in X) when probing for Y. This should lead to better accuracy.

#2011 ["Probe feedrate"] The feed rate to be used when probing. Probing is done by a G01 command in relative (G91) mode

Figures

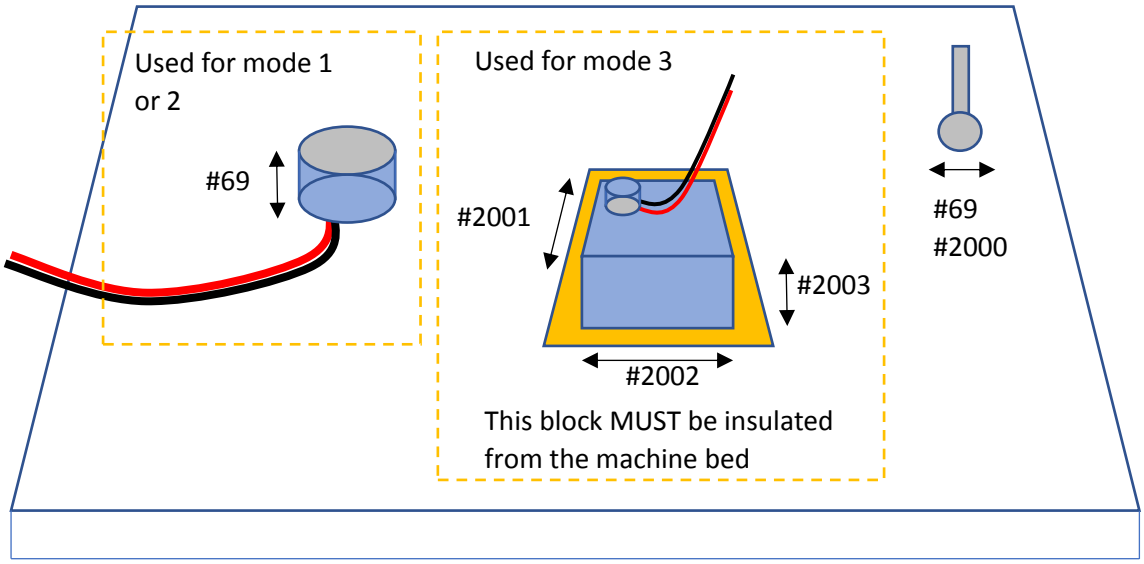

Fig. 1: Basic parameter definitions

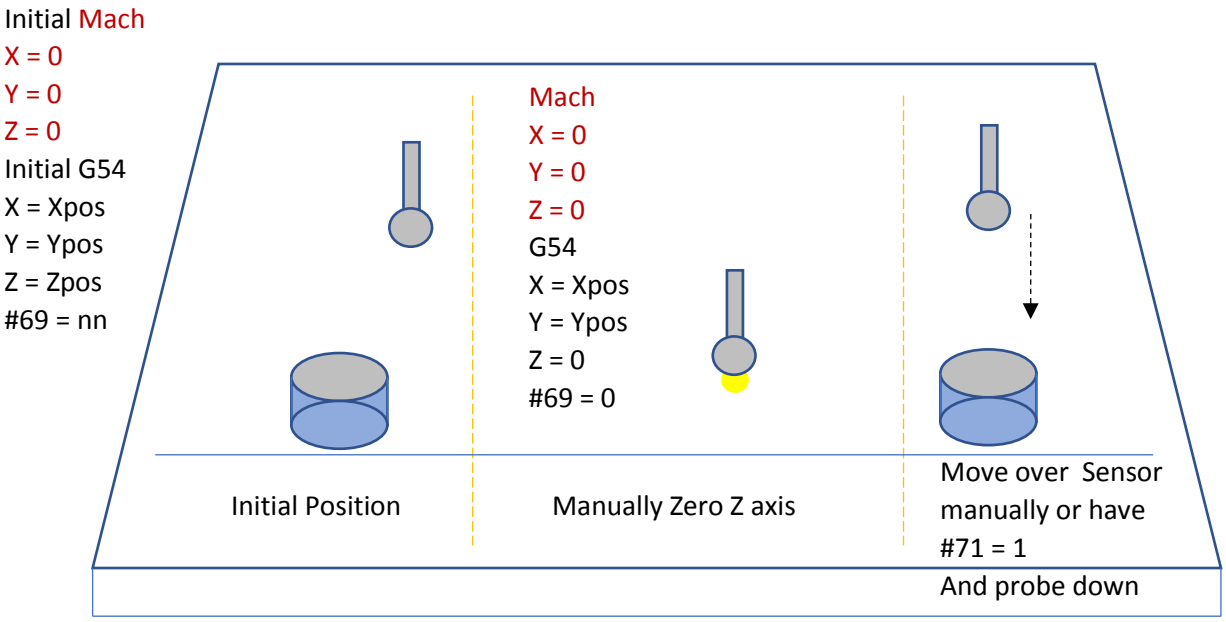

Fig. 2a: Mode 1 Probing Sequence for Z axis

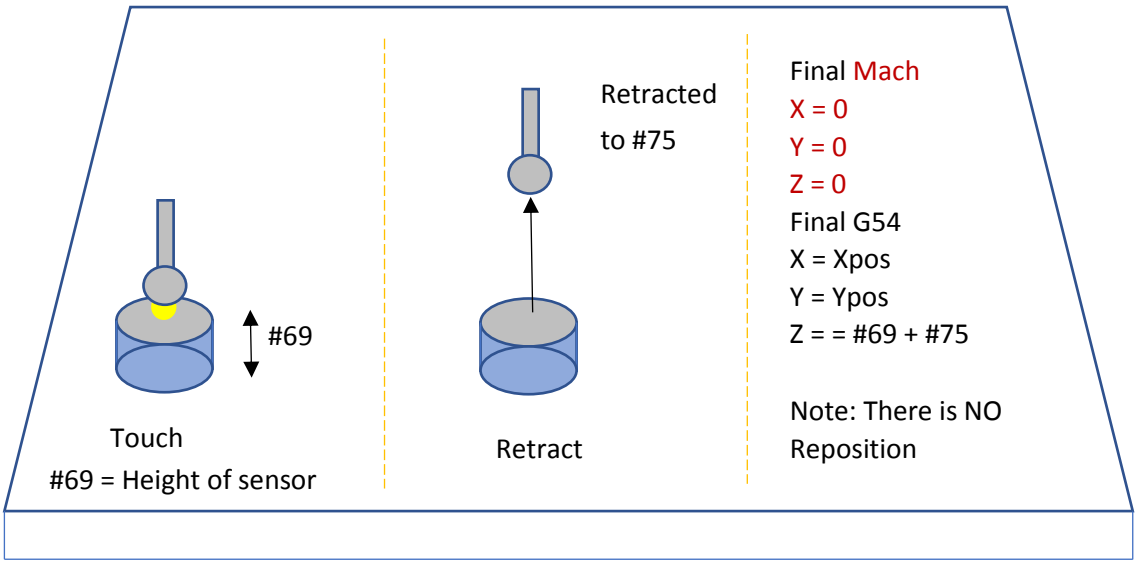

Fig. 2b: Mode 1 Probing Sequence for Z axis

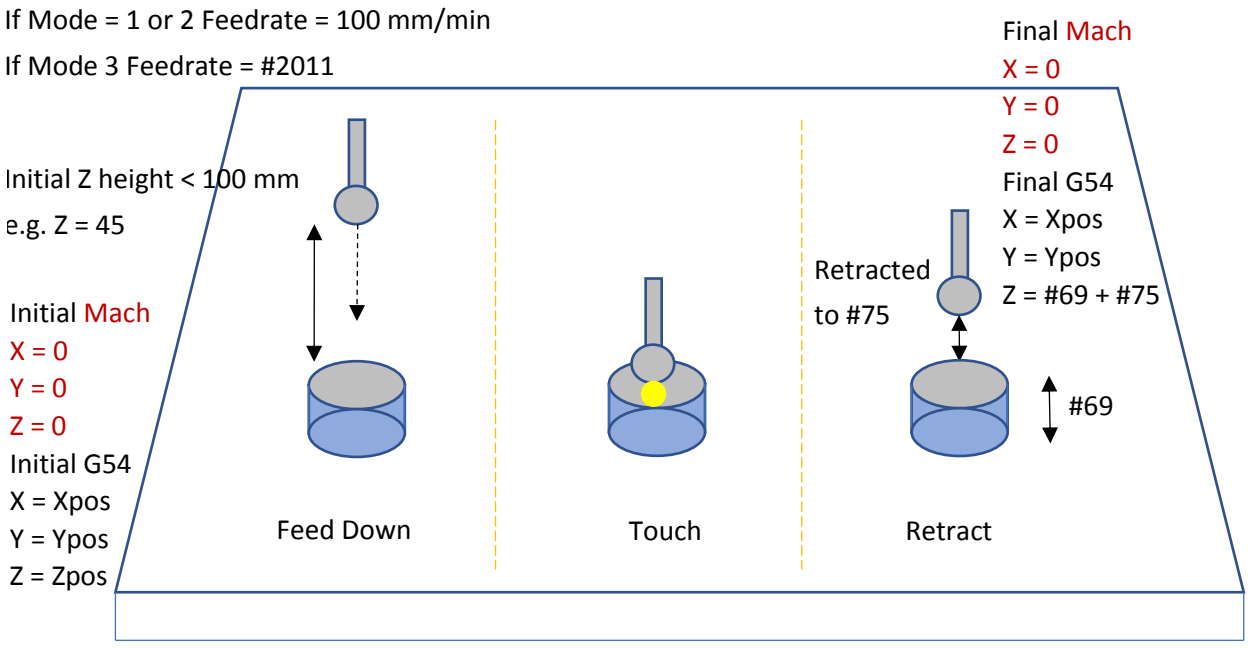

Fig. 3: Mode 2 Probing Sequence for Z axis #71 = 0: Current position

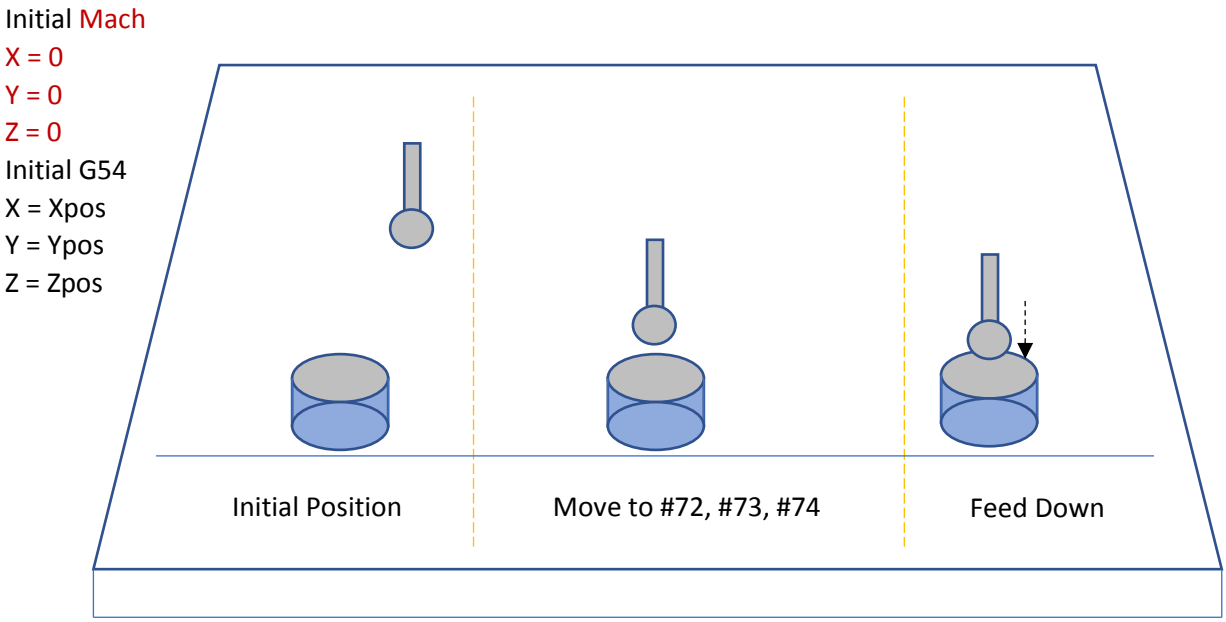

Fig. 4a: Mode 2 Probing Sequence for Z axis #71 = 1 Fixed position

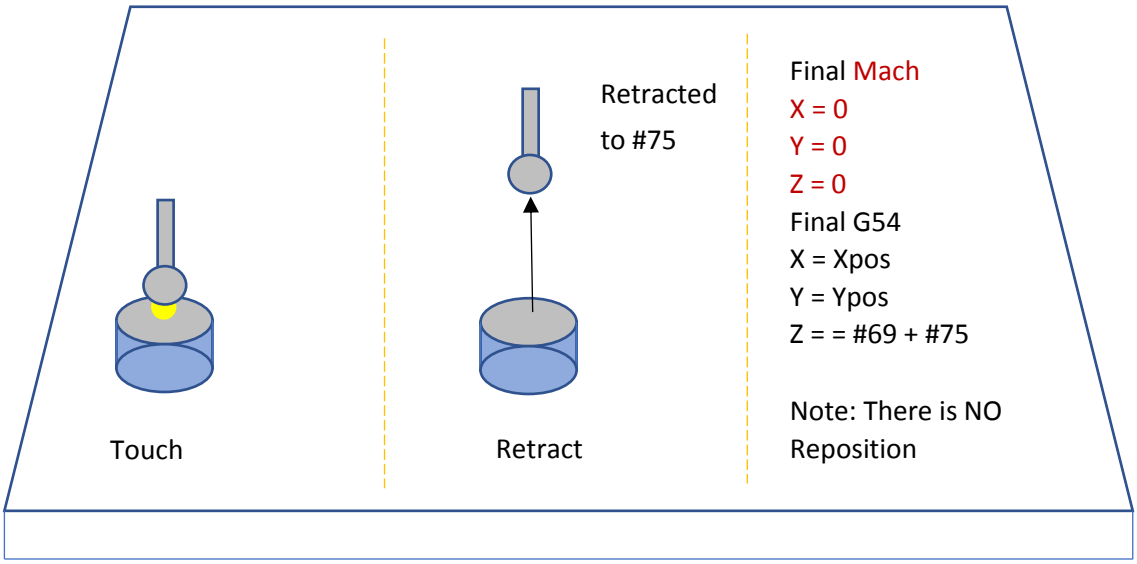

Fig. 4b: Mode 2 Probing Sequence for Z axis #71 = 1 Fixed position

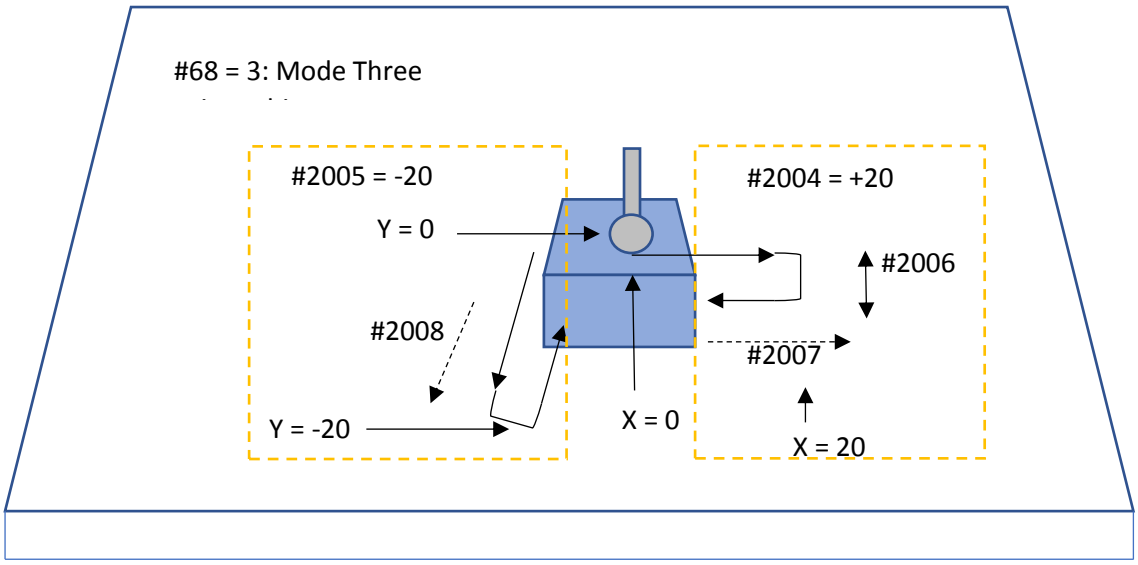

Fig. 5: Mode 3 Probing Directions

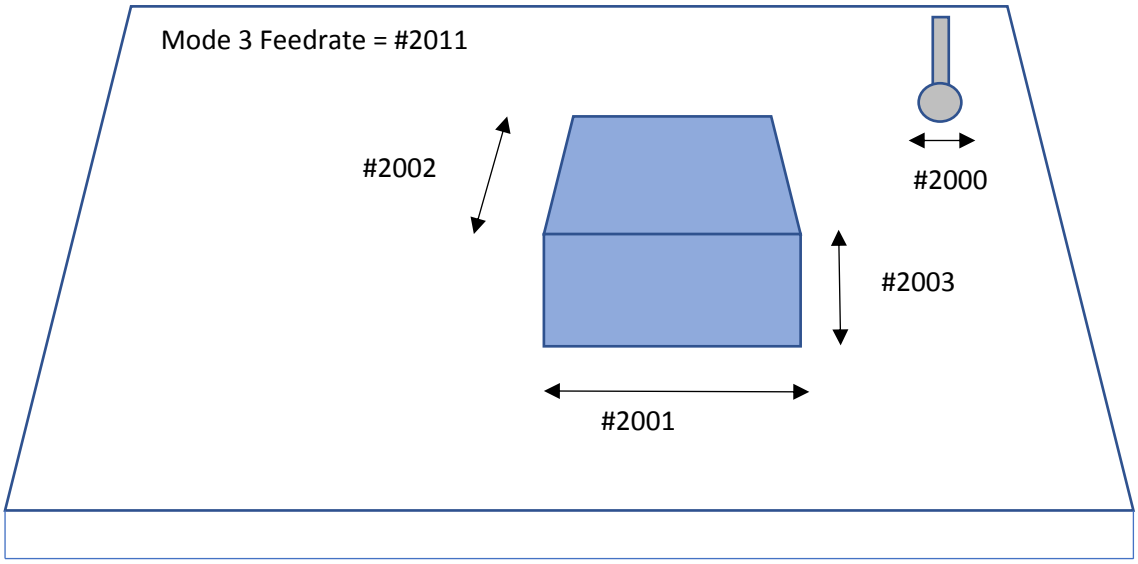

Fig. 6a: Mode 3 Test block parameters

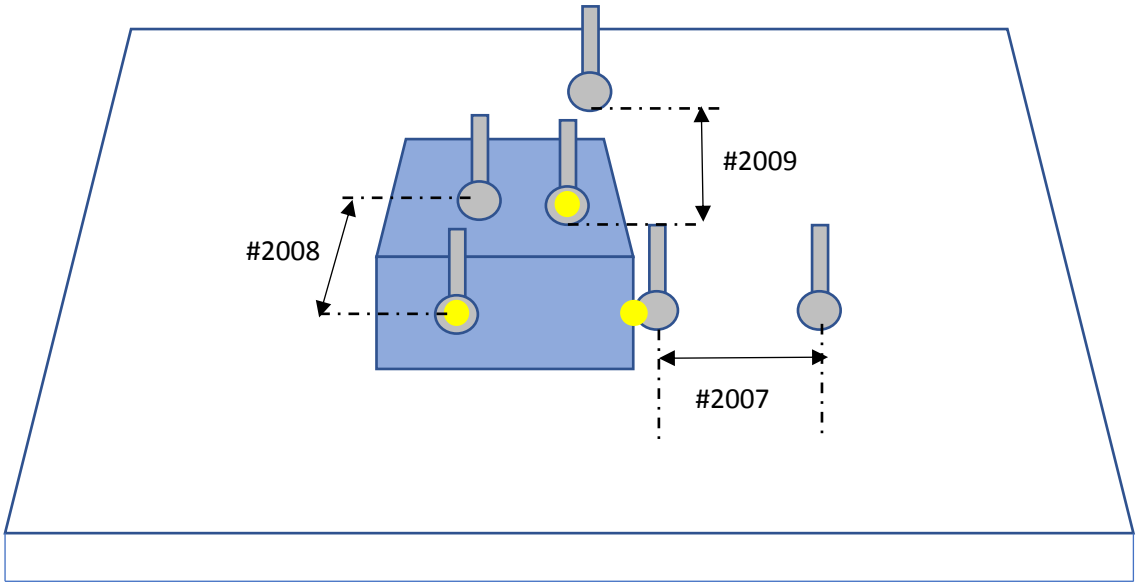

Fig. 6b: Mode 3 Test block retract parameters

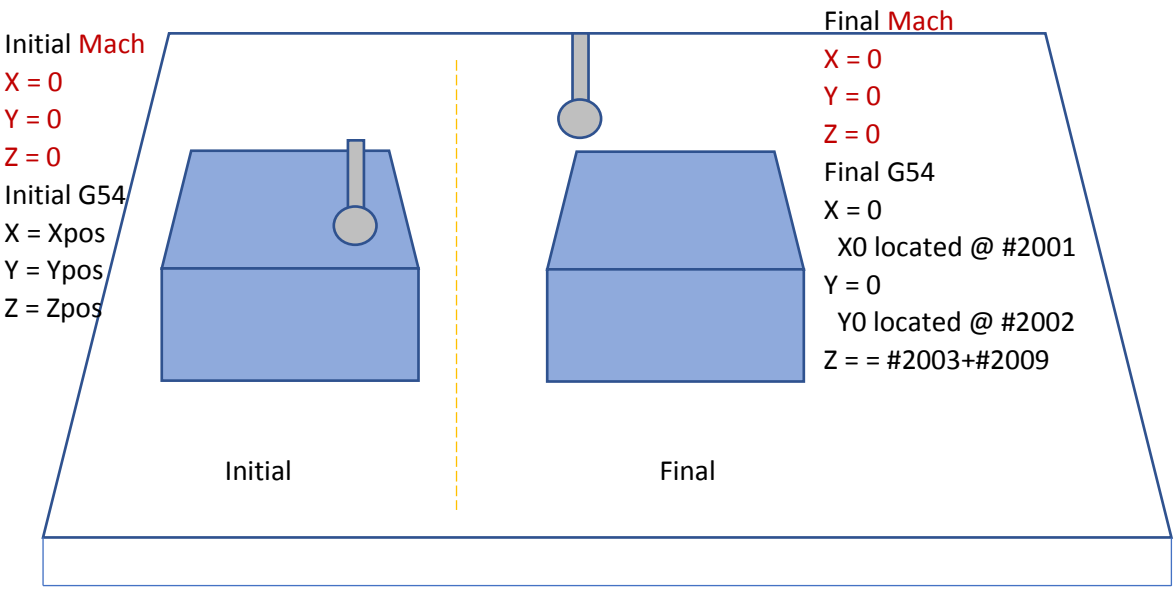

Fig. 7: Mode 3 Initial and Final Positions

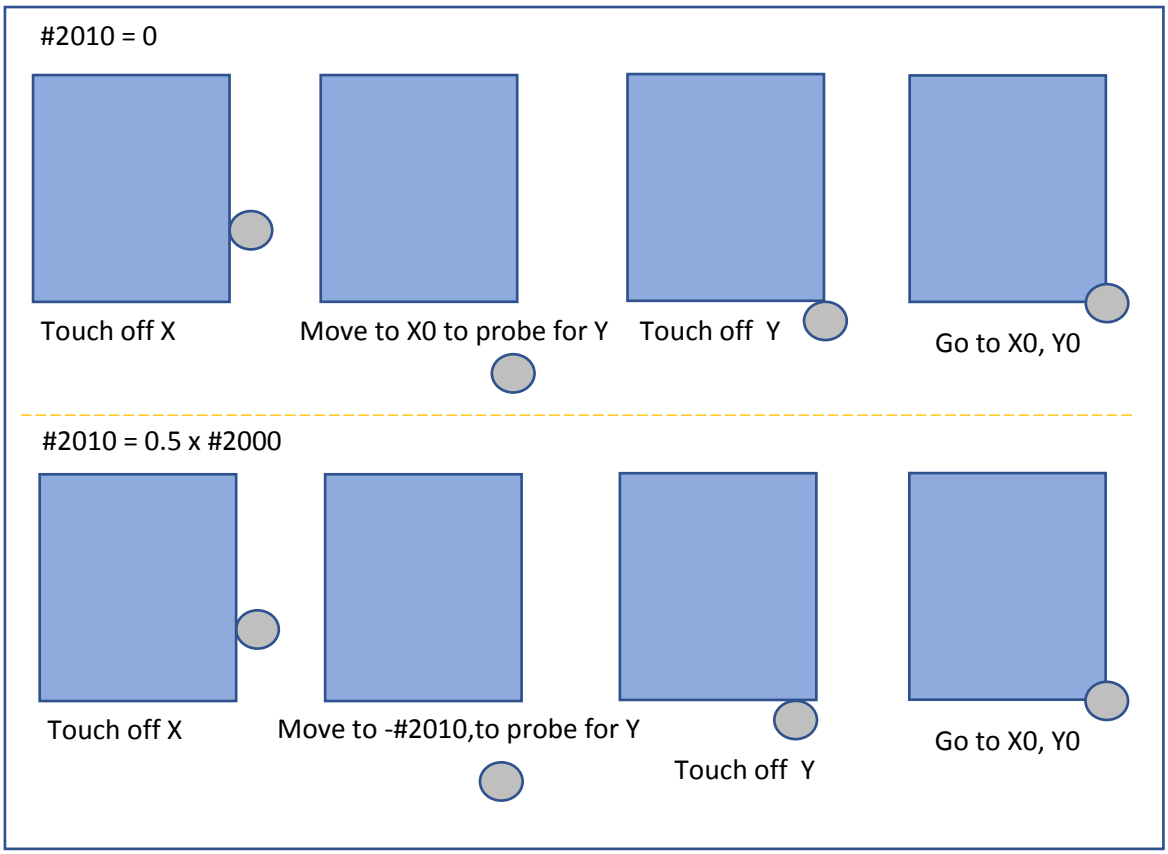

Fig. 8: Use of #2010 in Mode 3

Eng file syntax notes.

The chs or eng file configures the user-defined system parameters.

For example:

#2000 -t1 -s1"Cutter diameter" -s2"in" -m9 -min=0.000 -max=9.000

#n n is the parameter number

-t is used to specify the type of parameter, t0 is an integer, t1 is a real number, and t2 is a selection type

-s1 defines the name of the parameter

-s2 defines the parameter unit

-m parameter group, m0 that the parameters in the parameter configuration page does not show

-min minimum parameter

-max maximum parameter

-i The text of the relevant entry is configured through it when the parameter type is selection type

Tool Changing.

The system supports automated tool changing. It recognizes the M6 Tnn g-code.

The controller then tries to find a system file T.nc. If it exists the code is run, if not nothing happens TBC

Before looking at writing T.nc we need to understand the various parameters that may be used in it's code:

These are the tool height and diameter parameters:

#267 "H00 tool offset" -- #282 "H15 tool offset" Offset height from what??

#283 "D00 tool offset" -- #298 "D15 tool offset" Offset tool diameter from what??

The system keeps track of the current and required tool with these parameters:

#592: The current tool, if no tool selected yet it is 0

#593: The request tool as in M06 T02 this would be set to 02

After the tool change is completed, the current tool number will be updated: #592=#593

We also need to understand the parameters that define the various co-ordinate system in use:

We will define the position of the probe sensor in terms of MACH co-ordinates:

#864 X, #865 Y & #866 Z TBD

We store the position in #72, #73,#74

We need to know the current position in the current workspace (e.g. G54)

So for G54:

X = #451, Y = #452, Z = #453

T.NC Objective:

The objective is to support a manual tool change for a tool of unknown length(so a probe is done to establish the Z height) and integrate it into the g-code workflow with minimum operator intervention.

T.NC Requirements

The requirements for T.nc can be stated as follows

Given a tool sensing probe located at #72, #73,#74

Given that the spindle is currently at #451, #452, #453 and may or may not have a tool fitted and may or may not be spinning:

Stop Spindle, stop any coolants etc

Go to tool sensor position

Pause to allow manual tool change

User presses start

Probe the tool onto the sensor plate

Retract to safe Z

Return to initial position

Restart coolant etc

Restart spindle

Pause for safety check

User presses start

End Macro

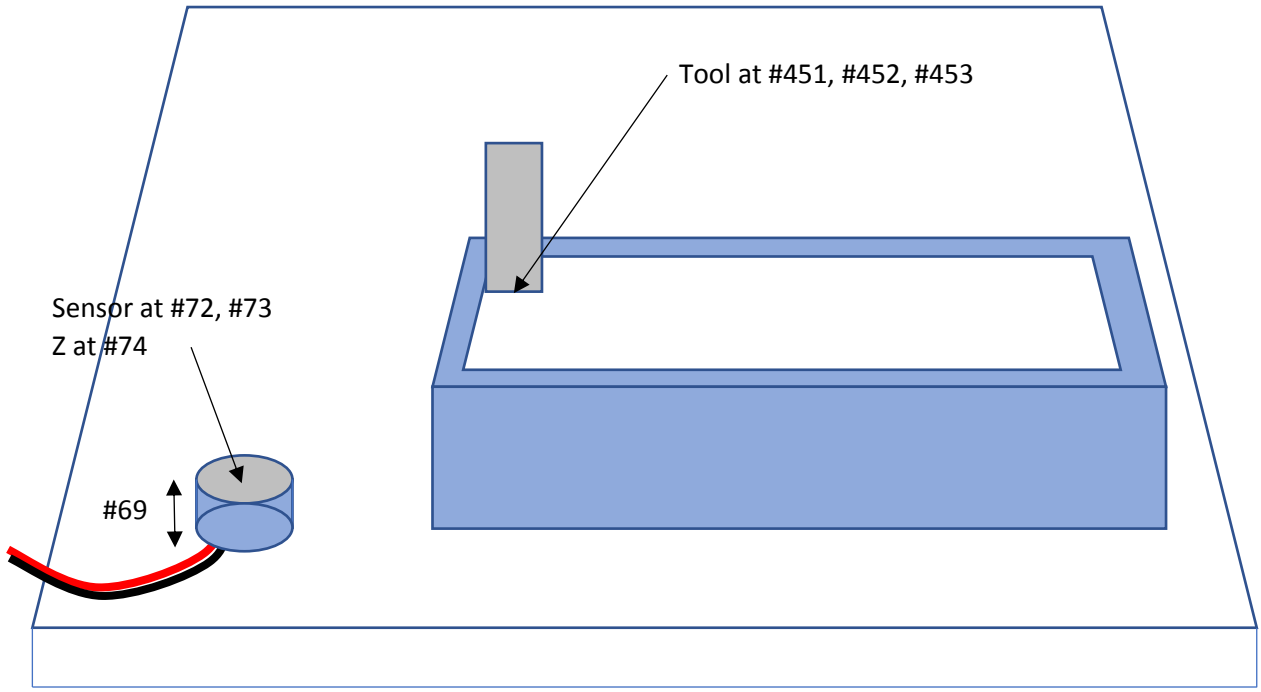

Fig. 1: Initial set ups

Some code snipets:

;Probe.nc

```
04P0; Pause 0s, read the current machine coordinate position correctly for subsequent procedures
M5;Close the spindle
(Read the current tool mechanical coordinate position)
#20=#864
#21=#865
#22=#866
IF#571EQ0GOTO1;Discrimination system using a fixed position on the knife mode or the current position of the knife mode
(Fixed feed mode for X, Y, Z)
#1=#572-#20
#2=#573-#21
#3=#574-#22
GOTO2
(In the current tool setting mode, X, Y and Z feeds are cleared)
N1#1=0
#2=0
#3=0
(Move to the initial position of the knife)
N2G91G00Z#3
G91G00X#1Y#2
(At 100 speed probe 100mm detection tool signal)
N1M101
G91G01Z-100F100
M102
G04P0;Pause 0s
#402=#400;Save the Z axis zero offset of the coordinate system
#403=1;Set automatic correction coordinate system logo
#404=-#870;Save the tool thickness, if the previous tool thickness parameter is 0, the system will use this variable to modify the tool thickness parameter to complete the first knife
G91G01Z#575F#578;The knife is completed, Z axis back
```

The 2 sub-routines

```
(O101 sub program: safez)
O101
IF#516EQ0GOTO 0
IF#516EQ1GOTO 1
IF#516EQ2GOTO 2
IF#516EQ3GOTO 3
IF#516EQ4GOTO 4
IF#516EQ5GOTO 5
IF#516EQ6GOTO 6
```

```
IF#516EQ7GOTO 7
GOTO 30
N0 #1 = 868
GOTO 10
N1 #1 = 840
GOTO 10
N2 #1 = 844
GOTO 10
N3 #1 = 848
GOTO 10
N4 #1 = 852
GOTO 10
N5 #1 = 856
GOTO 10
N6 #1 = 860
GOTO 10
N7 #1 = 864
GOTO 10
N10 #2 = #[#1+2]
#5 = #582-#2
IF#5LT0GOTO 20
G00 Z#582
N20 #3 = 0
M99

(O102 operation record track)
O102
#1 = 0
WHILE #1 <= #108 DO2
RDRECODE[#1]
G01 X#104 Y#105 Z#106 A#107
#1 = #1+1
END2
M99
```

From ytliu:

The user can write T.nc according to the actual demand and copy it to the system directory (/mnt/nand1-1/) to complete the extension definition of tool change instruction M6.

Example 1:

Manual tool change while encountering M6 command in machining, T.nc is implemented as follows:

G04P-1; external pause

Example 2:

Automatic Tool Changer, T.nc achieved as follows:

#1=105; In-line magazine first knife X position, set according to the actual situation
#2=50; The distance between each knife in the row magazine is set according to the actual situation
M05; Close the spindle

(The following is the retraction process)
IF [#592 <1] GOTO1; 592 variables for the current folder tool number, if less than 1, the description is empty knife, skip the retraction process, enter the folder knife process
G53 G00 G90 X[#1+[#592-1]*#2] Y2500 Z-126; Rapid traverse to the machine coordinate tool gate
G04 P5000; Pause until the spindle is completely stopped
G53 G00 G90 X[#1+[#592-1]*#2] Y2550 Z-126; Enter tool magazine
G04 P500; Pause to the holder on the holder
M08; Open the spindle tool change pneumatic valve
G04 P500; pause 0.5 seconds
G53 G00 G90 Z-20; raise the spindle
M09; Close the spindle tool change pneumatic valve
GOTO2

N1 G04 P5000; Pause until the spindle stops completely

(The following is the folder knife process)
N2 G53 G00 G90 X[#1+[#593-1]*#2] Y2500 Z-126; Rapid traverse to the machine coordinate tool gate
G53 G00 G90 X[#1+[#593-1]*#2] Y2550 Z-126; Enter tool magazine
G04 P500; Pause to the holder on the holder
M08; Open the spindle tool change pneumatic valve
G04 P500; pause 0.5 seconds
G53 G00 G90 Z-20; raise the spindle
M09; Close the spindle tool change pneumatic valve

G04P0; Synchronization
#592=#593; After the tool change is completed, the current tool number will be updated

Related instructions:
The #593 parameter is used to receive the tool number to be replaced. If the T3M6 instruction is executed, the system assigns 3 to #593.

#592 The parameter is the number of the currently used tool. When it is 0, it indicates that the tool is not currently installed.

Sorry, there was a problem with the previous statement.
If the G04P-1 or M0 command appears in the program header, it will be ignored, so the question you said will appear.
T.nc can be written like this
M5
G04P-1
or
M5
M0
In this way, when executing M6, the spindle will be shut down first, and then the program will be suspended. When the start key is pressed again, the program will continue to execute.

See also:
D:\CNC_Stuff\Probe_Test\DDCSV self-made tool auto-checking.pdf

From Gerald Ohlms

```
%
N0010 G40 G17 G90 G54
N0020 (TOOL PATH: EXTERNAL_THREADS)
N0030 (TOOL CHANGE LOAD TOOL T01 )
N0040 M98 P6666
N0050 M00
N0060 G54
N0070 G00 X-.4848 Y-.5222
N0080 G43 Z.5 H01
N0090 Z.1
N0100 G02 X-.6121 Y-.0227 I1.0939 J.5448 F10.
N0110 G01 X-.6088 Y.0759 Z.0992
N0120 X-.5887 Y.1725 Z.0985
N0130 X-.5534 Y.2648 Z.0977
~
~
~
N4020 X-.5222 Y.4848 I1.2212 J.0453
N4030 G00 Z.5
N4040 G49 G53 Z-.1
N4050 M02
```

M98P6666 calls my simple tool change program which just moves to a specific location for now.

G43Z.5H01 moves the tool tip to .5 inches above the Z Zero

The most important thing is when you set your Z zero, typically (the way I do it) there is no Tool Length offset set so you have to set Z zero relative to the part of the of spindle that locates the tool which is usually the gage line of the spindle. My spindle doesn't have a gage line you can see, so I set my Z with a too that I know the exact length of and that length is what I set my Z to.

I never tried calling tool length offset before setting Z so I am not sure if that would work or not, but my way works, so that's how I do it.

G49 G53 Z-.1 cancels the offset and moves the spindle to mach Z-.1. I do this because G49 by itself causes motion on this control (moves Z the amount of the current tool length offset). You might be able to do a G91G49 as well but I have never tried that...

Sorry this was so long. Seemed shorted in my head hahaha

O6666

(TOOL CHANGE POSITION)

G53G49G00Z-.1

G53G00X-11.5Y-3.5

M99

I just added this to slib.nc