Tool Changing on the DDCSV 3.1

Applicable to version 2019-10-19-112 and later

Change Log:

24/10/19 Version 1

Contents

Introduction

This document explores a way to semi automate manual tool changes with the existing and possibly "to be added" features of the DDCSV controller.

My Sieg SX2Plus mill has 3 axis control but cannot support auto tool changing as manual work with a spanner is needed to re-tighten the tooling into the R8 spindle

Objectives

Reduce the amount of probing when changing tools.

Support a tool library

The Tormach™ System: TTS and Mach 3

The proposed system is based on the Tormach™ Tool holder system referred to as TTS

Here are links to the Tormach™ site for the tool holders:

https://www.tormach.com/store/index.php?app=ecom&ns=catshow&ref=TOOLING_MILL_TTS_HOLDER&portrelay=1

https://www.tormach.com/store/index.php?app=ecom&ns=catshow&ref=ER_collet_TTS&portrelay=1

Essentially this system allows for a tool to permanently mounted in a holder (either ER or Weldon or Jacobs chuck), measured and it's length entered into the CNC controller's tool library. The collets can then be REPEATABLY re-fitted into the mill spindle. This is achieved by a special ¾" R8 arbour which does not protrude beyond the spindle nose so that the tool holder body registers onto the spindle nose. This means that the collect body seats onto the spindle itself and so ensure repeatability.

A side benefit is that the R8 arbour stays in the machine and so only a quarter or half turn of the drawbar is needed to release and re-clamp the tool holder

Here is a u-tube link to the system in use:

https://www.youtube.com/watch?v=nG96bIyHIQk

Now on a Mach3 controller you can set the up the tool lengths and when running G-Code the tool lengths will be changed when you command a tool change in the code. Importantly you can also tell the controller you have changed the tool when running the machine manually.

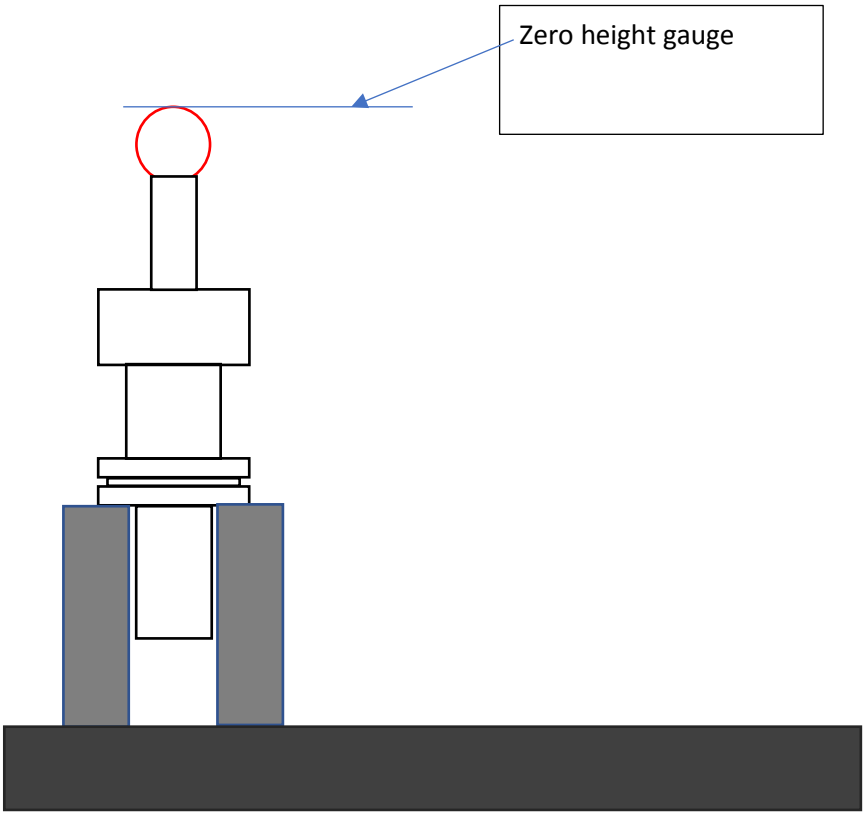
Here is a (rather long) video explaining all this:

https://www.youtube.com/watch?v=yE2NWeIzuXA

How it should work:

Tool Zero is going to establish the initial Z height (and also X & Y datums as well if required)

You fit it into its holder set on 1-2-3 blocks of similar and zero your height gauge on its tip:



For the other tools we need to know the difference in length from T0

So again, set them on the 1-2-3 blocks, and measure the difference in heights from T0

Note: Shorter tools are given a negative Hnn offset

Repeat this procedure for your other tools and record the offsets in parameters #268 to #282

Sample Code and Simple tests

These instructions apply to version 2019-10-19-112 and later

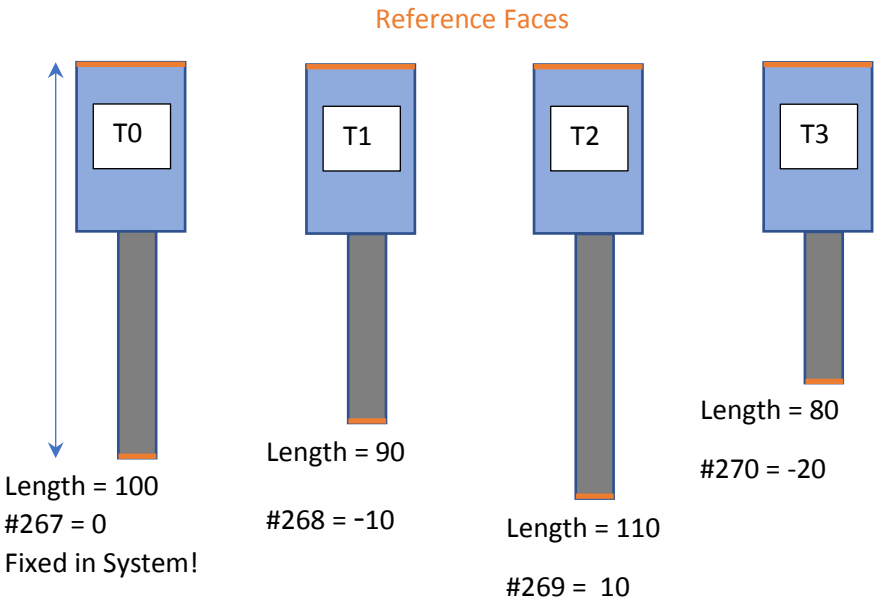
Manual Tool Setting

This release contains a dedicated T.nc file to support manually resetting tools already held in accurate fixtures that allow repeatability.

First let us review our tools:

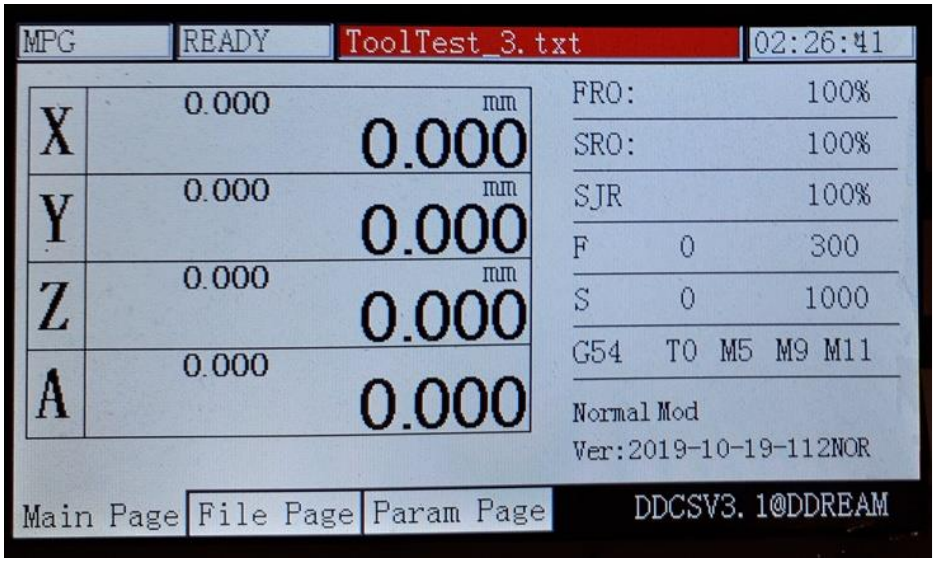
T0, is our reference tool. It is suggested to use an edge finder or similar like a Haimer to establish work X, Y and Z zeros

We measure our other tools against its length and record the difference, shorter tools are given a negative Hnn offset



When finished measuring enter the vales into the tool Length offset parameters

Next make sure Tool 0 is selected and displayed on the screen



If not:

Tab down to the T window next to the selected workspace and press the A- button to select the contents and use the Y+ and Y- buttons to select tool 0.

Press enter [Z-], wait for the 3 beeps, then press start.

Then fit our Tool 0 and use it to establish Z0 (and X and Y if you want to)

Manually raise Z to a comfortable place to change the tools (given the length of the holders)

Remove T0 and fit another tool as above. Confirm that the tool is in place by pressing Start after the 3 beeps,

So now zero out Tool 0 onto the selected workface (the workpiece itself or the vice or whatever)

So Z (G54) is now 0

When this is done select the chosen tool following the guidelines above and you should see that the Z coordinate values changes to allow for the new tool.

For example:

Select Tool 1 and Z should change to Z-10

Select Tool 2 and Z should change to Z+10

Select Tool 3 and Z should change to Z-20

G-Code Tool Changing

Always make sure Tool 0 is shown on the screen.

For this example, the 3 tools are longer than T0.

```
N010 G17 G21 G49 G54 G90

(Set #268 = +05.0 T1)
(Set #269 = +10.0 T2)
(Set #270 = +15.0 T3)

(Establish X,Y,Z with tool 0)
(For this code top of workpiece is Z0)

N020 G00 X0.0 Y0.0 Z0.0

     F 250
     M03
     S 1000

N030 G00 Z30
N040 T1 (is longer than T0)
N050 M06 (4mm long end mill)
         (Z increments by 5mm)

N070 G01 Z0.0
N080 G01 X10.0
N090 G01 X0.0
     M00 (Check for a 5mm gap)


N130 G00 Z30
N140 T2 (is longer than T0)
N150 M06 (2mm long end mill)
         (Z increments by 5mm)

N170 G01 Z0.0
N180 G01 X10.0
N190 G01 X0.0
     M00 (Check for a 10mm gap)


N230 G00 Z30
N240 T3 (is longer than T0)
N250 M06 (2mm long end mill)
         (Z increments by 5mm)

N270 G01 Z0.0
N280 G01 X10.0
N290 G01 X0.0
     M00 (Check for a 15mm gap)

M30
%
```