

Adding a NEXT tool call to Heidenhain iTNC530 NC Formats

June 5, 2007

1.0 Overview

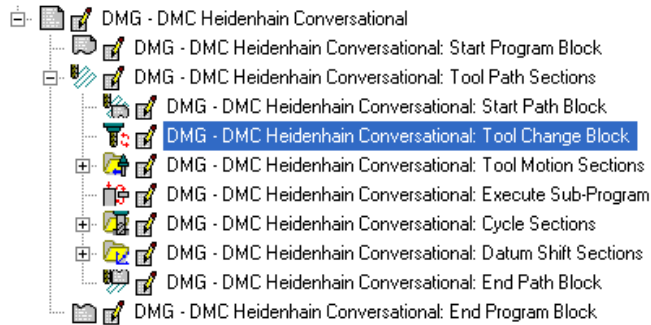
This document explains how you can add a NEXT tool call to a Heidenhain NC Format so that the tool changer can “look ahead” to select the next tool. This is useful in machines with very large tool changer magazines. The output G-Code would look like this:

```
...
590 TOOL CALL "1200" Z S700
591 TOOL DEF "1350"
...
```

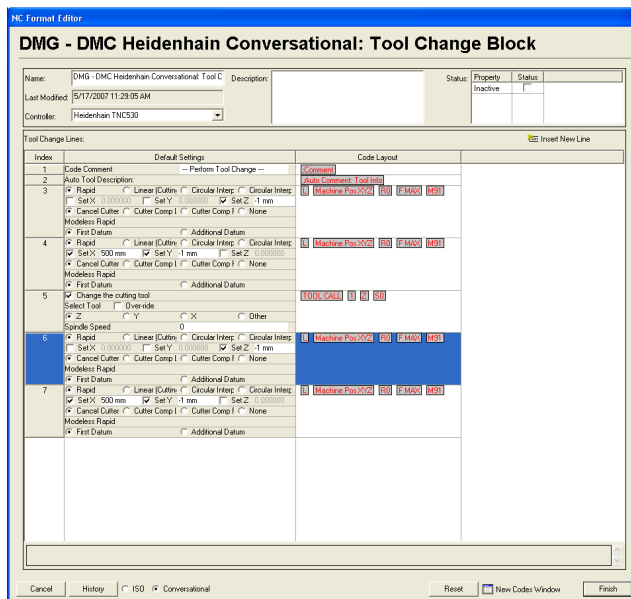
2.0 Modifying the NC Format

In order to output the NEXT tool call, you need to modify your NC Format(s) to control where this code will appear. Normally, you would place this code in your TOOL CHANGE block, to have it appear after the tool call for the current tool:

- 1) Start the CAMplete TruePath application
- 2) Go to Tools > Edit NC Formats
- 3) Select the NC Format section you wish to use
- 4) Find the Tool Change section



- 4) Click Edit
- 5) Insert a new line after the existing TOOL CALL line. Do this by selecting the line AFTER the TOOL CALL and click Insert New Line:

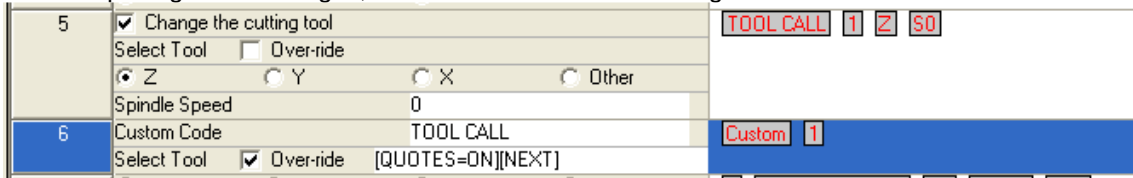


- 6) Click New Codes Window

Adding a NEXT tool call to Heidenhain iTNC530 NC Formats

June 5, 2007

- 7) Select the Custom Code and drag it onto the new line
- 8) Select the T<Tool ID> code and drag it next to the Custom Code you just added
- 9) For the custom code, enter TOOL CALL as the text
- 10) For the Select Tool code, click Over-ride and enter [QUOTES=ON][NEXT] as the text
- 11) After completing these changes, it should look like the following:



- 12) Click Finish to commit your changes
- 13) Now when you load your project, you should get the next tool call appearing