

# Adding Spindle Speed changes to Heidenhain iTNC530 NC Formats

February 25, 2011

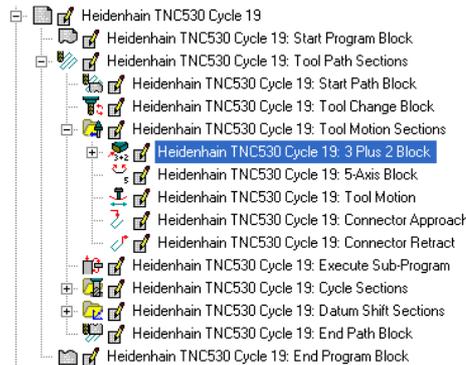
## 1.0 Overview

This document explains how you can add spindle speed output to each operation in a Heidenhain NC Format. Normally, the default NC Format only output the spindle speed in the TOOL CALL line whenever the tool changes (The Heidenhain always requires that a TOOL CALL be issued when the spindle speed changes). By adding a second TOOL CALL line at the beginning of the operation, you can ensure that every operation can have a unique spindle speed. The repeated TOOL CALL will not initiate a tool change.

## 2.0 Modifying the NC Format

In order to output the speed change, you need to modify your NC Format(s) to control where this code will appear. Normally, you would place this code in your 3+2 Block and 5-axis Block sections. The extra TOOL CALL code will not initiate a tool change, but will update the speed change.

- 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 3+2 Block section



- 4) Click Edit
- 5) Insert a new line near the beginning of the program. Do this by selecting the line AFTER the new location and click Insert New Line:

Index	Default Settings	Code Layout
1	Code Comment	-- Approach 3+2 Plane --
2		[Comment]
3	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp. <input type="radio"/> Circular Interp. <input checked="" type="checkbox"/> Set A [NEXT] <input checked="" type="checkbox"/> Set B [NEXT] <input checked="" type="checkbox"/> Set C [NEXT] <input type="checkbox"/> Cancel Cutter <input type="checkbox"/> Cutter Comp I <input type="checkbox"/> Cutter Comp F <input type="checkbox"/> None Modeless Rapid <input type="checkbox"/> Disabled <input checked="" type="checkbox"/> Go Shortest Distance	L [Machine Pos ABC] R0 F MAX M126
4	WP CS Shift Enable Trans	CYCL DEF 7.0 DATUM SHIFT
5	WP CS Shift Set X Trans	CYCL DEF 7.1 X
6	WP CS Shift Set Y Trans	CYCL DEF 7.2 Y
7	WP CS Shift Set Z Trans	CYCL DEF 7.3 Z
8	WP CS Set Plane Enable Set Plane	CYCL DEF 19.0 WORKING PLANE
9	WP CS Set Plane Set Rotations	CYCL DEF 19.1 [A=NEXT][C=NEXT]
10	Path Tolerance Enable Path Tolera	CYCL DEF 32.0 TOLERANCE
11	Path Tolerance Path Tolerance Lin	CYCL DEF 32.1 T0
12	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp. <input type="radio"/> Circular Interp. <input type="checkbox"/> Stop <input checked="" type="checkbox"/> Clockwise <input type="checkbox"/> Counter Clockwise	L M03
13	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp. <input type="radio"/> Circular Interp. <input checked="" type="checkbox"/> Set X [NEXT] <input checked="" type="checkbox"/> Set Y [NEXT] <input type="checkbox"/> Set Z 0.000000 Modeless Rapid <input type="checkbox"/> Coolant OFF <input type="checkbox"/> Flood <input type="checkbox"/> Mist <input type="checkbox"/> Through Spinc	L [Machine Pos XYZ] F MAX Coolant

- 6) Click New Codes Window
- 7) Select the Custom Code and drag it onto the new line. Enter TOOL CALL for the text
- 8) **OPTIONAL: Some versions of the iTNC require the T code and the axis code Z.** Select the T<Tool ID> code and drag it next to the Custom Code you just added
- 9) **OPTIONAL:** Select the Tool Axis Definition code and drag it onto the new line
- 10) Select the Spindle Speed code and drag it onto the new line
- 11) It should now look like this:

# Adding Spindle Speed changes to Heidenhain iTNC530 NC Formats

February 25, 2011

Index	Default Settings	Code Layout
1	Code Comment --- Approach 3+2 Plane ---	.Comment
2	Custom Code TOOL CALL Select Tool <input type="checkbox"/> Over-ride <input checked="" type="radio"/> Z <input type="radio"/> Y <input type="radio"/> X <input type="radio"/> Other Spindle Speed 0	Custom 1 Z S0
3	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp. <input type="radio"/> Circular Interp. <input checked="" type="checkbox"/> Set A [NEXT] <input checked="" type="checkbox"/> Set B [NEXT] <input checked="" type="checkbox"/> Set C [NEXT] <input type="checkbox"/> Cancel Cutter <input type="checkbox"/> Cutter Comp L <input type="checkbox"/> Cutter Comp F <input type="checkbox"/> None Modeless Rapid <input type="radio"/> Disabled <input checked="" type="radio"/> Go Shortest Distance	L Machine Pos ABC R0 F MAX M126
4	WP CS Shift Enable Trans	CYCL DEF 7.0 DATUM SHIFT
5	WP CS Shift Set X Trans [FROM_PATH_CS]	CYCL DEF 7.1 X
6	WP CS Shift Set Y Trans [FROM_PATH_CS]	CYCL DEF 7.2 Y
7	WP CS Shift Set Z Trans [FROM_PATH_CS]	CYCL DEF 7.3 Z
8	WP CS Set Plane Enable Set Plane	CYCL DEF 19.0 WORKING PLANE
9	WP CS Set Plane Set Rotations [A=NEXT][C=NEXT]	CYCL DEF 19.1 [A=NEXT][C=NEXT]
10	Path Tolerance Enable Path Tolera	CYCL DEF 32.0 TOLERANCE
11	Path Tolerance Path Tolerance Lin 0.011	CYCL DEF 32.1 T0
12	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp. <input type="radio"/> Circular Interp. <input type="radio"/> Stop <input checked="" type="radio"/> Clockwise <input type="radio"/> Counter Clockwise	L M03
13	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp. <input type="radio"/> Circular Interp. <input checked="" type="checkbox"/> Set X [NEXT] <input checked="" type="checkbox"/> Set Y [NEXT] <input type="checkbox"/> Set Z 0.000000 Modeless Rapid <input type="checkbox"/> Coolant OFF <input type="checkbox"/> Flood <input type="checkbox"/> Mist <input type="checkbox"/> Through Spindle	L Machine Pos XYZ F MAX Coolant

12) Click Finish to commit your changes

13) Now when you load your project, you should get spindle speed and tool call re-issued for every operation