

1.0 Overview

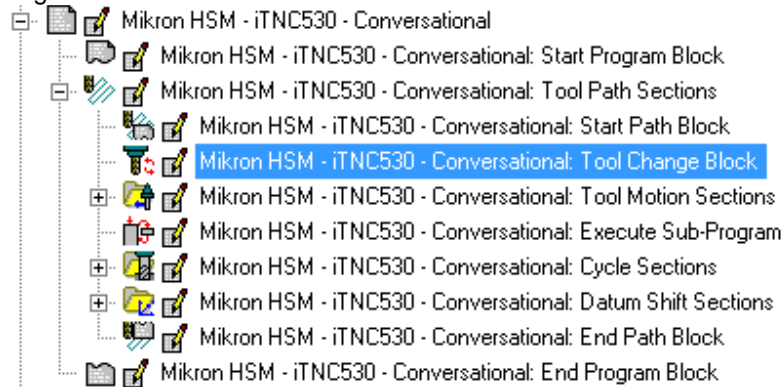
This document explains how you can add a NEXT tool call to a Heidenhain NC Format so that the tool changer can “look ahead” to select the next tool. This is useful in machines with very large tool changer magazines. The output G-Code would look like this:

```
...
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:

Tool Change Lines:

Index	Default Settings	Code Layout
1	Code Comment --- Perform Tool Change ---	:Comment
2	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cuttin: <input type="radio"/> Circular Interp: <input type="radio"/> Circular Interp: Tool Axis Retract	L Tool Axis Retract
3	Auto Tool Description:	:Auto Comment: Tool Info
4	<input checked="" type="checkbox"/> Change the cutting tool 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	TOOL CALL 1 Z S0
5	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cuttin: <input type="radio"/> Circular Interp: <input type="radio"/> Circular Interp: Tool Axis Retract [MAX]	L Tool Axis Retract
6	<input type="radio"/> Stop <input checked="" type="radio"/> Clockwise <input type="radio"/> Counter Clockwise	M03
7	Path Tolerance Enable Path Tolera ▾	CYCL DEF 32.0 TOLERANCE
8	Path Tolerance Path Tolerance Lin ▾ [DEF_LINEAR_TOL=C	CYCL DEF 32.1 T0
9	Path Tolerance Path Tolerance Ro ▾ [DEF_ROTARY_TOL=	CYCL DEF 32.2 HSC-MODE:0 TA0

Adding a NEXT tool call to Heidenhain iTNC530 NC Formats

June 30, 2008

- 6) Click New Codes Window
- 7) Select the T<Tool ID> code and drag it to the new line you just added
- 8) For the Select Tool code, click Over-ride and enter [NEXT] as the text
- 9) After completing these changes, it should look like the following:

Tool Change Lines:

Index	Default Settings	Code Layout
1	Code Comment --- Perform Tool Change ---	;Comment
2	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp <input type="radio"/> Circular Interp Tool Axis Retract	L Tool Axis Retract
3	Auto Tool Description:	;Auto Comment: Tool Info
4	<input checked="" type="checkbox"/> Change the cutting tool 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	TOOL CALL 1 Z S0
5	Select Tool <input checked="" type="checkbox"/> Over-ride [NEXT]	1
6	<input checked="" type="radio"/> Rapid <input type="radio"/> Linear (Cutting) <input type="radio"/> Circular Interp <input type="radio"/> Circular Interp Tool Axis Retract [MAX]	L Tool Axis Retract
7	<input type="radio"/> Stop <input checked="" type="radio"/> Clockwise <input type="radio"/> Counter Clockwise	M03
8	Path Tolerance Enable Path Tolera	CYCL DEF 32.0 TOLERANCE
9	Path Tolerance Path Tolerance Lin [DEF_LINEAR_TOL=C	CYCL DEF 32.1 T0
10	Path Tolerance Path Tolerance Ro [DEF_ROTARY_TOL=	CYCL DEF 32.2 HSC-MODE:0 TA0

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