

Reading the CYCLE 247 Datum from MasterCAM

October 2008

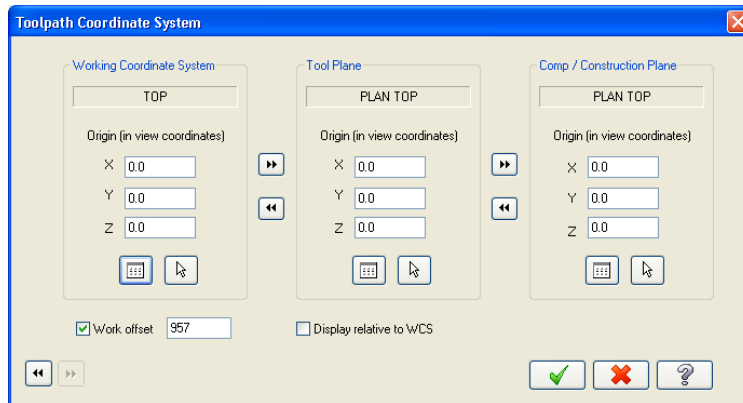
1.0 Overview

This document explains how you can read the CYCLE 247 datum from MasterCAM NCI and use that to post the G-Code. It also explains how to change the datum settings through the CAM Wizard.

2.0 MasterCAM Setup

In MasterCAM you must select the appropriate datum for each tool path (operation). To do this, g

- 1) In MasterCAM, go to the Parameters for the operation
- 2) Click Planes...
- 3) In the Planes dialog select the appropriate "view" for your Working Coordinate System. You can also type in the offset number.



- 4) Now when you create an NCI file, the datum information will be used.

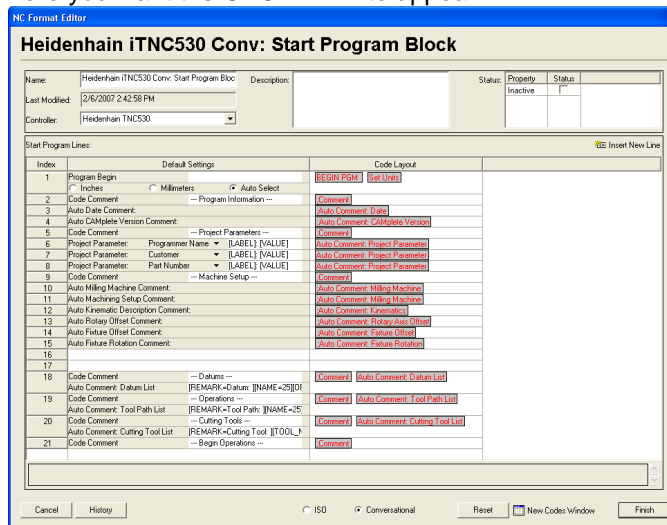
Reading the CYCLE 247 Datum from MasterCAM

October 2008

3.0 Modifying the NC Format

In order to output the CYCLE 247, you need to modify your NC Format(s) to control where this code will appear. If you want it to appear once per NC Program, then you should place the codes shown below in the *Start Program Block* of your NC Format. If you want a different CYCLE 247 for every tool path (operation) then place the code in the *Start Path Block*.

- 1) Start the CAMplete TruePath application
- 2) Go to Tools > Edit NC Formats
- 3) Select the NC Format section you wish to use
- 4) Click Edit
- 5) Insert two new lines where you want the CYCLE 247 to appear:



- 6) Click New Codes Window
- 7) Add a Cycle 247 Set Origin command code to each line
- 8) Add a Comment Code and Custom Code to the first line
- 9) Add a Line Number Control code to the last line
- 10) Set the drop list for the second Cycle 247 Set Origin to be Get Origin From Path
- 11) For the Custom Code add the character ~ (this is the line continuation code on the Heidenhain)
- 12) For the Line Number Control check Over-ride and set the option to [SKIP]
- 13) When you are done, it should look like the following:

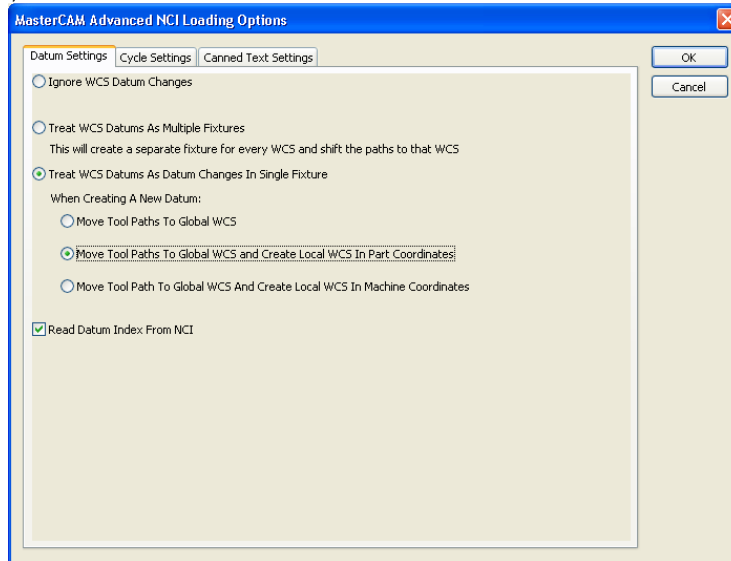
16	WP CS Set Origin	Enable Set Origin	Work Piece Coord System Set Origin
	Custom Code	~	
17	WP CS Set Origin	Get Origin from Patl	Work Piece Coord System Set Origin
	Line Number Control	<input checked="" type="checkbox"/> Over-ride [SKIP]	Line Number Control

- 14) Click Finish to commit the changes to the NC Format

4.0 Modifying the MasterCAM NCI Loader

You need to modify the File loader for Mastercam to read the datum index

- 1) Start the CAMplete TruePath software
- 2) Go to Tools > Options
- 3) Click File Types
- 4) Select Mastercam and click "Advanced Properties"
- 5) In "Datum Settings", Enable "Read Datum Index From NCI"



- 6) Close the dialog
- 7) Close the CAMplete TruePath application

5.0 Modifying the Datum From the CAM Wizard

After following these steps, when you load a MasterCAM NCI file with a datum, it will output that value for the CYCLE 247. If needed, you can modify the datum using the CAM Wizard.

- 1) Click CAM Wizard
- 2) Click Coord Sys
- 3) You will see each tool path listed with its current CYCLE 247 datum
- 4) To select a different datum, click the drop list in the Type column and select a new one

