1.0   Overview

This document explains how to modify a CAMplete TruePath Machining Setup so that the output for D and H Codes can be overridden or completely suppressed. This feature is available starting in CAMplete TruePath v4.6.424.
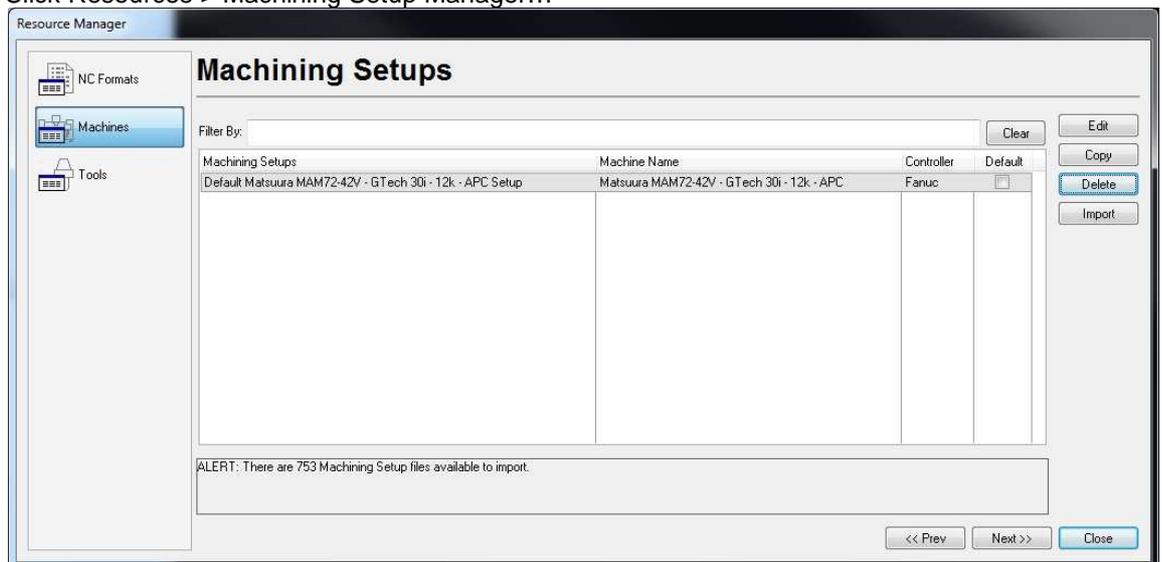
The information in this document applies to milling machines with FANUC controls. Overriding D and H codes on these controls can be useful when the user wishes to use a macro to retrieve the correct values when changing tools. Suppressing the output altogether can be useful when using Tool Life Management.

2.0   Modifying the Machining Setup

The override for both D and H code output are controlled by the Controller which is part of the machining setup in CAMplete TruePath.
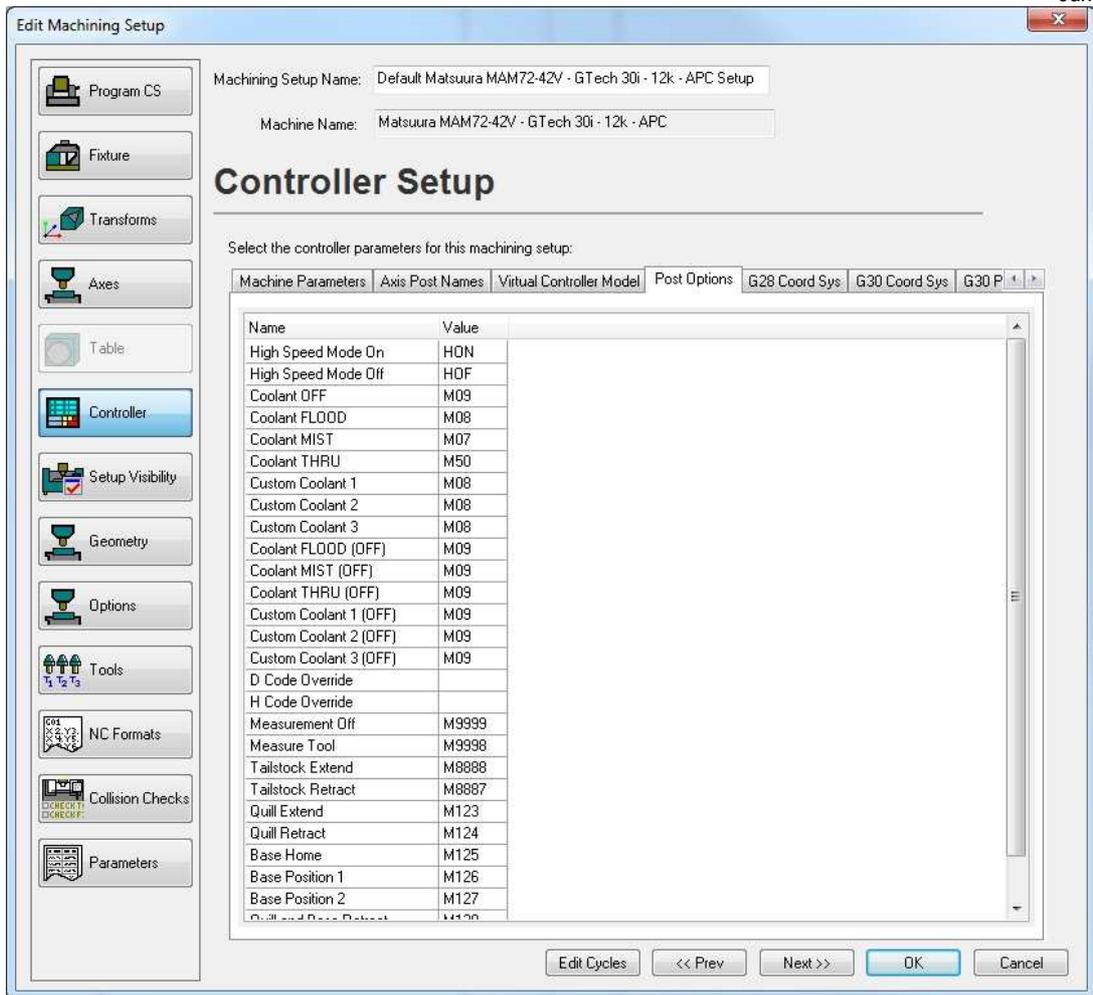
To modify the machining setup:

1) Click Resources > Machining Setup Manager…

2) Select the Machining Setup to edit and click Edit while holding the Shift key
3) In the Edit Machining Setup window, click Controller then select the Post Options tab

4) Double click the Value beside DCode Override and H Code Override to set the override values
   a. If the Value is empty (the default) the D and H codes will be posted normally using the values set in the tool path
   b. If the value is a specific value (e.g. D999 or H999), the codes will ALWAYS be posted with those values and the value set in the toolpath will be ignored (Note that the override must include the 'D' or 'H')
   c. If the value is set to [NO_OUTPUT] then the code will produce no output in the program
5) After editing the Values click OK
6) When prompted about overwriting the file click Yes
7) Select the newly modified Machining Setup in the Import Wizard or CAM Wizard and when reposted, the newly set override values will be used