

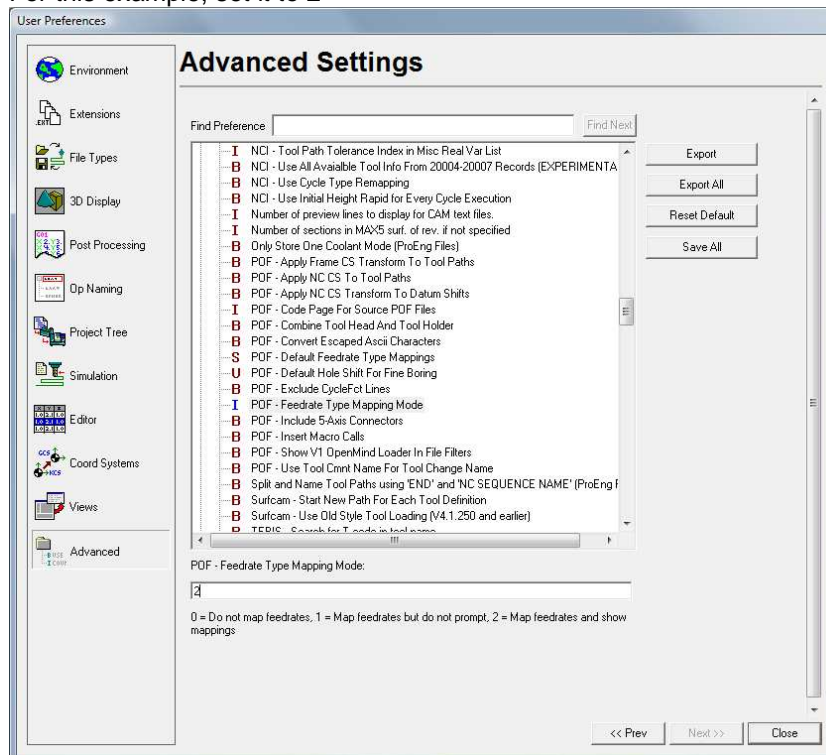
1.0 Overview

This document explains how to use CAMplete TruePath v4.5 with the new GFAC Master format to load and parse POF files.

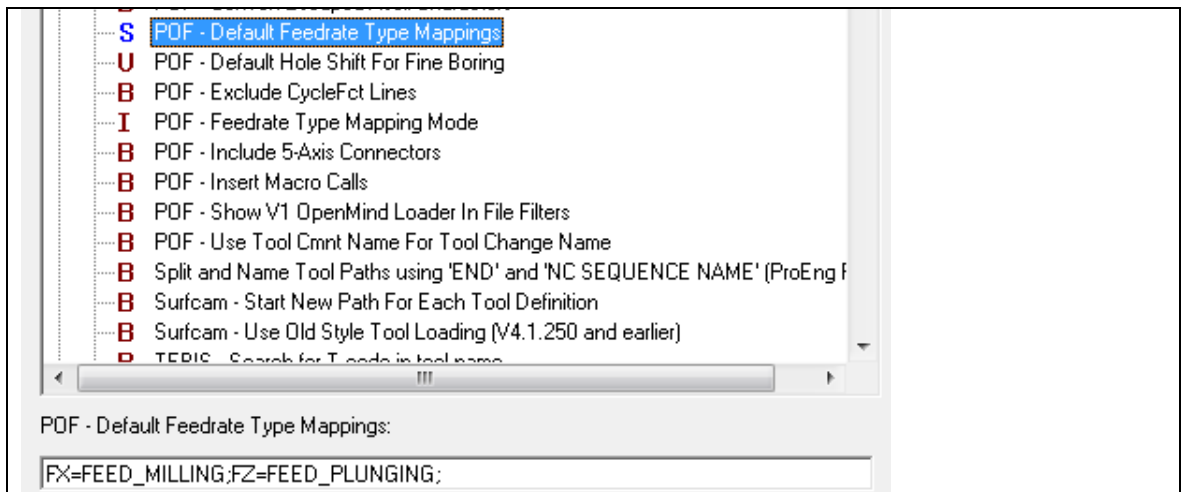
2.0 Setting the POF Loading Options

CAMplete can map POF feed rates to specific Q-variables in the NC Format. To enable this reading follow these steps:

1. Start the CAMplete TruePath application
2. Go to Tools > Options
3. Click Advanced and ignore the warning
4. Open the CAM File IO Prefs folder
5. Go to the option POF – Feedrate Type Mapping Mode
6. If you set it to 0, not Q-variable mapping will occur
7. If you set it to 1, it will attempt to map whatever feedrates it understands but will not prompt you if there are ones that it does not understand
8. If you set it to 2, it will prompt you if there are any feedrates that it does not understand
9. For this example, set it to 2



10. Next go to the POF – Default Feedrate Type Mappings
11. This allows you to map text from the POF files to specific feedrate types understood by CAMplete TruePath.



12. In this case, the text FX in the POF file will be mapped to the type FEED_MILLING and the text FZ in the POF file will be mapped to the type FEED_PLUNGING. The available feed types in TruePath are:
 - a. FEED_MILLING
 - b. FEED_PLUNGING
 - c. FEED_MILLING_REDUCED
 - d. FEED_MACRO
 - e. FEED_MILLING_OVERRIDE
 - f. FEED_MILLING_TCPC
 - g. FEED_PLUNGING_TCPC
 - h. FEED_MILLING_TCPC_MAX
 - i. FEED_DRILLING
13. These mappings are used in the NC Format to pre-assign Q-variables to specific feed types. (See section 3)
14. Close the User Preferences dialog
15. Close CAMplete TruePath

3.0 Importing the Master NC Format

Start by importing the new GFAC Master NC format:

1. Start the CAMplete TruePath application
2. Go to Tools > Edit NC Formats
3. Click Import
4. Select the GFAC Mikron Master Format.cs4 file
5. You should now have this in your NC Format list
6. Find the GFAC Mikron Master Format: Execute Sub-Program NC Format section and edit it
7. On line 6 is the Define Variable Code
8. Each mapped Q-variable is controlled with a Variable Assign command code

Variable: 2	Value: [FEED_VAR=FEED_MILLING][FEED_COMMENT=Milling Feed]	Q2 = [FEED_VAR=FEED_MILLING][FEED_COMMENT=Milling Feed]
Variable: 3	Value: [FEED_VAR=FEED_PLUNGING][FEED_COMMENT=Plunging Feed]	Q3 = [FEED_VAR=FEED_PLUNGING][FEED_COMMENT=Plunging Feed]

 - a. Select the variable number
 - b. In the Value use [FEED_VAR=FEED_MILLING] to select the feed type (as mapped by Section 2 - Step 12)
 - c. Set the [FEED_COMMENT=Milling Feed] – This sets the comment variable that will be output for this feed rate
9. Close the NC Format editor

4.0 Setting your default NC Formats

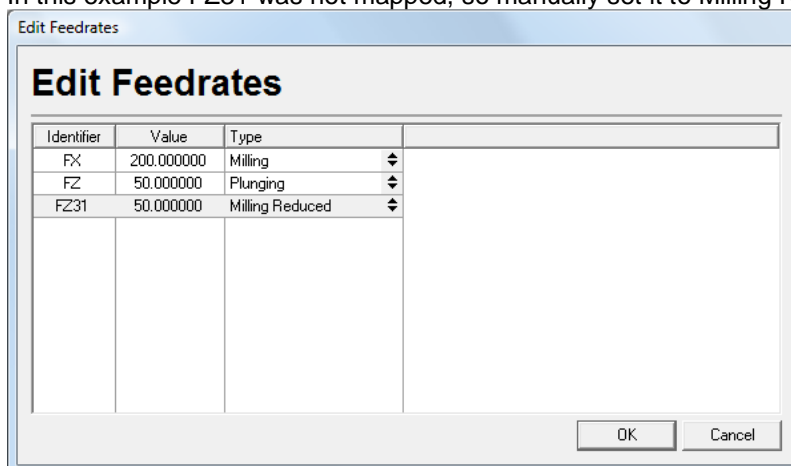
To set your default NC Formats for the machining setup, follow these steps:

1. Start CAMplete TruePath
2. Go to Resources > Machining Setup Manager
3. Select your machine and click edit
4. On the NC Formats page select the GFAC Mikron Master Format NC Format in the Default NC Format For Master Programs list
5. Close the Edit Machining Setup dialog

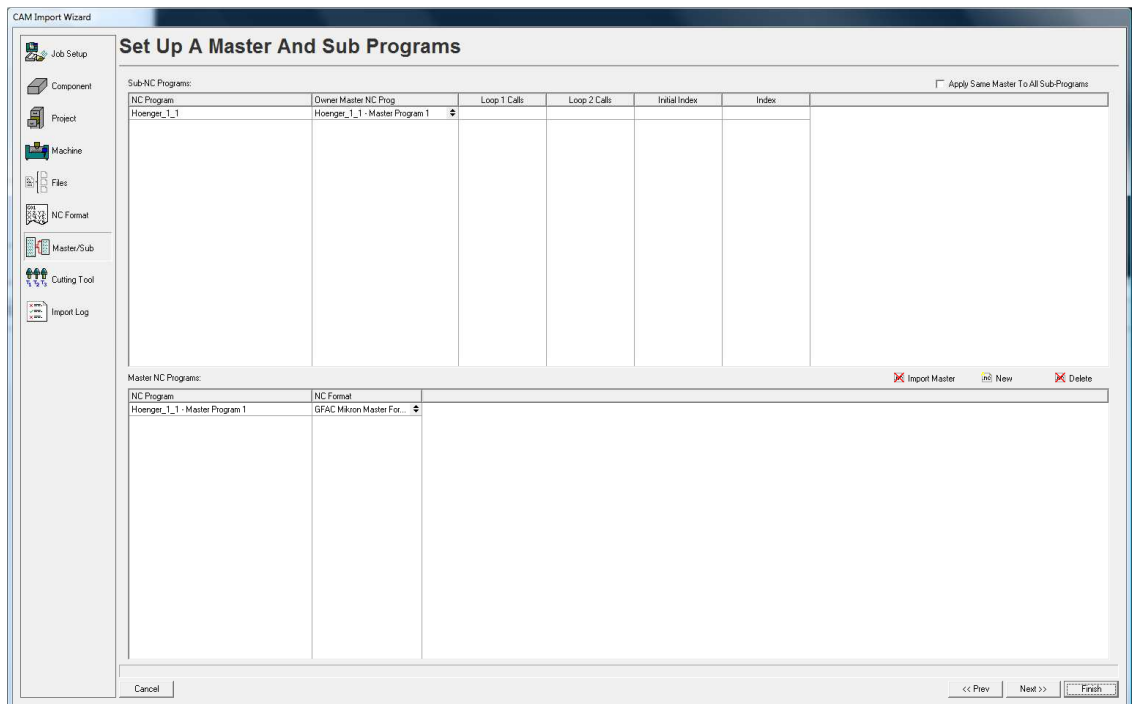
5.0 Creating a Master Program

To use this NC Format follow these steps:

1. Load the POF file as normal
2. If you have the POF feed mapping mode set to 2, you will get a prompt if some of the mappings were not made
3. In this example FZ31 was not mapped, so manually set it to Milling Reduced:



4. In the CAM Wizard go to the Master/Sub page
5. Click New
6. Answer Yes when prompted about adding the programs to the Master
7. The NC GFAC Mikron Master Format Format should already be selected
8. Click Finish to close the wizard.



9. You should now have a proper Master Program with Q-Variable mappings:

```

BEGIN PGM Hoenger_1_1 - Master Program 1 INCH
1 BLK FORM 0.1 Z X+0 Y+0 Z+0
2 BLK FORM 0.2 X+0 Y+0 Z+0
3 ;*****
4 ;Project:      xxxx-xxx
5 ;Machine:     HSM600U
6 ;Operator:    xxx
7 ;Hardness:    --
8 ;Material:    xxxxxxxx
9 ;Datum:      X=spec., Y=spec.
10 ;Datum:     Z=spec., see cad/cam
11 ;Preset:     #32
12 ;Date:      02.03.2009
13 ;*****
14 ;*****
15 * - RECHTS_1_SR
16 TOOL CALL SCHAFTFRAESER D10 SR Z S2000
17 ;-----
18
19
20 Q3 = 500 ;Plunging Feed
21 Q2 = 2000 ;Milling Feed
22 ;-----
23 CYCL DEF 32.0 TOLERANCE
24 CYCL DEF 32.1 T0
25 CYCL DEF 32.2 HSC-MODE:0 TA0
26 ;-----
27 CYCL DEF 301 Impulse Unilube ~
Q396 = +20 ;Pulse Unilube [/min]
28 ;-----
29 CALL LBL 999
30 L M140 MB MAX
31 L B+0 C+0 R0 F MAX M03 M126
32 L X+0 Y+0 R0 F MAX M36
33 L M10 M15
34 L M134
35 L M38
36 CALL PGM Hoenger_1_1
37 L M140 MB MAX
38 L M01
39 ;*****
40 * - Ende
    
```

```

41 L M140 MB MAX M37
42 L B+0 C+0 R0 F MAX M35
43 L X+0 R0 F MAX
44 L M30
45 ;*****
46 LBL 999
47 L M129
48 CYCL DEF 19.0 WORKING PLANE
49 CYCL DEF 19.1 B+0 C+0
50 CYCL DEF 19.0 WORKING PLANE
51 CYCL DEF 19.1
52 PLANE RESET STAY
53 CYCL DEF 247 BEZUGSPUNKT SETZEN ~
Q339 = +32 ;BEZYGSPUNKT-NUMMER
54 CYCL DEF 7.0 DATUM SHIFT
55 CYCL DEF 7.1 X0.0000
56 CYCL DEF 7.2 Y0.0000
57 CYCL DEF 7.3 Z0.0000
58 CYCL DEF 7.4 A
59 CYCL DEF 7.4 B
60 CYCL DEF 7.5 C
61 LBL 0
62 END PGM Hoenger_1_1 - Master Program 1 INCH
    
```