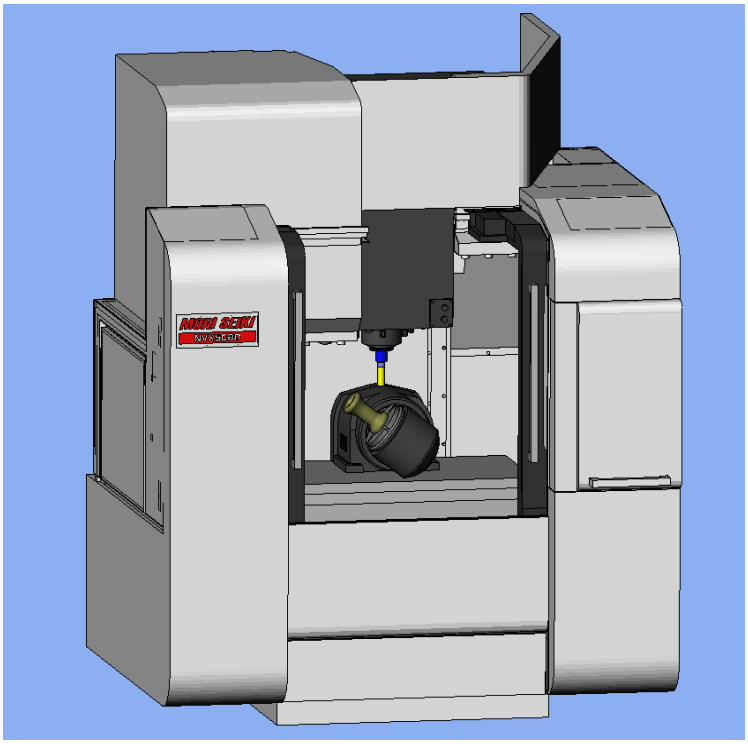
POST PROCESSOR MANUAL FOR

MORI SEIKI

NVX and NV



This manual was prepared with the assumption that the intended reader does have working knowledge of Esprit and NVX or NV programming experience so that he fully understands the information it contains.

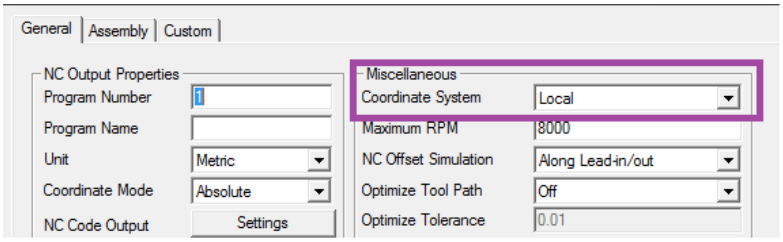
Contents

1. Introduction

This manual documents the Esprit NVX and NV post processors. The E12MSP_NVX_2012-03-15 post supports the NVX machine tool. The E12MSP_NVD_2012-03-16 post supports the NV machine tool.
These post processors require ESPRIT 2012 or higher.

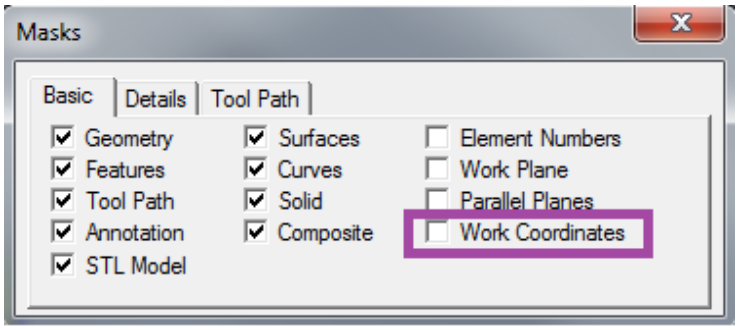
2. Defining Work Coordinates

Program Datums on the machine can be defined as Work Coordinates in Esprit. Esprit supports both Local and World Coordinate Systems. To set the Coordinate System in the milling environment, go to ***Common Machining> Machine Setup> Coordinate Systems*** and select between ***Local*** and ***World***.



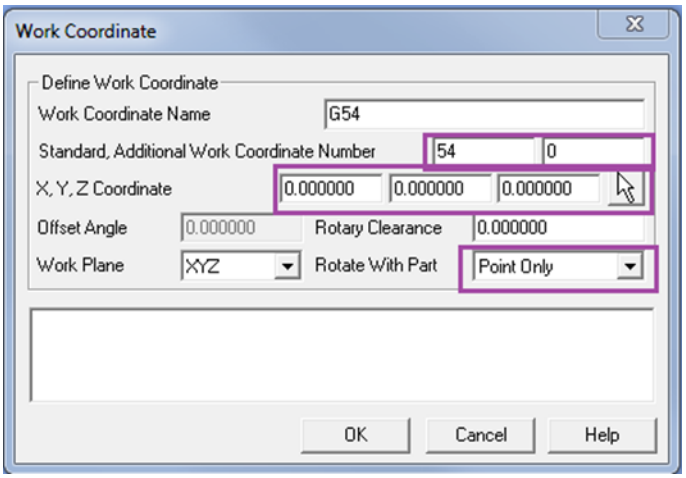
For most applications this will be set to Local.

To view your Work Coordinates in Esprit check the Work Coordinate box in in the Masks box.



2.1. Standard Work Coordinate System G54 to G59

To define a Standard Work Coordinate set the Work Coordinate page as:

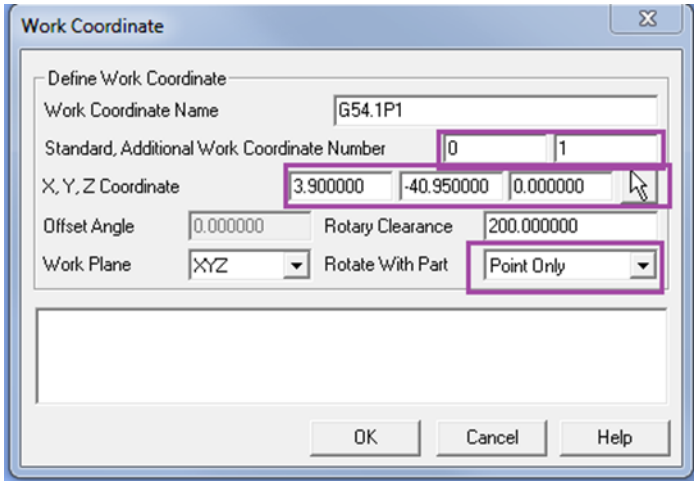


Standard is the Work Coordinate Number and Additional Work Coordinate Number is set to **0**. XYZ is the location of the Work Coordinate in the file relative to the Work Plane. Rotate with Part needs to be set to **Point Only**.

```
M06
T7
G90 G94 G54 G17 G00 X109.5 Y100.0
G43 Z-3.0 H17 S7767 M03
M08
```

2.2. Extended Work Coordinate System G54.1 P1-300

To define an Extended Work Coordinate set the Work Coordinate page as:



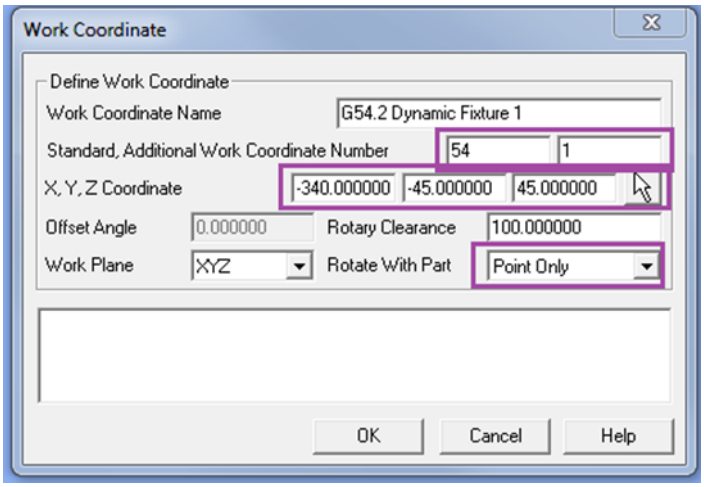
Standard is set to **0** and Additional Work Coordinate Number is set to the P value. XYZ is the location of the Work Coordinate in the file relative to the Work Plane. Rotate with Part needs to be set to **Point Only**.

```
M06
```

```
G90 G94 G54.1 P1 G17 G00 X-2.5 Y-6.5383
G43 Z10.0 H1002 S11650 M03
M08
```

2.3. Dynamic Fixture Offset Work Coordinate System G54.2 P

To define a Dynamic Fixture Offset Coordinate System, set the Work Coordinate page as:

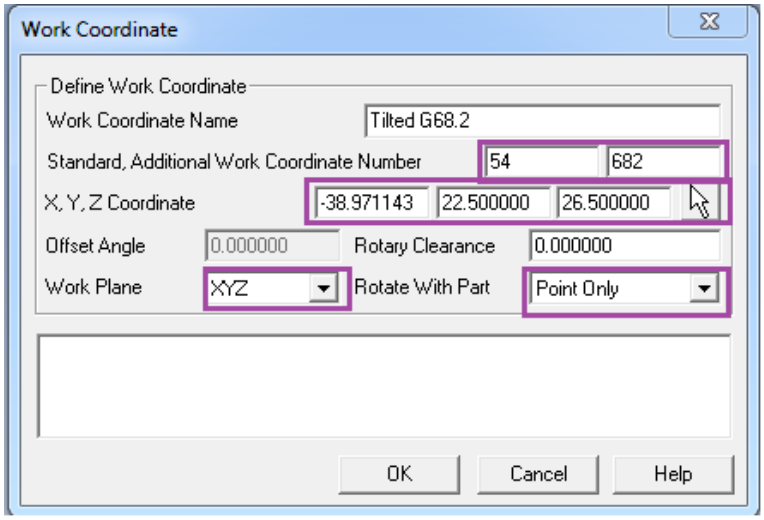



```
:
(SETTING FOR FIXTURE OFFSET G54.2)
G90 G10 L21 P1 X* Y* Z* A0 ( X-340.0 Y-45.0 Z45.0 )
:
:
M06
T2
G90 G54
G54.2 P1
M11
G00 A180.0
M10
G90 G94 G17 G00 X28.6 Y-46.4047
G43 Z92.0 H66 S1500 M03
M08
```

Note that the command line for the Dynamic Fixture Offset will be output near the beginning of the NC file.

2.4 Tilted Plane Work Coordinate System G68.2

To define a Tilted Plane Work Coordinate System, set the Work Coordinate page as follows. This example shows how to create the G68.2 can be defined in Esprit relative to P0. In this case we define the Work Coordinate page as:



Standard can be set to an existing Work Coordinate and Additional Work Coordinate Number is set to **682**. XYZ is the location of the Work Coordinate in the file relative to the current Work Plane which is **XYZ**. The XYZ Coordinate values in this scenario are most commonly entered using the button on the far right, and digitizing the point in Esprit where the new Work Coordinate will be located. The Work Plane is set to **XYZ**. Rotate with Part needs to be set to **Point Only**.

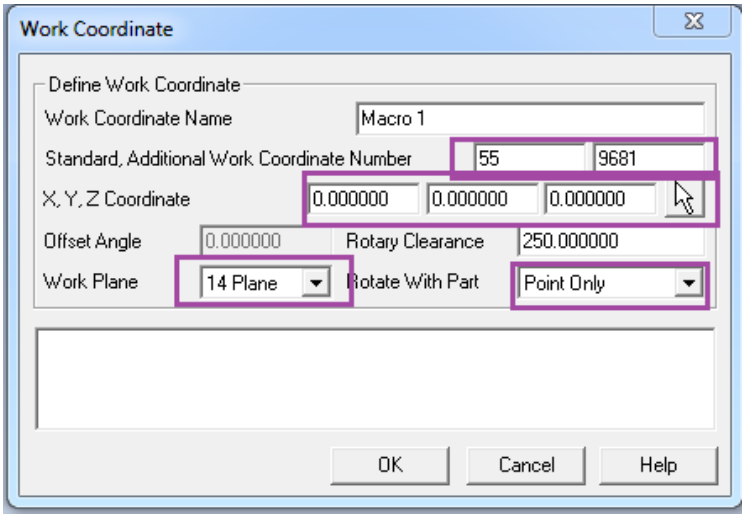
```
:
M06
M69
M11
G00 B-90.0 C150.0
M68
M10
G49
G68.2 X-38.9711 Y22.5 Z26.5 I-60. J90. K-90.
G53.1
G17 G94 X0.0 Y38.9711
G43 Z24.5 H29 S3001 M03
M08
```

Note: This option is not available on the NV.

2.5 Macro Work Coordinate System G65 P9681

This item is available only on the NVX machine as an option and may not be installed. Please contact Mori Seiki to check on its configuration and availability.

To define a Macro Work Coordinate System, set the Work Coordinate page as:



Standard is typically set to **54**, but it can be set to an existing Work Coordinate. Additional Work Coordinate Number is set to **9681**. XYZ is the location of the Work Coordinate in the file relative to the current Work Plane. The current Work Plane can be any Work Plane in Esprit. In this example the Work Plane is set to an existing plane '14 Plane'. Rotate with Part needs to be set to **Point Only**.

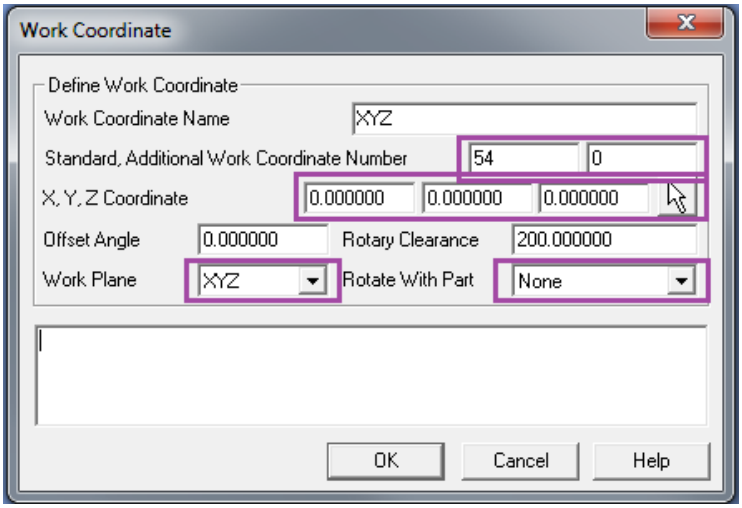
```
:
M06
G55
G65 P9681 X50.0 Y10.0 Z138.0 B60.0 C180.0 M55. Q59.
G59
M69
M11
G00 B60.0 C180.0
M68
M10
G17 G94 X-14.1421 Y8.5
G43 Z2.0 H69 S2000 M03
M08
```

3. Simultaneous 5 Axis Machining Settings

Creating 5x axis operations have specific Work Coordinate settings for the NVX and NV machines.

3.1 Work Coordinate Setting

To define a Work Coordinate System for 5x milling operations, set the Work Coordinate page as:

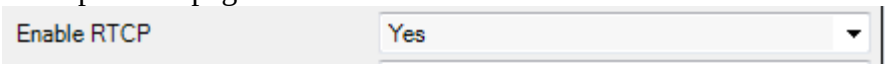


Standard is the Work Coordinate Number, typically **54** and Additional Work Coordinate Number is set to **0**. The Work Plane is set to **XYZ**. Rotate with Part needs to be set to **None**.

```
M06
M11
M69
G00 G54 G17 X0.0 Y0.0
G90 B0.0 C0.0
G49
G05 P10000
G43.4 H122 S329 M03
G00 X0.0 Y0.0 Z0.0 B0.0 C0.0
M08
```

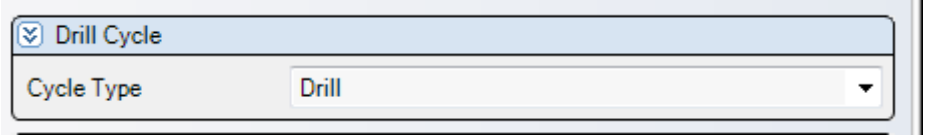
3.2 RTCP

For the NVX and NV, we recommend setting RTCP ON for 5 axis operations with DDRT2, and setting RTCP OFF for 5 axis operations with DDRT1. RTCP can be enabled on the operation page



4. Drill Cycles

Drilling, Boring and Tapping cycles on the NVX can be set using the Drill operation page. Selecting Cycle Type on the Drill operation page determines the output in the NC file.



Any cycle can output the PM or PR feedrate. By default the PM feedrate is output. Enter 95 into Custom Setting 10 on the operation to output a G95 and a PR feedrate value.

4.1 Drill

G99 G81 Z-3.5829 R2.0 F687.0

4.2 Peck

G99 G83 Z-11.7305 R-3.0 Q16.0 F845.0

4.3 Tap

G98 G84 Z-15.0 R-3.0 P500 F796.0

4.4 Bore

G98 G85 Z-18.862 R-3.0 F2959.0

4.5 Peck2

G99 G73 Z-38.6265 R-3.0 P5000 Q10.0 F952.0

NOTE: P value is for NVX only.

4.6 Tap2

G98 G74 Z-15.613 R-3.0 F954.0

4.7 Bore2

G98 G86 Z-25.0 R-3.0 P500 F8877.0

NOTE: P value is for NVX only.

4.8 Bore3

G98 G87 Z-36.295 R-8.0 I0.34 J0.43 F2959.0

NOTE: I and J values are for the NVX only. The NV will have a Q value.

4.9 Bore4

G98 G88 Z-18.006 R-3.0 P500 F8877.0

4.10 Bore5

G98 G89 Z-15.0 R-3.0 P500 F8877.0

4.11 Bore6

G98 G76 Z-21.75 R-3.0 I0.12 J0.21 F4439.0

NOTE: I and J values are for the NVX only. The NV will have a Q value.

4.12 Bore7

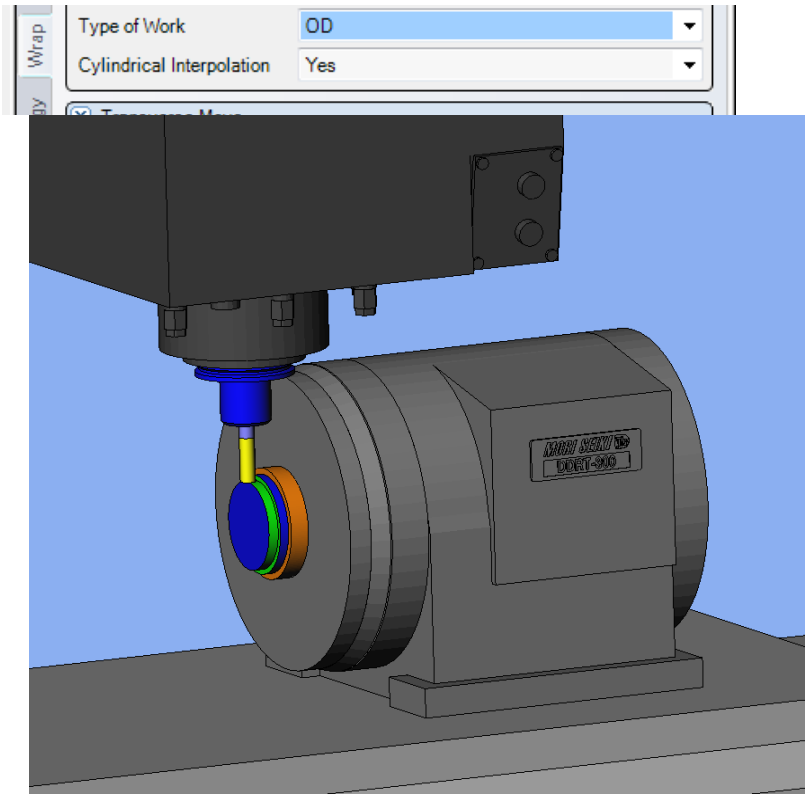
```
G98 G82 Z-18.006 R-3.0 P500 F2959.0
```

5. 4th Axis Wrap Milling Cycles

Esprit supports wrap milling operations on the diameter or face of a part. Cylindrical Interpolation can be used to shorten the length of the NC file.

5.1 Cylindrical Interpolation

Cylindrical Interpolation is available on Wrap Contour and Wrap Pocket operations.



NC output with Cylindrical Interpolation ON:

```
:
M06
M11
G00 A450.0
G90 G94 G54 G17 G00 X-44.5 Y0.0
G43 Z52.0 H1001 S2907 M03
M08
G07.1 A50.0
G01 Z45.0 F1551.0
X-35.0 F2067.0
A810.0
X-44.5
:
```

NC output with Cylindrical Interpolation OFF:

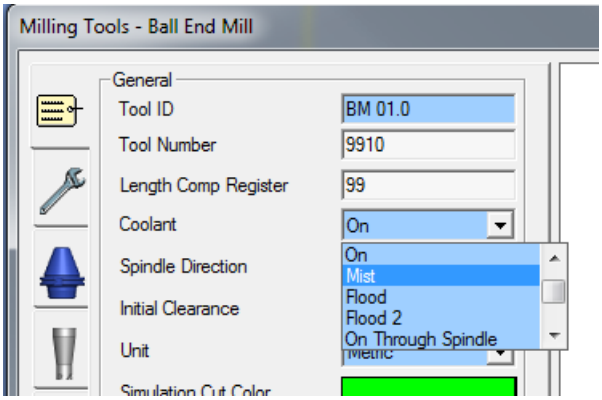
```
:
M06
T1
M11
G00 A90.0
G90 G94 G54 G17 G00 X-44.5 Y0.0
G43 Z52.0 H1001 S2907 M03
M08
G01 Z45.0 F1551.0
X-35.0 F2067.0
A210.0 F2632.0
A330.0
A450.0
X-44.5 F2067.0
:
```

6. Coolant Codes

Coolant codes can be selected inside Esprit for output in the NC file. Up to two coolant codes can be output per tool.

6.1 Primary Coolant Code

The primary coolant code is set by using the pull down on the tool page.



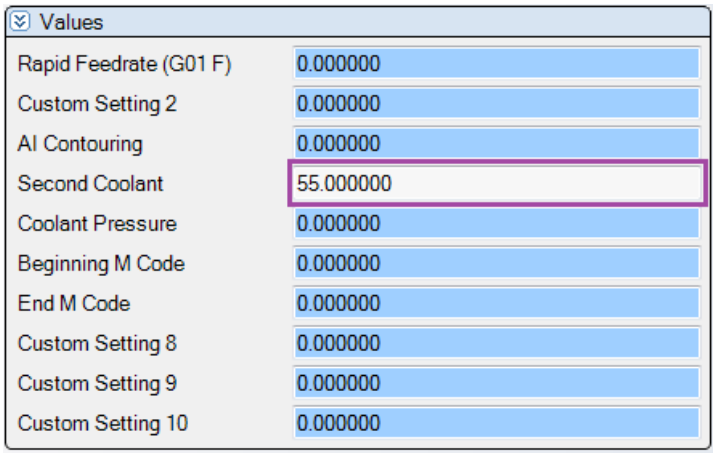
Here is a chart showing the available coolant codes in Esprit and M codes they are mapped to for the NVX and NV machines.

	M Code	
Esprit Pull down	**NVX**	**NV**
On	M08	M08
Mist	M07	M55
Flood	M51	M07
Flood 2	M53	M51
On Through Spindle	**M55**	**M88**

Mist Through Spindle	**M80**	**M72**
Flood Through Spindle	**M164**	**M52**
Flood2 Through Spindle	**M165**	**M80**

6.2 Secondary Coolant Code

The secondary coolant code is set by using **Second Coolant** on the Custom Tab of the operation page.

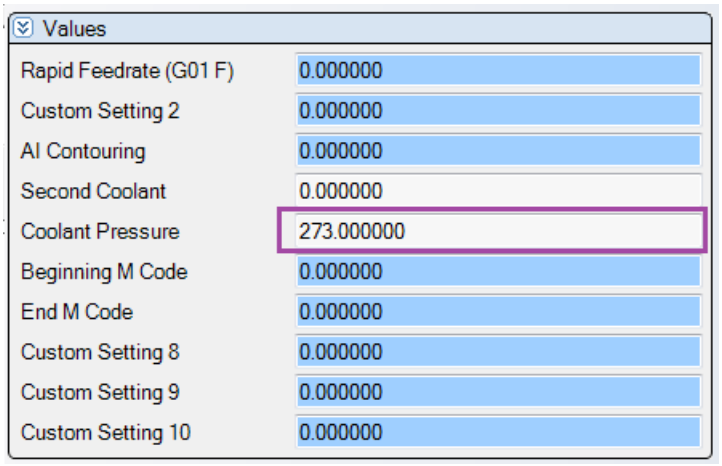


Here is a chart showing the available Second Coolant codes in Esprit and M codes they are mapped to for the NVX and NV machines.

	M Code	
Custom Code 4	**NVX**	**NV**
7	M07	M07
8	M08	M08
50	na	M50
51	M51	M51
52	na	M52
53	M53	M53
55	M55	M55
72	na	M72
80	M80	M80
88	na	M88
164	M164	M164
165	M165	M165
180	na	M180
226	na	M226

6.3 Coolant Pressure Code

The coolant pressure code is set by using **Coolant Pressure** on the Custom Tab of the operation page.



	M Code	
Custom Code 5	**NVX**	**NV**
270	M270	M270
271	M271	M271
272	M272	M272
273	M273	M273
274	M274	M274
275	M275	M275
276	M276	M276
277	M277	M277

Here is a sample of NC code with the 3 coolant related codes highlighted.

```
M11
G00 A810.0
G90 G94 G54 G17 G00 X-44.5 Y0.0
M270    // Coolant Pressure
G43 Z52.0 H1001 S2907 M03
M08     // First Coolant
M55     // Second Coolant
G01 Z45.0 F1551.0
```

7. Custom Codes

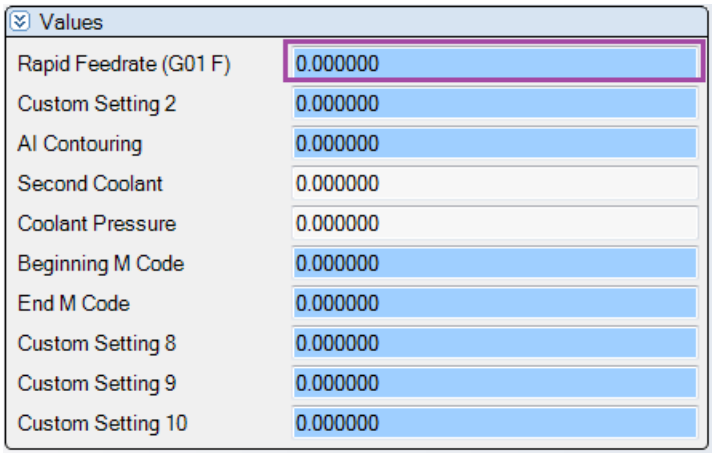
The Custom tab in Esprit allows the user to set machine specific functions. Some common to the operation pages are detailed here.

7.1 Rapid Moves / Tapping Cycles / Boring Cycles

Field #1 can be used set different values depending on what type of cycle is being output.

a) **Contouring type operations**

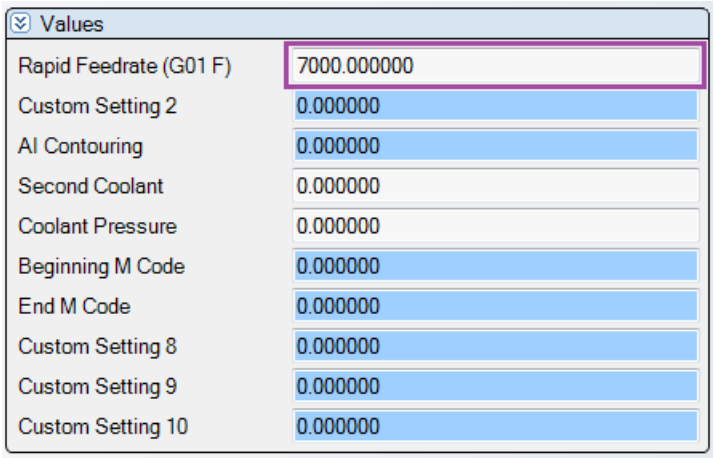
When this is set to 0, rapid moves inside the operation will be output as G00. If this value is not 0, the rapid moves will be output as linear feed moves using the entered value as the feedrate.



Ex. 1.

```
M08
G01 Z-5.0 F4735.0
X-109.5 F6313.0
Z-3.0 F4735.0
G00 X109.5 Y88.0
G01 Z-5.0 F4735.0
X-109.5 F6313.0
Z-3.0 F4735.0
G00 X109.5 Y76.0
G01 Z-5.0 F4735.0
```

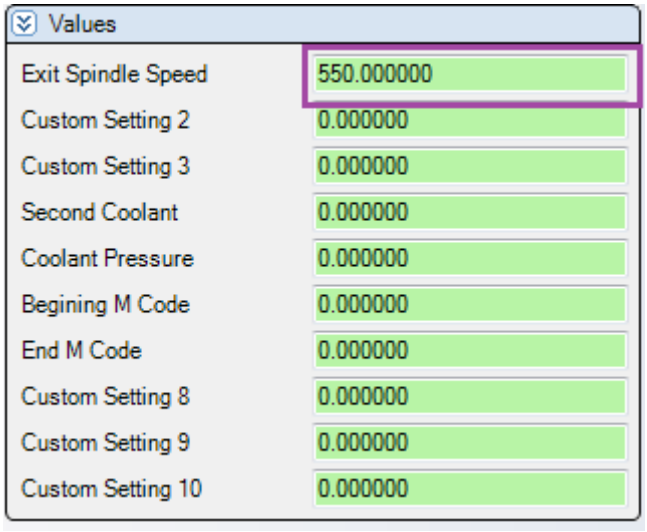
Ex. 2.



```
M08
G01 Z-5.0 F4735.0
X-109.5 F6313.0
Z-3.0 F4735.0
X109.5 Y88.0 F7000.0
Z-5.0 F4735.0
X-109.5 F6313.0
Z-3.0 F4735.0
X109.5 Y76.0 F7000.0
Z-5.0 F4735.0
```

b) Tapping operations

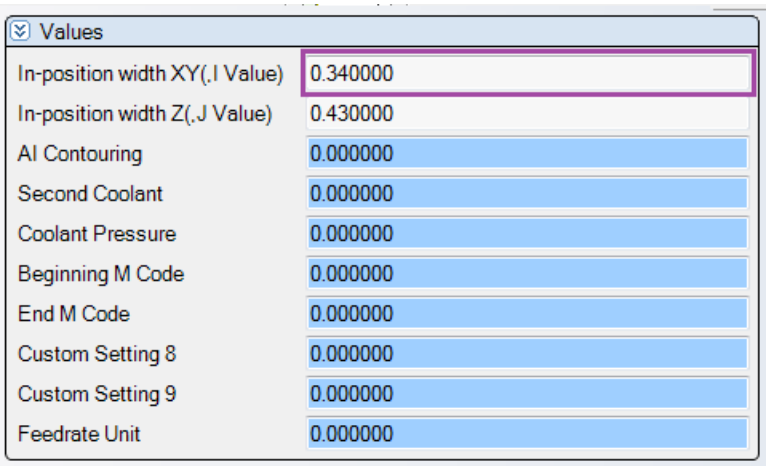
When this is set, an exit spindle speed is output on the G84 line.
NOTE: This is for the NVX only.




```
M06
G90 G94 G54 G17 G00 X60.878 Y-67.934
G43 Z500.0 H21 S318 M03
M08
G98 G84 Z-36.935 R-3.0 P550 F795.0 S550
G80
M09
```

c) Boring operations

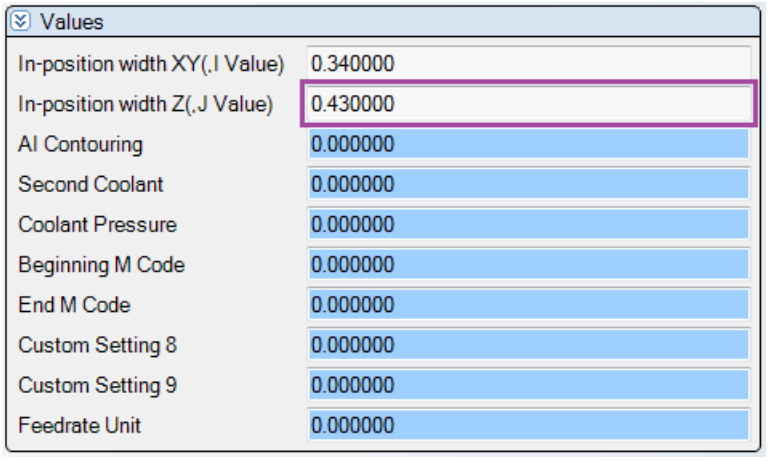
When this is set, an I value offset is output on the G87 and G86 line (Bore3 and Bore6 in Esprit).
NOTE: This is for the NVX only.




```
M08
G98 G87 Z-36.295 R-8.0 I0.34 J0.43 F2959.0
G80
M09
```

7.2 Boring Cycles

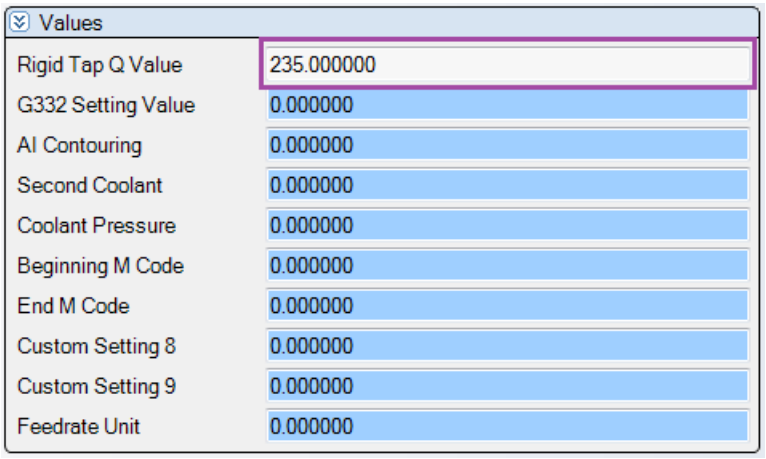
J Offset can be used set the J value offset output on the G87 and G86 line (Bore3 and Bore6 in Esprit). This can be used in conjunction with the I value mentioned above.
NOTE: This is for the NVX only.




```
M08
G98 G87 Z-36.295 R-8.0 I0.34 J0.43 F2959.0
G80
M09
```

7.3 Rigid Tapping

A Q value for rigid Tap2 cycles can be entered here.
NOTE: This is for the NV only.



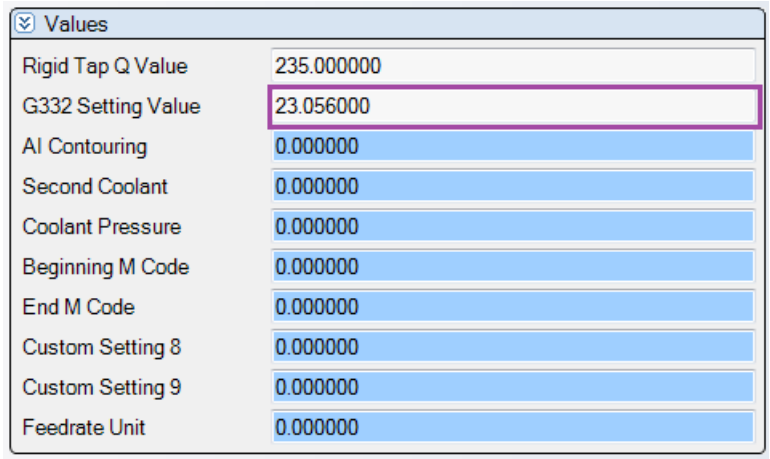

```
G90 G94 G56 G17 G00 X0.0 Y0.0
G43 Z0.0 H01 S0 M03
M08
M226
M29 S0
G98 G84 Z1.2185 Q235.0 R2.0685 F0.0
G80
```

7.4 Cutting Mode (G332)

To use Cutting Mode during a cycle, place the corresponding value into Cutting Mode on the Operation Page. To use G332 with T or T and A parameters, integers cannot be entered. For a T only value output, enter a decimal value. For a T and A values, enter the T value to the left of the decimal point, and the A value to the right of the decimal point.
NOTE: This is for the NV only.

Here is a table defining the mapping.

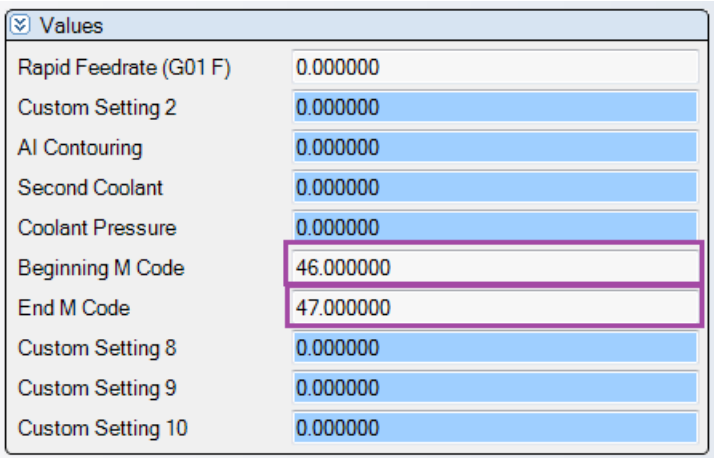
Cutting Mode	G332 Setting Value
G332 R1	**1**
G332 R2	**2**
G332 R3	**3**
G332 R4	**4**
G332 T	**.001-.999**
G332 T A	**1.001-200.999**




```
M06
G91 G28 Z0
G90 G94 G55 G17 G00 X13.0 Y-7.0
G332 T0.056 A23.0
G05 P10000
G43 Z2.0 H01 S0 M03
M08
G01 Z-2.0 F0.0
```

7.5 M Codes at the Beginning and End of an Operation

Beginning M code can be used to output any M code near the beginning of an operation. End M Code can be used to output any M code near the end of an operation.




```
:
M11
G00 A90.0
G90 G94 G54 G17 G00 X-44.5 Y0.0
M46
G43 Z52.0 H1001 S2907 M03
M08
G01 Z45.0 F1551.0
```

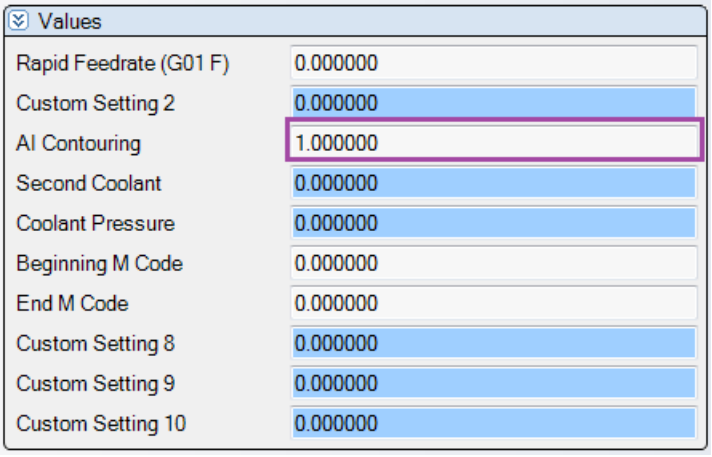
```
X-35.0 F2067.0
A210.0 F2632.0
:
:
X-37.0 F2067.0
Z52.0 F1551.0
M47
M10
M09
G49
G91 G28 Z0
G91 G28 X0 Y0
M01
```

7.6 AI Contour Control

To use AI Contour Control, enter the corresponding value into AI Contouring.

Here is a table defining the mapping:

	AICC	
AI Contouring	**NVX**	**NV**
1	G05 P10000	G05 P10000
2	G05.1 Q1	G05.1 Q1
3	na	G05.1 Q3
4	G08 P1	G08 P1



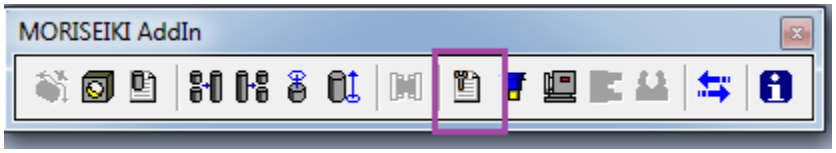
Example NC showing the placement of the AI Code:

```
M11
G00 A-30.0
M10
G90 G94 G54 G17 G00 X-45.25 Y0.0
G05 P10000
G43 Z200.0 H02 S8722 M03
M08
```

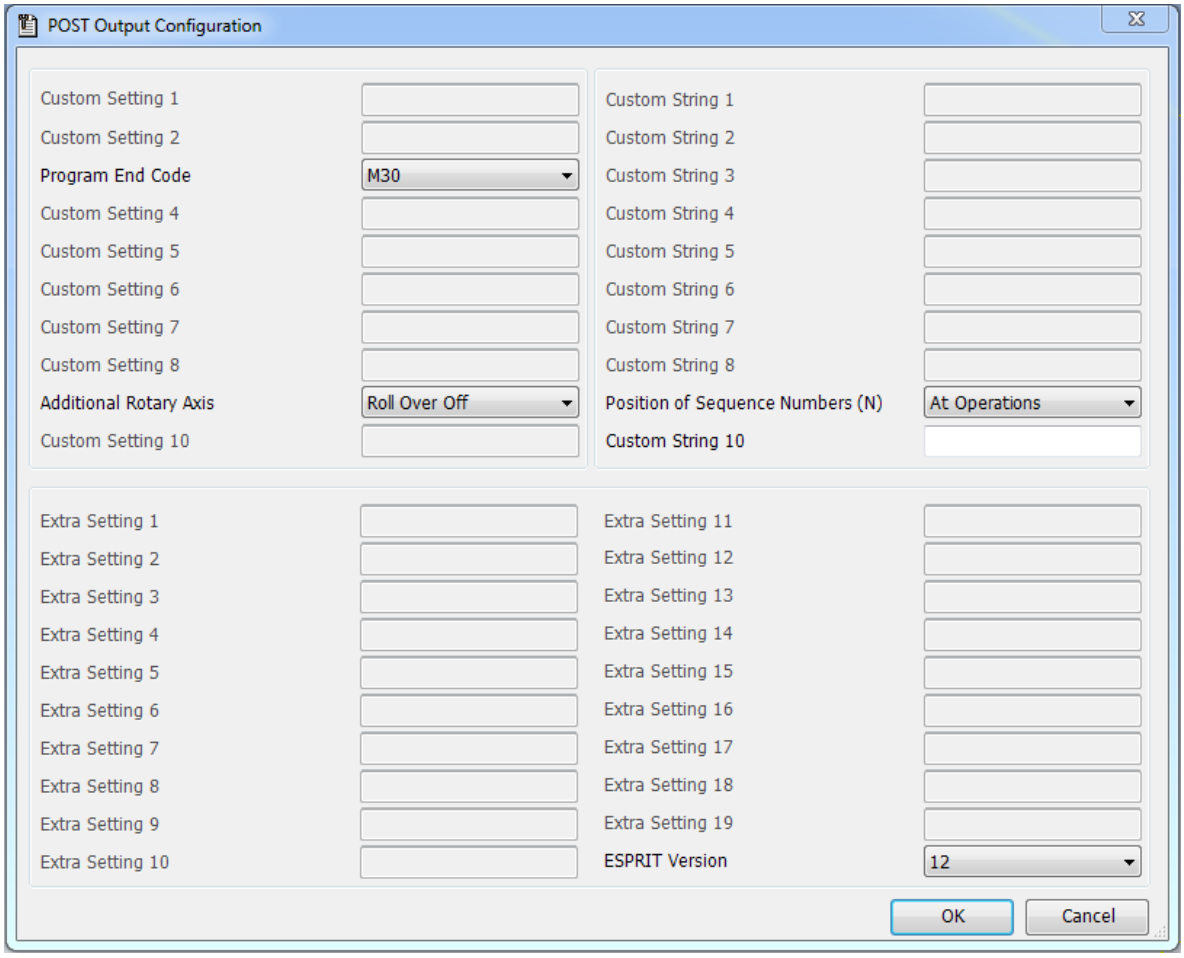
```
Z47.0G01 Z30.0 F2658.0
```

8. Post Output Configuration

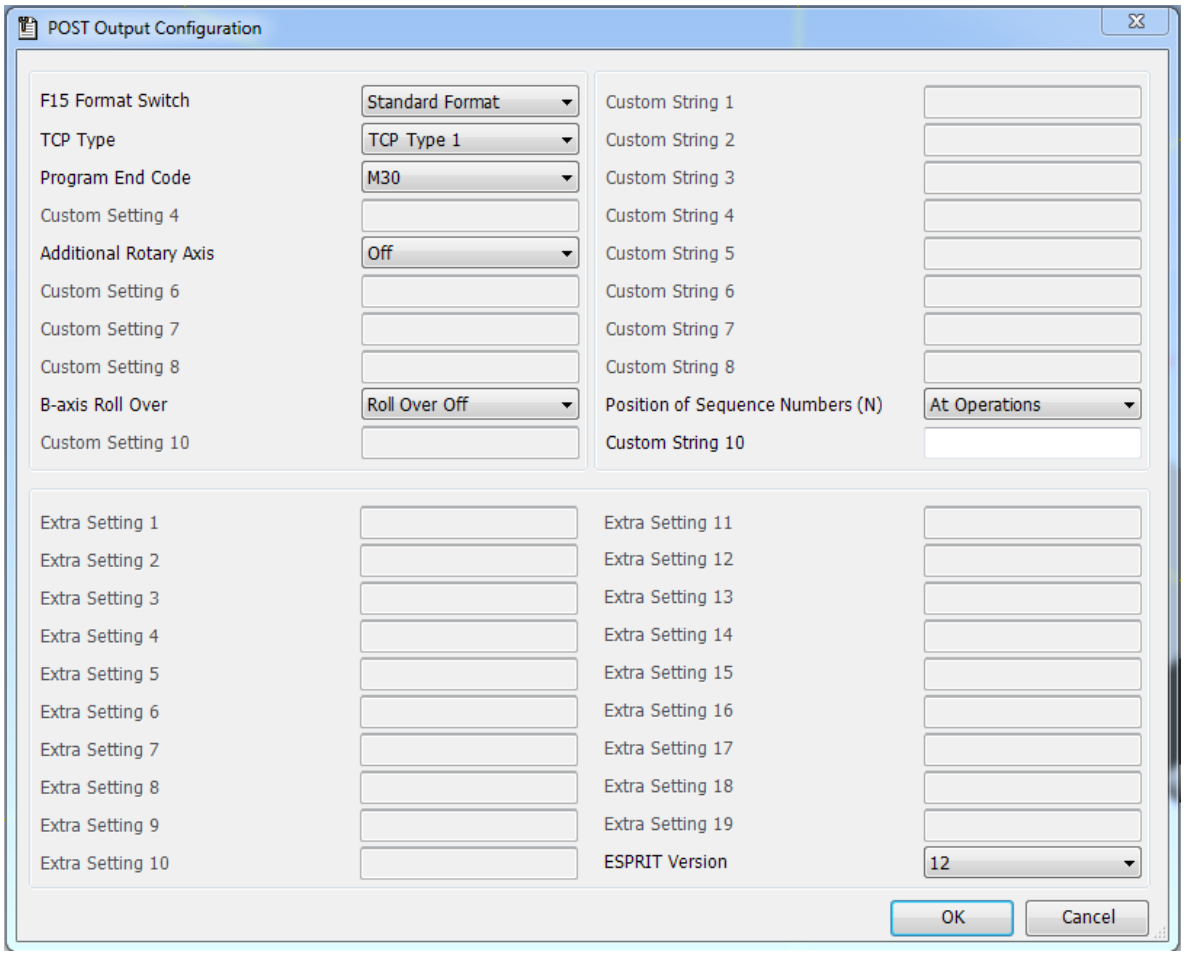
The Post Output Configuration program on the Mori Seiki Addin toolbar is a tool for setting values specific to your NVX or NV machine. The Post Output Configuration program is available for all Mori Seiki machines in Esprit, but the values may change depending on the machine.



For the NVX:



For the NV:



8.1. F15 Format Switch

Switches from standard Fanuc format (16/18) to F15 format

8.1. TCP TYPE

Select the type of RTCP output for 5 axis operations; 1 (XYZ BC) or 2 (XYZ IJK).

8.2. Program End Code

Select the type of code for end of Program; M02, M30 or M99

8.3. Additional Rotary Axis (NV)

Previously this was used to tell the post processor the rotary axis in the machine configuration. Now the post reads the Machine Setup and determines the number of rotary axis automatically.

8.4. Axis Roll Over (NV)

Controls the output of the Rotary axis.

Roll Over Off: Rotary axis Values in the NC file will between -99999.999 and +99999.999 (Parameter 1008 on Control must be set to 0 also).

Roll Over On: Rotary axis Values in the NC file will between -360. and +360. (Parameter 1008 on Control must be set to 1 also).

8.5. Position of Sequence Numbers (N)

Controls the output of the N block sequence numbers in the NC file.

At Operations: An N block will be output at the start of every operation.

NTCOnly: An N block will only be output at the start of an operation in which a tool change has occurred.

8.6. Custom String 10

Allows the user to enter text commands to control other features of the machine. Please see chapter 9 for a complete and detailed list.

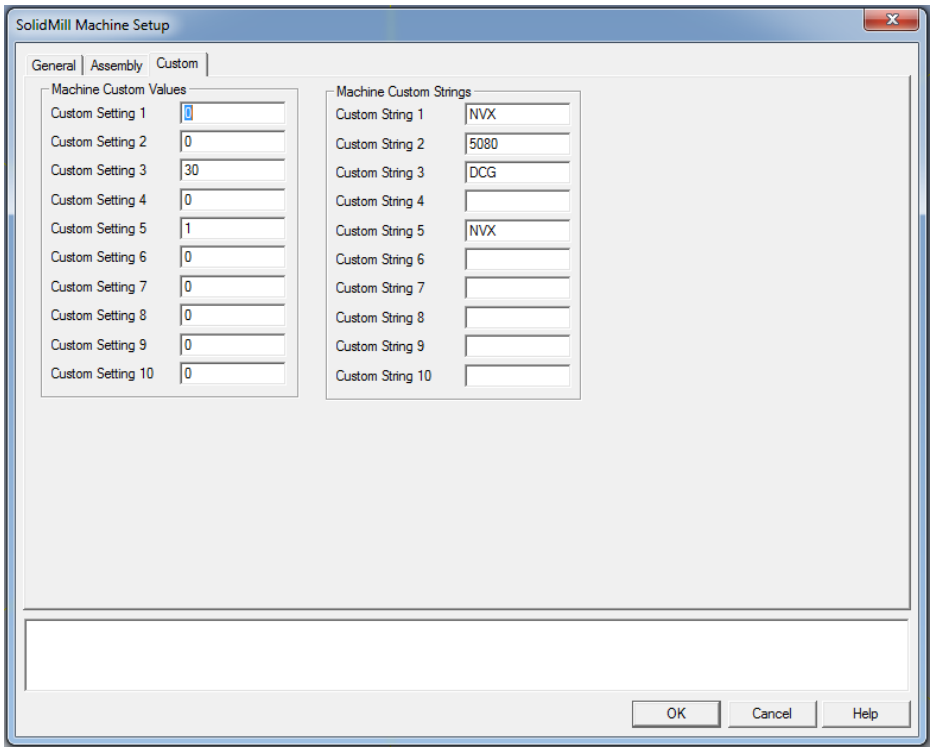
8.7. ESPRIT Version

Displays the version of Esprit being used.

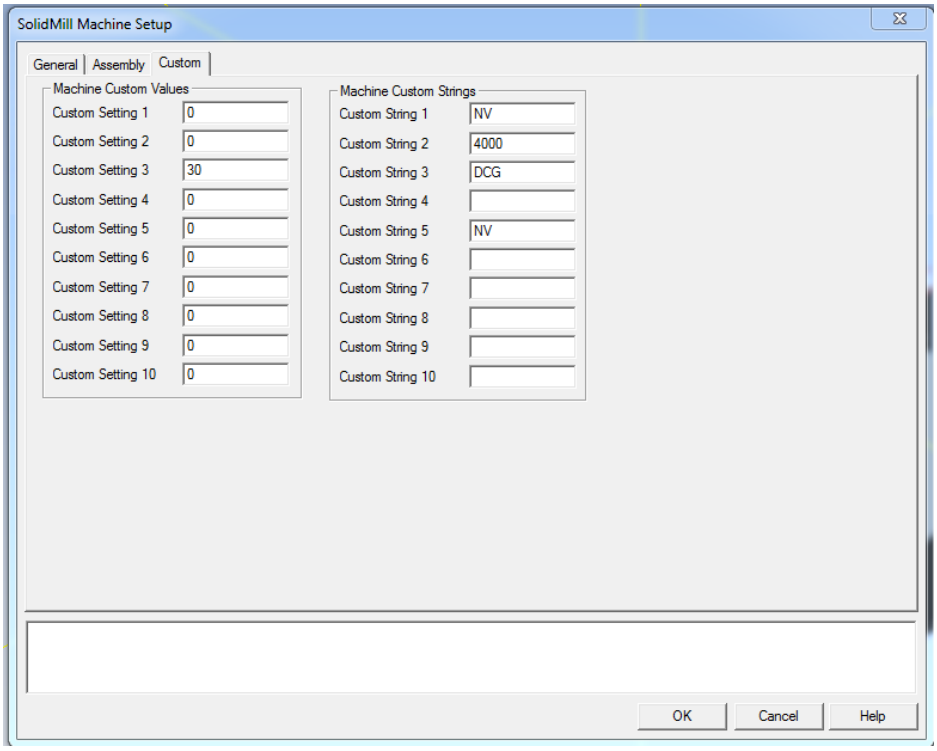
9. Machine Setup Custom Settings

Your system already has a configuration for the NV or NVX in it which has all of the axis and solids defined. In addition to the basic user settings on the General tab there are some user configurable settings on the Custom Tab

NVX:



NV:



9.1. Fanuc Output Style

Custom Setting 1 changes the NC code output format from Fanuc 16/18 to Fanuc 15. The default (0) is 16/18. To change to Fanuc 15 enter a 1. (NV only)

9.2. RTCP Output

Custom Setting 2 changes the RTCP output format from type I to type II. The default (0)is type I. To change type II enter 435. (NV only)

9.3. Rollover for Rotary Axis

Custom Setting 9 determines the rollover for rotary axis. The default (0) will output rotary axis values on the Machine will between -99999.999 and +99999.999 (Parameter 1008 on Control must be set to 0 also). Enter 1 and rotary axis values on the Machine will between -360. and +360. (Parameter 1008 on Control Must be set to 1 also).

9.4. Machine Name

Custom String 1 sets the name of the machine to be programmed. This should already be set.

9.5. Machine Model Number

Custom String 2 sets the model number name of the machine to be programmed. The NVX post processor supports 5060, 5080 and 5100 machines. The NV post processor supports 1800, 4000, 5000, 6000 and 7000 machines.

9.6. Tool Holder Type

Custom String 3 sets the tool holder type on the name of the machine to be programmed. This does not affect the output of the post.

9.7. TOOLTABLE

Entering TOOLTABLE into Custom String 10 creates a listing of all the tools used in the Esprit file at the beginning of the NC file.

```
%
O0001 ( )
(ESPRIT)
( MACHINE: NVX5080DCG )
(POST PROCESSOR : MoriSeikiNVX-2012-03-15)
(3/19/2012 2:05:38 PM)

(T1001 H1001 EM 15.0)
(T1 H1 DR 10.2)
(T2 H2 EM 05.0)

M200 (CHIP CONVEYOR START)
G00 G21 G40 G80 G94
M05
T1001
G91 G28 Z0
M11
G91 G28 A0
M10
G91 G28 X0
G91 G28 Y0
M01
```

9.8. StopTcOnly

By default, the post will output a stop code ‘M01’ at the end of every operation. Entering StopTCOnly (stop code for tool changes only), into Custom String 10 will tell the post to only output the stop code at the end of an operation when a tool change will occur. This data is not case sensitive.

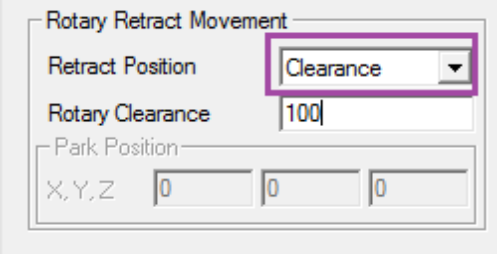
9.9. NTCOnly

By default, the post will output an N block at the beginning of every operation. Entering NTCOnly (N block for tool changes only), into Custom String 10 will tell the post to only

output an N block at the beginning of an operation when a tool change occurs. This data is not case sensitive. (NV only)

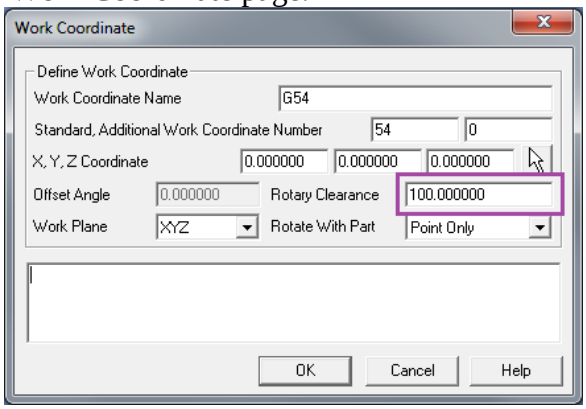
10. Rotary Retract Movement Setup

Retract Position on the Machine Setup lets the user set the retract location of the tool when the part is indexing between 2 operations. Currently the NVX and NV support Clearance and Tool Change.



10.1. Clearance

If the Retract Position is set to Clearance, the tool will retract to the Rotary Clearance value set on the active Work Coordinate page.




```
:
X-35.0
X-44.5
G00 Z47.0
Z100.0
M01

N2 (CUT2 OUTSIDE )
(EM 15.0 T69 H69 )
M11
```

```
M69
G00 B-90.0 C120.0
M10
M68
G90 G94 G54 G17 G00 X-44.5 Y-2.5
G43 Z47.0 H69 S1676 M03
M08
Z30.0
G01 X-35.0 F268.0
:
```

10.2. Tool Change

If the Retract Position is set to Tool Change, the tool will be sent home using ‘G91 G28 Z0.; G90 G53 X__ Y__’, where the X and Y values are set on the tool page. The Rotary Clearance value set on the Work Coordinate page will not be used.

```
:
X-35.0
X-44.5
G00 Z47.0
G91 G28 Z0
G90 G53 X-100.0 Y100.0
M01

N2 (CUT2 OUTSIDE )
(EM 15.0 T69 H69 )
M11
M69
G00 B-90.0 C120.0
M10
M68
G90 G94 G54 G17 G00 X-44.5 Y-2.5
G43 Z47.0 H69 S1676 M03
M08
Z30.0
G01 X-35.0 F268.0
X-27.5
Y2.5
X-35.0
:
```

11. Custom Settings Reference List

Here is a list of all of the Custom Settings for the NVX and NV post processor.

11.1. Machine Setup

	NVX	NV
Custom Setting 1		Switches from standard format to F15 format: 0 is standard 1 is F15
Custom Setting 2		Turns on TCP type 2: 435 outputs TCP Type 2; 0 Outputs Type 1
Custom Setting 3	End of Program Code; 0:M30, 99:M99, Anything Else M02	
Custom Setting 4		
Custom Setting 5	Rotary Axis addition: 0 is no addition axis, 1 is One Additional Rotary axis (4th axis); 2 is Two Additional Rotary axis (5th axis)	Rotary Axis addition: 1 is rotary axis addition; 0 is no addition
Custom Setting 6		
Custom Setting 7		
Custom Setting 8		
Custom Setting 9	Rollover for Rotary Axis; 0: Caxis Values on the Machine will between -99999.999 and +99999.999 (Parameter 1008 on Control Must be set to 0 also) 1: Rotary axis Values on the Machine will between -360. and +360. (Parameter 1008 on Control Must be set to 1 also)	
Custom Setting 10		
Custom String 1	NVX	NV(D)
Custom String 2	5060, 5080, 5100	1500, 4000, 5000, 6000
Custom String 3		DCG, DCG-a1A-40, DCG-a1A-50, DCG-a1B-40, DCG-a1B-50, DCG-40, DCG-50
Custom String 4		
Custom String 5		
Custom String 6		
Custom String 7		
Custom String 8		
Custom String 9		
Custom String 10	TOOLTABLE; Outputs a Listing of Tools at the top of the NC file	

	NTCOnly ; Limits the Output of N blocks for operations which start with a Tool Change

11.2. Operation Page

	NVX	NV
Custom Setting 1	For Contouring type operations, = 0 Rapid Moves output, <>0 Feedrate moves are output with a G01 and the enter value as a feedrate.	
	Q-Value for synchronized tapping	
	I value for G87	
	I value for G76	
Custom Setting 2	Cutting Mode G332; Value Entered as an R value(integer) or as T and A values(real)	
Custom Setting 3	AI Contouring; 0:Off, 1: G05 P10000, 2:G05 Q1, 3:G05.1 Q3 X0 Y0 Z0, 4:G08 P1	
Custom Setting 4	Secondary Coolant Codes; Value Entered Appears as M Code	
Custom Setting 5	Coolant Pressure Levels ; Value Entered Appears as M Code	
Custom Setting 6	Beginning of Operation M Code; Value Entered Appears as M Code	
Custom Setting 7	End of Operation M Code; Value Entered Appears as M Code	
Custom Setting 8		
Custom Setting 9		
Custom Setting 10	Sets the Feedrate Unit for Drilling Cycles. 95=G95.	

11.1. Tool Page

Currently no Custom Settings are being used on the Tool Page.