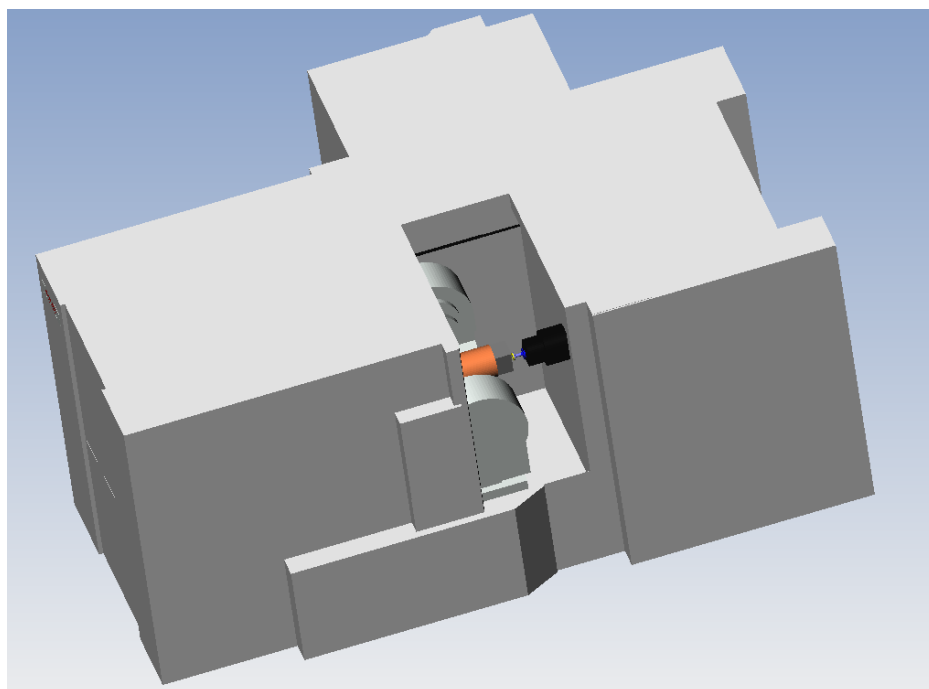




**POST PROCESSOR
MANUAL
FOR**

MORI SEIKI

NMH



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

Table Of Contents

1. Introduction	4
2. Defining Work Coordinates	4
2.1 Defining Global and Local Work Coordinates	4
2.2 Standard Work Coordinate System G54 to G59	5
2.3 Extended Work Coordinate System G54.1 P1-300	6
2.4 Dynamic Fixture Offset Work Coordinate G54.2 P	7
2.5 Tilted Plane Work Coordinate System G68.2.....	8
2.6 Macro Work Coordinate System G65 P9681	8
3. Drill Cycles	9
3.1 Drill – Standard Drill	9
3.2 Peck – Standard Peck	10
3.3 Peck2 – High Speed Peck.....	10
3.4 Tap – Tap	10
3.5 Tap2 – Deep Hole Tap	10
3.6 Bore – Standard Bore	10
3.7 Bore2 – Standard Bore	10
3.8 Bore3 – Back Bore	10
3.9 Bore4 – Manual Tool Extraction.....	10
3.10 Bore5 – Dwell	10
3.11 Bore6 – Fine Bore.....	10
3.12 Bore7 – Counter Bore	10
3.13 Changing Feed Units (Feed per Minute Feed per Revolution).....	11
4. 4th Axis Wrap Milling Cycles	12
4.1 Cylindrical Interpolation.....	12
4.2 Side Arcs with Cylindrical Interpolation	13
5. 5 Axis Milling Settings	14
5.1 RTCP	14
5.2 Calculate Links.....	16
6. NURB Output	17
7. Custom and Coolant Codes	18
7.1 Primary Coolant Code	18
7.2 Secondary Coolant Code	19
7.3 Coolant Pressure Code.....	19
7.4 Custom First and Second Coolant Position	20
7.5 M-Codes at Beginning and End of Operation	20
7.6 AI Contouring	21
7.7 Cutting Mode (G332 Setting Value)	22
8. Custom Settings	23
A. Machine Custom Settings	24
8.1 F15 Format Switch	24
8.2 TCP Type.....	24
8.3 Program End Code	25
8.4 Rotary Axis Addition.....	25

8.5 Output Tool Return	25
8.6 Output Tool List.....	25
8.7 SuppressClamp.....	25
8.8 Rollover for B-Axis.....	25
B. Machine Custom Strings	26
8.9 Machine Name	26
8.10 Machine Model Number	26
8.11 Tool Holder Type.....	26
8.12 Position of Sequence Numbers (N).....	27
C. Machine Extra Settings.....	27
8.13 ESPRIT Version	27
D. Operation Page Custom Settings	28
8.14 Rapid Feedrate (G01 F)	28
E. Tool Page Custom Settings.....	29
9. Rotary Retract Movement Setup.....	29
9.1 Clearance.....	30
9.2 Tool Change.....	30

1. Introduction

This manual documents the Esprit NMH post processor version 03/02/12.
The E12 MSP_NM_H_2012-03-02.pst post supports the NMH machine tool.
The post processor requires ESPRIT 2011 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.

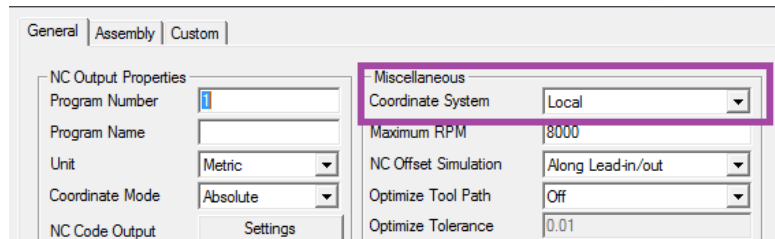


Fig. 1: Setting a Local Coordinate System

For most applications this will be set to Local.

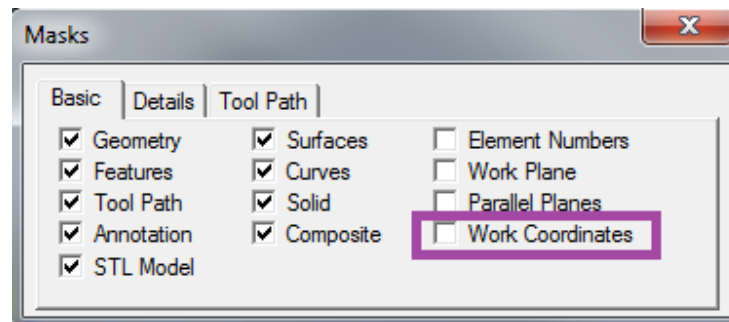


Fig. 2: Displaying Work Coordinate on Workspace

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

2.1 Defining Global and Local Work Coordinates

Depending on the application, Work Coordinates can be defined as 'Local,' that is, movements will be in reference to the Work Coordinate location -- effectively making the position of the Work Coordinate the origin:

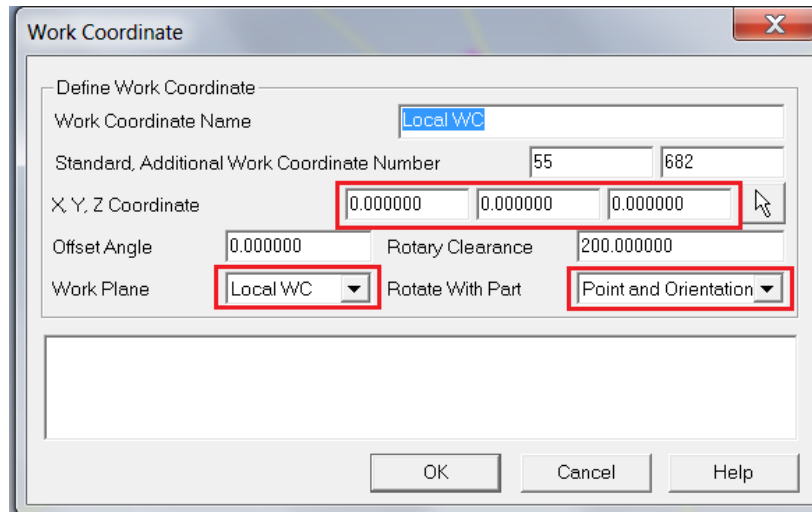


Fig. 3: Defining a Local Work Coordinate

Note: The Work Plane needs to be where the Work Coordinate is defined, and Rotate with Part needs to be set to Point and Orientation

Alternatively, the Work Coordinate can be defined as 'Global' whereby movements will be in reference to a location that is standard to all Work Coordinates in a file (For instance, the origin [0,0,0]):

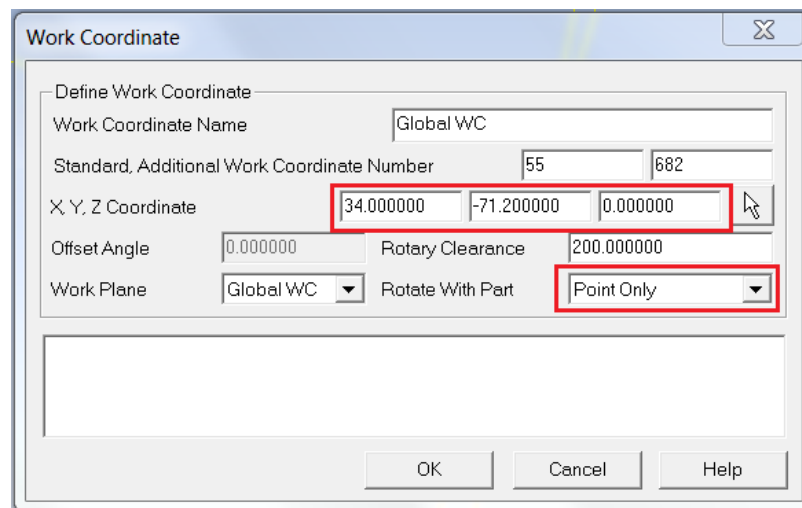


Fig. 4: Defining a Global Work Coordinate

Note: Rotate with Part needs to be set to Point Only

2.2 Standard Work Coordinate System G54 to G59

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

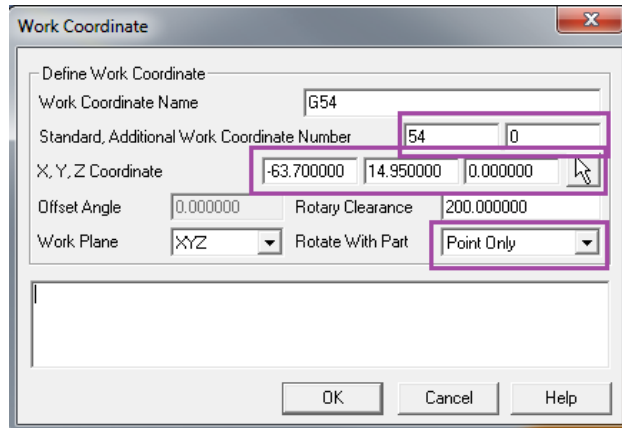


Fig. 5: Defining a Standard Work Coordinate System

NC Code Example:

```

N1 (0 FACING )
  (EM 15.0 T17 H17 )
G49
M05
G91 G30 Z0
G91 G30 X0 Y0
T17
M06
G90 G94 G54 G17 G00 X-109.5 Y100.0
M11
M79
G00 A-90.0 B-90.0
[....]

```

2.3 Extended Work Coordinate System G54.1 P1-300

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

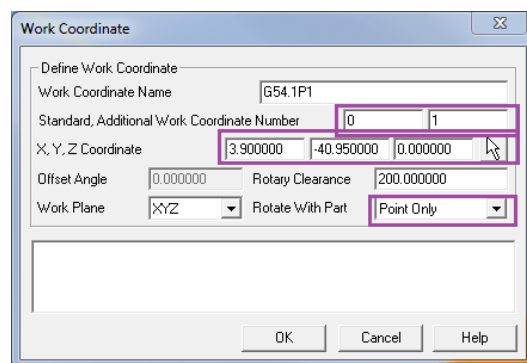


Fig. 6: Defining an Extended Work Coordinate System

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**.

NC Code Example:

```

N1 ( FACING )
  ( FACEMILL 75 T3 H03 )
G49
M05
G91 G30 Z0
G91 G30 X0 Y0
T3
M06
G90 G94 G54.1 P1 G17 G00 X-139.5 Y100.0
M11
M79
G00 A-90.0 B-90.0
M10

```

2.4 Dynamic Fixture Offset Work Coordinate G54.2 P

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

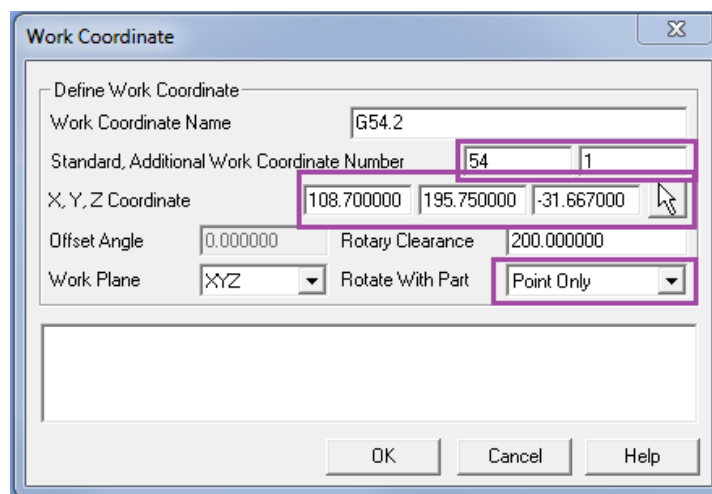


Fig. 7: Defining a Dynamic Fixture Offset

Standard is set to **54** 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**.

NC Code Example:

```

N2 ( OPEN POCKET )
  ( EM 20.0 T66 H66 D66 )
G49
M05
G91 G30 Z0
  G91 G30 X0 Y0
  T66
  M06
  T3
G90 G54

```

G54.2 P2
M11

2.5 Tilted Plane Work Coordinate System G68.2

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

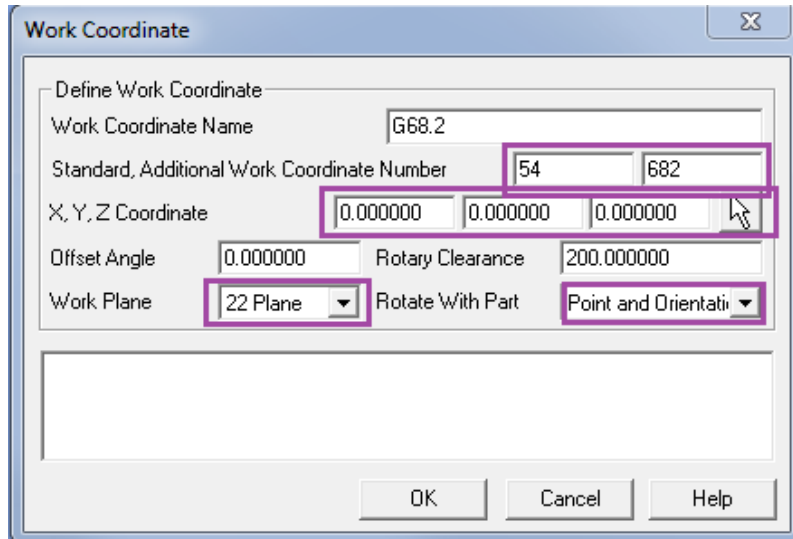


Fig. 8: Defining a Tilted Plane Work Coordinate

Standard is set to **54** and Additional Work Coordinate Number is set to **682**. XYZ is the location of the Work Coordinate in the file relative to the Work Plane. The Work Plane is the Work Plane of the feature to be machined. Rotate with Part needs to be set to **Point and Orientation**

NC Code Example:

```
N1 (FACING )
(FACEMILL 75 T3 H03 )
G49
M05
G91 G30 Z0
G91 G30 X0 Y0
[...]
G68.2 X0.0 Y0.0 Z0.0 I180. J90. K90.
G53.1
M10
```

2.6 Macro Work Coordinate System G65 P9681

This item is an option on your machine 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:

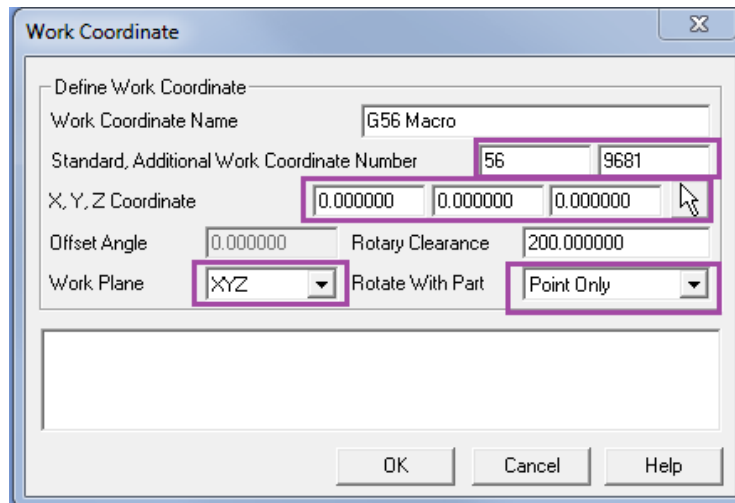


Fig. 8: Defining a Macro Work Coordinate

Standard is set to **54** and Additional Work Coordinate Number is set to **682**. XYZ is the location of the Work Coordinate in the file relative to the Work Plane. The Work Plane is the Work Plane of the feature to be machined. Rotate with Part needs to be set to **Point and Orientation**

NC Code Example:

```

N1 (FACING )
(FACEMILL 75 T3 H03 )
G49
M05
G91 G30 Z0
G91 G30 X0 Y0
T3
M06
T66
G90 G54
G65 P9681 X0.0 Y0.0 Z0.0 A-90.0 B-90.0 M54. Q59.
G59
M11

```

3. Drill Cycles

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

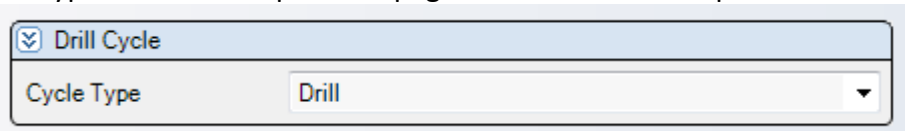


Fig. 9: Selecting Drill Cycle Type

3.1 Drill – Standard Drill

```
G99 G81 Z-3.583 R2.0 F687.0
```

3.2 Peck – Standard Peck

G99 G83 Z-3.583 R2.0 Q2.5 F687.0

3.3 Peck2 – High Speed Peck

G99 G73 Z-3.583 R2.0 Q2.5 F687.0

3.4 Tap – Tap

Standard Format

G99 G84 (74) Z-3.583 R2.0 F687.0

F15 Format

G99 G84.2 (84.3) Z-3.583 R2.0 F687.0

3.5 Tap2 – Deep Hole Tap

Standard Format

G99 G84 (74) Z-3.583 Q2.5 R2.0 F687.0

F15 Format

G99 G84.2(84.3) Z-3.583 Q2.5 R2.0 F687.0

3.6 Bore – Standard Bore

G99 G85 Z-3.583 R2.0 F687.0

3.7 Bore2 – Standard Bore

G99 G86 Z-3.583 R2.0 F687.0

3.8 Bore3 – Back Bore

G99 G87 Z2.0 R-3.583 Q2.5 P100000 F687.0

3.9 Bore4 – Manual Tool Extraction

G99 G88 Z-3.583 R2.0 P100000 F687.0

3.10 Bore5 – Dwell

G99 G89 Z-3.583 R2.0 P100000 F687.0

3.11 Bore6 – Fine Bore

G99 G76 Z-3.583 R2.0 Q2.5 P100000 F687.0

3.12 Bore7 – Counter Bore

G99 G82 Z-3.583 R2.0 P100000 F687.0

Name of Drill Cycle	Description	Unique Setting	Custom Codes
Drill -- G81	Standard Drill	N/A	N/A
Peck -- G83	Standard Peck	Peck Increment (Q)	*Set Peck Increment on Operation Page
Peck2 -- G73	High Speed Peck	Peck Increment (Q)	*Set Peck Increment on Operation Page
Tap -- G84, 74, 84.2, 84.3	Tap	*Standard and F15 *Fanuc Format	*Select ' Standard Format ' in F15 Format Switch of Global Custom Setting Utility to use Fanuc 16/18 *Select ' F15 Format ' in F15 Format Switch of Global Custom Setting Utility to use Fanuc 15

Tap2 -- G84, 74, 84.2, 84.3	Deep Hole Tap	*Standard Fanuc *F15 Fanuc Format *Peck Increment (Q)	*Select ' Standard Format ' in F15 Format Switch of Global Custom Setting Utility to use Fanuc 16/18 *Select ' F15 Format ' in F15 Format Switch of Global Custom Setting Utility to use Fanuc 15 *Set ' Rigid Tap Q Value ' on Operation Page to a Peck Increment
Bore -- G85	Standard Bore	N/A	N/A
Bore2 -- G86	Standard Bore	N/A	N/A
Bore3 -- G87	Back Bore	*Peck Increment(Q) *Dwell Time(P)	*Set ' Rigid Tap Q Value ' on Operation Page to a Peck Increment *Set Dwell Time on Operation Page
Bore4 -- G88	Manual Tool Extraction	*Dwell Time(P)	*Set Dwell Time on Operation Page
Bore5 -- G89	Dwell	*Dwell Time(P)	*Set Dwell Time on Operation Page
Bore6 -- G76	Fine Bore	*Peck Increment(Q) *Dwell Time(P)	*Set ' Rigid Tap Q Value ' on Operation Page to a Peck Increment *Set Dwell Time on Operation Page
Bore7 -- G82	Counter Bore	*Dwell Time(P)	*Set Dwell Time on Operation Page

3.13 Changing Feed Units (Feed per Minute | Feed per Revolution)

The feed unit for Drill Cycles can be changed from the default Feed per Minute (G94) to Feed per Revolution (G95) by **inputting 95 into 'Feedrate Unit' on the Operation Page.**

With 'Feedrate Unit' set to 0 : [...]

```
G90 G94 G54 G17 G00 X0.0 Y0.0
M11
M79
G00 A-90.0 B-90.0
M10
M78
G43 Z0.0 H01 S2000 M03
M07
G98 G81 Z37.0 R52.0 F100.0
G80
[...]
```

With 'Feedrate Unit' set to 95: [...]

```

G90 G94 G54 G17 G00 X0.0 Y0.0
M11
M79
G00 A-90.0 B-90.0
M10
M78
G43 Z0.0 H01 S2000 M03
M07
G95
G98 G81 Z37.0 R52.0 F0.05
G80
[...]
```

4. 4th Axis Wrap Milling Cycles

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

4.1 Cylindrical Interpolation

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

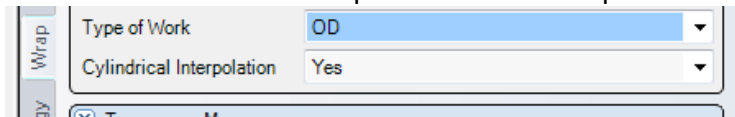
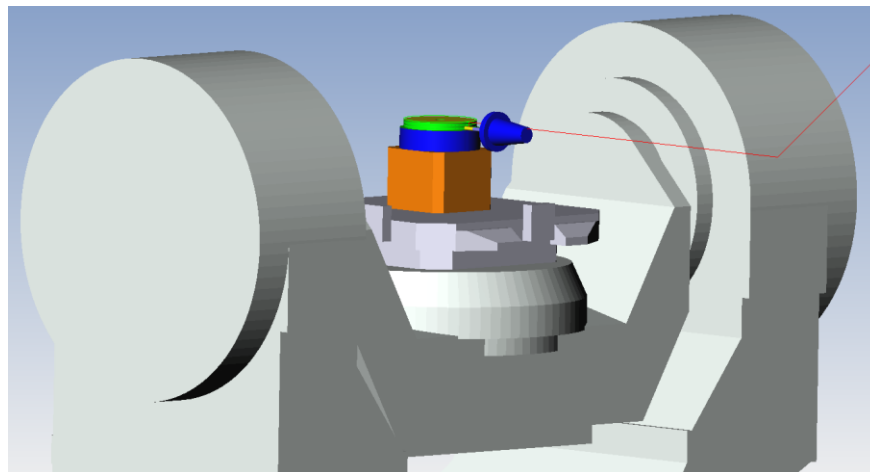


Fig. 10: Enabling Cylindrical Interpolation



NC output with Cylindrical Interpolation ON:[...]

```

M06
G90 G94 G54 G17 G00 X0.0 Y2.0
M11
M79
G00 A0.0 B90.0
```

```

M78
G43 Z102.0 H02 S1000 M03
M08
G07.1 B100.0
G01 Z90.0 F1000.0
B90.0
Z102.0
G00 Y-6.0

```

NC output with Cylindrical Interpolation OFF:[...]

```

M06
G90 G94 G54 G17 G00 X0.0 Y2.0
M11
M79
G00 A0.0 B90.0
M78
G43 Z102.0 H02 S1000 M03
M08
G01 Z90.0 F1000.0
B210.0 F637.0
B330.0
B90.0
Z102.0 F1000.0
G00 Y-6.0

```

4.2 Side Arcs with Cylindrical Interpolation

The post processor handles the output of Side Arcs with Cylindrical Interpolation depending on the angle the Side Arc sweeps. Currently for arcs less than 180 degrees the output will follow the expected movement seen in simulation.

However, for arcs greater than 180 degrees, an error message will be output in place of the movement:

```

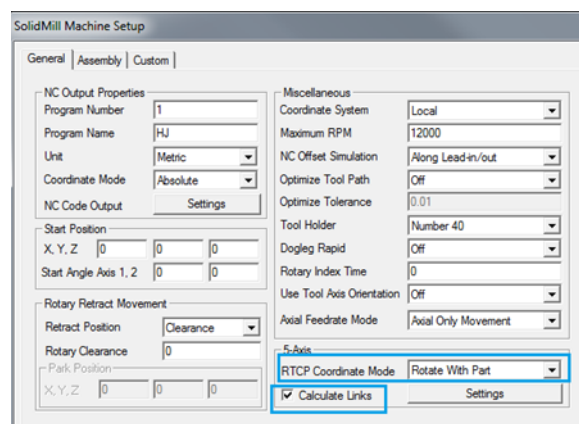
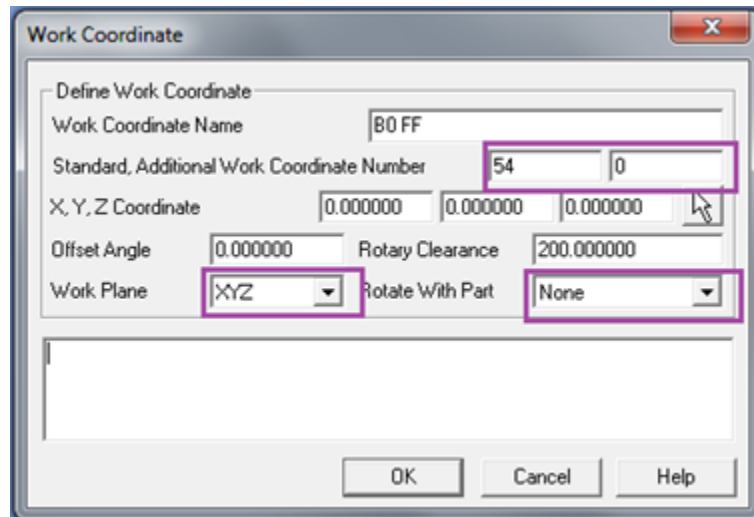
[...]
M08
G07.1 B100.0
G01 Z90.0 F7374.0
Y246.896 B32.664 F9831.0
(Error - MACHINE DOES NOT SUPPORT ARCS OVER 180 WITH
CYLINDRICAL INTERPOLATION ON)
Y419.914 B356.962
Y417.982 B357.871
Z102.0 F7374.0
G07.1 B0
M09
G49
[...]

```

5. 5 Axis Milling Settings

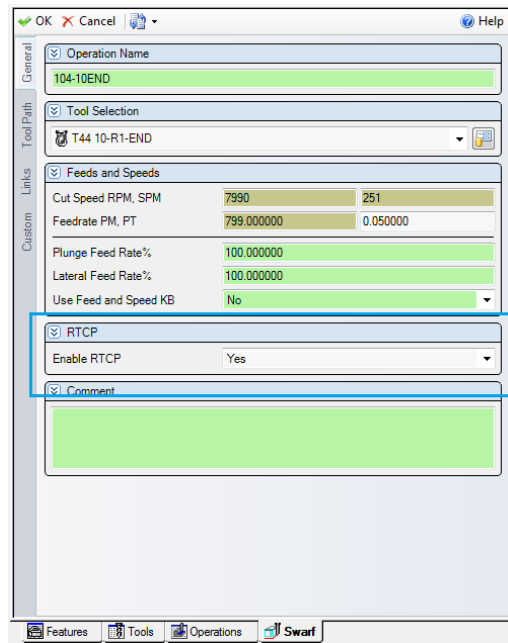
There are a few crucial settings and considerations that need to be set before programming a 5-axis part.

ESPRIT Work Coordinate for 5-axis simultaneous milling operations must be set the following way:



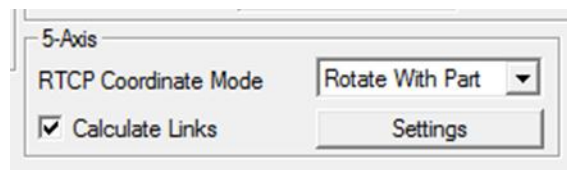
5.1 RTCP

- On the **Machine Setup Page** > **RTCP Coordinate Mode** needs to be set to 'Rotate With Part'
- On the **Operation Page**, always remember to 'Enable RTCP'.



• RTCP Type I (G43.4)

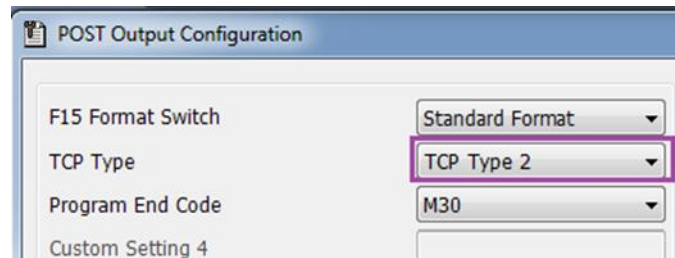
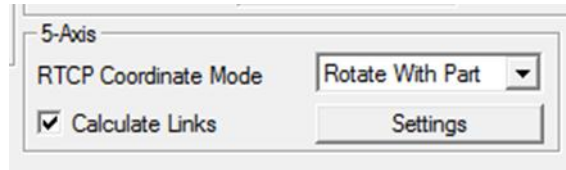
To use Type I, set Parameter 19696 #5=0 on the machine tool. On the Machine Setup Page set RTCP Coordinate Mode to Rotate with Part. Note we also recommend having Calculate Links checked.



```
[...]
M06
T1
M11
M79
G90 G94 G54 G17 G00 X58.1 Y-49.5
G90 A0.0 B0.0
G49
G05 P10000
G43.4 H02 S3024 M03
G00 X58.1 Y-49.5 Z-24.0 A-90.0 B-90.0
M08
G01 X58.088 Y-51.5 F1000.0
X53.1 F2000.0
X53.089 Z20.565
[...]
```

• RTCP Type II (G43.5)

To use Type II, set Parameter 19696 #5=0 on the machine tool. On the Machine Setup Page set RTCP Coordinate Mode to Rotate with Part. Note we also recommend having Calculate Links checked.



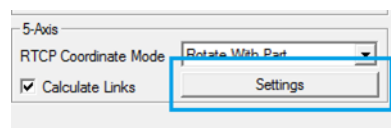
In the Post Output Configuration Tool, on the Mori Seiki tool bar, select TCP Type 2.

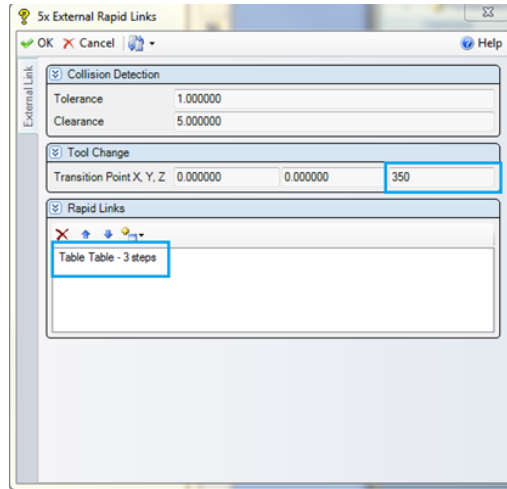
```
[...]
M06
T1
M11
M79
G90 G94 G54 G17 G00 X58.1 Y-49.5
G90 A0.0 B0.0
G49
G05 P10000
G43.5 H02 S3024 M03
G00 X58.1 Y-49.5 Z-24.0 I0 J1. K0
M08
G01 X58.088 Y-51.5 I0 J1. K0 F1000.0
X53.1 I0 J1. K0 F2000.0
X53.089 Z20.565 I0 J1. K0
```

5.2 Calculate Links

On the **Machine Setup Page** > **Calculate Links** Box needs to be **Checked**.

After the Links Box is Checked, the Settings button becomes active. Press the Settings button to define your links. A recommended Z Transition Point value is something larger than your part in any axis direction (i.e., outside the machining envelope of the part). There are many different type of Rapid Links Available; Table-Table is simple, straightforward and recommended. The Esprit Manual has through explanation of all the Rapid Links.





6. NURB Output

The NMH post processor has NURB and non NURB Output. Select the type of output you want on the **Machine Setup Page** > **Optimize Tool Path**. Optimize Tool Path needs to be set before the operation is created. If you forget to set it before creating the operation, simply rebuild the operation and current Optimize Tool Path setting will be applied.

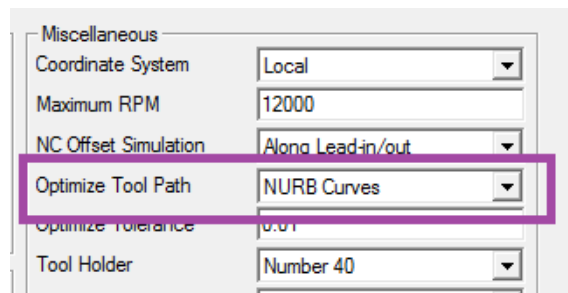


Fig. 11: Enabling NURB Curve Optimization in Machine Setup

NC output with NURB Output ON: [...]

```

M08
Z6.498
G01 Z4.498 F122.54
X126.049 Y113.257 Z-0.001 F490.14
X126.137 Y113.316 Z4.498
X126.049 Y113.257 Z-0.001
G06.2 P4 K0 X56.049 Y393.257 Z189.779 R1.
F490.14
K0 Y388.859 Z189.78
K0 Y379.137 Z189.779
K0 Y349.554 Z189.78
K.149 Y324.369 F350.1
K.329 Y304.507
[...]
```

NC output with NURB Output OFF: [...]
M08
Z6.498
G01 Z4.498 F122.54
X126.049 Y113.257 Z-0.001 F490.14
Y107.007 Z0.0
Y24.5 F350.1
X126.137 Z4.499 F122.54
G00 Z6.499
X127.537 Y112.945
Z6.329
[...]

7. Custom and Coolant Codes

Coolant codes can be selected in two ways.

For the primary coolant, going to the **General tab of the Tool Page**, coolants are selected from the **Coolant drop-down**.

For the secondary coolant, going to the **Custom tab of the operation page**, a coolant code can be input in '**Second Coolant.**' Up to two coolant codes can be output per tool.

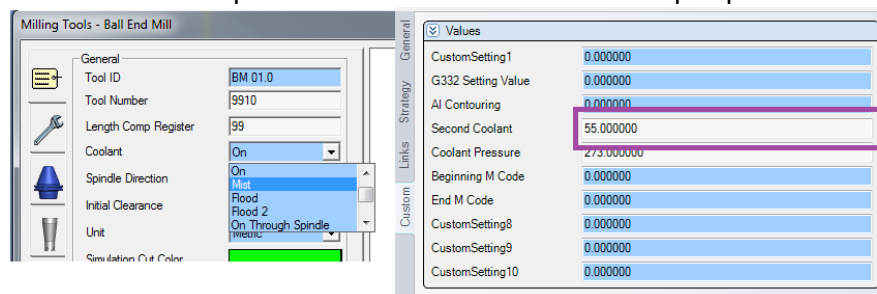


Fig. 11: Setting Coolant Codes – Primary on Tool Page (Left) | Secondary on Operation page (Right)

7.1 Primary Coolant Code

Esprit Pull down	NMH M Code	
	Start	Stop
On	M08	M09
Mist	M50	M09
Flood	M50	M09
Flood 2	M51	M59
On Through Spindle	M88	M89
Mist Through Spindle	M204	M205
Flood Through Spindle	M52	M58
Flood2 Through Spindle	M80	M81

7.2 Secondary Coolant Code

Second Coolant	NMH M Code	
	Start	Stop
7	M07	M09
8	M08	M09
50	M50	M09
51	M51	M59
52	M52	M58
53	M53	M58
55	M55	M09
80	M80	M81
88	M88	M89
92	M92	M09
204	M204	M205

Note: When using Oil Shot 1 (M07), Sensor Air Blow (M53), Oil Shot 2 (M55), or Oil Mist as the only coolant; the primary coolant type should be set to off.

7.3 Coolant Pressure Code

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

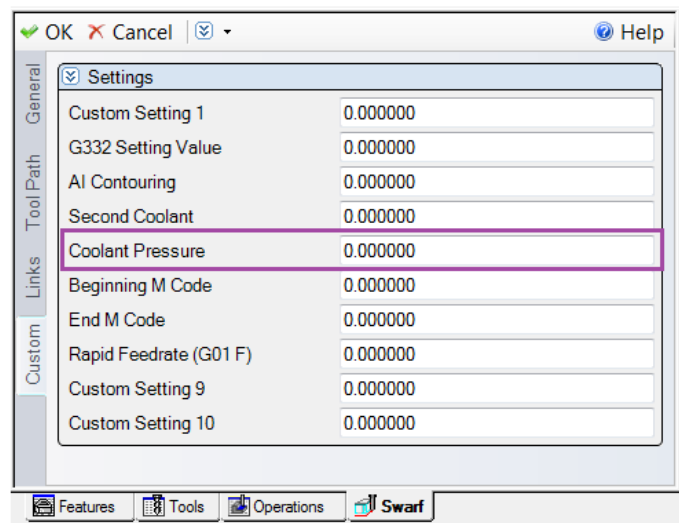


Fig. 12: Setting the Coolant Pressure

Coolant Pressure	NMH M Code
270	M270
271	M271
272	M272
273	M273
274	M274
275	M275
276	M276
277	M277

7.4 Custom First and Second Coolant Position

Entering '**COOLANTORDER1**' in **Custom String 10** moves the position of the First and Second Coolants to before the first movement of the operation to ensure a coolant is being used before the machine begins cutting.

Without COOLANTORDER1 in Custom String 10

```

N7 (OIL SHOT 2 )
(DR 0.3 T7 H07 )
G49
M05
G91 G30 Z0
G91 G30 X0 Y0
T7
M06
T8
G90 G94 G54 G17 G00 X0.0 Y0.0
M11
M79
G00 A-90.0 B-90.0
M10
M78
M276 <----- Coolant Pressure
M12345 <----- Beginning M-Code
G43 Z0.0 H07 S1668 M03
M50 <----- First Coolant
M55 <----- Second Coolant
G98 G81 Z37.0 R52.0 F83.0
G80
M09
M09
G49
G91 G30 Z0
G91 G30 X0 Y0
M01

```

With COOLANTORDER1 in Custom String 10

```

N7 (OIL SHOT 2 )
(DR 0.3 T7 H07 )
G49
M05
G91 G30 Z0
G91 G30 X0 Y0
T7
M06
T8
G90 G94 G54 G17 G00 X0.0 Y0.0
M11
M79
G00 A-90.0 B-90.0
M10
M78
M276 <----- Coolant Pressure
M50 <----- First Coolant
M55 <----- Second Coolant
M12345 <----- Beginning M-Code
G43 Z0.0 H07 S1668 M03
G98 G81 Z37.0 R52.0 F83.0
G80
M09
M09
G49
G91 G30 Z0
G91 G30 X0 Y0
M01

```

7.5 M-Codes at Beginning and End of Operation

Custom codes can be used to output any M code near the beginning of an operation and at the end of an operation. '**Beginning M Code**' defines the code at the **beginning of an operation** and '**End M Code**' can define one **at the end**.

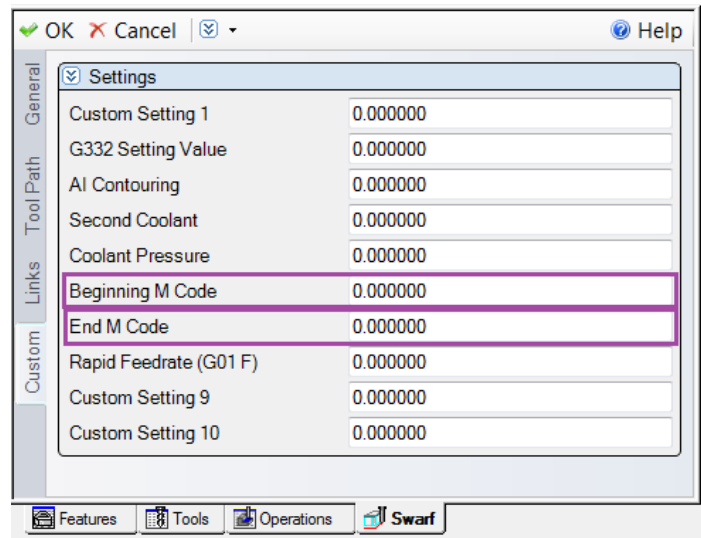


Fig. 13: Setting Custom codes at Beginning and End of an operation

M-Code Placement:

```
G90 G94 G54 G17 G00 X0.0 Y0.0
M11
M79
G00 A-90.0 B-90.0
M10
M78
M276 ## Coolant Pressure
M12345 ## Beginning M-Code
G43 Z500.0 H79 S115 M03
M55 ## Secondary Coolant
G98 G81 Z-50.0 R2.0 F6.0
G80
M09
G49
G91 G30 Z0 M05
```

7.6 AI Contouring

To use AI Contour Control, enter the corresponding value into **AI Contouring** in the **operation page**.

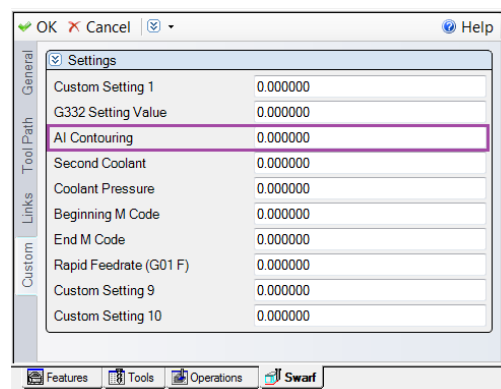


Fig. 14. Setting AI Contour Control

AICC	AI Contouring
G05 P10000	1
G05.1 Q1	2
G05.1 Q3	3
G08 P1	4

Placement of the AI Code: [...]
M11
G00 B0.0
M10
G94 G17 G00 X-7.0 Y140.0
G05 P10000
G43 Z2.0 H04 S11650 M03
M08
G01 Z0.0 F7101.84
X140.0 F9469.12
[...]

7.7 Cutting Mode (G332 Setting Value)

To use Cutting Mode during a cycle, place the corresponding value into '**G332 Setting Value**' 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.

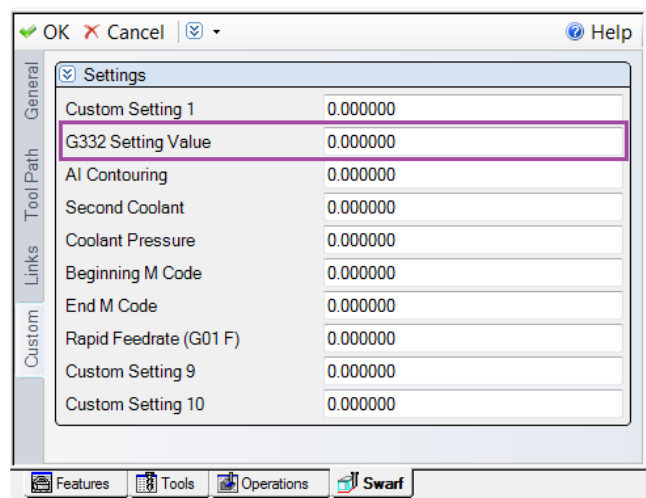


Fig. 15. Setting Cutting Mode

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

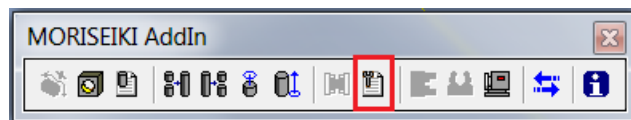
Placement of Cutting Mode Code: [...]

```

M11
G00 B0.0
M10
G94 G17 G00 X-7.0 Y140.0
G332 T0.056 A23.0
G05 P10000
G43 Z2.0 H04 S11650 M03
M08
G01 Z0.0 F7101.84
[...]
```

8. Custom Settings

Your system already has a configuration for the NMH 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 of the operation. You can also access Machine-based Custom options through the Mori Seiki Add-In Toolbar seen here:



Through this Custom page you can toggle settings through the drop-downs or text fields. Each option's functionality and enabling/disabling options are described below:

A. Machine Custom Settings

F15 Format Switch	Standard Format	Custom String 1	
TCP Type	TCP Type 2	Custom String 2	
Program End Code	M30	Custom String 3	
Custom Setting 4		Custom String 4	
Rotary Axis Addition	Off	Custom String 5	
Output Tool Return M Code	Off	Custom String 6	
Output Tool List	Off	Custom String 7	
Suppress Clamp Mcode	Off	Custom String 8	
B-axis Roll Over	Roll Over Off	Position of Sequence Numbers (N)	At Operations
Custom Setting 10		Custom String 10	

Custom Setting 1	F15 Format Switch	8.1
Custom Setting 2	TCP Type	8.2
Custom Setting 3	Program End Code	8.3
Custom Setting 4	OPEN	----
Custom Setting 5	Rotary Axis Addition	8.4
Custom Setting 6	Output Tool Return	8.5
Custom Setting 7	Output Tool List	8.6
Custom Setting 8	Suppress Clamp	8.7
Custom Setting 9	B Axis Roll Over	8.8
Custom Setting 10	OPEN	----

8.1 F15 Format Switch

On the *Global Custom Setting Utility* page, **F15 Format Switch** changes the NC code output format from Fanuc 16/18 to Fanuc 15. The default **'Standard Format'** is Fanuc 16/18. To change to Fanuc 15 **click the drop down menu and select 'F15 Format.'**

8.2 TCP Type

On the *Global Custom Setting Utility* page, **'TCP Type'** changes the G-Code for TCP depending on the type. If **TCP Type 2** is selected, "G43.5" will be output. If **TCP Type 1** is selected, "G43.4" will be output.

8.3 Program End Code

On the *Global Custom Setting Utility* page, '**Program End Code**' determines the M code output at the end of the NC file. The default is M30. Select '**M02**' from the drop-down menu to output M02; select '**60:M60**' to output M60; select '**M99**' to output M99.

8.4 Rotary Axis Addition

On the *Global Custom Setting Utility* page, selecting '**1: Suppress A Axis**' from the drop-down menu into '**Rotary Axis Addition**' will prevent the A-axis from being released on linear moves; selecting '**0: Off**' will turn this option off.

8.5 Output Tool Return

On the *Global Custom Setting Utility* page, '**Output Tool Return**' outputs M34 at the end of the NC file if '**1: M01/M34**' is entered

8.6 Output Tool List

On the *Global Custom Setting Utility* page, selecting '**1: On**' from the drop-down menu into '**Output Tool List**' creates a listing of all the tools used in the Esprit file at the beginning of the NC file.

8.7 SuppressClamp

On the *Global Custom Setting Utility* page, if '**1: On**' is selected in '**SuppressClamp**', clamping and unclamping codes are suppressed.

8.8 Rollover for B-Axis

On the *Global Custom Setting Utility* page, '**B Axis Roll Over**' determines the rollover for B axis. The default '**Roll Over Off**' will output B axis Values on the Machine will between -99999.999 and +99999.999 (Parameter 1008 on Control Must be set to 0 also). Select '**Roll Over On**' and B axis Values on the Machine will between -360. and +360. (Parameter 1008 on Control must be set to 1 also).

B. Machine Custom Strings

Custom String 1	Machine Name	8.9
Custom String 2	Machine Model Number	8.10
Custom String 3	Tool Holder Type	8.11
Custom String 4	OPEN	----
Custom String 5	OPEN	----
Custom String 6	OPEN	----
Custom String 7	OPEN	----
Custom String 8	OPEN	----
Custom String 9	Position of Sequence Numbers (N)	8.12
Custom String 10	Custom String 10	7.4

8.9 Machine Name

On the **Machine Setup Page**, **Custom String 1** sets the name of the machine to be programmed. This should already be set to NMH.

8.10 Machine Model Number

On the **Machine Setup Page**, **Custom String 2** sets the model number name of the machine to be programmed. The NMH post processor supports 5000, 6300, and 10000 machines.

8.11 Tool Holder Type

On the **Machine Setup Page**, **Custom String 3** sets the tool holder type on the name of the machine to be programmed. The word **Indexing** can also appear in this string.

Indexing determines the output of the B values. If Indexing is present, the values will be rounded to integers. This string is not case sensitive.

8.12 Position of Sequence Numbers (N)

By default, the NMH post will output an N block at the beginning of every operation. Selecting '**NTCOnly:At Tool Changes**' (N block for tool changes only) from the drop-down of **Custom String 9 – 'Position of Sequence Numbers (N)'** will tell the post to only output an N block at the beginning of an operation when a tool change occurs.

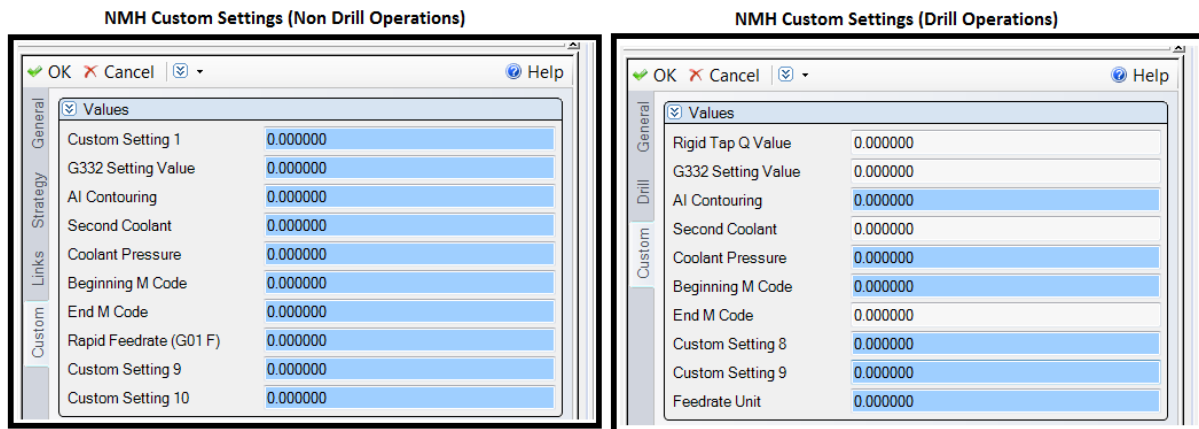
C. Machine Extra Settings

Extra Setting 1	<input type="text"/>	Extra Setting 11	<input type="text"/>
Extra Setting 2	<input type="text"/>	Extra Setting 12	<input type="text"/>
Extra Setting 3	<input type="text"/>	Extra Setting 13	<input type="text"/>
Extra Setting 4	<input type="text"/>	Extra Setting 14	<input type="text"/>
Extra Setting 5	<input type="text"/>	Extra Setting 15	<input type="text"/>
Extra Setting 6	<input type="text"/>	Extra Setting 16	<input type="text"/>
Extra Setting 7	<input type="text"/>	Extra Setting 17	<input type="text"/>
Extra Setting 8	<input type="text"/>	Extra Setting 18	<input type="text"/>
Extra Setting 9	<input type="text"/>	Extra Setting 19	<input type="text"/>
Extra Setting 10	<input type="text"/>	ESPRIT Version	12

8.13 ESPRIT Version

Displays the version of Esprit being used.

D. Operation Page Custom Settings

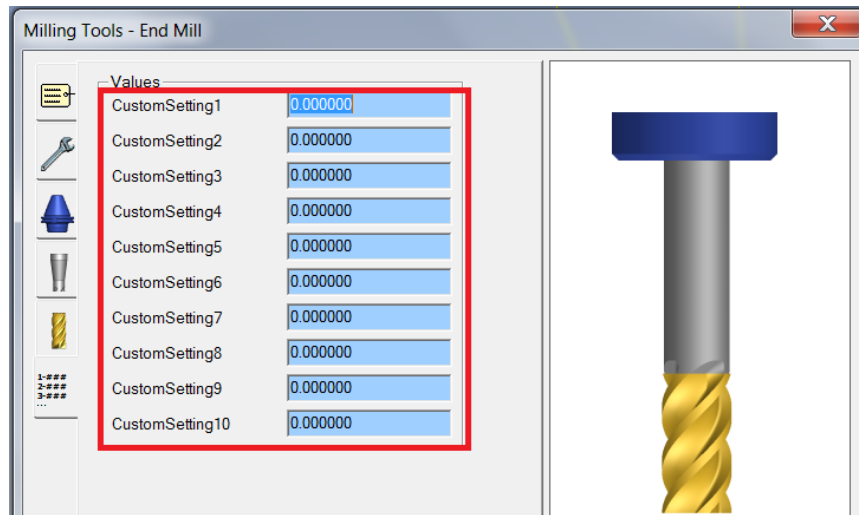


Custom Setting 1	Rigid Tap Q Value	3.5, 3.8, 3.11
Custom Setting 2	G332 Setting Value	7.6
Custom Setting 3	AI Contouring	7.5
Custom Setting 4	Secondary Coolant	7.2
Custom Setting 5	Coolant Pressure	7.3
Custom Setting 6	Beginning M Code	7.4
Custom Setting 7	End M Code	7.4
Custom Setting 8	Rapid Feedrate (G01 F)	8.14
Custom Setting 9	OPEN	----
Custom Setting 10	Feedrate Unit	3.13

8.14 Rapid Feedrate (G01 F)

*On the Custom Tab of the Operation Page, 'Rapid Feedrate (G01 F)' allows the user to change rapid moves (G00) to feed moves (G01) for the entirety of the operation. By entering a **Feed value**, the switch will be enabled with the number input as the Feed.*

E. Tool Page Custom Settings



Custom Setting 1	OPEN	----
Custom Setting 2	OPEN	----
Custom Setting 3	OPEN	----
Custom Setting 4	OPEN	----
Custom Setting 5	OPEN	----
Custom Setting 6	OPEN	----
Custom Setting 7	OPEN	----
Custom Setting 8	OPEN	----
Custom Setting 9	OPEN	----
Custom Setting 10	OPEN	----

9. 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 NMH post processor supports **Clearance** and **Tool Change**.

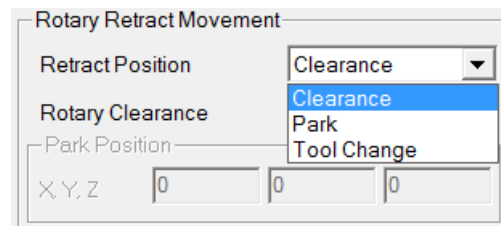


Fig. 16: Setting the Rotary Retract position

9.1 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.

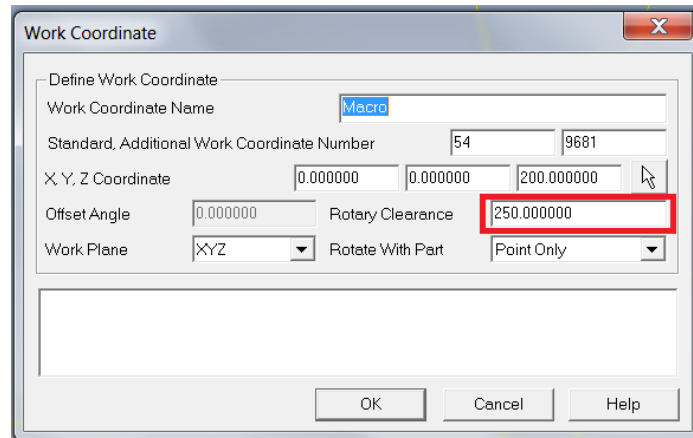


Fig. 17: Setting the Rotary Clearance position

```
[...]
G01 Z-165.0 F18.579
Y334.8 F157.92
B843.894 F1.774
B921.163 F1.774
B998.433 F1.774
Z-158.0 F45.12
G00 Z-148.0
Z250.0
M10
G49
G05 P0
M01

N2 (MACRO WITH WRAP )
(TOOL 1 T2 H02 )
G90 G54
G65 P9681 X0.0 Y0.0 Z200.0 A0.0 B1126.625 M54. Q59.
G59
[...]
```

9.2 Tool Change

If the Retract Position is set to Tool Change, the tool will be sent home using 'G91 G28 Z0'. The Rotary Clearance value set on the Work Coordinate page will not be used.

```
[...]
X6.065
X0.0 Y23.914
X-6.065 Y31.868
X0.0
```

Y33.868 Z2.0 F1519.0

G00 Z22.0

G91 G28 Z0

G49

G54.2 P0

M01

N3 (OPEN POCKET)

(EM 20.0 T66 H66 D66)

G90 G54

G54.2 P1

[...]