



CNC Mill Program Structure Specification

Global Common

SD-1042

ISSUED January 31, 2017
REVISED November 4, 2019

Table of Contents

1. Scope.....	3
2. Program Structure.....	3
3. Main Program Name and Number	3
4. Program Header.....	4
5. Safe Start	4
6. Setting the Work Coordinate System.....	4
7. Offset Linking and Checks.....	5
8. Macro Variable and Parametric Programming Organization and Description.....	5
9. M-Code Organization and Description	6
10. Subprograms	6

1. Scope

- 1.1 The intent of this document is to provide specifications to Original Equipment (OEM) Manufacturers in their design and implementation of CNC programs for CNC mills. This document establishes program templates and numbering systems to improve engineer and team member familiarity and understanding of the programs.
- 1.2 This specification assumes Fanuc control. Machines that utilize a different CNC manufacturer should follow similar guidelines.
- 1.3 Any deviation to the specification can be approved if the purchasing Manufacturing Engineer approves the deviation.
- 1.4 The use of the word "shall" indicates requirements and the use of the word "should" indicates recommendations. The use of the word "may" indicates permission or allowance and the use of the word "can" indicates a possibility.

2. Program Structure

- 2.1 CNC programs for mills for Nexteer Automotive shall be structured in the following manner:

- 2.1.1 Main Program Number and Title
- 2.1.2 Header
- 2.1.3 Macro Variable List and Description
- 2.1.4 Offset Linking (if necessary)
- 2.1.5 Machining Program
- 2.1.6 Program Completion.

2.2 Absolute programming shall be used throughout the program. Small incremental moves may be used but shall be denoted by U and W.

2.3 All Programs shall be in metric units

2.4 Constant RPM and Feed per Minute should be used

2.5 Tool description and features machined shall be included at tool call per the following example:
T0101 (14MM ENDMILL);
(SPOTFACE FOR MOUNTING HOLES);

3. Main Program Name and Number

- 3.1 The main program number should follow a four digit naming convention. It shall start with an 8 and be followed by the last 3 digits of the part number. See below for Subprogram Numbering Conventions.

08321;

4. Program Header

- 4.1 The header of the main program shall include the following information at a minimum and be structured in the following format.

4.1.1 Complete Part Number, Rev Level, Rev Date, Brief description of program, Incoming blank or part number, person who wrote the most recent version of the program, and the date the last program version was written.

```
O8742;  
(PART # 38005742);  
(REV 006 20APR16);  
(OP 20; FACE, DRILL, AND CHAMFER STEM);  
(BLANK #38001033);  
(PROGRAMMER NAME);  
(PROGRAM DATE: 2AUG16);
```

- 4.2 Additional human-readable information may be included after the main header where required or requested.

5. Safe Start

- 5.1 Program shall automatically start at the beginning of the program regardless of where the cursor is or where it was stopped during the last cycle.
- 5.2 Provisions made so machine will not cycle start in the middle of a subroutine.
- 5.3 5.3 Programs for safe recovery from an interrupted cycle shall be provided.

6. Setting the Work Coordinate System

- 6.1 For all programmable axis, a written procedure shall be provided for home position setup, home position shall be marked with indicators pinned in position.
- 6.2 Labels shall be installed showing the coordinate system, and +/- direction shown (CNC)
- 6.3 The zero point for the work coordinate system shall be centerline of part for the A-B center line or "X" axis and should be the finished face of part for the Z-Axis per datum structure on part print. Z coordinates may be negative if needed.

7. Offset Linking and Checks

7.1 Wear offset limiting shall be used.

7.1.1 The incremental wear offset limit should be set to the total of the lowest tolerance being machined in this process or a maximum of 0.100 mm, whichever is.

7.1.2 The cumulative wear offset limit should be set to allow a maximum of 10 times the max incremental allowance in one direction before faulting the machine.

7.2 Wear offsets shall be linked so rough to finish depth of cut is maintained. If for some reason offsets are not linked, they shall have a measurable diameter and height feature on the finished part.

```
T0101  
G00 X[25.10+#2002] Z[20.00+#2102];  
G01 X[25.10+#2002] Z[0.00+#2102];  
T0202  
G00 X25.00 Z20.00;  
G01 X25.00 Z0.00;
```

```
#2001=#2002  
#2101=#2102
```

```
#614=#2104-#604(CHECK DIFF OF T04, Z-WEAR OFFSET)  
#2107=#2107+#614(CO-RELATE T04 and T07)
```

7.3 Tool offsets should be linked so that if that feature is shifted, all other related geometry will shift with it.

7.4 Geometry offsets shall be locked out from operator adjustment.

8. Macro Variable and Parametric Programming Organization and Description

8.1 All variables used in the program shall be listed at the beginning of the program under the program header. See following example:

```
O8742;  
(PART # 38005742);  
(REV 006 20APR16);  
(OP 20; FACE, DRILL, AND CHAMFER STEM);  
(BLANK #38001033);  
(PROGRAMMER NAME);  
(PROGRAM DATE: 2AUG16);
```

```
(#100: ROUGH PASS COUNTER);  
(#101: X CLEAR STARTING POSITION);  
(#102: Z CLEAR STARTING POSITION);
```

8.2 All variables and variable mathematics must include a description of what they are being used for.

```
#100 = 1 (INITIALIZE ROUGH PASS COUNTER);  
GO X[#101] Z[#102] (RAPID TO CLEAR STARTING POSITION);  
#100 = #100+1 (INCREMENT ROUGH PASS COUNTER);
```

8.3 All conditional branching and loops shall include a description of the process they are controlling
IF [#1 EQ 5] GOTO 100 (AFTER 5 ROUGH PASSES GOTO LINE N100);

9. M-Code Organization and Description

9.1 M-Code descriptions shall be included in program for machine specific processes when used.

```
M08 (COOLANT ON)  
M10 (CHUCK CLAMP)  
M137 (PART EJECT)  
M138 (PART EJECTOR RETRACTS)
```

10. Subprograms

10.1 Numbering convention for subprograms shall be:

```
6000 = All subroutines for non-machining functions; example warm-up program.  
7000 = All subroutines for machining functions  
8000 = All main part machining programs.  
9000 = All subroutines created by the machine manufacturer for machine specific operations.
```

10.2 All subprograms shall have a header including a description of the program, programmer name, and date of last revision. Additional information may be included but after the header.

```
O7001;  
(FACING MACRO PROGRAM);  
(PROGRAMMER NAME);  
(PROGRAM DATE: 14OCT16);
```

10.3 If using a Macro B subprogram call, description of the variables and routine testing of mandatory arguments shall be included in the subprogram. Below is an example of how this may be done but does not mean it shall be done in this manner.

```
G65 P7001 A135.0 D3.5 H76.3 (CALL FACING MACRO);
```

```
IN START OF PROGRAM O7001;
```

```
(A IS DATUM A ON PART PRINT)  
(D IS AMOUNT OF STOCK TO REMOVE FROM PART FACE);  
(H IS PART OD);
```

```
IF [#1 EQ #0] GOTO 100 (VERIFY VALUE IN A);  
IF[#7 EQ #0] GOTO100 (VERIFY VALUE IN D);  
IF[#11 EQ #0] GOTO100 (VERIFY VALUE IN H);  
.
```

M99;

N100 #3000 = 150 (DATA MISSING IN MACRO CALL);

10.4 Tool change programs shall position the turret in an ergonomically correct position for the operator. Each tool shall have its own tool change subprogram that will position the turret and tool in an ergonomically correct position. After the tool is changed the tool life counter shall automatically reset and wear offsets set back to zero.

10.5 The warm-up program shall be its own unique program. The operator or engineer shall not have to make any adjustments to the main program to use it as a warm-up program.

10.6 Tool Management option shall be utilized.

A. Common "G" Codes for CNC Mill Machine Controls

% - Start of Program

The character % must be inserted as first instruction of the Program. Normally it is written alone. In the same block is however accepted a comment . It forces in the CNC the following initial conditions:

- Axes moving in Rapid Traverse (G0) .
- Tool Radius Compensation reset (G40)
- Axes Feed = 100 mm/min (F100)
- Spindle programming at constant Speed (G94)
- Spindle rotation = 100 giri/min (S100)
- Reset of Length and Radius over stock (DRA = DLN=0)
- Reset Tool Length compensation (D=0)
- Precise Positioning and Machining mode (G60)
- Reset "Floating" mode (G30)
- Reset special interpolations G62, G67 and G69 (G68)
- Metric Programming (G71)
- Absolute Programming (G90)
- Deactivation off all Static and Dynamic Transformation Array (TCT/DCT :OFF)
- Reset of all Canned Cycles (G80)
- Reset of all Measurement Cycles (G200)
- Reset of all Macro cycles (G100-G150-G250)
- Reset of spindle speed limitation (G92)
- Reset of TRANSMIT transformation (G36)
- Reset of Working field limitations (G25/G26)
- Reset of Mirroring function (MIR:OFF)
- Activation of the contouring Plane associated to the first two configured axes (if, as usual, X and Y - G17)
- Activation of origin 1 for all configured axes (G54.01)
- Reset of all Rot-Translation eventually applied to the origins (G58/G59)

Notes: The above list reports the default initial conditions set by ECS. If necessary, all the start-up parameters, can be however, differently initialized by the Machine Builder.

B. Typical Tooling Commands

G798 - Loads / Modifies Geometrical Parameters of a Tool

The complete syntax of the command is the following:

```
<TPC=..> <TTC=..> <LUN=..> <RTA=..> <RAD=..> G798
```

Where

TPC = Tool Physical Code

TTC = Tool Logical Code (T)

LUN = Tool Length (in mm)

RTA = Cutting radius. Valid only in case of milling Tool . For cylindrical tool RTA = 0. For spherical tool RTA=RAD.

RAD = Tool Radius (in mm).

Example:

```
<TPC=3> <TTC=3> <LUN=100.000> <RAD=5.000> G798
```

It defines the tool T3 with Length = 100 mm, and Radius 5 mm.

G792 - Associates a comment to a Tool

The complete syntax of the command is the following:

```
<TPC=..> <TTC=..> (Comment) G792
```

Where

TPC = Tool Physical Code

TTC = Tool Logical Code (T)

Comment= the associated comment string (max 30 chrs)

Example:

```
<TPC=3> <TTC=3> (Spherical Tool with Radius 5 mm) G792
```

G799 - Loads Wear & Life parameters

The complete syntax of the command is the following:

```
<TPC=..> <TTC=..> <ATL=..> <WTL=..> <MXL=..> <MXR=..> <MXP=..> <MXU=..> G799
```

Where

TPC = Tool Physical Code

TTC = Tool Logical Code (T)

ATL = Tool Expected Life (in sec)

WTL = Life Warning Threshold (in sec) - Optional parameter

MXL = Max Length Wear (in mm)

MXR = Max Radius Wear (in mm)

MXP = Max Wear first time (in mm) – Optional parameter

MXU = Max one time Wear (in mm) – Optional parameter

Example:

<TPC=16> <TTC=16> <ATL=10000> <WTL=9900> <MXL=0.012> <MXR=0.011> <MXP=0.016>
<MXU=0.014> G799

Note:

The "Wear & Life Management Option" must be enabled.

RECORD OF REVISIONS

Revision No	Date	Section	Description
001	31JA17	ALL	Initial release.
002	31OC19	2,4,5,6,7,8,10	Update to reflect Safety Checklist, Revised offset limiting, and update wording to reflect SD 1041. Reformatted document to match Global Specifications.
003			
004			
005			
006			
007			
008			
009			
010			
011			
012			
013			
014			
015			
016			
017			
018			
019			
020			