CNC Mill Program
Structure Specification

Global Common

SD-1042

ISSUED 31JA17
REVISED 31JA17

TABLE OF CONTENTS

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:

1.) Main Program Number and Title

2.) Header

3.) Macro Variable List and Description

4.) Offset Program (If used)

5.) Machining Program

a. Basic instructions to build a Part Program:

2.2. Absolute programming shall be used through the program. If small incremental moves may be used but shall be denoted by U and W.

6.) Sub-Programs

a. Tool Setter

b. Tool Broken detection shall be used. (This System functionality shall considering a the minimum impact on the machine cycle time.)

c. Warm Up (short and Long Term)

d. Thermal Comp (If used)

e. Machine ID Program

f. Check for unload parts.

7.) Tool Management shall be utilized.

a. Tool life monitoring
   i. Target Tool Life
   ii. Actual Tool Life

b. Tool Wear Offset: Length and Diameter
c. Tool Geometry Offset.
d. Tool description shall be included at tool call per the following example:

T021 (PCD MILLING TOOL DIA 50)
(INNER SEAL SURFACE ABOVE -DATUM A-)

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

9.) Program Completion
   a. Cycle Time Monitoring
   b. Good Parts
   c. Rejects (if possible)

3. MAIN PROGRAM NAME & 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.

3.2. In the case where more than one part number can share the same program, this program shall be developed as a subprogram. Each part number shall have a unique Main Program that calls the subprogram when required.

3.3. If the last 3 digits of different part numbers are the same. The purchasing ME shall decide how to handle the naming convention depending on the situation.

3.4. PROGRAMS HEADER

3.4.1. Main Program. The header of the main program shall include the following information at a minimum and be structured in the following format.

Ex: O8312

3.4.1.1. Nexteer Manufacturing Location; Complete Part Number, Rev Level at time of vendor runoff, 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.

```
8004 (Main Program);
(================================================)
(NEXTEER PLT:     SAGINAW PLT 7)
(PART:            K2XX BALL NUT / 38003312;)
(DRAWING NO:      38005742, REV 006 20APR16;)
(PROGRAM NAME:    OP 50; PORT HOLES, BALL TRACK RETURN MILL;)
(VERSION:         2.0 / 02MAY2012)
(ENGINEER:        JOHN DOE;)
(================================================)
```

3.4.2. Other Programs. The header of the other programs shall include the following information at a minimum and be structured in the following format.

```
8104 (Machining Program)
(================================================)
(NEXTEER PLT:     SAGINAW PLT 7)
(PART:            K2XX BALL NUT / 38003312;)
(DRAWING NO;      38005742, REV 006 20APR16;)
(VERSION:         2.0 / 02AUG2016)
(ENGINEER:        JOHN DOE;)
(================================================)
```

3.4.3. Additional man readable information may be included after the main header where required or requested.

4. SAFE START

4.1. 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.

4.2. Provisions made so machine will not cycle start in the middle of a subroutine.

5. UNITS AND SETTING THE WORK COORDINATE SYSTEM

5.1. All programs shall be in metric units; with minimum of (3) decimal places.

5.2. 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.

5.3. G54 TO G59 - SELECT WORK OFFSET COORDINATE SYSTEM

5.3.1. Most machine startup with G54 selected. It's a good practice to put a G54 into the safety line at the top of all of your g-code programs to make sure you know

what work offset is being used unless you have reason to want to leave that aside.

```
(Sample Program G54EX19:)
(Workpiece Size: X8, Y5, Z2)
(Tool: Tool #6, 3/4" HSS Drill)
(Tool Start Position: X0, Y0, Z1)
(Workpiece Coordinate system 2: X1, Y1, Z0)
(Workpiece Coordinate system 3: X5, Y1, Z0)

N2 G90 G80 G40 G54 G20 G17 G50 G94 G64 (safety block)
N5 G90 G80 G20
N10 M06 T6 G43 H6
N15 M03 S1300
N20 G55 G00 X1.0 Y1.0 (Rapid to X1, Y1 of work coordinate
system 2)
N25 Z0.5
N30 G82 Z-0.25 R0.125 P1 F5
N35 Y2
N40 X2
N45 Y1
N50 X1.5 Y1.5
N60 G80 G00 Z1
N65 G56 G00 X1.0 Y1.0 (Rapid to X1, Y1 of work coordinate system 3)
N70 Z0.5
N75 G82 Z-0.25 R0.125 P1 F5
N80 Y2
N85 X2
N90 Y1
N95 X1.5 Y1.5
N100 G80 G00 Z1
N105 X0 Y0
N110 M05
N115 M30
```

6. OFFSET LINKING AND CHECKS

6.1. Max incremental wear offset of 0.050 mm.

6.2. Max cumulative wear offset of 0.500 mm.

6.3. 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 length and height feature on the finished part.

6.4. Tool offsets should be linked to part DATUMS structure so that if that feature is shifted, all other related geometry will shift with it.

6.5. Geometry offsets should be locked out from operator adjustment.

6.6. The Offset Program shall include the following information at a minimum and be structured in the following format:

```
O1904(Offset Program)
(=================================)
(  Customer:   NEXTEER            )
(         Part:   K2XX RACK HOUSING    )
(   Drawing:   28309904F          )
(    Version:   1.0 / 23JAN12       )
(  Engineer:   M.Hipp             )
(=================================)

M123(DISABLE TOOL COUNT)
G53
/M124(ENABLE TOOL COUNT)
#1100=1
G53

(=================================)
(LOAD WORK-OFFSET G54 - F / G DATUM - A-90, C180)
G90G10L2P1X-379.937Y-332.900Z-348.199A0C0
(=================================)
(LOAD WORK-OFFSET G55 - K DATUM - A-90, C0)
G90G10L2P2X-349.894Y-332.742Z-334.5935A0C0
(=================================)
(LOAD WORK-OFFSET G56 - A DATUM  - A-90 , C72 )
G90G10L2P3X-251.562Y-310.911Z-412.006A0C0
(=================================)
```

7. MACRO VARIABLE AND PARAMETRIC PROGRAMMING ORGANIZATION AND DESCRIPTION

7.1. All variables used in the program shall be listed at the beginning of the program under the program header.

```
8004 (Main Program);
=============================================
NEXTEER PLT:     SAGINAW PLT 7
PART:            K2XX BALL NUT / 38003312;
DRAWING NO:      38005742, REV 006 20APR16;
PROGRAM NAME: OP 50; PORT HOLES, BALL TRACK RETURN MILL;
```

```
VERSION:        2.0 / 02MAY2012
ENGINEER:       JOHN DOE;
================================================

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

7.2. All variables must include a description of what they are being used for.

```
N1
#3002=0(RESET TIMER)
(0-PRODUCTION 1-WARM-UP)
#702=1
G53
/#702=0
G53
M260(CLAMPING DEVICE ACTIVE?)
IF[#702EQ1]GOTO3
G53G0Z0G49
G53G0Y0
G53G0A-10C0
N1000M98P9746(CHECK FOR PART)
IF[#1031EQ1]GOTO9046
#3000=1(== REMOVE PART  ==)
N9046
G53
IF[#1006EQ1]GOTO2
#1100=0
N1300M98P5904(MACHINE ID)
G53
```

7.3. 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);
```

8. M-CODE ORGANIZATION AND DESCRIPTION

8.1. 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)

8.2. Writing comments can be helpful in keeping track of each line. Comments will also let you "comment-out" a line, so that the line is ignored.

Nxx - Block identification number

Block (Instruction) identification number. It is mandatory only in some instructions associated to the Flow Program control

(..) - Comment for machine Operator

This message will be displayed on the CNC display.

{..} or '.. - Comment for Programmer

The CNC manage this format as a not operative Instructions

Examples : { This is a Comment } or ' This is a Comment

9. SUBPROGRAMS

9.1. 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.

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

9.3. If using a Macro B subprogram call, description of the variables and routine testing of mandatory arguments shall be included in the subprogram

APPENDIX 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.

APPENDIX 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 #	Date	Section	Description
001	31JA17	ALL	Initial Release
002			
003			
004			
005			
006			
007			
008			
009			
010			
011			
012			
013			
014			
015			
016			
017			
018			
019			
020			