M- and G-Code Sample Files

The following sample files illustrate how M and G Codes are used to create a file. Click here for additional information on M and G Codes.

Sample File 1

The following file is for a single head system with manual tool change to complete a 5" square. Rapid level 0.5" above material, feed down 60 IPM, cut feed 200 IPM, rapid down to 0.1 above material, depth 0.25". The manual tool change is indicated with an asterisk (*) below and is optional.

M90 Program Start
G90 Absolute Coordinate Mode
G75 Follow the XY coordinate arc mode (Standard).
G97 S18000 Set Spindle Speed to 18,000 RPM.
G00 T1 Tool #1 call. Following moves will use Tool #1.
*G00 CE-MILL .2500 2FL. The keypad will display "E-MILL .2500 2FL." The user will now be able to load
the tool manually and Z reference the tool. This line can be sent multiple times in the same file to support manual
tool changes.
G00 Z-0.5 Position the Z-axis 0.5" above Z=0 or above the material.
G00 X0. Y0. Position X=0.0 and Y=0.0.
M12 The M12 command should be called before the Z-axis. This lowers the tool into the material or at the start of a
new contour.
- The Auxiliary output for the selected tool is turned on. (This output can be wired to operate a tool misting or
cooling unit.)
- The Motion will change from Slew mode (high speed rapid) to Machine mode (Cutting Mode).
- If this is the first M12 called in the program, the spindle output will come on for the current selected tool.
G00 Z-0.1 Position the Z-axis 0.1" above Z=0.0 or above the material.
G01 Z0.25 F1. Position the Z-axis 0.25" below Z=0.0 or into the material at the feedrate of 1"/sec (60"/min).
G01 X5. F3.333 Position the X-axis 5.0" at the feedrate of 3.333"/sec (200"/min).
G01 Y5. Position the Y-axis 5.0" (feedrate will continue at last set speed).
G01 X0. Position the X-axis 0.0".

G01 Y0. Position the Y-axis 0.0".
G00 Z-0.5 Position the Z-axis 0.5" above Z=0.0 or above the material.
M22 The M22 command should be called after the Z-axis lifts the tool out of the material and complete the current
contour.
- The Auxiliary output for the selected tool is turned off..
- The Motion will change from Machine mode (Cutting Mode) to Slew mode (high speed rapid).
G00 X0. Y0. Position X=0.0 and Y=0.0.
M02 End of job code. Spindle is turned off.

Sample File 2 (ATC or Single Tool)

This is the same file for an ATC system with the ATC tool change command. There is no "G00 C" command.

```
M90
G90
G75
G97 S18000
G00 T1
G00 Z-0.5
G00 X0. Y0.
M12
G00 Z-0.1
G01 Z0.25 F1.
G01 X5. F3.333
G01 Y5.
G01 X0.
G01 Y0.
G00 Z-0.5
M22
G00 X0. Y0.
M02
```

Sample File 3 (ATC Tool 3)

The following is a 5" circle clockwise, center at 2.5,2.5, 0.5 rapid level, feed down 60 IPM, cut feed 120 IPM, rapid down to 0.1, depth of 0.4". ATC system uses Tool #3.

```
M90
G90
G75
G97 S18000
G00 T3
G00 Z-0.5
G00 X2.5 Y5.
M12
G00 Z-0.1
G01 Z0.4 F1.
G02 I2.5 J2.5 F2.
G00 Z-0.5
M22
G00 X0. Y0.
M02
```

Sample File 4 (Sample Files 2 and 3 together)

```
M90
G90
G75
G97 S18000
G00 T1
G00 Z-0.5
G00 X0. Y0.
M12
G00 Z-0.1
G01 Z0.25 F1.
G01 X5. F3.333
G01 Y5.
G01 X0.
G01 Y0.
G00 Z-0.5
M22
G97 S18000
G00 T3
G00 Z-0.5
```

G00 X2.5 Y5.
M12
G00 Z-0.1
G01 Z0.4 F1.
G02 I2.5 J2.5 F2.
G00 Z-0.5
M22
G00 X0. Y0.
M02

Sample Boring File

M90
G90
G75
G00 X6.0394 Y6.0394
M38 Lowers the drill bank into place and turns on the drill motor.
G00 X6.0394 Y6.0394 Z-0.5 Moves X and Y so that boring drill 1 is centered at X = 6.0394 Y = 6.0394 and
Z is 0.5" above the material surface.
G98 P300 D496 Engages drills 5, 6, 7, 8, and 9.
G00 Z-0.1 Slew move in Z to 0.1" above the material.
G01 Z0.4 F1.333 Plunge the Z axis 0.4" into the material at 80 IPM.
G00 Z-0.5 Slew move in Z to 0.5" above the material.
G00 X6.0394 Y8.5591 Move X and Y and continue to drill more holes with the same engaged drills.
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X6.0394 Y11.0787
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X6.0394 Y12.3386
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X6.0394 Y13.5984
G00 Z-0.1
G01 Z0.4 F1.333

G00 Z-0.5
G00 X6.0394 Y14.8583 Z-0.5
G98 P300 D480 Engages drills 6, 7, 8, and 9.
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X6.0394 Y16.1181 Z-0.5
G98 P300 D448 Engages drills 7, 8, and 9.
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X6.0394 Y17.378 Z-0.5
G98 P300 D384 Engages drills 8 and 9.
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X6.0394 Y18.6378 Z-0.5
G98 P300 D256 Engages drill 9.
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X12.3386 Y12.3386 Z-0.5
G98 P300 D31 Engages drills 1, 2, 3, 4, and 5.
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X12.3386 Y6.0394
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X13.5984 Y6.0394
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X14.8583 Y6.0394
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X16.1181 Y6.0394
G00 Z-0.1
G01 Z0.4 F1.333

G00 Z-0.5
G00 X17.378 Y6.0394
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
G00 X18.6378 Y6.0394
G00 Z-0.1
G01 Z0.4 F1.333
G00 Z-0.5
M48 Retract all boring drills, retract drill bank, and turn off drill bank.
G97 S24000
G00 T3
G00 X18. Y12.875 Z-0.5
M12
G00 Z-0.1
G01 Z0.5 F1.667
G02 I18. J12. F3.333
G00 Z-0.5
M22
G00 X22. Y8.375 Z-0.5
M12
G00 Z-0.1
G01 Z0.5 F1.667
G02 I22. J7. F3.333
G00 Z-0.5
M22
G97 S18000
G00 T1
G00 X0. Y-0.25 Z-0.5
M12
G00 Z-0.1
G01 Z0.77 F1.667
G01 X25. F13.333
G03 X25.25 Y0. I25. J0.
G01 Y15.
G03 X25. Y15.25 I25. J15.
G01 X0.
G03 X-0.25 Y15. I0. J15.
G01 Y0.
G03 X0. Y-0.25 I0. J0.
G00 Z-0.5

M22
G00 X35.0 Y0.0
M02

Cutter Comp G40\G41\G42 File

Before any cutter compensation (G41\G42) will work, the code for the cut **must** start with a lead-in and end with a lead-out. The type of compensation that is applied depends on the direction selected. If the cut is going clockwise in the inside of a pocket, then the machine should be compensated to the **right** (G42); if it is going clockwise on the outside of a contour, then it should be compensated to the **left** (G41), etc.

N10 M90 Program Start
N20 G90 Absolute Coordinate System
N40 G75 Use absolute mode (G90 from above) for arcs.
N50 G97 S24000 Set spindle speed to 24,000 RPM.
N60 G00 T1 Select Tool #1.
N70 G00 Z-0.5 Rapid move to safe rapid level.
N80 G00 X-0.25 Y-0.25 Rapid move
N90 M12 Device on code.
N100 G00 Z-0.1 Rapid move
N110 G01 Z0.5 F1. Feed move to final depth with feedrate 1"/sec.
N120 G42 Set Cutter Comp Right.
N130 G02 X-0.125 Y-0.125 I-0.125 J-0.25 F3.333 Lead-in arc
N140 G01 X10.125 Feed move
N150 G01 Y10.125 Feed move
N160 G01 X-0.125 Feed move
N170 G01 Y-0.125 Feed move
N180 G02 X-0.25 Y-0.25 I-0.25 J-0.125 Lead-out arc
N190 G40 Set Cutter Comp Off.
N200 G00 Z-0.5 Retract to safe rapid level.
N210 M22 Device off code.
N220 G00 X0. Y0. Move to Home position.
N230 M02 End of job code

N10 M90 Program Start
N20 G90 Absolute Coordinate System
N30 G75 Use absolute mode (G90 from above) for arcs.
N40 G97 S18000 Set spindle speed to 18,000 RPM.

N50 G00 T2 Select Tool #2.
N60 G00 Z-0.5 Rapid move to safe Z level.
N70 G00 X5.125 Y10.25 Rapid move
N80 M12 Device on code
N90 G00 Z-0.1 Rapid move
N100 G01 Z0.5 F1. Feed move to final depth with feedrate 1"/sec.
N110 G42 Set Cutter Comp Right on.
N120 G02 X5. Y10. I5. J10.25 F3.333 Lead-in arc
N130 G03 I5. J5. Feed move circle
N140 G02 X4.75 Y10.25 I5. J10.25 Lead-out arc
N150 G40 Set Cutter Comp off.
N160 G00 Z-0.5 Retract to safe rapid level.
N170 M22 Device off code
N180 G00 X0. Y0. Move to Home position.
N190 M02 End of job code

N10 M90 Program Start
N20 G90 Absolute Coordinate System
N30 G75 Use absolute mode (G90 from above) for arcs.
N40 G97 S24000 Set spindle speed to 24,000 RPM.
N50 G00 T2 Select Tool #2.
N60 G00 Z-0.5 Rapid move to safe Z level.
N70 G00 X5.125 Y7.75 Rapid move
N80 M12 Device on code
N90 G00 Z-0.1 Rapid move
N100 G01 Z0.5 F1. Feed move to final depth with feedrate 1"/sec
N110 G41 Set Cutter Comp Left on.
N120 G03 X5. Y7.875 I5. J7.75 F6.667 Lead-in arc
N130 G03 I5. J5. Feed move
N140 G03 X4.875 Y7.75 I5. J7.75 Lead-out arc
N150 G40 Set Cutter Comp off.
N160 G00 Z-0.5 Retract to safe rapid level.
N170 M22 Device off code
N180 G97 S18000 Set spindle speed to 18,000 RPM.
N190 G00 T1 Select Tool #1.
N200 G00 Z-0.5 Rapid move to safe Z level.
N210 G00 X-0.375 Y-0.125 Rapid move
N220 M12 Device on code
N230 G00 Z-0.1 Rapid move
N240 G01 Z0.5 F1. Feed move to final depth with feedrate 1"/sec.
N250 G42 Set Cutter Comp Right on.

N260 G01 X0. F3.333 Lead-in line feedrate 200"/min.
N270 G01 X10. Feed move line
N280 G03 X10.125 Y0. I10. J0. Feed move arc
N290 G01 Y10. Feed move line
N300 G03 X10. Y10.125 I10. J10. Feed move arc
N310 G01 X0. Feed move line
N320 G03 X-0.125 Y10. I0. J10. Feed move arc
N330 G01 Y0. Feed move line
N340 G03 X0. Y-0.125 I0. J0. Feed move arc
N350 G02 X0.125 Y-0.25 I0. J-0.25 Lead-out arc
N360 G40 Set Cutter Comp off.
N370 G00 Z-0.5 Rapid move to safe Z level.
N380 M22 Device off code
N390 G00 X0. Y0. Move to Home position.
N400 M02 End of job code

G-Code File

Drilling, Peck drilling, cutter comp, no cutter comp

N10 M90 Program Start
N20 G90 Absolute Coordinate System
N30 G75 Use absolute coordinates for arc.
N40 G97 S10000 Set spindle speed to 10,000 RPM.
N50 G00 T1 Select Tool #1.
N60 G00 Z-0.5 Rapid Z move
N70 G00 X1. Y1. Rapid XY move
N80 M12 Device on code
N90 G83 R0.1 Z0.75 D0.2 F1.667 Peck drill
N100 G00 Z-0.5 Rapid Z move
N110 M22 Device off code
N120 G00 X2. Y1. Rapid XY move
N130 M12 Device on code
N140 G83 R0.1 Z0.75 D0.2 F1.667 Peck drill
N150 G00 Z-0.5 Rapid Z move
N160 M22 Device off code
N170 G00 X3. Y1. Rapid XY move
N180 M12 Device on code
N190 G83 R0.1 Z0.75 D0.2 F1.667 Peck drill
N200 G00 Z-0.5 Rapid Z move

N210 M22 Device off code
N220 G00 X4. Y1. Rapid XY move
N230 M12 Device on code
N240 G83 R0.1 Z0.75 D0.2 F1.667 Peck drill
N250 G00 Z-0.5 Rapid Z move
N260 M22 Device off code
N270 G00 X5. Y1. Rapid XY move
N280 M12 Device on code
N290 G83 R0.1 Z0.75 D0.2 F1.667 Peck drill
N300 G00 Z-0.5 Rapid Z move
N310 M22 Device off code
N320 G97 S12000 Set spindle speed to 12,000 RPM.
N330 G00 T11 Select Tool #11 (a pneumatic drill).
N340 G00 Z-0.5 Rapid Z move
N350 G00 X1. Y4. Rapid XY move
N360 M11 Fire drill
N370 M21 Retract drill
N380 G00 X2. Y4. Rapid XY move
N390 M11 Fire drill
N400 M21 Retract drill
N410 G00 X3. Y4. Rapid XY move
N420 M11 Fire drill
N430 M21 Retract drill
N440 G00 X4. Y4. Rapid XY move
N450 M11 Fire drill
N460 M21 Retract drill
N470 G00 X5. Y4. Rapid XY move
N480 M11 Fire drill
N490 M21 Retract drill
N500 G97 S18000 Set spindle speed to 18,000 RPM.
N510 G00 T2 Select Tool #2.
N520 G00 Z-0.5 Rapid Z move
N530 G00 X-0.5 Y3. Rapid XY move
N540 M12 Device on code
N550 G00 Z-0.1 Rapid Z move
N560 G01 Z0.375 F1. Feed Z move
N570 G01 X0. F6.667 Linear feed XY move
N580 G01 X12. Linear feed XY move
N590 G01 X12.5 Linear feed XY move
N600 G00 Z-0.5 Rapid Z move
N610 M22 Device off code

N620 G97 S24000 Set spindle speed to 24,000 RPM.
N630 G00 T5 Select Tool #5.
N640 G00 Z-0.5 Rapid Z move
N650 G00 X-0.563 Y-0.187 Rapid XY move
N660 M12 Device on code
N670 G00 Z-0.1 Rapid Z move
N680 G01 Z0.75 F1. Feed down Z move.
N690 G42 Set Cutter Comp Right on.
N700 G01 X0. Y-0.188 F8.333 Linear feed XY move
N710 G01 X12. Linear feed XY move
N720 G03 X12.188 Y0. I12. J0. Arc feed XY move
N730 G01 Y6. Linear feed XY move
N740 G03 X12. Y6.188 I12. J6. Arc feed XY move
N750 G01 X0. Linear feed XY move
N760 G03 X-0.188 Y6. I0. J6. Arc feed XY move
N770 G01 Y0. Linear feed XY move
N780 G03 X0. Y-0.188 I0. J0. Arc feed XY move
N790 G02 X0.188 Y-0.375 I0. J-0.375 Lead-out arc
N800 G40 Set Cutter Comp off.
N810 G00 Z-0.5 Rapid Z move
N820 M22 Device off code
N830 G00 X0. Y0. Rapid XY move
N840 M02 End of job code

then here is some more info

G Codes and M Codes with Software Ports

All lines of machine code must start with either a G Code or an M Code, and software ports can be used to define the operation. The timeline for additions and updates to the codes and software ports is included, and the following tables provide additional information on each type of code and on software ports. Multiple G or M Codes cannot be put on the same line.

Timeline for Additions and Updates

The following timeline shows what changes were made to the G and M Codes and to the software ports since 2003.

Date

Additions or Updates

02 Nov 2010

Update port 159

03 Jun 2010

Add port 111

01 Jun 2010

Add M17 information

10 Jun 2009

Added port 183

11 Jan 2008

Added Material Handling

31 Jul 2007

Added font ports

23 Mar 2007

Added ports in 200 range

01 Mar 2007

Clarified Park commands, when they were added

18 Dec 2006

Added M50

27 Jul 2006

Updated Port 137 operation

09 Feb 2006

Added ports 126 and 127

11 Jan 2006

Add G84 commands (converted to atap_cycle)

29 Jul 2005

Added C-axis ports

28 Sep 2004

Added M18 and M19

01 Jul 2004

Allow G83 and G81to be modal

12 May 2004

Added in Plasma Library ports

21 Oct 2003

Fix G81 pecking more than once if retract height < 0

19 Aug 2003

Add G81 and allowed G83 to have X and Y

27 May 2003

Added M95, M96, M97, M98, and VP(133,142)

13 Jan 2003

Added Homing and Set Surface Virtual Ports

G Codes

Operators creating G-Code files from their post processor must end the file with a .cnc or .anc file extension so that the DNC program can recognize the file. The following table lists the supported G Codes for the MultiCam controller. Parameters within brackets are optional. The fields represented by “d.d” may be any decimal number, and fields represented by “d” may be any positive integer.

Code

Description

Notes (if applicable)

G00

[Xd.d] [Yd.d] [Zd.d] [Fd.d] [Td] [Ctext string]

High speed move (slew)

G01

[Xd.d] [Yd.d] [Zd.d] [Fd.d]

Linear move (machine)

G02

[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]

CW 2D circular move

G03

[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]

CCW 2D circular move

G04

Fd.d

Dwell (seconds)

G17

Specify XY Plane for Helical

G18

Specify ZX Plane for Helical

G19

Specify YZ Plane for Helical

G37

Find Home

G40

Cancel Tool Compensation

G41

Left Tool Compensation

G42

Right Tool Compensation

G62

Clear Soft Home

G70

English Programming (inches)

G71

Metric Programming (millimeters)

G72

[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]

CW 3D circular move

G73

[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]

CCW 3D circular move

G74

Incremental Mode for G02/03 Arcs

G75

G90/91 Mode for G02/03 Arcs

G81

[Xdd.d] [Ydd.d] [Rdd.d] [Zdd.d] [Fdd.d]

One-stroke drill cycle

G83

[Xdd.d] [Ydd.d] [Rdd.d] [Zdd.d] [Ddd.d] [Fdd.d]

Peck drill cycle with router

G84

[Xdd.d] [Ydd.d] [Zdd.d] [Rdd.d] [Fdd.d]

Tap cycle

G90

Absolute Coordinate Mode

G91

Incremental Coordinate Mode

G92

[Xd.d] [Yd.d] [Zd.d]

Set Soft Home

G97

Sd

Set spindle speed (RPM)

G98

Plasma Systems Only 12. P133 D0

1. P145 Dd 23. P300 Dd

By default, plasma systems ignore feedrates sent in the job file. 1. Feedrates will be used from now on. 2. Feedrates will be ignored from now on. 1. Go to pre-recorded Home position (e.g., D1=Home 1, D2=Home 2) 2. Park X-axis (e.g., D0=X Min, D1=X Max) 3. Boring unit drill select

Innovation. Quality. Performance. Best in Industry. Worldwide. 3

Routers - Lasers - Plasmas - Water Jets - Knife Cutters

www.multicam.com 16 August 2011

The following table lists the letters used to denote various arguments in Etc CNC version 1.0.

Argument

Description

Location of Use (if applicable)

C

Tool Change Operator message

G00

D

Peck Drill Data

G83, Data selection in G98

F

Feedrate in Units per Second

G00, G01, G02, G03, G72, G73, G83

G

Preparatory Function

I

Circular Interpolation Value in X Dimension

G02, G03, G72, G73

J

Circular Interpolation Value in Y Dimension

G02, G03, G72, G73

K

Circular Interpolation Value in Z Dimension

G02, G03, G72, G73

M

Miscellaneous or Control Function

N

Sequence Number

R

Beginning Z Motion Dimension

G83

S

Spindle RPM

G97

T

Tool Change

G00

X

X Motion Dimension

Y

Y Motion Dimension

Z

Z Motion Dimension

Innovation. Quality. Performance. Best in Industry. Worldwide. 4

Routers - Lasers - Plasmas - Water Jets - Knife Cutters

www.multicam.com 16 August 2011

M Codes

The following table lists the available M Codes and how they should be configured for JobServer. Operators can use Device 199 Inactive to ignore M Code.

Code

Description

Device #

State*

Graphics

Notes

M00

Program Pause

-97

I

n/a

0 prg_pause (needed for Suite4)

M01

Optional Program Pause

-96

I

n/a

1 prg_pause (needed for Suite4)

M02

End of Job

-98

I

n/a

end_plot (needed for Suite4)

M03

Start Spindle Clockwise

113

A

n/a

Spindle ON clockwise

M04

Start Spindle Counter-clockwise

114

A

n/a

Spindle ON counter-clockwise

M05

Spindle OFF

105

A

n/a

spindle_off

M11

2D Device ON

-1 or 101

A

ON

-1= current tool number is passed 101= current tool number is selected by Init file

M12

3D Device ON

-1 or 102

A

ON

-1= current tool number is passed 101= current tool number is selected by Init file

M13

2D Device ON, no Z down

113

A

ON

-1= current tool number is passed 101= current tool number is selected by Init file **M13 does not lower the pneumatic Z and is used in Plasma only.**

M17

Turn on Z Tracking

118

A

OFF

Turns on Z Tracking for the current contour in Plasma only

M18

Turn off Z Tracking

118

I

OFF

Turns off Z Tracking for the current contour in Plasma only

M19

Disable Arc Out Pause

119

I

OFF

Disables the Pause feature when the arc goes out and becomes enabled at the next contour (i.e., M11 or M12) in Plasma only

M21

2D Device OFF

-1 or 101

I

OFF

-1= current tool number is passed 101= current tool number is selected by Init file

M22

3D Device OFF

-1 or 101

I

OFF

-1= current tool number is passed 101= current tool number is selected by Init file

M23

2D Device OFF, no Z up

123

I

OFF

-1= current tool number is passed 101= current tool number is selected by Init file **M23 does not lower the pneumatic Z and is used in Plasma only.**

M25

Start of Sheet

-99

A

OFF

Starts sheet and is available as nsheet in H4LDR version 4.50 and later

M30

Fire Enabled Drill

130

A

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #130 (Active) to 66 PD

M31

Drill 1 ON (Enable & Offset)

131

A

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56 and above convert device #131 to 66 PD tool_change

M32

Drill 2 ON (Enable & Offset)

132

A

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #132 to 67 tool_change

M38

Gang Drill 1 ON

138

A

OFF

M41

Drill 1 OFF (Disable)

131

I

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #131 (Inactive) to 66 PU

M42

Drill 2 OFF (Disable)

132

I

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #131 (Inactive) to 67 PU

M48

Gang Drill 1 OFF

138

I

OFF

Turns off and raises gang drill 1; resets surface back to surface for current tool. For JobConsole, change “Post” tab under CNC settings. Under XMI Settings, do the following:

1.

Select CNC tab

2.

Select Post tab

3.

Add tool number -138

4.

Add tool description as Gang Drill

5.

Add Tool Change M38

6.

Add Tool Unload M48

7.

Leave Tool Activation blank

8.

Leave Tool Deactivation blank

M50

Material Handler (Panel Pusher)

150

I

OFF

Requires RIO_pusher.uc module. Starts and ends a material unload process with the following:

1.

Moves to X start.

2.

Lowers pusher pins.

3.

Moves to X stop.

4.

Raises pusher pins.

M60

Put Away Tool

104

A

OFF

Available in H4LDR version 4.55 and later **M60 is only available on ATC machines.**

M90

Program Start

n/a

n/a

n/a

Opens start_plot or cycle_start **For Suite4 set device number to -90.**

M91

Program Replay

n/a

n/a

n/a

Not supported

M92

ALL Mode

192

A

OFF

Available in H4LDR version 4.58 and later **M92 is only available on standard machines.**

M93

Return to Auto Mode

192

I

OFF

Available in H4LDR version 4.58 and later **M93 is only available on standard machines.**

M94

Disable Spindle Offset

194

A

n/a

Available in H4LDR version 4.71 and later **M94 makes the spindle offset between heads 0,0 and is only available on standard machines.**

M95

Enable Marking Mode

195

A

OFF

Plasma only

M96

Disable Marking Mode

195

I

OFF

Plasma only

M97

Double Velocity

197

I

OFF

Sets for faster lead-outs in Plasma only

M98

Turn off Z Tracking, Disable Arc Out Pause, then Turn off Plasma Arc

198

I

OFF

Disables the arc before the end of the contour in Plasma only

M99

Exit CNC Interpreter

n/a

n/a

n/a

M150

Material Handling Pusher Cycle

150

A

n/a

Runs a complete pusher cycle

M151

Material Handling Lifter Cycle

151

A

n/a

Runs a complete lifter cycle

M152

Material Handling Dust Collector Blast Gate ON

152

A

n/a

Turns ON the dust collector blast gate

M153

Material Handling Dust Collector Blast Gate OFF

153

A

n/a

Turns OFF the dust collector blast gate

M154

Material Handling Sweeper Blast Gate ON

154

A

n/a

Turns ON the sweeper blast gate

M155

Material Handling Sweeper Blast Gate OFF

155

A

n/a

Turns OFF the sweeper blast gate

M156

Material Handling Air Knife ON

156

A

n/a

Turns ON the air knife

M157

Material Handling Air Knife OFF

157

A

n/a

Turns OFF the air knife

* A = Active, I = Inactive

M Codes can directly control M24 devices when used to configure the M-Code table for JobServer. Devices 200-299 provide direct control while devices 300-399 provide exclusive device control that turns other devices off when a particular device has been turned on.

Device

Device Output

M24 Header / Location

200

Spindle 1 Output

H2: 1&2

201

Mister 1 Output

H2: 3&4

202

Spindle 2 Output

H2: 5&6

203

Mister 2 Output

H2: 7&8

204

Spindle 3 Output for M24 revision 2, 3, and 4 Spindle Enable for M24 Revision 5

H2: 9&10

205

Mister 3 Output for M24 revision 2, 3, and 4 Drill Enable for M24 revision 5

H2: 11&12

206

Drill 1 Output

H3: 1&2

207

Drill 2 Output

H3: 3&4

208

Caution Output

H3: 5&6

209

TC Chuck Output

H4: 1&2

210

TC Blast

H4: 3&4

211

Dust Collector

H4: 5&6

212

Misc 1 ** DO NOT USE **

H4: 7&8

213

Misc 2 ** DO NOT USE **

H4: 9&10

214

Misc 3 ** DO NOT USE **

H4: 11&12

250 – 269

General M-Code Mapping to Devices Mapped using MCode_Device_Map file.

Software Ports

Software ports are used to expand either HPGL or CNC language. Operators can use a software port in the job file with either a G98 or Z0 command as shown below:

CNC Job G98 P147 D1 will park the machine at X Max.

HPGL Job Z0147, 1 will park the machine at X Max.

Virtual ports 1-49 are reserved for setting physical ports while virtual ports 50-99 are reserved for clearing physical ports. The following 2 strings are defined in relation to their associated port, location or data, and description while the third and fourth strings are more complex.

G98 P<n> X<x.x> Y<y.y> Z<z.z> S<string>

G98 P<n> [X<x.x>] [Y<y.y>] [Z<z.z>] [S<string> --- Xlated to --- <n> info_string <string> The XYZ are only used for JobPreviewer.

Port

XYZS

Description

1020

Any

Specifies the bounding box of sheet. XYZ are read in Xlate version 3.85 and sent to JobPreviewer but not the controller; S is read in Xlate version 3.9.24 and converted to <n> info_string.

1040

Tool Prompt

G98 P<n> S<string>

G98 P<n> [S<string> --- Xlated to --- <n> info_string <string>

Port

String

Description

140

Any

Displays the string but does not wait for the operator. Requires XLate v3.85 or later.

141

Any

Displays the string and waits for operator response. Requires Xlate v3.85 or later.

G98 P<p> D<d> [E<e.e>]

G98 P<p> D<d> --- Xlated to --- <d> <p> set_port G98 P<p> D<d> E<e.e> --- Xlated to --- <d> <e.e> <p> set_port_ex

Port

Data

Description

100

Spindle Speed in RPM

Sets spindle RPM. Mainly used for HPGL jobs. In CNC jobs, operator should use G97 S<spindle speed>.

101

Marker Identifier

1=character 2=word 3=line 4=part 5=start of part 6=end of part

110

Max Z Increment for Multipass

Specifies in 1/1000th

Port 110 is not applicable to M2521.

111

Feedrate Override

0=disable, set to 100%

This command is immediate and will change the feedrate override to the percent specified (i.e., 0% to 100%).

120

Percentage of Laser Power

Sets laser power

121

Absolute Index

Move to abs. Position, int value *HPGL Resolution

122

Solenoid ON/OFF

0=off 1=on

123

Auto Spindle Enable (BOJ, EOJ)

0=disable auto spindle 1=enable auto spindle 2=enable manual spindle control using M03, M04, M05

125

Z Probe Disable/Enable

0=disable probing 1=enable probing on PD

Port 125 is reset at the start of each job (added to ZSurf_mod.uc v2.22).

126

Tapping Mode

0=off The next tool change will be a tapping tool to be used with G84 commands. 1=on

Tapping mode gets converted into µCito job file command atap_cycle and requires either JobConsole 4.0.10 or later or PSS Xlate.dll v3.9.27.

127

Linear Encoder Mode

0=off 1=on

Completes a second adjustment move at the end of each AC or U based on readings from linear encoders.

128

Z Probe Location (relative to current surface)

Specifies in 1/1000th; indicates the expected location of the top of the material when using the Z surface probe

Port 128 is reset to 0.0 at the start of each job (ZSurf_Mod.uc).

129

Touch Off Radius

Specifies in 1/1000th; uses the radius during Z probe use to determine if the probe should touch off again

Port 129 is reset back to the keypad settings at the start of each job (ZSurf_Mod.uc).

130

Spindle Control

0=spindle off 1=spindle on

131

Job Type

1=start of vector job 2=start of raster job

132

Data Ignored

Sets Home at the current position

133

Use Feedrates in Job

0=ignore feedrates in the job 1=use feedrates in the job

Port 133 is only applicable in Plasma v1.44 and is not available to routers.

134

Use Z Values in Job

0=ignore Z values 1=use Z values

Port 134 is only available in Water Jets (v3.01.10).

135

Spindle Reverse Direction

1=spindle 1 reverse (CCW) 2=spindle 2 reverse (CCW)

Port 135 orients the bit to reverse spindle and is available in version 8.02.24.

136

Mister Configuration

0=disable mister 1=re-enable mister

If Port 136 is set to 0, then the mister will not turn on for an M11 or M12. The mister is ON by default. At the start of every job, the mister is re-enabled unless the keypad option MisterOpt (added in v8.09.05) indicates the mister is disabled.

137

Manual Dust Collector Control

0=raise dust collector (v8.02.27) 1=lower dust collector 2=disable dust collector (v8.02.60)

Use G98 P137 D0 to raise and re-enable dust collector operation.

138

Low RPM Mode for Motor #2

0=disable low RPM mode 1=enable low RPM mode for Motor #2

Used for slow speed drilling. Port 138 requires a special F7 Inverter programming that will switch to Motor #2 in the inverter (added v8.02.32).

140

Drill Hole

D=<mode> E=<Hole Diameter>

Hole is drilled at current location.

Innovation. Quality. Performance. Best in Industry. Worldwide. 11

Routers - Lasers - Plasmas - Water Jets - Knife Cutters

www.multicam.com 16 August 2011

142

Host Jog

Starts host Jog and ignores data

143

Set Surface

0=auto (added in v.6.10) 1=current position

144

Find Limits

Ignores data (added in v6.10) Port 144 is the same as using the keypad to find limits, but Port 144 leaves the machine in motion to allow the job file commands to follow.

145

Fixture Number

Sets Home at the fixture location

146

Park Z Location

Ignores data but always parks Z to Z0.

147

Park Command

0=X Home 1=X Max 2=X Home, Y Home (v8.02.46)

Port 147 parks Z first and then parks X (e.g., G98 P147 D1 {parks machine at X Max}).

150

Z Lift

Sets the Z lift in 1/1000th (v3.12) in Plasma only. See Port 271.

151

Contour Acceleration

D value = axis bit value (1=X, 2=Y, 3=Z) E value = acceleration in user units (e.g., G98 P151 D3 E1.0 sets acceleration of XY to 1.0 for the following contours)

Port 151 command must precede contour and is used on WaterJets ONLY.

152

Pierce Type

D=<type> E=<exit angle> (value -999 means unknown)

<type> 0=standard, 1=dynamic, 2=drill

Operators should input the final angle relative to the start point to finish the pierce for dynamic pierce methods. This sets the desired direction for the next move.

153

Set Dual Head Control (dual heads, not dual gantry axes)

D=<enable mask> Ha=Head a (either Xa or Ya) Hb= Head b (either Xb or Yb)

D=1 activate Ha, park Hb D=2 activate Hb, park Ha D=3 activate both heads with offset specified

E=<Yb offset> Hb=Y offset + Ha

154

Set Dual-head Mirror

D=<enable> 0 to disable, 1 to enable E=<mirror Pt> Hb=Hmirror - Ha

155

Contour Deceleration

D=<mask> E=<deceleration value> for those axes

D value = axis bit value (1=X, 2=Y, 3=Z) E value = deceleration in user units

156

Torch Height Voltage

Sets the torch height voltage in Plasma only

157

User Rate (1 – 10)

1=slowest 10=fastest

Port 157 determines the user feedrate/acceleration factor in Plasma only.

158

Max Volt Gap

Sets the maximum voltage gap in 1000 (v3.26.02) in plasma only

159

Bevel Angle and Mode

D=<mode> -1=turn off bevel mode 0=turn on bevel mode

E=<angle> Bevel angle in degrees (a positive angle will angle outward)

Examples – G98 P159 E3 D1 = set bevel angle for following contour to 3° G98 P159 E0 D-1 = turn off bevel mode

160

Material Handling Suction Cup Pod Vacuum

D<bitmask> = suction cup to activate (e.g., G98 P160 D1 turns on suction cup #1; G98 P160 D0 turns off suction cups)

Port 160 calls DCN #324 and was added to Router Inits in version 8.06.10.

161

Material Handling Suction Cup Pod Blower

D<bitmask> = blower to activate

Port 161 calls DCN #325 and was added to Router Inits in version 8.06.10.

162

Material Handling Sheet Size

D<Sheet Size> (value in 1/1000th) (e.g., G98 P162 D96000 {sets the sheet size to 96})

Available if the Material_Handling.uc module is loaded

163

Material Handling Start of Sheet

D<Start Location> (value in 1000th) (e.g., G98 P163 D12000 {sets the start of sheet location to 12})

Available if the Material_Handling.uc module is loaded.

164

Conveyor Index

D<Mode> E<X Distance> E value is required in all cases

<mode> 0=end of panel 1=E value specifies center of panel 5=index panel (E value ignored) 10=set grab position (reset at EOJ) 11=set release position (reset at EOJ)

180

Dual Y Configuration (deprecate)

1=Ya 2=Yb 3=Both 4=Both Mirrored

Port 180 is available in WaterJet v3.01.21 and Router v8.03 (e.g., G98 P180 D3 {sets Dual Y to Both mode}).

181

Dual Y Yb Offset (deprecate)

Yb Offset value in 1000th for Water Jet v3.01.21 and Router v8.03 (e.g., G98 P181 D20000 {sets Yb offset to 20})

183

Chip Break Value

D value not used and does not need to be included.

E=relative lift height for chip breaking and is currently only implemented for the Big Drill machine. Port 183 is called before a G83 command to indicate it should only lift by the Chip Break value instead of retract height between pecks. If the chip break value is ≤ 0.0, then the chip break feature will be ignored (e.g., G98 P183 E0.1 sets the chip break value to 0.1).

** Port 183 value will be used for all following G83 commands and so does not need to be specified for each G83 command. The value is set back to 0.0 at the start of each job.**

184

Y Brake Control

D=0 turns OFF Y brake D=1 turns ON Y brake

Y Brake is controlled by DCN 152 and is currently only implemented for the Max-40 machine.

Ports 200 – 251 are used to set job values.

200

Cut Feedrate

Machine Feedrate converted to <E value> SDF

201

Slew Feedrate

Slew Feedrate

202

Z Up Feedrate

203

Z Down Feedrate

Value is in MM/sec based on the tool. If the knife tool is currently selected, then this will affect the Knife Plunge Feedrate (e.g., G98 P203 E25.4 sets the Z down feedrate to 25.4mm {1.0”} per second).

210

Pen Down Delay

211

Pen Up Delay

220

Pen Down Height

Equivalent to SPD in µCito and ZD in HPGL

221

Pen Up Height

Equivalent to SPU in µCito and ZU in HPGL

231

Contour Group ID

240

Material Library Information

D=1 S=Material Name D=2 S=Process Name

Coding used G98 P240 D1 SMS │0.375 (3/8)” G98 P240 D2 S60A │ SHLD Will get translated into 1 240 set_port 240 info_string MS │ 0.375 (3/8)” 2 240 set_port 240 info_string 60A │ SHLD

241

Material Library Cut Parameters

D=<Parameter ID> E=<Parameter value>

250

Set Raster Scan Line Start

Set the zero-based index (inclusive) of the first scan line in the following PICT

251

Set Raster Scan Line End

Set the zero-based index (inclusive) of the last scan line in the following PICT

Ports 252-299 are reserved for per tool settings.

252

Select Tool

Used for port numbers 253 – 299

253

Reserved

Reserved for more tool settings.

254

Pause at Tool Change

Pause on next tool change

255

Laser Power for Tool

D=0:0% ~ 100:100%

256

Feedrate for Tool

Used to set other calibrations for Laser and Plasma

257

Focus Offset for Tool

258

Pierce Time for Tool

D=value in milliseconds

259

Pierce Power for Tool

D=Power 0=0% 100=100%

260

Gas Pressure for Tool

261

Nozzle Type for Tool

262

User Acceleration for Tool

263

Rotary Diameter for Tool

E=diameter

264

Laser Power Control Mode

D=0 for fixed PWM D=1 for pulsed D=2 for analog output D=3 for variable PWM

265

VPPI or VPPmm

E=VPPunit

266

PWM Frequency

E=frequency in kHz

267

PWM Min Power (Pulsed Mode)

D=Min Power 0=0% 100=100%

268

Analog Output Voltage, No Slew

E=volts

269

Analog Output Voltage, Slew Over Move

E=volts

270

Cut Height

E=distance in user units

271

Lift Height

E=distance in user units

272

Pierce Height

E=distance in user units

273

Reference Voltage

E=voltage (Z tracking)

274

Max Voltage Gap

E=voltage Port 274 tells Z tracking to inhibit because of the hole.

275

Abrasive

D=Abrasive Selection 0=do not use abrasive 1=use abrasive

E=0.0 E value is not used, so operators should set to 0.0.

276

Abrasive Delay

D=Abrasive Delay -value= abrasive is turned on before the water (Abrasive Delay -500 allows for a 500 millisecond delay after the abrasive is turned on before the water is turned on.) +value=

abrasive is turned on after the water (Abrasive Delay 300 allows for a 300 millisecond delay after the water is turned on before the abrasive is turned on.)

E=0.0 E value is not used, so operators should set to 0.0

277

Abrasive Flow Rate

D=units 0=lb/min 1=g/min

E=Abrasive Rate in units specified by D value

278

Pierce Pressure

D=Pierce Pressure 0=Low 2=High

E=0.0 E value is not used, so operators should set to 0.0.

279

Cut Pressure

D=Cut Pressure 0=Low 1=High

E=0.0 E value is not used, so operators should set to 0.0.

280

Fiber Laser Clock Frequency

281-294

Reserved

Reserved for more tool settings.

295

Set Material Name

S<Material> as a string

296

Set Material Type

S<Material Type> as a string

Innovation. Quality. Performance. Best in Industry. Worldwide. 16

Routers - Lasers - Plasmas - Water Jets - Knife Cutters

www.multicam.com 16 August 2011

297

Set Material Layer

S<Layer Name> as a string

298

Reserved

Material settings

299

Set Material Type and Current

D<current> in amps E<thickness> in mm S<material> as a string, of specified list

300

Gang Drill Number

D<bitmask>=gang drill to lower (e.g., G98 P300 D4 {lowers the third gang drill})

301

Rotary Control <data=degree to move>

Moves the rotary to a specified degree

302

Rotary RPM <data=RPM>

Turns the rotary at a specified RPM

303

No Spin Tool

D<tool>=tool to enable no spin (v6.31)

304

Bitmask of OPTO EB Ports To Turn ON and Wait

Not used on M2521 controllers

305

Enable Lathe

1=enable 2=disable 3=move to lathe offset 4=find lathe Home 5=adjust feedrate of lathe based on diameter (v8.19 mh_lathe_along_x module) 6=don’t adjust feedrate based on diameter; default (v8.19 mh_lathe_along_x module) 7=G01 B value is in degrees; default (v8.19 mh_lathe_along_x module) 8=G01 B value is in user units (v8.19 mh_lathe_along_x module)

306

Lathe Diameter

D value specifies the diameter in 1/1000ths of user units (for metric system, G98 P306 D65000 = 65mm)

E value specifies the diameter in user units (for metric system, G98 P306 E65 = 65mm)

307

C-axis

0=disable C-axis 1=enable C-axis in Automatic Knife mode

Added to mh_caxis.uc module v8.12

310

OPTO EB Port to turn ON

Available in version 6.00

311

OPTO EB Port to turn OFF

Available in version 6.00

Innovation. Quality. Performance. Best in Industry. Worldwide. 17

Routers - Lasers - Plasmas - Water Jets - Knife Cutters

www.multicam.com 16 August 2011

G98 P<p> D<d> E<e.e> G10 L<n> P<p> R<r.r>

G98 P<p> D<d> E<e.e> --- Xlated to --- <e.e> <d> <p> set_port_ex G10 L<n> P<p> R<r.r> --- Xlated to --- <r.r> <p> <n> set_port_ex

Port G98 P<p> G10 L<n>

Parameter G98 D<d> G10 P<p>

Description

410

Tool #

Sets the tool length for the specified tool number. Added to Router Inits v8.05. Requires JobConsole v4.0.32 or later.

411

Tool #

Adjusts the tool length by the tool length wear. Added to Router Inits v8.05. Requires JobConsole 4.0.32 or later.

412

Tool #

Sets the tool comp value. Added to Router Inits v8.05. Requires JobConsole v4.0.32 or later.

413

Tool #

Adjusts the tool comp value. Added to Router Inits v8.05. Requires JobConsole v4.0.32 or later.

4000 – 4999

Reserved

Reserved for Job Info commands sent from JobConsole (Job Info Table)

G98 P<p> D<d>

G98 P<p> D<d> --- Xlated to --- <d> <p> set_port

The following ports are processed by JobConsole and/or Xlate and are not passed to the controller.

Port

Description

Notes

148

Resets HPGL Resolution in the Translator

XY to 1021 and Z to 1 – does NOT send a command

1000

X Soft Home in 1/1000th

Set X Soft Home position

1001

Y Soft Home in 1/1000th

Set Y Soft Home position

1002

Z Soft Home in 1/1000th

Set Z Soft Home position

1003

Reserved

1004

Reserved

1005

Perform Program Pause (prg_pause)

0 for M00 1 for M01

1006

Diameter in 1/1000th

Specify diameter of workpiece

1007

Reserved

1008

Angle in Degrees

Set rotational angle

1009

X Rotational Point of Origin in 1/1000th

Specify X rotation point of origin

1010

Y Rotational Point of Origin in 1/1000th

Specify Y rotation point of origin

1011

X Letter Base Coordinate in 1/1000th

Specify X letter base location

1012

Y Letter Base Coordinate in 1/1000th

Specify Y letter base location

1013

Z Letter Base Coordinate in 1/1000th

Specify Z letter base location

1015

Lead-in Length XY Length per Unit Z Depth

Not implemented. Reserved for future use.

1020

Bounding Box of Sheet

This port is sent as <port> info_string.

1030

Skip Pre Process

1031

1=Negative Z down

1032

Pass Information to Controller and Change Presentation Graphics

0x01= Digitized 0x02= R-Z-Theta (mapped from XYZ) 0x04= XYZUV

1040

Tool Prompt

This port is sent as <port> info_string.

1050

Font Name (JobConsole v4.0.68 or later)

G98 P1050 S<name of font> Optional: default font is specified in XMI.

Port 1050 specifies the name of the font to be used for all text that follows or until another G98 P1050 is used (e.g., G98 P1050 SRomans). If the name of the font does not match a supported font, then the default font is used. (DEXYZ are not used.)

1051

Text (JobConsole v4.0.68.0 or later)

G98 P1051 [D<subst>] E<size> S<string> specifies the text string to be output at the current location

D<subst> = substitution number, 0 = no substitution E<size> = height of text in user units S<string> = text to be displayed

The substitution number comes from the pre_job_table and allows the CNC job file to substitute text (e.g., G98 P1051 E0.25 S09863). Operators can use the keyword TEXT_nnn to specify the substitution string.

For example: The pre_job_table specifies TEXT_001 09863. The job file includes G98 P1051 D1 E0.25. The results would be the same as G98 P1051 E0.25 S09863.

1053

Character Identifier (JobConsole v4.0.68.0 or later)

G98 P1053 D<ascii #> S<character> Used for defining fonts by identifying the character to be defined. (EXYZ are not used.)

D= the ascii value of the character (e.g., 97 = a) S= text representation of the character

For example: G98 P1053 D97 Sa

1054

Character Minimum Extents (JobConsole v4.0.68 or later)

G98 P1054 X<Min Extent> Y<Min Extent> Used for defining fonts.

XY = the minimum extents of the character

For example: G98 P1054 X0.0 Y-0.30

1055

Character Maximum Extents (JobConsole v4.0.68 or later)

G98 P1054 X<Max Extent> Y<Max Extent> Used for defining fonts.

XY = the maximum extents of the character

For example: G98 P1055 X0.6 Y1.0

2008

Rotation angle (internal use only)

2009

X of the center of rotation (internal use only)

2010

Y of the center of rotation (internal use only)

2200 – 2399

Reserved

Reserved

G Codes and M Codes with Software Ports

All lines of machine code must start with either a G Code or an M Code, and software ports can be used to define the operation. The timeline for additions and updates to the

codes and software ports is included, and the following tables provide additional information on each type of code and on software ports. Multiple G or M Codes cannot be put on the same line.

Timeline for Additions and Updates

The following timeline shows what changes were made to the G and M Codes and to the software ports since 2003.

Date

Additions or Updates

02 Nov 2010

Update port 159

03 Jun 2010

Add port 111

01 Jun 2010

Add M17 information

10 Jun 2009

Added port 183

11 Jan 2008

Added Material Handling

31 Jul 2007

Added font ports

23 Mar 2007

Added ports in 200 range

01 Mar 2007

Clarified Park commands, when they were added

18 Dec 2006

Added M50

27 Jul 2006

Updated Port 137 operation

09 Feb 2006
Added ports 126 and 127
11 Jan 2006
Add G84 commands (converted to atap_cycle)
29 Jul 2005
Added C-axis ports
28 Sep 2004
Added M18 and M19
01 Jul 2004
Allow G83 and G81to be modal
12 May 2004
Added in Plasma Library ports
21 Oct 2003
Fix G81 pecking more than once if retract height < 0
19 Aug 2003
Add G81 and allowed G83 to have X and Y
27 May 2003
Added M95, M96, M97, M98, and VP(133,142)
13 Jan 2003
Added Homing and Set Surface Virtual Ports

G Codes
Operators creating G-Code files from their post processor must end the file with a .cnc or .anc file extension so that the DNC program can recognize the file. The following table lists the supported G Codes for the MultiCam controller.

Parameters within brackets are optional. The fields represented by “d.d” may be any decimal number, and fields represented by “d” may be any positive integer.

Code

Description

Notes (if applicable)

G00

[Xd.d] [Yd.d] [Zd.d] [Fd.d] [Td] [Ctext string]

High speed move (slew)

G01

[Xd.d] [Yd.d] [Zd.d] [Fd.d]

Linear move (machine)

G02

[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]

CW 2D circular move

G03

[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]

CCW 2D circular move

G04

Fd.d

Dwell (seconds)

G17

Specify XY Plane for Helical

G18

Specify ZX Plane for Helical

G19

Specify YZ Plane for Helical

G37

Find Home

G40
Cancel Tool Compensation
G41
Left Tool Compensation
G42
Right Tool Compensation
G62
Clear Soft Home
G70
English Programming (inches)
G71
Metric Programming (millimeters)
G72
[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]
CW 3D circular move
G73
[Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d]
CCW 3D circular move
G74
Incremental Mode for G02/03 Arcs
G75
G90/91 Mode for G02/03 Arcs
G81
[Xdd.d] [Ydd.d] [Rdd.d] [Zdd.d] [Fdd.d]
One-stroke drill cycle
G83
[Xdd.d] [Ydd.d] [Rdd.d] [Zdd.d] [Ddd.d] [Fdd.d]
Peck drill cycle with router
G84

[Xdd.d] [Ydd.d] [Zdd.d] [Rdd.d] [Fdd.d]
Tap cycle
G90
Absolute Coordinate Mode
G91
Incremental Coordinate Mode
G92
[Xd.d] [Yd.d] [Zd.d]
Set Soft Home
G97
Sd
Set spindle speed (RPM)
G98
Plasma Systems Only 12. P133 D0
1. P145 Dd 23. P300 Dd
By default, plasma systems ignore feedrates sent in the job file. 1. Feedrates will be used from now on. 2. Feedrates will be ignored from now on. 1. Go to pre-recorded Home position (e.g., D1=Home 1, D2=Home 2) 2. Park X-axis (e.g., D0=X Min, D1=X Max) 3. Boring unit drill select

The following table lists the letters used to denote various arguments in Etc CNC version 1.0.
Argument
Description
Location of Use (if applicable)

C

Tool Change Operator message

G00

D

Peck Drill Data

G83, Data selection in G98

F

Feedrate in Units per Second

G00, G01, G02, G03, G72, G73, G83

G

Preparatory Function

I

Circular Interpolation Value in X Dimension

G02, G03, G72, G73

J

Circular Interpolation Value in Y Dimension

G02, G03, G72, G73

K

Circular Interpolation Value in Z Dimension

G02, G03, G72, G73

M

Miscellaneous or Control Function

N

Sequence Number

R

Beginning Z Motion Dimension

G83

S

Spindle RPM

G97

T

Tool Change

G00

X

X Motion Dimension

Y

Y Motion Dimension

Z

Z Motion Dimension

M Codes

The following table lists the available M Codes and how they should be configured for JobServer. Operators can use Device 199 Inactive to ignore M Code.

Code

Description

Device #

State*

Graphics

Notes

M00

Program Pause

-97

I

n/a

0 prg_pause (needed for Suite4)
M01
Optional Program Pause
-96
I
n/a
1 prg_pause (needed for Suite4)
M02
End of Job
-98
I
n/a
end_plot (needed for Suite4)
M03
Start Spindle Clockwise
113
A
n/a
Spindle ON clockwise
M04
Start Spindle Counter-clockwise
114
A
n/a
Spindle ON counter-clockwise
M05
Spindle OFF
105
A

n/a

spindle_off

M11

2D Device ON

-1 or 101

A

ON

-1= current tool number is passed 101= current tool number is selected by Init file

M12

3D Device ON

-1 or 102

A

ON

-1= current tool number is passed 101= current tool number is selected by Init file

M13

2D Device ON, no Z down

113

A

ON

-1= current tool number is passed 101= current tool number is selected by Init file **M13 does not lower the pneumatic Z and is used in Plasma only.**

M17

Turn on Z Tracking

118

A

OFF

Turns on Z Tracking for the current contour in Plasma only

M18

Turn off Z Tracking

118

I

OFF

Turns off Z Tracking for the current contour in Plasma only

M19

Disable Arc Out Pause

119

I

OFF

Disables the Pause feature when the arc goes out and becomes enabled at the next contour (i.e., M11 or M12) in Plasma only

M21

2D Device OFF

-1 or 101

I

OFF

-1= current tool number is passed 101= current tool number is selected by Init file

M22

3D Device OFF

-1 or 101

I

OFF

-1= current tool number is passed 101= current tool number is selected by Init file

M23

2D Device OFF, no Z up

123

I

OFF

-1= current tool number is passed 101= current tool number is selected by Init file **M23 does not lower the pneumatic Z and is used in Plasma only.**

M25

Start of Sheet

-99

A

OFF

Starts sheet and is available as nsheet in H4LDR version 4.50 and later

M30

Fire Enabled Drill

130

A

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #130 (Active) to 66 PD

M31

Drill 1 ON (Enable & Offset)

131

A

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56 and above convert device #131 to 66 PD tool_change

M32

Drill 2 ON (Enable & Offset)

132

A

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #132 to 67 tool_change

M38

Gang Drill 1 ON

138

A

OFF

M41

Drill 1 OFF (Disable)

131

I

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #131 (Inactive) to 66 PU

M42

Drill 2 OFF (Disable)

132

I

OFF

Available in H4LDR version 4.55 and later; JobConsole v4.0.56.0 and above convert device #131 (Inactive) to 67 PU

M48

Gang Drill 1 OFF

138

I

OFF

Turns off and raises gang drill 1; resets surface back to surface for current tool. For JobConsole, change “Post” tab under CNC settings. Under XMI Settings, do the following:

1.

Select CNC tab

2.

Select Post tab

3.

Add tool number -138

4.

Add tool description as Gang Drill

5.

Add Tool Change M38

6.

Add Tool Unload M48

7.

Leave Tool Activation blank

8.

Leave Tool Deactivation blank

M50

Material Handler (Panel Pusher)

150

I

OFF

Requires RIO_pusher.uc module. Starts and ends a material unload process with the following:

1.

Moves to X start.

2.

Lowers pusher pins.

3.

Moves to X stop.

4.

Raises pusher pins.

M60

Put Away Tool

104

A

OFF

Available in H4LDR version 4.55 and later **M60 is only available on ATC machines.**

M90

Program Start

n/a

n/a

n/a

Opens start_plot or cycle_start **For Suite4 set device number to -90.**

www.multicam.com 16 August 2011

M91

Program Replay

n/a

n/a

n/a

Not supported

M92

ALL Mode

192

A

OFF

Available in H4LDR version 4.58 and later **M92 is only available on standard machines.**

M93

Return to Auto Mode

192

I

OFF

Available in H4LDR version 4.58 and later **M93 is only available on standard machines.**

M94

Disable Spindle Offset

194

A

n/a

Available in H4LDR version 4.71 and later **M94 makes the spindle offset between heads 0,0 and is only available on standard machines.**

M95

Enable Marking Mode

195

A

OFF

Plasma only

M96

Disable Marking Mode

195

I

OFF

Plasma only

M97

Double Velocity

197

I

OFF

Sets for faster lead-outs in Plasma only

M98

Turn off Z Tracking, Disable Arc Out Pause, then Turn off Plasma Arc

198

I

OFF

Disables the arc before the end of the contour in Plasma only

M99

Exit CNC Interpreter

n/a

n/a

n/a

M150

Material Handling Pusher Cycle

150

A

n/a

Runs a complete pusher cycle

M151

Material Handling Lifter Cycle

151

A

n/a

Runs a complete lifter cycle

M152

Material Handling Dust Collector Blast Gate ON

152

A

n/a

Turns ON the dust collector blast gate

M153

Material Handling Dust Collector Blast Gate OFF

153

A

n/a

Turns OFF the dust collector blast gate

M154

Material Handling Sweeper Blast Gate ON

154

A

n/a
Turns ON the sweeper blast gate
M155
Material Handling Sweeper Blast Gate OFF
155
A
n/a
Turns OFF the sweeper blast gate
M156
Material Handling Air Knife ON
156
A
n/a
Turns ON the air knife
M157
Material Handling Air Knife OFF
157
A
n/a
Turns OFF the air knife
* A = Active, I = Inactive

M Codes can directly control M24 devices when used to configure the M-Code table for JobServer. Devices 200-299 provide direct control while devices 300-399 provide exclusive

device control that turns other devices off when a particular device has been turned on.

Device

Device Output

M24 Header / Location

200

Spindle 1 Output

H2: 1&2

201

Mister 1 Output

H2: 3&4

202

Spindle 2 Output

H2: 5&6

203

Mister 2 Output

H2: 7&8

204

Spindle 3 Output for M24 revision 2, 3, and 4 Spindle Enable for M24 Revision 5

H2: 9&10

205

Mister 3 Output for M24 revision 2, 3, and 4 Drill Enable for M24 revision 5

H2: 11&12

206

Drill 1 Output

H3: 1&2

207

Drill 2 Output
H3: 3&4
208
Caution Output
H3: 5&6
209
TC Chuck Output
H4: 1&2
210
TC Blast
H4: 3&4
211
Dust Collector
H4: 5&6
212
Misc 1 ** DO NOT USE **
H4: 7&8
213
Misc 2 ** DO NOT USE **
H4: 9&10
214
Misc 3 ** DO NOT USE **
H4: 11&12
250 – 269
General M-Code Mapping to Devices Mapped using MCode_Device_Map file.

www.multicam.com 16 August 2011

Software Ports

Software ports are used to expand either HPGL or CNC language. Operators can use a software port in the job file with either a G98 or Z0 command as shown below:

CNC Job G98 P147 D1 will park the machine at X Max.

HPGL Job Z0147, 1 will park the machine at X Max.

Virtual ports 1-49 are reserved for setting physical ports while virtual ports 50-99 are reserved for clearing physical ports.

The following 2 strings are defined in relation to their associated port, location or data, and description while the third and fourth strings are more complex.

G98 P<n> X<x.x> Y<y.y> Z<z.z> S<string>

G98 P<n> [X<x.x>] [Y<y.y>] [Z<z.z>] [S<string> --- Xlated to --- <n> info_string <string> The XYZ are only used for JobPreviewer.

Port

XYZS

Description

1020

Any

Specifies the bounding box of sheet. XYZ are read in Xlate version 3.85 and sent to JobPreviewer but not the controller; S is read in Xlate version 3.9.24 and converted to <n> info_string.

1040

Tool Prompt

G98 P<n> S<string>

G98 P<n> [S<string> --- Xlated to --- <n> info_string <string>

Port

String

Description

140

Any

Displays the string but does not wait for the operator. Requires XLate v3.85 or later.

141

Any

Displays the string and waits for operator response. Requires Xlate v3.85 or later.

G98 P<p> D<d> [E<e.e>]

G98 P<p> D<d> --- Xlated to --- <d> <p> set_port G98 P<p> D<d> E<e.e> --- Xlated to --- <d> <e.e> <p> set_port_ex

Port

Data

Description

100

Spindle Speed in RPM

Sets spindle RPM. Mainly used for HPGL jobs. In CNC jobs, operator should use G97 S<spindle speed>.

101

Marker Identifier

1=character 2=word 3=line 4=part 5=start of part 6=end of part

110

Max Z Increment for Multipass

Specifies in 1/1000th

****Port 110 is not applicable to M2521.****

111

Feedrate Override

0=disable, set to 100%

This command is immediate and will change the feedrate override to the percent specified (i.e., 0% to 100%).

120

Percentage of Laser Power

Sets laser power

121

Absolute Index

Move to abs. Position, int value *HPGL Resolution

122

Solenoid ON/OFF

0=off 1=on

123

Auto Spindle Enable (BOJ, EOJ)

0=disable auto spindle 1=enable auto spindle 2=enable manual spindle control using M03, M04, M05

125

Z Probe Disable/Enable

0=disable probing 1=enable probing on PD

****Port 125 is reset at the start of each job (added to ZSurf_mod.uc v2.22).****

126

Tapping Mode

0=off The next tool change will be a tapping tool to be used with G84 commands. 1=on

Tapping mode gets converted into µCito job file command atap_cycle and requires either JobConsole 4.0.10 or later or PSS Xlate.dll v3.9.27.

127

Linear Encoder Mode

0=off 1=on

****Completes a second adjustment move at the end of each AC or U based on readings from linear encoders.****

128

Z Probe Location (relative to current surface)

Specifies in 1/1000th; indicates the expected location of the top of the material when using the Z surface probe

****Port 128 is reset to 0.0 at the start of each job (ZSurf_Mod.uc).****

129

Touch Off Radius

Specifies in 1/1000th; uses the radius during Z probe use to determine if the probe should touch off again

****Port 129 is reset back to the keypad settings at the start of each job (ZSurf_Mod.uc).****

130

Spindle Control

0=spindle off 1=spindle on

131

Job Type

1=start of vector job 2=start of raster job

132

Data Ignored

Sets Home at the current position

133

Use Feedrates in Job

0=ignore feedrates in the job 1=use feedrates in the job

****Port 133 is only applicable in Plasma v1.44 and is not available to routers.****

134

Use Z Values in Job

0=ignore Z values 1=use Z values

****Port 134 is only available in Water Jets (v3.01.10).****

135

Spindle Reverse Direction

1=spindle 1 reverse (CCW) 2=spindle 2 reverse (CCW)

****Port 135 orients the bit to reverse spindle and is available in version 8.02.24.****

136

Mister Configuration

0=disable mister 1=re-enable mister

If Port 136 is set to 0, then the mister will not turn on for an M11 or M12. The mister is ON by default. At the start of every job, the mister is re-enabled unless the keypad option MisterOpt (added in v8.09.05) indicates the mister is disabled.

137

Manual Dust Collector Control

0=raise dust collector (v8.02.27) 1=lower dust collector 2=disable dust collector (v8.02.60)
Use G98 P137 D0 to raise and re-enable dust collector operation.
138
Low RPM Mode for Motor #2
0=disable low RPM mode 1=enable low RPM mode for Motor #2
Used for slow speed drilling. Port 138 requires a special F7 Inverter programming that will switch to Motor #2 in the inverter (added v8.02.32).
140
Drill Hole
D=<mode> E=<Hole Diameter>
Hole is drilled at current location.

142
Host Jog
Starts host Jog and ignores data
143
Set Surface
0=auto (added in v.6.10) 1=current position
144
Find Limits

Ignores data (added in v6.10) Port 144 is the same as using the keypad to find limits, but Port 144 leaves the machine in motion to allow the job file commands to follow.

145

Fixture Number

Sets Home at the fixture location

146

Park Z Location

Ignores data but always parks Z to Z0.

147

Park Command

0=X Home 1=X Max 2=X Home, Y Home (v8.02.46)

****Port 147 parks Z first and then parks X (e.g., G98 P147 D1 {parks machine at X Max}).****

150

Z Lift

Sets the Z lift in 1/1000th (v3.12) in Plasma only. See Port 271.

151

Contour Acceleration

D value = axis bit value (1=X, 2=Y, 3=Z) E value = acceleration in user units (e.g., G98 P151 D3 E1.0 sets acceleration of XY to 1.0 for the following contours)

****Port 151 command must precede contour and is used on WaterJets ONLY.****

152

Pierce Type

D=<type> E=<exit angle> (value -999 means unknown)

<type> 0=standard, 1=dynamic, 2=drill

Operators should input the final angle relative to the start point to finish the pierce for dynamic pierce methods. This sets the desired direction for the next move.

153

Set Dual Head Control (dual heads, not dual gantry axes)

D=<enable mask> Ha=Head a (either Xa or Ya) Hb= Head b (either Xb or Yb)

D=1 activate Ha, park Hb D=2 activate Hb, park Ha D=3 activate both heads with offset specified

E=<Yb offset> Hb=Y offset + Ha

154

Set Dual-head Mirror

D=<enable> 0 to disable, 1 to enable E=<mirror Pt> Hb=Hmirror - Ha

155

Contour Deceleration

D=<mask> E=<deceleration value> for those axes

D value = axis bit value (1=X, 2=Y, 3=Z) E value = deceleration in user units

156

Torch Height Voltage

Sets the torch height voltage in Plasma only

157

User Rate (1 – 10)

1=slowest 10=fastest

Port 157 determines the user feedrate/acceleration factor in Plasma only.

158

Max Volt Gap

Sets the maximum voltage gap in 1000 (v3.26.02) in plasma only

159

Bevel Angle and Mode

D=<mode> -1=turn off bevel mode 0=turn on bevel mode

E=<angle> Bevel angle in degrees (a positive angle will angle outward)

Examples – G98 P159 E3 D1 = set bevel angle for following contour to 3° G98 P159 E0 D-1 = turn off bevel mode

160

Material Handling Suction Cup Pod Vacuum

D<bitmask> = suction cup to activate (e.g., G98 P160 D1 turns on suction cup #1; G98 P160 D0 turns off suction cups)

Port 160 calls DCN #324 and was added to Router Inits in version 8.06.10.

161

Material Handling Suction Cup Pod Blower

D<bitmask> = blower to activate

Port 161 calls DCN #325 and was added to Router Inits in version 8.06.10.

162

Material Handling Sheet Size

D<Sheet Size> (value in 1/1000th) (e.g., G98 P162 D96000 {sets the sheet size to 96})

Available if the Material_Handling.uc module is loaded

163

Material Handling Start of Sheet

D<Start Location> (value in 1000th) (e.g., G98 P163 D12000 {sets the start of sheet location to 12})

Available if the Material_Handling.uc module is loaded.

164

Conveyor Index

D<Mode> E<X Distance> E value is required in all cases <mode> 0=end of panel 1=E value specifies center of panel 5=index panel (E value ignored) 10=set grab position (reset at EOJ) 11=set release position (reset at EOJ)

180

Dual Y Configuration (deprecate)

1=Ya 2=Yb 3=Both 4=Both Mirrored

****Port 180 is available in WaterJet v3.01.21 and Router v8.03 (e.g., G98 P180 D3 {sets Dual Y to Both mode}).****

181

Dual Y Yb Offset (deprecate)

Yb Offset value in 1000th for Water Jet v3.01.21 and Router v8.03 (e.g., G98 P181 D20000 {sets Yb offset to 20})

183

Chip Break Value

D value not used and does not need to be included. E=relative lift height for chip breaking and is currently only implemented for the Big Drill machine. Port 183 is called

before a G83 command to indicate it should only lift by the Chip Break value instead of retract height between pecks. If the chip break value is ≤ 0.0, then the chip break feature will be ignored (e.g., G98 P183 E0.1 sets the chip break value to 0.1).

**** Port 183 value will be used for all following G83 commands and so does not need to be specified for each G83 command. The value is set back to 0.0 at the start of each job.****

184

Y Brake Control

D=0 turns OFF Y brake D=1 turns ON Y brake

Y Brake is controlled by DCN 152 and is currently only implemented for the Max-40 machine.

Ports 200 – 251 are used to set job values.

200

Cut Feedrate

Machine Feedrate converted to <E value> SDF

201

Slew Feedrate

Slew Feedrate

202

Z Up Feedrate

203

Z Down Feedrate

Value is in MM/sec based on the tool. If the knife tool is currently selected, then this will affect the Knife Plunge Feedrate (e.g., G98 P203 E25.4 sets the Z down feedrate to 25.4mm {1.0”} per second).

210

Pen Down Delay

211

Pen Up Delay

220

Pen Down Height

Equivalent to SPD in µCito and ZD in HPGL

221

Pen Up Height

Equivalent to SPU in µCito and ZU in HPGL

231

Contour Group ID

240

Material Library Information

D=1 S=Material Name D=2 S=Process Name

Coding used G98 P240 D1 SMS │0.375 (3/8)" G98 P240 D2 S60A │ SHLD Will get translated into 1 240 set_port 240 info_string MS │ 0.375 (3/8)" 2 240 set_port 240 info_string 60A │ SHLD

241

Material Library Cut Parameters

D=<Parameter ID> E=<Parameter value>

250

Set Raster Scan Line Start

Set the zero-based index (inclusive) of the first scan line in the following PICT

251

Set Raster Scan Line End

Set the zero-based index (inclusive) of the last scan line in the following PICT

Ports 252-299 are reserved for per tool settings.

252

Select Tool

Used for port numbers 253 – 299

253

Reserved

Reserved for more tool settings.

254

Pause at Tool Change

Pause on next tool change

255

Laser Power for Tool

D=0:0% ~ 100:100%

256

Feedrate for Tool

Used to set other calibrations for Laser and Plasma

257

Focus Offset for Tool

258

Pierce Time for Tool

D=value in milliseconds

259

Pierce Power for Tool

D=Power 0=0% 100=100%

260

Gas Pressure for Tool

261

Nozzle Type for Tool

262

User Acceleration for Tool

263

Rotary Diameter for Tool

E=diameter

264

Laser Power Control Mode

D=0 for fixed PWM D=1 for pulsed D=2 for analog output D=3 for variable PWM

265

VPPI or VPPmm

E=VPPunit

266

PWM Frequency

E=frequency in kHz

267

PWM Min Power (Pulsed Mode)

D=Min Power 0=0% 100=100%

268

Analog Output Voltage, No Slew

E=volts

269

Analog Output Voltage, Slew Over Move

E=volts

270

Cut Height

E=distance in user units

271

Lift Height

E=distance in user units

272

Pierce Height

E=distance in user units

273

Reference Voltage

E=voltage (Z tracking)

274

Max Voltage Gap

E=voltage Port 274 tells Z tracking to inhibit because of the hole.

275

Abrasive

D=Abrasive Selection 0=do not use abrasive 1=use abrasive

E=0.0 E value is not used, so operators should set to 0.0.

276

Abrasive Delay

D=Abrasive Delay -value= abrasive is turned on before the water (Abrasive Delay -500 allows for a 500 millisecond delay after the abrasive is turned on before the water is turned on.)

+value= abrasive is turned on after the water (Abrasive Delay 300 allows for a 300 millisecond delay after the water is turned on before the abrasive is turned on.)
E=0.0 E value is not used, so operators should set to 0.0
277
Abrasive Flow Rate
D=units 0=lb/min 1=g/min
E=Abrasive Rate in units specified by D value
278
Pierce Pressure
D=Pierce Pressure 0=Low 2=High
E=0.0 E value is not used, so operators should set to 0.0.
279
Cut Pressure
D=Cut Pressure 0=Low 1=High
E=0.0 E value is not used, so operators should set to 0.0.
280
Fiber Laser Clock Frequency
281-294
Reserved
Reserved for more tool settings.
295
Set Material Name
S<Material> as a string
296
Set Material Type
S<Material Type> as a string

297

Set Material Layer

S<Layer Name> as a string

298

Reserved

Material settings

299

Set Material Type and Current

D<current> in amps E<thickness> in mm S<material> as a string, of specified list

300

Gang Drill Number

D<bitmask>=gang drill to lower (e.g., G98 P300 D4 {lowers the third gang drill})

301

Rotary Control <data=degree to move>

Moves the rotary to a specified degree

302

Rotary RPM <data=RPM>

Turns the rotary at a specified RPM

303

No Spin Tool

D<tool>=tool to enable no spin (v6.31)

304

Bitmask of OPTO EB Ports To Turn ON and Wait

Not used on M2521 controllers

305

Enable Lathe

1=enable 2=disable 3=move to lathe offset 4=find lathe Home 5=adjust feedrate of lathe based on diameter (v8.19 mh_lathe_along_x module) 6=don't adjust feedrate based on diameter; default (v8.19 mh_lathe_along_x module) 7=G01 B value is in degrees; default (v8.19 mh_lathe_along_x module) 8=G01 B value is in user units (v8.19 mh_lathe_along_x module)

306

Lathe Diameter

D value specifies the diameter in 1/1000ths of user units (for metric system, G98 P306 D65000 = 65mm)

E value specifies the diameter in user units (for metric system, G98 P306 E65 = 65mm)

307

C-axis

0=disable C-axis 1=enable C-axis in Automatic Knife mode Added to mh_caxis.uc module v8.12

310

OPTO EB Port to turn ON

Available in version 6.00

311

OPTO EB Port to turn OFF

Available in version 6.00

G98 P<p> D<d> E<e.e> G10 L<n> P<p> R<r.r>

G98 P<p> D<d> E<e.e> --- Xlated to --- <e.e> <d> <p>
set_port_ex G10 L<n> P<p> R<r.r> --- Xlated to --- <r.r> <p> <n> set_port_ex
Port G98 P<p> G10 L<n>
Parameter G98 D<d> G10 P<p>
Description
410
Tool #
Sets the tool length for the specified tool number. Added to Router Inits v8.05. Requires JobConsole v4.0.32 or later.
411
Tool #
Adjusts the tool length by the tool length wear. Added to Router Inits v8.05. Requires JobConsole 4.0.32 or later.
412
Tool #
Sets the tool comp value. Added to Router Inits v8.05. Requires JobConsole v4.0.32 or later.
413
Tool #
Adjusts the tool comp value. Added to Router Inits v8.05. Requires JobConsole v4.0.32 or later.
4000 – 4999
Reserved
Reserved for Job Info commands sent from JobConsole (Job Info Table)

www.multicam.com 16 August 2011

G98 P<p> D<d>

G98 P<p> D<d> --- Xlated to --- <d> <p> set_port

The following ports are processed by JobConsole and/or Xlate and are not passed to the controller.

Port

Description

Notes

148

Resets HPGL Resolution in the Translator

XY to 1021 and Z to 1 – does NOT send a command

1000

X Soft Home in 1/1000th

Set X Soft Home position

1001

Y Soft Home in 1/1000th

Set Y Soft Home position

1002

Z Soft Home in 1/1000th

Set Z Soft Home position

1003

Reserved

1004

Reserved

1005

Perform Program Pause (prg_pause)

0 for M00 1 for M01

1006

Diameter in 1/1000th

Specify diameter of workpiece

1007

Reserved

1008

Angle in Degrees

Set rotational angle

1009

X Rotational Point of Origin in 1/1000th

Specify X rotation point of origin

1010

Y Rotational Point of Origin in 1/1000th

Specify Y rotation point of origin

1011

X Letter Base Coordinate in 1/1000th

Specify X letter base location

1012

Y Letter Base Coordinate in 1/1000th

Specify Y letter base location

1013

Z Letter Base Coordinate in 1/1000th

Specify Z letter base location

1015

Lead-in Length XY Length per Unit Z Depth

Not implemented. Reserved for future use.

1020

Bounding Box of Sheet

This port is sent as <port> info_string.

1030

Skip Pre Process

1031

1=Negative Z down

1032

Pass Information to Controller and Change Presentation Graphics

0x01= Digitized 0x02= R-Z-Theta (mapped from XYZ) 0x04= XYZUV

1040

Tool Prompt

This port is sent as <port> info_string.

1050

Font Name (JobConsole v4.0.68 or later)

G98 P1050 S<name of font> Optional: default font is specified in XMI.

Port 1050 specifies the name of the font to be used for all text that follows or until another G98 P1050 is used (e.g., G98 P1050 SRomans). If the name of the font does not match a supported font, then the default font is used. (DEXYZ are not used.)

1051

Text (JobConsole v4.0.68.0 or later)

G98 P1051 [D<subst>] E<size> S<string> specifies the text string to be output at the current location

D<subst> = substitution number, 0 = no substitution E<size> = height of text in user units S<string> = text to be displayed

The substitution number comes from the pre_job_table and allows the CNC job file to substitute text (e.g., G98 P1051 E0.25 S09863). Operators can use the keyword TEXT_nnn to specify the substitution string.
For example: The pre_job_table specifies TEXT_001 09863. The job file includes G98 P1051 D1 E0.25. The results would be the same as G98 P1051 E0.25 S09863.

1053

Character Identifier (JobConsole v4.0.68.0 or later)
G98 P1053 D<ascii #> S<character> Used for defining fonts by identifying the character to be defined. (EXYZ are not used.)
D= the ascii value of the character (e.g., 97 = a) S= text representation of the character
For example: G98 P1053 D97 Sa

1054

Character Minimum Extents (JobConsole v4.0.68 or later)
G98 P1054 X<Min Extent> Y<Min Extent> Used for defining fonts.
XY = the minimum extents of the character
For example: G98 P1054 X0.0 Y-0.30

1055

Character Maximum Extents (JobConsole v4.0.68 or later)
G98 P1054 X<Max Extent> Y<Max Extent> Used for defining fonts.
XY = the maximum extents of the character
For example: G98 P1055 X0.6 Y1.0

2008

Rotation angle (internal use only)

2009

X of the center of rotation (internal use only)
2010
Y of the center of rotation (internal use only)
2200 – 2399