MultiCam CNC Language Summary

The MultiCam CNC controller can interpret 2 languages HPGL and EIA-274D (G Codes). This command summary serves as a guide for both new and experienced EIA-274D (G Codes) users. The following table lists the supported G and M codes for the MultiCam CNC controller interface. You will also find many samples.

Uses For This Document:

• Post Processor or driver creation to interface CAM or Graphics software output with the MultiCam Controller. (Note many CAM and Graphics software already have post or driver support. Please check with your distributor or email the support department if you have any questions.)

• Code level trouble shooting and file creation.

M and G Code Table

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 number.

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
G40	CANCEL Tool Compensation
G41	LEFT Tool Compensation
G42	RIGHT Tool Compensation
G62	Clear soft home
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/G91) mode for G02/03 arcs
G83 Rd.d Zd.d Dd.d [Fd.d]	Peck drill (With Router)
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 P145 Dd	Go to pre-recorded home position, D(1) Home 1, D(2) Home 2, etc.
G98 P147 Dd	Park X axis , D(0) X-Min, D(1) X-Max
G98 P300 Dd	Boring Unit Drill Select.
Plasma Systems only	The plasma system ignors feed rates sent in the job file by default.
G98 P133 D1	Feedrate will be used from now on.
G98 P133 D0	Feedrates will be ignored from now on.

The following table lists the letters used to denote various arguments.

C - Tool change operator message (G00 required)
D - Peck drill delta (used in G83), Data selection in G98.
F - Feed rate (used in G00, G01, G02, G03, G72, G73, G83) Units per Second
F - Dwell (used in G04)
G - Preparatory function
I - Circular interpolation value in X dimension (used in G02, G03, G72, G73)
J - Circular interpolation value in Y dimension (used in G02, G03, G72, G73)
K - Circular interpolation value in Z dimension (used in G02, G03, G72, G73)
M - Miscellaneous function (control function)
N - Sequence number
R - Beginning Z motion dimension (used in G83)
S - Spindle rpm (used in G97)
T - Tool change (used in G00)
X - X motion dimension
Y - Y motion dimension
Z - Z motion dimension

M-Codes:

The table below lists the available M-Codes and how they should be configured for Job Server.

		Job Server Settings				
Code	**Description**	**Device #**	**State**	**Graphics**	**Notes**	**Init File Compatibility**
M00	Program Pause	N/A	N/A	N/A	0 prg_pause	
M01	Optional Program Pause	N/A	N/A	N/A	1 prg_pause	
M02	End of Job	N/A	N/A	N/A	end_plot	
M03	Start Spindle Clockwise	113	Active	N/A	Spindle ON Clockwise	
M04	Start Spindle Counter-Clockwise	114	Active	N/A	Spindle ON Counter-Clockwise	
M05	Spindle Off	105	Active	N/A	spindle_off	
M11	2D Device ON	-1 or 101	Active	ON	If you use -1, the current tool number is passed for M11. If you use 101, then the INIT file handles selecting the current tool number. Plasma Arc On	
M12	3D Device ON	-1 or 102	Active	ON	See note for M11	
M21	2D Device OFF	-1 or 101	Inactive	OFF	See note for M11 Plasma Arc Off	
M22	3D Device OFF	-1 or 102	Inactive	OFF	See note for M11	
M25	Start of Sheet	-99	Active	OFF	Pause and load next sheet	H4LDR Ver 4.50 and later
M30	Fire Enabled Drill	130	Active	OFF		H4LDR Ver 4.55 and later
M31	Drill 1 ON (Enable + Offset)	131	Active	OFF		H4LDR Ver 4.55 and later
M32	Drill 2 ON (Enable + Offset)	132	Active	OFF		H4LDR Ver 4.55 and later
M38	Gang Drill 1 ON	138	Active	OFF		
M41	Drill 1 OFF (Disable)	131	Inactive	OFF		H4LDR Ver 4.55 and later
M42	Drill 2 OFF (Disable)	132	Inactive	OFF		H4LDR Ver 4.55 and later
M48	Gang Drill 1 OFF	138	Inactive	OFF		
M60	Put Away Tool	104	Active	OFF	This is ONLY available on ATC machine.	H4LDR Ver 4.55 and later
M90	Program Start	N/A	N/A	N/A	start_plot CYCLE_START	
M92	ALL Mode On	192	Active	OFF	(Multi Head)	H4LDR Ver 4.58 and later. STD

						ONLY
M93	ALL Mode Off, Return to Auto Mode	192	Inactive	OFF	(Multi Head)	H4LDR Ver 4.58 and later. STD ONLY
M94	Auto Mode, Disable Spindle Offset	194	Active	N/A	(Multi Head) Make the Spindle Offset between heads 0.0.	H4LDR Ver 4.71 and later. STD ONLY
M95	Enable Marking Mode	195	Active	OFF	Plasma Only	
M96	Disable Marking Mode	195	Inactive	OFF	Plasma Only	
M97	Double Velocity	197	Inactive	OFF	Plasma Only, used for faster lead-outs.	
M98	Turn off Z-Tracking, then plasma arc.	198	Inactive	OFF	Plasma Only. Used to turn off the arc before the end of the contour.	
M99	Exit CNC Interpreter	N/A	N/A	N/A		

General Post Processor Notes

- Z axis parameters
  - Material Surface (Z = 0.0)
  - Tool Lift (-Z) (Z= -0.75 would raise the cutter .75 inches above the material.)
  - Depth (+Z) (Z=1.5 would lower the cutter 1.5 inches into the material.)
- **G00 C[tool change text]**
  - This command will display the text that follows the "C" on the keypad display. It can be used to perform tool changes on machines that do not have automatic tool changers.
  - The user will have that ability to surface the new tool when this command is called.
  - Do not use this command on systems that do have an automatic tool changer.
  - The keypad pad display has 2 lines, 20 characters each line, 40 characters total.. To provide the user with the best possible visual display spaces can be used to force some of the characters to the next line.
  - Example: **G00 CInsert .25 inch end mill** will prompt the user with the following text: "Insert .25 inch end mill".
- When creating a G-Code file from your post processor, the file must end with a **.cnc** or **.anc** file extension. This is important so the MultiCams DNC program knows that the file is a G-Code file.
- **File comments:** "//" is use for a comment example: // This line is a comment.
- Multiple G codes or M codes cannot be put on the same line.