CNC LASER MACHINE (AMNC-F)

PROGRAMMING MANUAL



LASER-AMNC-F PRO-E01-200406

PREFACE This manual describes the programming procedures for the laser machine. To increase the cutting efficiency of the laser machine, read the manual carefully before creating programs.

(For operating the laser machine, refer to the separate Operator's Manual.)

Programming Manual: CNC Laser Machine (AMNC-F) © 2004 by AMADA CO., LTD.

No part of this publication may be photocopied or otherwise reproduced without the prior written permission of AMADA CO., LTD.

CONTENTS

Part I

GENERA	L MACHINE COMMANDS	
G-Code li	sting	•••
M-Code li	sting	•••
Machine I	ayout	•••
Coordinat	es and dimensions	• • •
G20	Select INCH coordinates	• •
G21	Select METRIC coordinates	•••
G90	Absolute programming	•••
G91	Incremental programming	•••
G92	Establishing coordinate system	•••
G93	Origin point offset	•••
G120	Measurement probe (for LC-0)	•••
Motion ins	structions	•••
G00	Rapid traverse	•••
G01	Straight line motion	•••
G02	Circular arc CW	•••
G03	Circular arc CCW	•••
G09	Exact stop	••
G61	Exact stop check mode	••
G64	Contour cutting mode	••
G160	Space arc interpolation (for LC- θ)	••
General		••
0 P	rogram numbers	••
F F	eedrate code	••
DC	ffset code	••
N S	equence numbers	••
; Er	d of block	••
/ Blo	ock skip	••
(Com	ments)	••
G04		••
G25,	G27 Programmed repositioning (for LC- α)	••
G31	Assist gas selection	••
G50	Home return	••
G77	Measurement probe coordinate rotation (for LC- θ)	••
Laser bea	m compensation	••
- · ·	· · ·	

(Continued on next page.)

G41 Laser beam compensation-left I-22
G42 Laser beam compensation-right I-23
Laser control I-24
G24 Piercing mode I-24
M100 Laser mode ON I-24
M101 Laser mode OFF I-24
M102 Material designation I-25
M103 Start cutting mode I-25
M104 Cutting mode cancel I-25
M722, M723, M727 Tracking sensor calibration I-25
M758 Beam ON I-26
E1E10 Cut condition select I-26
E101E103 Pierce condition select I-26
E201E205 Edge condition select
Cutting parameter database I-28
U, V, W macro functions I-29
Macro number usage I-29
Macro memory (U, V) I-29
Macro recall (W) I-30
Nested macros I-31
Multiple part processing I-33
G98 Multiple part setup I-33
To cancel G98I-34
G75, G76 Multiple macro recall I-35
Multiple part example I-37
Multiple part processing on subcarriage side
of FO machine I-39
General M-codes I-42
M00 Program stop I-42
M02 Program end I-42
M30 Program end, return to start of program I-42
M80, M81 Work chute open/close (for LC- α) <i>I-42</i>
M96 Call subprogram I-43
M97 End of subprogram I-43
M99 End of subprogram (for FO) I-43
M150, M151, M152 Queue code (for FO) I-44
M180 Cycle work chute (for LC- α) <i>I-44</i>
Special I-45
G32, G33 Z-axis tracking sensor I-45
G65 Subprogram call (for FO) I-45
G95 Call program with parameters I-45

G96 Modal program call I-4	16
G97 Modal program call cancel I-4	16
G107 Pipe Interpolation I-4	<i>46</i>
G121, G122 HS-Edge detection I-4	1 6
G130 Axes retract I-4	17
G140, G141, G149 OVS I-4	17
G150 Scaling/Coordinate rotation	18
G161, G162 Space corner radius insertion (for LC-0) I-4	19
G163 3D coordinate conversion (for LC-θ) I-4	19
G164 3D coordinate conversion cancel (for LC-θ) I-4	19
G165 3D conversion (for LC-θ)	19
G166 3D conversion cancel (for LC-θ) <i>I-4</i>	19
G173 U-axis length compensation (for LC-0) I-4	19
M720, M721 Sensor ON/OFF (for LC-θ)	50
Loader control 1-5	51
G10 Pallet unload (for LC- β)	51
M10, M11 Workpiece clamp/release (for LC- α) <i>I-5</i>	51
M20 – M29 Detectable material thickenss (for LC- α) <i>I-5</i>	51
M33 Pallet load (for LC- β , FO)	
/Workpiece load (for LC- α) I-5	51
M34 Pallet unload (for LC- β)	51
M55 Cancel mirror image (for LC-β)	52
M65 Stock function (for LC- α)	52
M707, M772 – M774 Pallet change I-5	52
M790, M791 Pallet set (for LC- β , FO)	52
M792, M793 Pallet set pin (for LC- β , FO) <i>I-5</i>	52

Part II HOLES AND PATTERNS

G-codes fo	r holes and patterns	II-2
Standa	ard holes	<i>II-</i> 2
Standa	ard patterns	<i>II-</i> 2
G-codes fo	r standard holes	
G111 (with S	Square/Rectangle Square/Radius/Chamfered corners)	
G112	Round/Obround	
G113	Single D/Double D	
G114 (with S	Polygon Square/Radius/Chamfered corners)	II-10
G115	Arc slot (Radius ends)	II-12
		<i>.</i>

(Continued on next page.)

G116	Arc slot (Flat ends)	
G-codes fo	r standard patterns	II-16
Genera	al format of pattern call	II-16
G126	Bolt hole circle	II-17
G128	Line at angle	II-18
G129	Arc	II-19
G136	Grid- X	II-20
G137	Grid- Y	II-21

Part I General Machine Commands

G-C	ode lis	sting	<i>I-4</i>
M-C	ode lis	sting	
Machine layout		ayout	1-7
Coo	Coordinates and dimensions		1-7
	G20	Select INCH coordinates	1-7
	G21	Select METRIC coordinates	
	G90	Absolute programming	
	G91	Incremental programming	1-8
	G92	Establishing coordinate system	1-8
	G93	Origin point offset	1-9
	G120	Measurement probe (for LC-θ)	1-9
Motion instructions		tructions	I-10
	G00	Rapid traverse	I-10
	G01	Straight line motion	<i>I-11</i>
	G02	Circular arc CW	I-12
	G03	Circular arc CCW	<i>I-14</i>
	G09	Exact stop	<i>I-</i> 16
	G61	Exact stop check mode	<i>I-</i> 16
	G64	Contour cutting mode	<i>I-</i> 16
	G160	Space arc interpolation (for LC-0)	I-17
Gen	eral		<i>I-</i> 18
	O Pi	rogram numbers	<i>I-</i> 18
	F Fe	edrate code	<i>I-</i> 18
	D Of	ffset code	<i>I-</i> 18

(Continued on next page.)

N Sequence numbers	I-18
; End of block	I-18
/ Block skip	I-19
(Comments)	I-19
G04 Dwell	I-19
G25, G27 Programmed repositioning (for LC- α)	I-20
G31 Assist gas selection	I-20
G50 Home return	I-21
G77 Measurement probe coordinate rotation (for LC- θ)	I-21
Laser beam compensation	I-22
G40 Laser beam compensation-cancel	I-22
G41 Laser beam compensation-left	I-22
G42 Laser beam compensation-right	1-23
Laser control	I-24
G24 Piercing mode	I-24
M100 Laser mode ON	I-24
M101 Laser mode OFF	I-24
M102 Material designation	I-25
M103 Start cutting mode	I-25
M104 Cutting mode cancel	I-25
M722, M723, M727 Tracking sensor calibration	I-25
M758 Beam ON	I-26
E1E10 Cut condition select	I-26
E101E103 Pierce condition select	I-26
E201E205 Edge condition select	I-26
Cutting parameter database	1-28
U, V, W macro functions	1-29
Macro number usage	1-29
Macro memory (U, V)	1-29
Macro recall (W)	1-30
Nested macros	1-31
Multiple part processing	1-33
G98 Multiple part setup	1-33
To cancel G98	1-34
G75, G76 Multiple macro recall	1-35
Multiple part example	1-37
Multiple part processing on subcarriage side	1.20
	1-39 1 40
M00 Program stop	1-42
M02 Program and	۲ <u>+۲</u> -۱
NOL 1 1091011 610	······································

M30 Program end, return to start of program	<i>I-</i> 42
M80, M81 Work chute open/close (for LC- α)	1-42
M96 Call subprogram	1-43
M97 End of subprogram	1-43
M99 End of subprogram (for FO)	1-43
M150, M151, M152 Queue code (for FO)	
M180 Cycle work chute (for LC- α)	
Special	l-45
G32, G33 Z-axis tracking sensor	1-45
G65 Subprogram call (for FO)	1-45
G95 Call program with parameters	1-45
G96 Modal program call	1-46
G97 Modal program call cancel	I-46
G107 Pipe Interpolation	I-46
G121, G122 HS-Edge detection	I-46
G130 Axes retract	1-47
G140, G141, G149 OVS	1-47
G150 Scaling/Coordinate rotation	l-48
G161, G162 Space corner radius insertion (for LC- θ)	1-49
G163 3D coordinate conversion (for LC-θ)	
G164 3D coordinate conversion cancel (for LC-θ)	
G165 3D conversion (for LC-θ)	1-49
G166 3D conversion cancel (for LC- θ)	
G173 U-axis length compensation (for LC- θ)	
M720, M721 Sensor ON/OFF (for LC-θ)	1-50
Loader control	1-51
G10 Pallet unload (for LC-β)	1-51
M10, M11 Workpiece clamp/release (for LC- α)	1-51
M20 – M29 Detectable material thickenss (for LC- α)	1-51
M33 Pallet load (for LC- β , FO)/Workpiece load (for LC- α)) <i>I-51</i>
M34 Pallet unload (for LC-β)	<i>I-51</i>
M55 Cancel mirror image (for $1 \text{ C-}\beta$)	1-52
M65 Stock function (for $1 \text{ C-}\alpha$)	1-52
M707 M772 - M774 Pallet change	<i>I-52</i>
M790 M791 Pallet set (for LC-R EO)	<i>I-52</i>
M702 M703 Pallet set nin (for $1 C_{-B} E_{-D}$)	
(0, -p, 1, 0)	

G-CODE LISTING

The machine is controlled by various G-codes and M-codes. A listing of G-codes follows. For information about M-codes, see page I-6.

CODE ·····GROUP
G00 ·····Motion
G01 ·····Motion
G02 ·····Motion
G03 ·····Motion
G04 ·····General
G09 ·····Motion
G10 ·····Loader
G20, G21····· Select Inch/Metric coordinates ····· Coordinates, dimensions
G24 ·····Laser
G25, G27 (for LC- α) ···· Programmed repositioning ····· General
G31 ·····General
G32 · · · · · · · Special
G33 · · · · · · · Special
G40 ······ Laser beam path compensation cancel ····· Laser beam compensation
G41 ····· Laser beam path compensation
to LEFT of path ····· Laser beam compensation
G42 ······ Laser beam path compensation
to RIGHT of path ····· Laser beam
G50 ····· General
G53 ······ Setting in machine coordinate system ····· Coordinates, dimension
G61 ····· Motion
G64 ······ Motion
G65 (for FO) · · · · · · · · · · · · · · · · Subprogram call · · · · · Special
G75. G76······Multiple macro recall ·····Multiple
$G77$ (for L C- θ) Measurement probe coordinate rotation · · · · · General
G90 ······Coordinates dimension
G91 ·····Coordinates, dimension
G92 ······Coordinates, dimension
G93 ······Coordinates dimension
G95 ······ Call Program with parameters ····· Special
G96 ······Special
G97 ····· Special
G98 ······Multiple part setup ·····Multiple
G107 ···· Pipe interpolation (for rotary table option) ····· Special
G111·····Hole
G112Hole

G113 · · · · · Hole
G114 · · · · · · Hole
G115 · · · · · · Hole
G116 · · · · · · Hole
G120 (for LC-0) · · · · · · · · Measurement probe · · · · · Coordinates, dimensions
G121, G122 · · · · · · · · · · · · HS-edge detection · · · · · Special
G126 · · · · · · Pattern
G128 · · · · · · Pattern
G129 · · · · · · Pattern
G130 · · · · · · Special
G136 · · · · · · Pattern
G137 · · · · · · Pattern
G140 · · · · · · Special
G141 · · · · · · Special
G149 · · · · · · Special
G150 · · · · · · Special
G160 (for LC- θ) · · · · · · · · · Space arc interpolation · · · · · Motion
G161 · · · · · · Space corner radius insertion · · · · · Special
G162 · · · · · · Space corner radius cancel · · · · · Special
G163 · · · · · · Special
G164 · · · · · · · · 3D coordinate conversion cancel · · · · · Special
G165 · · · · · · Special
G166 · · · · · · Special
G173 · · · · · · Special

M-CODE LISTING

M-CODE ·····GROUP
M00 ····· General M-code
M02 ····· General M-code
M10, M11 (for LC- α) ····· Workpiece clamp/release ····· Special
M20 – M29 (for LC- α) Detectable material thickness · · · · · Special
M30 ····· Program end, return to start of program ····· General M-code
M33 (for LC- β /LC- α) · · · Pallet load/ Workpiece load · · · · · Loader
M34 (for LC-β) ·····Loader
M55 (for LC- β) · · · · · · · · Cancel mirror image · · · · · Loader
M65 (for LC- α) · · · · · · · · · · · · · Stock function · · · · · Loader
M80, M81 (for LC- α) · · · · · · Work chute open/close · · · · · General M-code
M96 ····· General M-code
M97 ····· General M-code
M99 (for FO) · · · · · · · End of subprogram · · · · · General M-code
M100 ·····Laser mode ON ·····Laser
M101 · · · · · · Laser mode OFF · · · · Laser
M102 · · · · · · Laser
M103 ····· Pierce material, start cutting mode ····· Laser
M104 ·····Laser
M150, M151, M152 (for FO)····· Queue code ····· General M-code
M180 (for LC- α) · · · · · · · Cycle work chute · · · · · General M-code
M707, M772 – M774 · · · · · · · Pallet change · · · · Loader
M720, M721 ·····Laser
M722, M723, M727 ···· Tracking sensor calibration ·····Laser
M758 · · · · · · Laser
M790, M791 (for LC- β) · · · · · · · · Pallet set · · · · Loader
M792, M793 (for LC- β) · · · · · · · Pallet set pin · · · · Loader

MACHINE LAYOUT

The LC- α machine and the LC- β machine are ahybrid system, with moving the material in the X-axis and moving the laser head in the Y-axis. The LC- α machine moves the material across a ball-transfer table, while the LC- β machine moves the pallet and material in the X-axis. The LC- θ or FO machine moves the laser head in the X-axis and Y-axis.

With the axes at the reference positions, the laser head is at the X+ and Y+ corner of the working area for the LC- α , - β , and - θ machines and is at the X+ and Y– corner of the working area for the FO machine.

COORDINATES AND DIMENSIONS

The NC used on these machines accepts information within certain ranges of values. The following table lists allowable values for various uses.

Numeric formats/allowable range of values

<u>Items</u>	<u>Metric</u>	Inch
X, Y, Z	+/- 99999.9999	+/- 9999.9999
G	1 to 9999	1 to 9999
N	0 to 99999	1 to 99999
0	0 to 9999	0 to 9999
R, I, J	+/- 99999.9999	+/- 9999.9999
Μ	1 to 999	0 to 999
X (as parameter)	.001 to 9999.999	.001 to 9999.999
Р	1 to 9999	1 to 9999

G20 Select INCH coordinates

May be used in MDI, or at the beginning of a program on a line by itself. After changing coordinate system, G92 must be re-set. This may be done by re-referencing the machine (using RETRACT mode) or by using the G92 or G130 instruction.

NOTE

In a program, must be followed by either a G92 statement for INCH coordinates, or a G130 instruction.

G21 Select METRIC coordinates

May be used in MDI, or at the beginning of a program on a line by itself. After changing coordinate system, G92 must be re-set. This may be done by re-referencing the machine (using RETRACT mode) or by using the G92 or G130 instruction.

G90 Absolute programming

When G90 is commanded, all coordinates in the program refer to current program origin or to the absolute origin point. G90 is MODAL and remains effective until G91 is commanded.

G91 Incremental programming

When G91 is commanded, all coordinates in the program are incremental distances from the previous coordinate.

G91 is MODAL and remains effective until G90 is commanded.

G92 Establishing coordinate system

The G92 command is optional, unless the system has been switched between INCH and METRIC. Once the machine has been powered up and referenced, the standard coordinate system is ready to use.

The G92 instruction may be used to establish an absolute origin point for programming.

The usual (default) absolute origin point for the X and Y axes corresponds to the corner of the sheet of material closest to the junction of the work clamps and the X-gauge block.

When the machine has been referenced and is at "home" reference position, the NC's position display (FUNC+POS keys) displays values which may be used in the G92 statement for the active units system (Inch/MM).

The following charts list dimensions for some common machines. If your machine does not appear here or the numbers do not seem to "match up", confirm with AMADA the correct values for your machine and write them in below.

NOTE

In a program, must be followed by either a G92 statement for METRIC coordinates, or a G130 instruction.

NOTE

[•] The G92 command must be immediately followed by the appropriate X, Y, and Z values, all on the same block of information.

Machine Type	X axis mm {in}	Y axis mm {in}	Z axis mm {in}
LC-1212 α	1270 {50.000}	1270 {50.000}	300 {11.8110}
LC-2415 α	2520 {99.2126}	1550 {61.0236}	300 {11.8110}
LC-2412 β	2520 {99.2126}	1270 {50.000}	300 {11.8110}
LC-3015 β	3070 {120.8661}	1550 {61.0236}	300 {11.8110}
LC-3015 θ	3050 {120.0787}	1530 {60.2362}	700 {27.5590}
FO2412	2520 {99.2126}	1270 {50.000}	200 {7.8740}
FO3015	3070 {120.8661}	1550 {61.0236}	200 {7.8740}

G93 Origin point offset

The G93 command establishes a reference origin point, relative to the absolute origin point, anywhere within the limits set by G92. This is done for ease of programming.

G93 X Y Z;

X...X-offset Y...Y-offset Z...Z-offset (normally zero)

Example

G93 X0.2 Y12.0 Z0;

Shifts the part-program reference point 0.2 inches in the plus-X direction and 12.0 inches in the plus-Y direction from the absolute origin point or current reference established by a G98 multiple part instruction (see page I-33).

To cancel the origin point offset:

G93 X0 Y0 Z0;

NOTE

- The G93 command must be immediately followed by the appropriate X, Y, and Z values, all on the same block of information.
- When programming multiple parts using G98, the G93 refers to each part origin as set by G98.
- If G91 (incremental coordinates) is effective when G93 is commanded, it becomes an incremental offset from the previous G93. Otherwise it replaces the previous G93.
- When using the cutting database, the system automatically corrects for material thickness. Unless cutting formed materials or not using the cutting database, use Z0.

G120 Measurement probe (for LC-θ)

Compensates the machine coordinate system and program origin point by using the optional measurement probe. For details, refer to the Operator's Manual.

MOTION INSTRUCTIONS

Motion instructions belong to two groups: rapid traverse, and contouring rapid traverse (G00) is strictly for positioning the material to a particular location. Contouring instructions (G01, G02, and G03) are used to move the material through a particular path under the laser head at a particular speed (feedrate).

The system defaults to absolute coordinates programming.

G00 Rapid traverse

This command is for positioning. It moves the table and laser head to the designated X, Y axis location at the current traverse speed. (default is maximum speed)

G00 X_Y_;

The Z-axis may also be positioned, but not on the same block as with X, Y axes.

Only axes included in the command are actually moved.

- The laser beam is OFF when G00 is active.
- Each axis moves independently, so the material path is usually not a straight line.
- G00 forces an in-position check at the commanded end point. This may be used to force a sharp corner during processing. (Use G00 on separate line.)
- Maximum rapid traverse speed is 40 m/min (1575 ipm). The RATE buttons on the CNC control panel can reduce travel speed to 50% or 25% of this.
- G00 is MODAL: Once commanded, it stays in effect until a G01, G02, or G03 is commanded.
- Absolute/incremental programming is available by G90/G91.



G01 Straight line motion

Moves the material from current position to commanded location via a straight line. Feedrate, assist gas selection, laser power, pulse rate, etc. are determined by the active material and table selection (M102, En) and by active operator overrides.

G01 X_Y_;

X...X-coordinate (mm or in.)

Y...Y-axis coordinate (mm or in.)

NOTE

- G01 is MODAL: once commanded, it remains effective until a G00, G02, or G03 is commanded.
- The machine is capable of moving all three axes at the same time, in this mode. Absolute/Incremental programming is available by G90/G91.
- A feedrate must be specified for G01, G02, G03. This is normally done by M102 and Enn, but may also be done using an Fnnnn feedrate command.
- During machine operation, feedrate may be overridden from 0% to 255% in 1% steps from the operator panel.
- The feedrate of the optional subcarriage of the FO machine is set by an NC parameter.

Example

G90 G00 X11.0 Y20.0;

```
G91 G01 X7.0 Y4.0;
```



G02 Circular arc CW

Moves the material from current position to commanded location via a clockwise arc at a commanded radius and feedrate.

G02 X Y R (or I J);

- X... X-coordinate (mm or in.)
- Y... Y-axis coordinate (mm or in.)
- R... Radius of arc (negative value creates an arc > 180°) (can use either R or I, J in instruction)
- I... Distance in the X-direction from the staring point to the arc center.
- J... Distance in the Y-direction from the staring point to the arc center.

Example of R format

G90 G02 X.5 Y25.0 R2.5;

Example of I, J format

G90 G02 X.5 Y25.0 I.5 J2.449;

- G02 is MODAL: once commanded, it remains effective until a G00, G01, or G03 is commanded.
- The parameter "R" has priority over "I" and/or "J", when used on the same line.
- The radius R (or that computed from I, J) must be non-zero.
- Absolute/incremental programming available by G90/G91 only affects the end point. The I, J values are always incremental from arc starting point.
- If the angle of the arc is greater than 180 degrees, the R value must be negative.
- The machine is capable of moving only two axes at the same time, in this mode.
- To cut a full circle, I and J must be used, rather than R.
- A feedrate must be specified for G01, G02, G03. This is normally done by M102 and En, but may also be done using an Fnnnn feedrate command.
- During machine operation, feedrate may be overridden from 0% to 255% in 1% steps from the operator panel.





G02 I-1.414 J1.414;

 $cut\ circle\ with\ 2"\ radius$

G03 Circular arc CCW

Moves the material from current position to commanded location via a counter-clockwise arc.

This command is used to cut material in a counter-clockwise arc at a commanded feedrate and radius. The X and Y values specified determine the endpoint of the cut. The R value specified determines the radius of the cut.

G03 X__ Y__ R__ (or I__ J__);

- X... X-coordinate (mm or in.)
- Y... Y-axis coordinate (mm or in.)
- R... Radius of arc (negative value creates an arc > 180°) (can use either R or I, J in instruction)
- I... Distance in the X-direction from the starting point to the arc center.
- J... Distance in the Y-direction from the starting point to the arc center.

- G03 is MODAL: once commanded, it remains effective until a G00, G01, or G02 is commanded.
- The parameter "R" has priority over "I" and/or "J", when used on the same line.
- The radius R (or that computed from I, J) must be non-zero.
- Absolute/incremental programming available by G90/G91 only affects the end point. The I, j values are always incremental from arc starting point.
- If the angle of the arc is greater than 180 degrees, the R value must be negative.
- The machine is capable of moving only two axes at the same time, in this mode.
- To cut a full circle, I and J must be used, rather than R.
- A feedrate must be specified for G01, G02, G03. This is normally done by M102 and En, but may also be done using an Fnnnn feedrate command.
- During machine operation, feedrate may be overridden from 0% to 255% in 1% steps from the operator panel.





G09 Exact stop

A command effective for a specified block only. Axis travel is decelerated at the ending point of the block and checked for in-position. The next block is then executed.

G09 (G01 X_Y_);

The command in parentheses may be G02 or G03.

NOTE

G61 Exact stop check mode

This command requires the machine to stop and wait for verification of each programmed position before moving toward the next programmed position.

NOTE

- G61 must be canceled before cutting any blended radii or using any of the standard hole commands (G111 to 115) either singly or in patterns.
- Once G61 is commanded, it will stay in effect until a G64 is commanded.

G64 Contour cutting mode

This is the default cutting mode for the machine. No position verification is required prior to movement towards the next programmed position. This mode is in effect until changed by a G61 command.

[•] The in-position check refers to the check made to see if the axis travel has reached the specified position (within the range set by a parameter).

G160 Space arc interpolation (for LC- θ)

G160 X<u>x1</u> Y<u>y1</u> Z<u>z1</u> V<u>a1</u> U<u>b1;</u> X<u>x2</u> Y<u>y2</u> Z<u>z2</u> V<u>a2</u> U<u>b2;</u>

The first block indicates the center point of the arc, and the second block indicates the ending point of the arc.

G160 is MODAL and remains effective until G00, G01, G02 or G03 is commanded.



When the center point and ending point are commanded, the arc to the ending point is obtained.

When the commands for the V- and U-axes are omitted, the nozzle moves in that attitude. When the ending point is not commanded and another code (e.g., G01) is commanded, the conditions for the arc are not met. In this case, the nozzle moves with the path from the starting point to the center point straightly interpolated.

When the center point is omitted, the attitude of the nozzle is automatically controlled according to the radius of the arc from the starting point to the ending point. When another space arc interpolation is commanded, the ending point becomes the starting point of the next arc.

GENERAL

O Program numbers

Each program must be assigned a program number. This number is used to separate the 200 different programs that can be stored in memory at one time. The program number must begin with the letter "O".



- Any number from 0 to 8999 can be used.
- Program 0 (zero) should be left vacant, as it can easily be overwritten during certain extended edit procedures.
- Programs numbered 8000 to 8999 can be protected by setting a parameter.

F Feedrate code

Cutting feedrates are normally specified by using M102 to select material type and thickness, and using E1-E9 to fine-tune or select for type of contour or detail. The F-code may be used to override a standard feedrate, or for material for which an entry does not exist in the database.

NOTE

- In Inch mode, the feedrate is in inches per minute.
- In Metric mode, the feedrate is in millimeters per minute.
- An F code is required only when the M102 functions are not used.

D Offset code

These codes are not used on the LC- α , β machine. Instead, laser beam offset amounts are kept in the cutting parameter database. See the section on laser beam compensation, beginning page I-22, and the section on standard holes, in Part II.

N Sequence numbers

Instruction blocks in a program may be marked or labeled using sequence numbers. When used, a sequence number must be the first address in the program block. The valid numeric range is from 1 to 99999. They do not need to be in numeric sequence.

; End of block

This symbol is used to separate one block of information from another.

/ Block skip

If the block skip button is illuminated, any block of information with this symbol at its beginning will be ignored.

(Comments)

Comments may be placed in a program by enclosing them in parentheses.

If a comment is placed on the first line after the program number, it will be displayed in the program directory listing of the CNC (machine control). Comments should not be mixed into the middle of program lines. Place each comment at the end of a program line or on a separate line.

O1234 (SAMPLE COMMENT);

(THIS COMMENT IS ON A LINE BY ITSELF);

If programming off-line, make sure to use all capitals for comments and instructions.

G04 Dwell

The dwell function stops the machine for a specified period of time, in seconds.

G04 Xnnnn;

Where .001<=*nnnn*<=99999.999

- CAUTION: Use of G04 between contouring motion instructions (G01, G02, G03) cancels laser beam compensation.
- When the time period is over, machine operation continues. Use extreme caution when using the dwell function.
- The minimum dwell value is 0.001 second. This is equivalent to G04 with no "X" value.

G25, G27 Programmed repositioning (for LC-a)

There may be times when you need to process sheets of material longer than the X-axis travel of the machine. This can be done by using the command G27, which causes the machine to release the worksheet, move the work clamps to a new position, and re-grip the material. This is done without loss of registration of the worksheet.

G25 does the same thing as G27 on this machine. It does not move the clamps away from the workpiece as it does on other machines. G25 is provided for program compatibility only.

An M104 and (if laser beam comp is active, G00 G40) must be commanded prior to the reposition.

Example

G00 G40 X29.0 Y15.0;Cancel laser beam compensation move to suitable location on sheeG27 X28.0;Reposition 28"	M104;	Cancel "cutting mode"
G27 X28.0; Reposition 28"	G00 G40 X29.0 Y15.0;	Cancel laser beam compensation, move to suitable location on sheet
	G27 X28.0;	Reposition 28"

Note the locations of the repositioning pads for your machine, and make sure to position the workpiece so that the pads are on the sheet when repositioning.

NOTE

• No other information can be on the G25 or G27 line except the X dimension.

G31 Assist gas selection

When there is NC assist gas control

G31 P__ T__;

The type and pressure of the assist gas are commanded.

A "P code" is used to specify the type of the assist gas.

The type of the assist gas is the same as indicated by the "gas type" number in the processing condition file.

Machine without NC assist gas control

- 1: Low-pressure oxygen
- 2: Medium-pressure oxygen
- 3: High-pressure oxygen
- 4: Nitrogen
- 5: Air
- 6: Easy cut (optional)
- 7: High-pressure assist gas (optional)

[•] The Work Holders must have solid material underneath them to insure a good hold on the worksheet. If necessary, you can move the material with a G00 prior to the reposition.

Machine with NC assist gas control

- 1: Low-pressure oxygen
- 2: Medium-pressure oxygen
- 3: High-pressure oxygen
- 4: Nitrogen
- 5: Air
- 6: Easy cut (optional)
- 7: High-pressure assist gas (optional)
- A "T code" is used to specify the pressure of the assist gas.
- T = Pressure setting $(1 = 0.01 \text{ MPa} \{0.1 \text{ kgf/cm}^2\})$

When the machine is not equipped with the NC assist gas control, pressure control is disabled.

3D processing **G31 L__;** G31 L0: Assist gas stopped G31 L1: Cutting assist gas discharged G31 L2: Piercing assist gas discharged

G50 Home return

The command G50 causes all axes to return to the Home Origin position and ends the program. The G92 values are reset to default values.

This command also cancels "Cutting mode", laser beam compensation, coordinate rotation, and scaling.

NOTE

G77 Measurement probe coordinate rotation (for LC- θ)

Rotates the coordinate system to suit the material measured using the optional measurement probe.

[•] No other command can be on the G50 line.

[•] If G50 is used in a program, M30 is not needed.

LASER BEAM COMPENSATION

Laser beam compensation is used to correct for the width of cut (kerf width) when laser contouring. The part program is normally written to the print dimensions, and laser beam compensation is used to correct for the width of the "cutting tool". (The offset amount should be 1/2 the total width of the cut.)

The functions use an offset amount held in the cutting parameter database. Each entry in a material type/thickness table (selected by M102 and "E" value) has an entry for laser beam offset.

The standard laser beam offset table also exists in the NC. Those entries are selected by "D" codes. The "D" address is not normally needed.

G40 Laser beam compensation-cancel

This command is used to cancel any previously commanded G41 or G42.

This command must be used with a G00 or G01 instruction. It is not necessary to include X or Y arguments. Starting or canceling laser beam compensation with G02 or G03 is not permitted.

G00 G40;

G41 Laser beam compensation-left

This command will offset the beam to the left of the programmed direction of travel. This is done to compensate for the width of the cut.

G00 G41;

The offset amount is determined by the active material type/thickness selection (from M102) and active "E" value. These select the table entry in the cutting parameter database, which has the offset distance to use.

[•] This command should be called on the same line as the positioning move prior to the start of cutting.

[•] This command must be used with a G00 or G01 instruction. It is not necessary to include X or Y arguments. Starting or canceling laser beam compensation with G02 or G03 is not permitted.

Actual Compensated Beam Path

offset distance

Programmed Beam Path

G42 Laser beam compensation-right

This command will offset the beam to the right of the programmed direction of travel. This is done to compensate for the width of the cut.

G00 G42;

NOTE
 This command should be called on the same line as the positioning move prior to the start of cutting.
 This command must be used with a G00 or G01 instruction. It is not necessary to include X or Y arguments. Starting or canceling laser beam compensation with G02 or G03 is not permitted.
Programmed Beam Path
Actual Compensated Beam Path

LASER CONTROL

The laser is controlled using selections from a cutting parameter database, which has up to ten sets of feedrates, gas selection, piercing information, and other necessary data for successful cutting of each standard material type/thickness combination.

For new or non-standard material types or thicknesses, the user can create new data files either as copies from existing ones or from scratch. The user's data files reside in the NC along with the AMADA cutting data, and are handled and used exactly the same as the ones that came from AMADA.

Only one material name can be active at a time – active material name is used for cutting, piercing, edge control, and restart of interrupted cut.

M102 is used to select the material/thickness, and an "E" code (from 1 to 10) is used to select a set of parameters for that material. "E" codes are also used to override default pierce and edge selections.

G24 Piercing mode

Turns on the laser beam at the specified power, pulse condition, and time.

G24 S__ P__ Q__ R__ ;

S... Specifies the power.

- P... Specifies the pulse frequency.
- Q... Specifies the pulse duty.
- R... Specifies the laser beam on time.

NOTE

• Assist gas select (G31) must be commanded before G24.

M100 Laser mode ON

Opens the mechanical shutter and enables laser operations.

M101 Laser mode OFF

Closes the mechanical shutter and disables laser operations.

[•] Always use M101 before opening the work chute (LC - α only).

M102 Material designation

Use to select material type and thickness from cutting parameter database.

M102 (typennn);

Where type*nnn* must match a material table in the cutting parameter database. The default cutting parameters for that material type and thickness will then be used.

M103 Start cutting mode

M103 An; If A0 is present, no pierce is made.

If "A" is missing or has a non-zero value, a pierce is made according to the selected cutting data table.

The head is lowered to the material surface, and the material is pierced, using the routine called out in the cutting parameter database. In this mode, the laser beam will be active during any contouring move (G01, G02, G03), and inactive during any rapid-traverse move (G00). Use M104 to cancel this mode.

M104 Cutting mode cancel

Cancels M103 mode, turns laser beam off, and retracts head to safe height.

M104 M__Z__;

- M... Optional code (M00/M180^{*}) for feed-hold or work chute. *Applicable to LC- α machines
- Z... Incremental retract distance. Overrides value stored in the control.

(Z-retract height is stored in parameter "setting values")

Example

M104 M00 Z50;

Cancels M103 mode, retracts head 50.0 mm above part surface, and applies an M00. (wait for operator to press START)

M722, M723, M727 Tracking sensor calibration

Commanded in the calibration operation of the Z-axis tracking sensor.

- Usually, not directly commanded.
- M722: Calibration ON

Initiates calibration.

- M723: Calibration OFF
 - Terminates calibration.
- M727: Calibration position

Specifies calibration position.

M758 Beam ON

Discharges to turn on laser beam.

Used by the AMADA service engineer during maintenance.

Not directly used in an ordinary program.

The laser beam cannot be emitted unless the laser mode is selected.

The discharge can be terminated by ending the program.

M758: Beam ON

Initiates discharge and emits laser beam.

E1...E10 Cut condition select

Thereafter, an "E" value may be commanded (E1 .. E10), to select any of the ten entries in that particular material/thickness table.

E101...E103 Pierce condition select

Select among preset piercing condition for the selected material. If piercing is not selected in the program, the default pierce (set within the cutting database for that material) is used.

Example

M102 (SUS1.5);	
E3;	$select\ condition\ 3$
E102;	select piercing #103

E201...E205 Edge condition select

Edge condition really refers to handling of sharp corners. The "sharpness" of a corner is determined by the angle between the two line segments at the corner. If the angle is the same or smaller (sharper) than the setting in the active edge table, then the system will use the feedrate and laser settings in the edge table for that corner. (Feedrate, power, pulse parameters, etc. for the distance set in the table.)

Select among the preset edge conditions for the active material name. Selected by the part program.

Notes on Sharp Corners

G00 may be used on a line by itself (without X, Y, or Z) to force a sharp corner and create a brief dwell. The laser beam will be OFF during this dwell, which may help to cool the material being cut. However, this defeats the edge (corner) handling of the control. If using the edge/corner control features, watch out for extra G00 or EOB lines in the program.

The above method will defeat the NC's edge/corner control feature. (See page I-28.) When cutting a material which requires the edge/corner control, don't use this method. Instead, use exact stop check mode (G61) on the section which requires sharp corners.

Cutting parameter database

The cutting parameter database is used to control piercing and cutting parameters. It also provides for special handling of sharp corners and recovering from an interrupted cut.

Only one material name can be active at a time – active material name is used for cutting, piercing, and edge control.

The listing below is a brief summary of the items settable.

Piercing variables

assist gas selection Laser power: initial Laser pulse frequency: initial Laser duty cycle: initial Laser power: step Laser pulse frequency: step Laser duty cycle: step number of steps time of each step total allowable time

Cutting variables

Feedrate assist gas selection Laser power Laser pulse frequency Laser duty cycle cutter offset

Edge variables

angle feedrate Laser pulse frequency Laser pulse duty

Recover variables

Pierce info on restart distance speed frequency duty

Other

Cap sensor start height Z-axis retract height no end Z-axis retract height for M00 Z-axis retract height for chute Assist gas "ON" height focus base height

For information on setup and maintenance of the cutting parameter database, see the Operator's Manual.

U, V, W MACRO FUNCTIONS

These functions permit storing portions of an NC program in a "macro" and recalling them one or many times later in the same program. Numbers from 01 to 99 can be used.

Macro number usage

NUMBERS	PURPOSE	
01 to 59	memorize instructions and execute them at the same time.	
60 to 89	memorize instructions but do not execute them at the time of storage.	
90 to 99	memorize multiple macros as a group.	
Macro numbers 90 to 99 can only memorize commands that are inside other macro instructions.		
If the BLOCK SKIP (slash) code is instructed between U and V, no memorization can be performed while the BLOCK SKIP key is ON.		

Macro memory (U, V)

Unn and Vnn mark the beginning and end of a block of instructions to be memorized. Each Unn must be paired with a corresponding Vnn, with nn being a number 01 to 99.

The instructions M02, M03, and G50 are not permitted in U-V macro blocks.

Each Unn block must be closed with matching Vnn before another U may be commanded.



Macro recall (W)

Once a macro has been stored using Unn, Vnn, it may be recalled as many times as necessary using Wnn.

To recall one or several macros in a grid, use the G98/G75/G76 Multiple Part functions.

See the section on Multiple part processing (page I-33) for more information.

U90	Begin macro 90 will store macros 60, 61	
U60	Begin macro 60 definition	
:		
V60	End definition of macro 60	
U61	Begin macro 61	
:		
V61	End of macro 61 definition	
G112 X5. Y5. I.5	Since the G112 is not inside a UV macro, it will not be stored in macro 90. Instead, it will be executed immediately. Since this is prior to the G93, it may not produce the intended result.	
V90	End of macro definition	
G93 X0.5 Y12.0		
W90		
Nested Macros

One macro can call another macro. The W instruction may be memorized inside a U-V block. Macros can be nested up to threedeep. The following example shows nested macros. The first macro (60) has code to contour two holes. The second macro calls the first, then contours the periphery of the part. The third macro sets up the laser cutting information and coordinate system offset and calls the second macro. It then cancels cutting mode and laser mode and drops the part out the work chute. Notice that, since all macros are numbered 60 and above, they are only memorized (not executed) until the W62 command is executed near the bottom of the program.

This example only shows the nesting of U-V macros. This programming technique will not be optimum in all circumstances.

N01 G90 G92 X98.425 Y49.213: N02 U60; begin first macro N03 G00 X2.175 Y2.: position for first hole N04 G01 X2.375 Y2. E002: lead-in N05 G03 X2.375 Y2. I-.375 J0; cut circle N06 G00 X4.175 Y2.: position for second hole N07 G01 X4.375 Y2.; N08 G00; N09 G03 X4.375 Y2. I-.375 J0; N10 V60; end first macro N11 U61; begin second macro N12 W60: call first macro to cut holes N13 G00 X3.8 Y4.0; position for periphery N14 G61: use exact stop check for sharp corners N15 G01 X4. Y4.0 E3; N16 G01 X4. Y3.5: N17 G01 X0 Y3.5: N18 G64; cancel exact stop check mode for corner N19 G01 X0 Y.6: N20 G03 X.6 Y0 I.6 J0; N21 G61: use exact stop check for sharp corners N22 G01 X7. Y0; N23 G01 X7. Y.1; N24 G01 X8. Y.1: N25 G01 X8. Y3.9; N26 G01 X7, Y3.9: N27 G01 X7. Y4.; N28 G01 X4. Y4.; N29 V61; end of second macro

N30 U62;	begin third macro
N31 M100;	set up laser
N32 M102(SUS0.078);	
N33 M103;	
N34 W61;	call second macro, which calls first macro
N35 M104	cancel cutting mode
N36 M101;	cancel laser mode before using work
	chute!
N37 M180;	drop the part through the work chute
N38 V62;	end of third macro
N39 G93 X.25 Y4.5;	set coordinate offsets for part
N40 W62;	call and run the whole thing
N41 G50	





MULTIPLE PART PROCESSING

The multiple part functions are like those used on AMADA's NCTs, rather than on previous AMADA lasers. All portions of a part program to be called out as a multiple must be stored in U-V macros, NCT style. See page I-29 for U, V, W macro usage.

As on the NCTs, the machine can either run a single part for checking, the remainder of a sheet (where the first part has been cut already), or all parts specified by G98. This selection is done on the machine's "Operator Panel". On many machines, this is a "soft panel", which may be displayed on the NC display screen.

The grid of parts may also be started (or resumed) on any part in the grid. See the G75, G76 commands for further information.

Setting	Result
"First process"	cuts only the first piece
"Others process"	cuts remaining pieces (all but first piece)
"All process"	cuts entire sheet according to the G98 setup

G98 Multiple part setup

Sets up a grid for multiple part processing. Specifies global offset, increment between parts, and numbers of parts in each direction. If a part contains G93 offsets (see page I-9), they will refer to the local part offset each time it is recalled within the G98 grid.



- P... number of additional parts in the X direction (zero or positive integer)
- K... number of additional parts in the Y direction (zero or positive integer)

For multiple part processing, the part-cutting program code must be memorized in U-V type macros. This is the same as used on AMADA's NCTs, and unlike previous laser programming.

Since the G98 provides offsets for X and Y, the part may be programmed in any convenient fashion, and the G98 may be used to place the parts on the sheet. The part may also include one or more G93 offsets as desired.

G98 sets up origin, increments, and numbers of pieces. G75 or G76 determines the macro(s) to be called and the starting quadrant and direction of processing.

To cancel G98

Command G98 with X, Y values of zero to cancel a previous G98.



G98 Z0 Y0;

G75, G76 Multiple macro recall

These are used to recall a U-V macro in a grid. The pattern origin, increment values and quantities must be first set using G98.

G75 W__Q__(P__); Grid-X

G76 W__ Q__ (P__); Grid-Y

- W...Specifies macro number to recall
- Q... Specifies a starting quadrant, 1-4
- P... (optional) Specifies part number to start (resume) with.
- When using G98 with P0, only use Q1 or Q3.
- When using G98 with K0, only use Q1, Q2.

NOTE

The above notes refer to the "P" value in the G98 instruction, not the "P" value which may or may not be present in the G75 or G76.



G75 W__ Q__ P__; (Grid-X)

Recalls a macro according to the current G98 grid settings. For single horizontal row of parts, use G98 with K0, G75 with Q1 or Q2.



G76 W__ Q__ P__; (Grid-Y)

Recalls a macro according to the current G98 grid settings. For single vertical row of parts, use G98 with P0, G76 with Q1 or Q3.



Multiple part example

The following is a part-program using G98 to cut a grid of pieces out of a sheet. In this case, instead of using G112, the holes were "hard-coded".

Note that "spaces" have been added between instruction words for readability. This should not be done in a program to be used on the machine.

(FORMULT) (36. X 36., .078 SUS) (4., 32.) M102 (SUS0.078) G90 G92 X98.425 Y49.213; G98 X1 Y4.5 I8.5 J4.3 P3 K6; U60: begin macro M100; G00 X2.175 Y2.; position for first hole M103; G01 X2.375 Y2. E002; G00; G03 X2.375 Y2. I-.375 J0; G00: G00 X4.175 Y2.; position for second hole G01 X4.375 Y2.; G00; G03 X4.375 Y2. I-.375 J0; G00; G00 X3.8 Y4.; position for periphery G01 X4. Y4.; G00; G01 X4.Y3.5; G00; G01 X0 Y3.5; G00; G01 X0 Y.6; G03 X.6 Y0 I.6 J0; G00; G01 X7. Y0; G00; G01 X7. Y.1; G00: G01 X8. Y.1; G00;

G01 X8. Y3.9; G00; G01 X7. Y3.9; G00; G01 X7. Y4.; G00; G01 X4. Y4.; M104; M180; V60; G75 W60 Q4; M101; G50; %



[1	Γ
	0 0		
00	00		

Multiple part processing on subcarriage side of FO machine

Multiple part processing on the subcarriage side of the FO machine is commanded by methods different from those for multiple part processing on the main carriage side.

Multiple part processing is set for two or more parts of the same type.

A subprogram created for multiple part processing is called by G65 and is executed by setting relevant values.

A part program is an arrangement of subprograms. The last program of the part program must be changed to "M99;".

G65 P9200 X Y_; Set reference point for multiple-part processing

G65 P9097 I__ J__ Q__ K__ H__ B__ A__ ; Set multiple-part processing

- P9200... Subprogram for setting the reference point for multiple part processing
- X... Reference point in the X direction (lower left corner of the part arranged at the upper right (program coordinate origin point))
- Y... Reference point in the Y direction (lower left corner of the part arranged at the upper right (program coordinate origin point))
- P9097... Subprogram for setting the method of arranging multiple parts
- I... Pitch of parts in the X direction
- J... Pitch of parts in the Y direction
- Q... Number of parts in the X direction (including the part in the reference position)
- K... Number of parts in the Y direction (including the part in the reference position)
- H... Starting line (defaults to 1; refer to the next page)
- B... Starting row (defaults to 1; refer to the next page)
- A... Processing program number (subprogram number)

NOTE

- Specify X and Y in absolute values.
- Specify Q, K, H, and B in positive values.
- Determine the reference point and the pitch of parts by considering the between-part scrap skeleton width, scrap skeleton edge width, worksheet size, and other relevant factors.

Example of layout

Parts arranged in five vertical rows and six horizontal lines

B5	B4	B3	B2	B1	
[25]	[19]	[13]	[7]	[1]	H1
H1 B5	H1 B4	H1 B3	H1 B2	H1 B1	
[26]	[20]	[14]	[8]	[2]	H2
H2 B5	H2 B4	H2 B3	H2 B2	H2 B1	
[27]	[21]	[15]	[9]	[3]	H3
H3 B5	H3 B4	H3 B3	H3 B2	H3 B1	
[28]	[22]	[16]	[10]	[4]	H4
H4 B5	H4 B4	H4 B3	H4 B2	H4 B1	
[29]	[23]	[17]	[11]	[5]	H5
H5 B5	H5 B4	H5 B3	H5 B2	H5 B1	
[30]	[24]	[18]	[12]	[6]	H6
H6 B5	H6 B4	H6 B3	H6 B2	H6 B1	

G65 P9097 ~ Q<u>5</u> K<u>6</u> H___ B___ H___ ~

The parts are processed in the order of the numbers bracketed in the table above.

When interrupting and resuming multiple part processing, specify the resumption line and row at H and B, respectively.

Examples of main programs

Parts arranged in four vertical rows and three horizontal lines G92 G90 X3070 Y1550;

G65 P9200 X350 Y160;

G65 P9097 I110 J70 Q4 K3 H1 B1 A123;

G50;



Parts arranged in only one horizontal line G92 G90 X3070 Y1550;

G65 P9200 X370 Y20;

G65 P9097 I70 J0 Q6 K1 H1 B1 A2;

G50;



Example of part program (subprogram)

The part size must be specified by setting the coordinate system in the part program. If the program starts at the origin point, the parts cannot be properly arranged.

The last command of the program must be "M99;". If G50, M02, or M30 is the last command, the program is not assumed to have ended, and processing does not proceed any further.

G92 G90 X100 Y50;



M99;

GENERAL M-CODES

M00 Program stop

Halts program execution until the START button is pressed.Must be on line by itself, except for (optional) sequence number.Used to permit clearing scrap from table, etc. during program run.

M02 Program end

Program execution ends, NC returns to an initialized status. Does not return control to top of program. (cursor remains on line following M02)

M30 Program end, return to start of program

Ends program, returns cursor to beginning of program. Does not return machine to HOME position.

Use instead of G50 when you want to end a program. Without sending the machine home.

M80, M81 Work chute open/close (for LC-α)

The M80 instruction causes the work chute to open. The M81 instruction causes the work chute to close.

NOTE

The DWELL instruction (G04 X_{-}) is normally used to keep the chute open for desired period.

M80;

G04 X__; M81;

These instructions may be commanded by MDI.

The system must not have M100 active when opening the work chute. (command M101 first)

M96 Call subprogram

This is used to execute a separate program, then return to current program. The separate program may be a special shape or pattern, or any other sequence of instructions. The called program can also make subprogram calls.

The effect is similar to that of macro storage/recall (U, V, W)

M96 P_L_;

P... Program number to call

L... number of times to run (up to 9999)

The parameter L optional. When omitted, the named program is called and executed once. (same as L1)

M97 End of subprogram

Means "return to calling program and continue execution". Use at end of subprogram only. If you select a program ending with M97 and run it, each time it reaches the M97 instruction it will return to the top of the program and continue execution.

See figure below for example



M99 End of subprogram (for FO)

Means "return to calling program and continue execution". **M99;**

M150, M151, M152 Queue code (for FO)

M150;

M151;

M152;

Command queue codes in the main carriage and subcarriage programs to queue in the specified blocks.

Three types of queue codes from M150 to M152 can be used.



In the above case, the program that has a queue code executed first stops until each corresponding queue code is executed.

M180 Cycle work chute (for LC-α)

The M180 instruction causes the work chute to open, then close. Like using M80, M81 with a one-second delay.

NOTE

The system must not have M100 active when opening the work chute. (command M101 first)

SPECIAL

This section covers commands which are less-frequently used in partprogramming. Some of them apply to only one machine, or are used only with certain options.

G32, G33 Z-axis tracking sensor

Turn on and off the optional Z-axis tracking sensor. G32 : ON G33 : OFF

G65 Subprogram call (for FO)

Calls a subprogram. Mainly used in multiple part processing commands for the subcarriage of the FO machine. For multiple part processing, refer to "Multiple part processing on subcarriage side of FO machine".

G65 P__;

P... subprogram number to call

G95 Call program with parameters

Similar to sub-program call using M98 P____, except that parameters (information) can be passed to the program being called. Previous lasers and standard machining centers use G65 for this function. Not used with any of the standard holes or patterns, but available for customer use as needed. Refer to G65, G66, G67 in the FANUC Operator's Manual for information about passing parameters, etc.

G95 P___ {parameters} L__;

- P... program number to call
- L... number of times to repeat the called program

{parameters} depends entirely on program being called. Consult the FANUC Operator's Manual for macro programming.

NOTE

[•] Some systems use G65 for this function.

G96 Modal program call

Sets up a modal program call where the selected program can be executed repeatedly by either single moves or by (possibly) a standard pattern call.

Previous lasers and standard machining centers use G66 for this function. Not used with any of the standard holes or patterns, but available for customer use as needed. Refer to G65, G66, G67 in the FANUC Operator's Manual for information about passing parameters, etc.

G96 P____ {parameters} L__;

P... program number to call

{parameters} data to be passed to the program being called.

NOTE

• Some systems use G66 for this function.

G97 Modal program call cancel

Previous lasers and standard machining centers use G67 for this function. Any system which uses G65, G66 must also use G67.

G97;

Cancels any active G96. No parameters are needed/used.

Example of G96/G97

G96 P8002;	Set up modal program call
X2500 Y2500;	moves to position, then executes $program \ 8002$
X2700 Y2500;	moves to position, then executes $program \ 8002$
G97;	cancels modal program call

G107 Pipe interpolation

G107 IPr: Initiates the pipe interpolation mode (enables pipe interpolation).

G107 IP0: Terminates the pipe interpolation mode.

NOTE

Command G107 IPr and G107 IP0 in separate blocks.

G121, G122 HS-Edge detection

Use the optional material edge detection function using the Z-axis tracking sensor.

For details, refer to the HS-edge detection system Operator's Manual.

G130 Axes retract

Automatically returns the X-, Y-, and Z-axes to the origin.

G140, G141, G149 OVS

Commanded to use the function of measuring the material position using the OVS $\rm III$ option.

For details, refer to the OVS III Operator's Manual.

G150 Scaling/Coordinate rotation

May be used to change the size or orientation of all or some portion of a part-program. Cancelled by repositioning commands (G25/G27), pallet commands, or program end.

G150 X Y E A B R;

- X...Center point to scale around in the "X" axis
- Y...Center point to scale around in the "Y" axis
- E...Scaling ratio (1=1:1) (0.00001 to 9.99999)
- A...Center point to rotate around in the "X" axis
- B...Center point to rotate around in the "Y" axis
- R...Rotating angle (-360.000 to 360.000)

For scaling only, just X, Y, and E are required. For rotation only, just A, B, and R are required.

Scaling and rotation cancel: G150 Also cancelled by:

> repositioning (G25, G27) program end (G50, M02, M30) by pressing the RESET button.



G161, G162 Space corner radius insertion (for LC-θ)

 G161 G01;
 Space corner radius insertion mode

 X_ Y_ Z_ V_ U_ R_ F_;

 X_ Y_ Z_ V_ U_;

 R_;

 X_ Y_ Z_ V_ U_;

 G162;
 G161 mode cancel

An arc of the radius specified by R is automatically inserted at each corner formed by the rows of points in the space enclosed with G161 and G162.

When the command R for the radius of the arc to be inserted is specified singly, the corners between the preceding and succeeding blocks assume the radius commanded by R.

When the R command is specified following the coordinates of the specified point, R is inserted at the corner between the block and the succeeding block.

NOTE

• Command by G01 the movement of the table and laser head in the G161 mode. An alarm occurs when a G code in the 01 group other than G01 is commanded.

G163 3D coordinate conversion (for LC-θ)

Pasts a 2D shape at an arbitrary point in the space.

G164 3D coordinate conversion cancel (for LC- θ)

Cancels a 3D coordinate conversion command.

G165 3D conversion (for LC- θ)

Moves a 3D shape to an arbitrary position in the space.

G166 3D conversion cancel (for LC-θ)

Cancels a 3D conversion command.

G173 U-axis length compensation (for LC- θ)

Compensates the U-axis length.

M720, M721 Sensor ON/OFF (for LC-θ)

Turn on and off the W/Z-axis tracking sensor as programmed during processing.

Used to process over a cut hole, for example.

M720: Turns on the W/Z-axis tracking sensor.

M721: Turns off the W/Z-axis tracking sensor.

LOADER CONTROL

G10 Pallet unload (for LC-β)

Unloads pallet.

M10, M11 Workpiece clamp/release (for LC-α)

Command the operation of the workclamp and workholder during auto repositioning. Not used in an ordinary program.

M10 : Lowers the workholders and opens the workclamps.

M11 : Closes the workclamps and raises the workholders.

M20 – M29 Detectable material thickness (for LC-α)

Specify the material thickness range to be checked with the worksheet thickness detector when the LC- α machine is equipped with the optional loader/unloader (α loader).

M20 : Thickness detector disabled

- M21 : Thickness range of up to 0.4 mm
- M22 : Thickness range of 0.5 to 0.6 mm
- M23 : Thickness range of 0.7 to 0.9 mm
- M24 : Thickness range of 1.0 to 1.3 mm
- M25 : Thickness range of 1.4 to 1.8 mm
- M26 : Thickness range of 1.9 to 2.4 mm
- M27 : Thickness range of 2.5 to 3.2 mm
- M28 : Thickness range of 3.3 to 4.2 mm
- M29 : Thickness range of 4.3 to 6.2 mm

M33 Pallet load (for LC-β, FO)/Workpiece load (for LC-α)

Retracts axes, swaps pallets, sets up mirror image/coordinate system for pallet operation. (for LC- β , FO) Loads materials on the machine when the machine is equipped with the optional loader/unloader (α loader). (for LC- α)

M34 Pallet unload (for LC-β)

Used inside G10 function. Not for use in part-program.

M55 Cancel mirror image (for LC-β)

Used inside G10 function. Not for use in part-program.

M65 Stock function (for LC-α)

Commanded to use the function of stacking unloaded materials in two rows one after the other on the pallet when the LC- α machine is equipped with the optional loader/unloader (α loader) with autostorage.

M707, M772 – M774 Pallet change

M707: Changes the pallets A and B at the same time. (for LC- β , LC- θ , FO) M772: Changes the pallet A alone. (for LC- θ)

- M773: Changes the pallet B alone. (for LC- θ)
- M774: Changes the pallet C alone. (for LC- θ)

M790, M791 Pallet set (for LC-β, FO)

Used to lock and unlock the pallet when the LC- β or FO machine is equipped with the optional loader/unloader.

M790 : Pallet set lock

Clamps the pallet to the table.

M791 : Pallet set unlock Releases the pallet from the table.

M792, M793 Pallet set pin (for LC-β, FO)

Raises and lowers the pallet positioning pin when the LC- β or FO machine is equipped with the optional loader/unloader.

M792 : Pallet pin in

Raises the pallet positioning pin and inserts it into the pallet.

M793 : Pallet pin out Lowers the pallet positioning pin and removes it from the pallet.

Part II

Holes and Patterns

G-codes fo	or holes and patterns	II-2
Standa	ard holes	<i>II-2</i>
Standa	ard patterns	<i>II-2</i>
G-codes fo	or standard holes	II-3
G111	Square/Rectangle	11_4
G112	Round/Obround	
G113	Single D/Double D	<i>II-</i> 8
	G114 Polygon (with Square/Radius/Chamfered corners)	II-10
G115	Arc slot (Radius ends)	II-12
G116	Arc slot (Flat ends)	II-14
G-codes fo	or standard patterns	II-16
Gener	al format of pattern call	II-16
G126	Bolt hole circle	II-17
G128	Line at angle	II-18
G129	Arc	II-19
G136	Grid- X	II-20
G137	Grid- Y	II-21

G-CODES FOR HOLES AND PATTERNS

Standard holes

- G111 Square/Rectangle
- G112 Round/Obround
- G113 Single D/Double D
- G114 Polygon
- G115 Arc slot (Round ends)
- G116 Arc slot (Flat ends)

Standard patterns

- G126 Bolt holes circle
- G128 Line at angle
- G129 Arc
- G136 Grid pattern X
- G137 Grid pattern Y

G-codes for standard holes

These are custom G-codes to cut standard kinds of holes at any sizes and angle. Various options are available for each one, such as corner radius or chamfer, retaining tab, and so forth.

These G-codes may be used to cut holes one-at-a-time, or in standard patterns such as bolt hole circle, grid, etc.

To cut holes one-at-a-time, use the hole-cutting G-code with the X and Y values of the desired hole. (The reference point for these holes is usually the center–See each G-code for specifies).

To use any of the standard patterns, first command the hole-cutting Gcode without X, Y values. Then use the pattern command, and the holes will be cut according to that pattern.

When setting up for patterns, note the SIGN of the length of the hole: it determines whether the hole rotates along with an angled line or grid, and whether the hole rotates with an arc or bolt hole circle.

G111 Square/Rectangle (with Square/Radius/Chamfered corners)

G111 X Y I J Q K R C H M A;

X, Y for cutting single hole.

$G111 I_J_Q_K_R_C_H_M_A_;$

X, Y omitted, becomes macro setup for arc or other pattern G-code.

- X... X-center of hole to cut
- Y... Y-center of hole to cut
 - If making a single hole, be sure to use both X and Y values.
- I... Diameter/length: If negative, the hole will rotate around an arc or bolt hole circle pattern origin, or along the lean angle of a line-at-angle or grid pattern. I must be greater than J.
- J... Width: If omitted, a square hole is cut.
- K... Lean angle
- R... Corner radius: Must be less than one half of the "J" value.
- C... Chamfer: Must be less than one half of the "J" value.
- Q... Entrance cut: Specify length. If omitted, the entrance cut will start in the middle or .2" from the edge, whichever distance is less. If "Q" value is negative, the cut will begin on the outside.
- H... Tab/micro joint: Specify tab length. If omitted, no tab is allowed for.
- M... M00 Shutter close/head up/stop program

M180 - Shutter close/head up/chute open-close

A0 Piercing override: If A0, then no piercing is done. If A_ is absent or non-zero, piercing is performed according to current material selection and active "E" code.





Example of G111:

```
O6111 (EXAMPLE OF SQUARE/RECTANGLE);

G92 X_Y_Z_;

M102 (CRS.063);

M100;

E4;

G111 X2.502 Y3.513 I1.063 J.626 H.010; Rectangle at 0 degrees

with tab

G111 X1.25 Y1.25 I.75 K45; Square at 45 degrees

M101;

G50;
```



G112 X Y I J Q K H M A;

G112 I__J_Q_K_H_M_A_;

X, Y omitted, becomes macro setup for arc or other pattern G-code.

- X... X-center of hole to cut
- Y... Y-center of hole to cut

If making a single hole, be sure to use both X and Y values.

- I... Diameter/length: If negative, the hole will rotate around an arc or bolt hole circle pattern origin, or along the lean angle of a line-at-angle or grid pattern. I must be greater than J.
- J... Width: If omitted, a round hole is cut.
- K... Lean angle
- Q... Entrance cut: Specify length. If omitted, the entrance cut will start in the middle or .2" from the edge, whichever distance is less. If "Q" value is negative, the cut will begin on the outside.
- H... Tab/micro joint: Specify tab length. If omitted, no tab is allowed for.
- M... M00 Shutter close/head up/stop program
 - M180 Shutter close/head up/chute open-close
- A0 Piercing override: If A0, then no piercing is done. If A.. is absent or non-zero, piercing is performed according to current material selection and active "E" code.



Example of G112:

```
O6112 (EXAMPLE OF ROUND/OBROUND);

G92 X_Y_Z_;

M102 (CRS.063);

M100;

E4;

G112 X2.875 Y2.513 I1.063 J.626 H.010 K45;

1.

ob
```

1.063" x 0.626" obround at 45 degrees with .010" tab

G112 X1.25 Y1.25 I.75; 0.75" round M101; G50;



G113 X Y I J Q K H M A;

G113 I__ J__ Q__ K__ H__ M__ A__;

X, Y omitted, becomes macro setup for arc or other pattern G-code.

- X... X-center of hole to cut
- Y... Y-center of hole to cut

If making a single hole, be sure to use both X and Y values.

- I... Diameter/length: If negative, the hole will rotate around an arc or bolt hole circle pattern origin, or along the lean angle of a line-at-angle or grid pattern. I must be greater than J.
- J... Distance across the flats: For single flat, "J" value must be negative. For two flats, use a positive "J" value.
- K... Lean angle
- Q... Entrance cut: Specify length. If omitted, the entrance cut will start in the middle or .2" from the edge, whichever distance is less. If "Q" value is negative, the cut will begin on the outside.
- H... Tab/micro joint: Specify tab length. If omitted, no tab is allowed for.
- M... M00 Shutter close/head up/stop program

M180 - Shutter close/head up/chute open-close

A0 Piercing override: If A0, then no piercing is done. If A.. is absent or non-zero, piercing is performed according to current material selection and active "E" code.





Example of G113:

```
O6113 (EXAMPLE OF SINGLE-D DOUBLE-D);

G92 X_Y_Z_;

M102 (CRS.063);

M100;

E4;

G113 X2.5 Y2.5 I1.50 J.75 H.010 K45; 1.5" x 0.75" single-D at

45 degrees with .010"

tab

G113 X1.50 Y1.0 I.75 J-.625 K-30; 0.75" x .625 double-D at -30

degrees

M101;

G50;
```

,00,



G114 Polygon (with Square/Radius/Chamfered corners)

G114 X Y I J Q K R C H M A;

G114 I__ J__ Q__ K__ R__ C__ H__ M__ A__;

X, Y omitted, becomes macro setup for arc or other pattern G-code.

- X... X-center of hole to cut
- Y... Y-center of hole to cut

If making a single hole, be sure to use both X and Y values.

- I... Distance from center of polygon to flat edge. If negative, the hole will rotate around an arc or bolt hole circle pattern origin, or along the lean angle of a line-at-angle or grid pattern.
- J... Number of sides (there must be at least 3 sides).
- K... Lean angle
- Q... Entrance cut: Specify length. If omitted, the entrance cut will start in the middle or .2" from the edge, whichever distance is less. If "Q" value is negative, the cut will begin on the outside.
- R... Corner radius: Must be less than one half of the "I" value.
- C... Chamfer: Must be less than one half of the "I" value. NOTE: If a combination of chamfer and number of sides would cause the chamfers to overlap, an error will occur.
- H... Tab/micro joint: Specify tab length. If omitted, no tab is allowed for.
- M... M00 Shutter close/head up/stop program
 - M180 Shutter close/head up/chute open-close



-Q

Example of G114:

```
O6114 (EXAMPLE OF POLYGON);

G92 X_Y_Z_;

M102 (CRS.063);

M100;

E4;

G114 X2.5 Y2.5 I.50 J3 K45 R.1; Triangle at 45 degrees with .1"

corner radius

G114 X1.50 Y1.0 I.375 J6; Hexagon measuring .75" across flats

(.375" * 2)

M101;

G50;
```



G115 X Y I J Q K H M A;

G115 I__ J__ Q__ K__ H__ M__ A__;

X, Y omitted, becomes macro setup for arc or other pattern G-code.

- X... X-center of hole to cut
- Y... Y-center of hole to cut

If making a single hole, be sure to use both X and Y values.

- I... Arc angle: If negative, the hole will rotate around an arc or bolt hole circle pattern origin, or along the lean angle of a line-at-angle or grid pattern.
- J... Width of slot.
- K... Lean angle.
- R... Arc slot radius: Must be positive and specified.
- Q... Entrance cut: Specify length. If omitted, the entrance cut will start in the middle or .2" from the edge, whichever distance is less. If "Q" value is negative, the cut will begin on the outside.
- H... TAB/MICRO JOINT: Specify tab length. If omitted, no tab is allowed for.
- M... M00 Close shutter, raise head, stop program (wait for START push-button)

M180 - Close shutter, raise head, open and close workchute, continue program.

A0 Piercing override: If A0, then no piercing is done. If A_ is absent or non-zero, piercing is performed according to current material selection and active "E" code.





Example of G115:

```
O6114 (EXAMPLE OF ARC SLOT RADIUS ENDS);

G92 X_Y_Z_;

M102 (CRS.063);

M100;

E4;

G115 X2.5 Y2.5 I90 J.5 K45 R1.0; 90° Arc slot at 45°, .5" wide, 1"

radius.

G115 X1.50 Y1.0 I36 J.375 K180 R1.5; 36° Arc slot at 180°, .375"

wide, 1" radius

M101;

G50;
```



G116 X Y I J Q K H M A;

G116 I__ J__ Q__ K__ H__ M__ A__;

X, Y omitted, becomes macro setup for arc or other pattern G-code.

- X... X-center of hole to cut
- Y... Y-center of hole to cut

If making a single hole, be sure to use both X and Y values.

- I... Arc angle: If negative, the hole will rotate around an arc or bolt hole circle pattern origin, or along the lean angle of a line-at-angle or grid pattern.
- J... Width of slot.
- K... Lean angle.
- R... Arc slot radius: Must be positive and specified.
- Q... Entrance cut: specify length. If omitted, the entrance cut will start in the middle or .2" from the edge, whichever distance is less. If "Q" value is negative, the cut will begin on the outside.
- H... TAB/MICRO JOINT: Specify tab length. If omitted, no tab is allowed for.
- M... M00 Shutter close/head up/stop program

M180 - Shutter close/head up/chute open/close

A0 Piercing override: If A0, then no piercing is done. If A.. is absent or non-zero, piercing is performed according to current material selection and active "E" code.




Example of G116:

```
O6114 (EXAMPLE OF ARC SLOT W/FLAT ENDS);

G92 X_Y_Z_;

M102 (CRS.063);

M100;

E4;

G116 X3 Y3 I90 J.5 K45 R1 Q.1; 90° Arc slot at 45°, 1"

radius, .5" wide, .1" lead-in

G116 X1.50 Y1.0 I36 J.375 K180 R1.5; 36° Arc slot at 180°, .375"

wide, 1.5" radius,

default lead-in.

M101;
```

G50;



G-codes for standard patterns

These are custom G-codes to allow any of the hole-cutting G-codes to be called out in regular patterns. In each case, the hole-cutting G-code must be set up first, then the pattern can be called.

General format of pattern call

Where

G____ is G-code

X____ is X-location for the pattern origin.

Y____ is Y-location for the pattern origin.

I___, J___, etc. specify increment, etc. according to the particular pattern requirements.

Example

G112 I.5;

G126 X12.5 Y14.32 I4.01 J45.0 K3; (cuts three 1/2" holes)

G126 Bolt hole circle

G126 X_Y_I_J_K_;

- X... Pattern origin X
- Y... Pattern origin Y
- I... Radius of the bolt hole.
- J... Starting angle.
- K... Number of holes.



G128 X_Y_I_J_K_;

- X... Pattern origin X
- Y... Pattern origin Y
- I... Increment or spacing
- J... Lean angle: If omitted, pattern will cut parallel to the "X" axis
- K... Number of additional holes in the "X" direction (K5 means 6 holes total)



X-axis

G129 X Y I J F K;

- X... Pattern origin X
- Y... Pattern origin Y
- I... Radius of the arc.
- J... Starting angle.
- P... Incremental angle between holes Positive: counter-clockwise Negative: clockwise
- K... Number of holes.



G136 Grid-X

Cuts a grid of holes in X-direction. See illustration below for execution sequence. Use with any of the standard hole-cutting G-codes.

G136 X Y I J P K Q;

X... Pattern origin X

- Y... Pattern origin Y
- I... Increment or spacing in the "X" axis Positive: +X direction

Negative: -X direction

J... Increment or spacing in the "Y" axis

Positive: +Y direction

Negative: -Y direction

- P... Number of additional holes in the "X" direction
- K... Number of additional holes in the "Y" direction (must be specified)
- Q... Lean angle: If omitted, pattern will cut parallel to the "X" axis



G137 Grid-Y

Cuts a grid of holes in Y-direction. See illustration below for execution sequence. Use with any of the standard hole-cutting G-codes.

G136 X_Y_I_J_P_K_Q_;

- X... Pattern origin X
- Y... Pattern origin Y
- I... Increment or spacing in the "X" axis Positive: +X direction

Negative: -X direction

J... Increment or spacing in the "Y" axis

Positive: +Y direction

Negative: -Y direction

- P... Number of additional holes in the "X" direction
- K... Number of additional holes in the "Y" direction (must be specified)
- Q... Lean angle: If omitted, pattern will cut parallel to the "X" axis





Printed on recycled paper.