PROGRAMMING MANUAL

AN OPERATION SUPPLEMENT MANUAL FOR THE

CINCINNATI CL-707, CL-7A, AND CL-800 LASER SYSTEM WITH PC-BASED CONTROL

CINCINNATI

CINCINNATI INCORPORATED

CINCINNATI, OHIO 45211

EM-423 (R-02/11)

PROGRAMMING CONTENTS

SECTION 1 STANDARD G-CODES

SECTION 2 CUSTOM G-CODES

SECTION 3 M-CODES

SECTION 4 CINCINNATI MACROS

SECTION 5 PROGRAM STRUCTURE

SECTION 6 PROGRAM VARIABLES

SECTION 7 AUXILIARY FUNCTIONS

SECTION 8 FILE TRANSFER & NETWORKING

SECTION 9 INDEX

CODE	DESCRIPTION	SEC.
G00	Rapid move to Work Coordinates	1.00
G01	Linear move to Work Coordinates	1.01
G02	Clockwise arc to Work Coordinates	1.02
G03	Counterclockwise arc to Work Coordinates	1.03
G04	Dwell	1.04
G09	Exact Stop (one block)	1.09
G20	Inch Mode	1.20
G21	Metric Mode	1.21
G31	Position Capture Move	1.31
G40	Cancel Kerf Compensation	1.40
G41	Kerf Compensation Left	1.41
G42	Kerf Compensation Right	1.42
G50	Cancel Scaling	1.50
G51	Work Coordinate System Scaling	1.51
G52	Temporary Local Work Coordinate System	1.52
G53	Rapid move to Machine Coordinates	1.53
G54 to G59	Work Coordinate Offset selection	1.54
G61	Modal Exact Stop	1.61
G64	Cancel Exact Stop Mode	1.64
G65	Sub-program call	1.65
G68	Work Coordinate System Rotation	1.68
G69	Cancel Rotation	1.69
G90	Absolute mode	1.90
G91	Incremental mode	1.91
G92	Set Work Coordinate Origin	1.92

These four G-codes move the cutting nozzle to commanded Work coordinates:

G00 Rapid Traverse move
G01 Linear move
G02 Clockwise Arc
G03 Counterclockwise Arc

These four G-codes form a modal group; the last G-code commanded in the group is active for all blocks until the program commands another G-code in the group. The default code when a program starts is G00. The leading zero can be omitted; G0, G1, G2 and G3 are the same as G00, G01, G02 and G03.

Each of these G-codes specifies the end of the move with X and Y in the Work coordinate system. X and Y are absolute coordinates when the program commands the block in G90 mode, and incremental distances when commanded in G91 mode. The command must specify at least one axis.

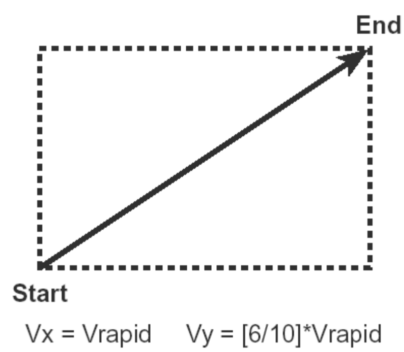
The G00 command moves the axes at the rapid traverse rate of the machine. G01, G02 and G03 move the axes at the contouring feedrate (optionally specified in the block with “F”). When the block does not command a feedrate, the program uses the last defined contouring feedrate. When the control applies the rapid traverse rate for a G00 move, it does not change the contouring feedrate used by the G01, G02, and G03 blocks.

1.00 G00 RAPID TRAVERSE MOVE

The G00 command moves the cutting nozzle to a work coordinate location (or incremental distance) using the rapid traverse rate.

G00 X__ Y__

Example: (G91) G00 X10 Y6



When the command requires both axes to move, the axis moving the longer distance uses the rapid traverse rate of the machine. The other axis moves at a lower velocity proportional to the distance required, so both reach their endpoints at the same time, approximating linear interpolation.

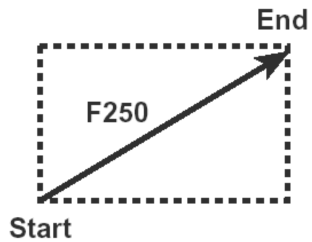
If the command syntax is incorrect, a Message window displays RAPID MOVE SYNTAX ERROR.

1.01 G01 LINEAR MOVE

This command moves the cutting head to the work coordinates (or incremental distance) defined by X and Y, at a contouring feedrate optionally specified by F.

G01 X__ Y__ (F_)

Example: (G91) G01 X6 Y4 F250



When the command requires both axes to change position, the machine moves each axis at a velocity required to produce a combined feedrate equal to the contouring feedrate. The move follows the linear path between start and end points.

If the command syntax is incorrect, a Message window displays LINEAR MOVE SYNTAX ERROR.

FEEDRATE

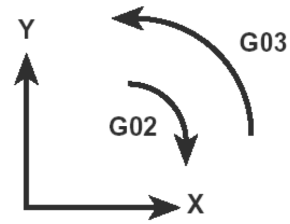
The program can specify contouring feedrate from a parameter library. For example, the program can command F#148 after a block calling G89 Pfilename.lib. The user can also configure the control to assign feedrate to a different variable than #148 (see "Common Variables", SECTION 6).

1.02 G02 CLOCKWISE ARC

1.03 G03 COUNTERCLOCKWISE ARC

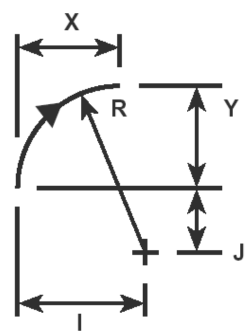
A program uses G02 or G03 to command a circular contouring move ending at the work coordinates (or incremental distances) specified by X and Y. The command defines the shape of the arc either by specifying incremental distances (with I and J) from the starting position to the center, or by specifying the radius (with R). The control software interprets "I" and "J" as distances in the X and Y directions (respectively) from the starting position to the center. When the command specifies radius "R", the control moves the nozzle along a circular path with that radius.

The machine maintains the modal contouring feedrate (F) along the circular path.



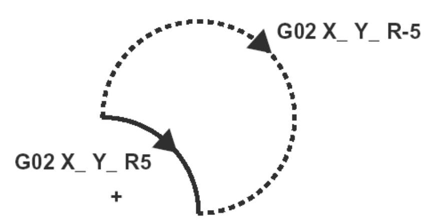
G02 X_ Y_ I_ J_ (F_)
G02 X_ Y_ R_ (F_)
G03 X_ Y_ I_ J_ (F_)
G03 X_ Y_ R_ (F_)

Example: (G91) G02 X5 Y4 I7 J-3



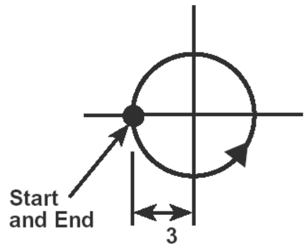
When the block uses "R" instead of "I and J", there are two possible arcs for a given direction (cw or ccw) and end coordinates. To specify which arc to contour, the block commands "R" with a positive or negative sign. To specify an arc that is less than 180 degrees, the block commands a positive "R" value. To specify an arc greater than 180 degrees, the G02 or G03 block commands "R" with a negative value.

Example:



When G02 or G03 specifies the same coordinates for the start and end of the arc, the machine contours a complete circle. For complete circles, the block must specify the center with I and J. Programming software must specify both coordinates accurately. If the ending coordinates for a circular move are not exactly the same as the starting coordinates, the path may be a very small arc instead of a complete circle. To avoid this problem, programs can omit X and Y from a G02 or G03 block to command a complete circle; the control will automatically apply the same starting and ending coordinates.

Example: (G91) G03 I3 J0



If the syntax is incorrect, the software will display the CIRCULAR INTERPOLATION SYNTAX ERROR message.

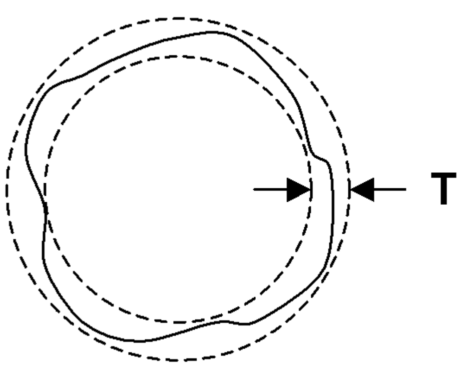
RECOMMENDED ARC FEEDRATE

Recommended maximum G02 or G03 feedrate depends on machine design, arc radius, and allowable roundness error. Use this equation to calculate the maximum feedrate for each arc:

$$F = K * \sqrt{R * (T - T_0)}$$

F = arc feedrate (IPM or mm/min.)
K = constant (1 / min.) See tables.
R = arc radius (inches or mm)
T = roundness tolerance (inches or mm)
T_0 = minimum radial error (inches or mm)

Roundness tolerance "T" is the radial distance between two concentric circles that enclose the contoured shape. To use this formula, the specified roundness tolerance must be greater than "T_0" and not more than "Tmax".



The maximum acceleration also determines the maximum feedrate for contouring an arc. The following tables include that requirement by specifying a maximum roundness "Tmax" for each value of K. If the roundness tolerance does not exceed Tmax, then the calculated feedrate will not command the machine to exceed the maximum acceleration.

CL-707 Arc Feedrate Programming (Original Drive Design)			
Model	**K**	**T_0**	**Tmax**
4x8	18,000	.0002 in. (.005 mm)	.006 in. (.152 mm)
5x10	18,000	.0002 in. (.005 mm)	.006 in. (.152 mm)
6x12	18,000	.0002 in. (.005 mm)	.005 in. (.127 mm)

Arc feedrate programming parameters in the following table apply to CL-707 laser systems with Serial Numbers: 51226, 51242, 51296, 51466, 51509, 51553, 51572, 51631 and higher:

CL-707 Arc Feedrate Programming ("Fast Pack" Drive Design)			
Model	**K**	**T_0**	**Tmax**
4x8	26,500	.0002 in. (.005 mm)	.004 in. (.102 mm)
5x10	26,500	.0002 in. (.005 mm)	.003 in. (.076 mm)
6x12	26,500	.0002 in. (.005 mm)	.003 in. (.076 mm)
8x20	18,000	.001 in. (.025 mm)	.005 in. (.127 mm)

CL-7A Arc Feedrate Programming			
Model	**K**	**T_0**	**Tmax**
4x8	6,000	.001 in. (.025 mm)	.005 in. (.127 mm)
5x10	6,000	.001 in. (.025 mm)	.005 in. (.127 mm)
6x12	6,000	.001 in. (.025 mm)	.005 in. (.127 mm)

To determine the feedrate for contouring an arc, compare the calculated maximum feedrate to a minimum arc feedrate (typically 30 IPM) and select the higher value. Then compare the selected value to the material feedrate, and use the lower value.

1.04 G04 DWELL

The G04 (or G4) command causes the CNC program to dwell for the time specified by the P argument (in milliseconds).

Example (to dwell for one second):

G04 P1000

This dwell time does not include the block processing time of the CNC command.

If the software finds a syntax error, a message window will display “DWELL SYNTAX ERROR”.

1.09 G09 EXACT STOP (ONE BLOCK)

The program commands G09 (or G9) in the same block as a G00, G01, G02 or G03 command. When the block commands G09, the control does not proceed to the next block until the axes reach zero feedrate. If the block does not command G09, the control proceeds to the next block when each axis position is within a specified distance of the commanded position. The specified distance is a system parameter.

Example: (G01 X_ Y_) G09

If the software finds a syntax error, a message window will display “PROGRAMMING SYNTAX ERROR”.

1.20 G20 INCH MODE

1.21 G21 METRIC MODE

The G20 command puts the CNC in the inch units mode. In G20 mode, the control interprets program coordinates and feedrates in inch system units. (Positions are in inches and feedrates are in inches per minute).

The G21 command puts the CNC in the metric units mode. In G21 mode, the control interprets program coordinates and feedrates in metric system units. (Positions are in millimeters and feedrates are in millimeters per minute).

The default mode is G20 when the CNC LASER application starts. After the control runs a program, the default mode is the same as the last program. To make sure the control interprets a program correctly, the program should begin by commanding G20 or G21 to specify units.

G20 and G21 do not change the units mode of CINCINNATI control windows. The windows display values in inch or metric units as selected by the VIEW, UNITS menu item.

1.31 G31 POSITION CAPTURE MOVE

When a program commands G31, the X and Y-axes move to the specified coordinates in the Work coordinate system. The G31 command uses the modal contouring feedrate (F). While the axes are moving, the control system monitors the Position Capture input. If the control system receives the Position Capture input, it records the X and Y-axis Machine coordinates at that time and stores the values in system variables #5061 and #5062.

G31 X_ Y_ (F_)

If the control detects more than one Position Capture input during the move, it only saves the coordinates of the first occurrence. If the control does not receive the Position Capture input, it stores the coordinates at the end of the move. The control always completes the move to the coordinates specified in the G31 block (unless an overtravel alarm stops motion).

Position Capture system variables:

#5061 = X axis Machine Coordinate
#5062 = Y axis Machine Coordinate

CINCINNATI macro programs use G31 to find coordinates associated with optional measurement functions (Workpiece Edge Detection or Optical Probe). The machine control does not accept the G31 command unless the machine configuration includes one of those options.

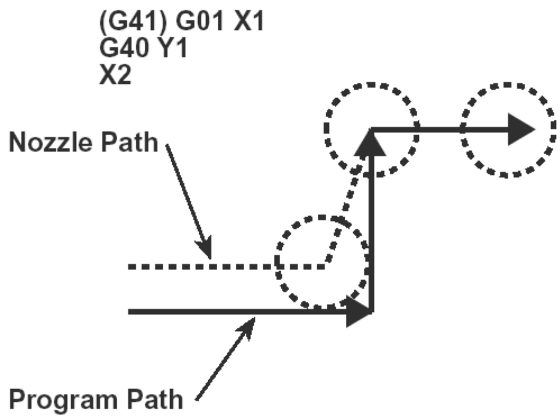
1.40 G40 CANCEL KERF COMPENSATION

1.41 G41 LEFT SIDE COMPENSATION

1.42 G42 RIGHT SIDE COMPENSATION

G40 cancels G41 or G42. The cutting nozzle moves from the compensated position to the commanded coordinates during the G40 move.

Example: G40

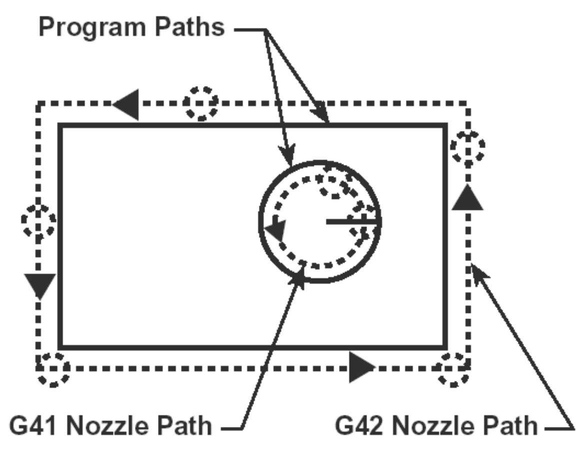


The control automatically cancels kerf compensation at the end of any G00 or G53 move if the program commands G00, G53, M02 or M30 in the next block.

If a program commands G40 in a block by itself, and then commands a move without G41 or G42, the control cancels compensation during that move.

A program commands kerf compensation with G41 or G42. When a G01, G02 or G03 block commands G41 or G42, the control begins that move with the nozzle offset to one side of the programmed path. If a block commands G41 or G42 without commanding a move, the control ends the previous move with the cutting nozzle offset to one side of the path.

Example: G41 and G42

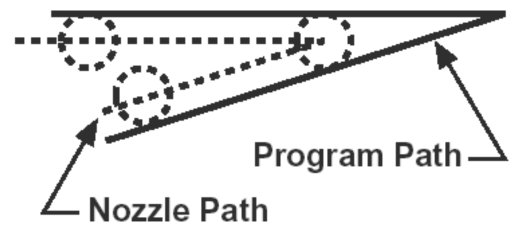


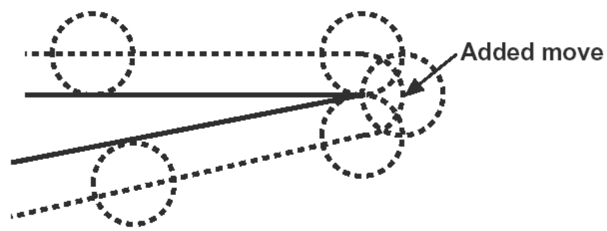
The CNC automatically offsets the cutting nozzle by half the kerf width specified by last G89 command. (See Section 2.89.)

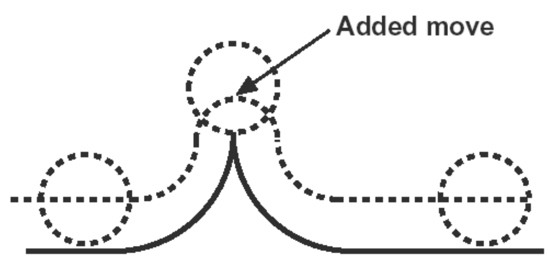
G40, G41 and G42 form a modal group; the last G-code commanded in the group is active for all blocks until the program commands another code in the group. When each program starts, the default code is G40.

The CNC automatically commands the closest possible position for the nozzle to contour the programmed shape with the specified kerf size. If necessary, the control inserts small moves so compensated paths intersect and do not over-cut the shape.

Examples:







1.50 G50 CANCEL SCALING

1.51 G51 WORK COORDINATE SYSTEM SCALING

G51 X__ Y__ P__
G51 X__ Y__ I__ J__

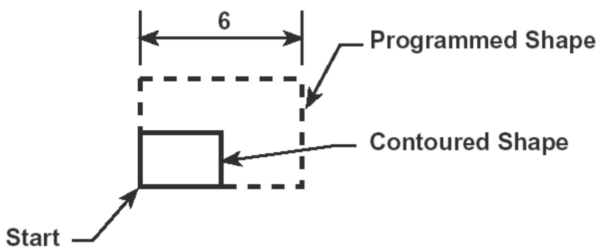
The control interprets the work coordinate system at a different scale or as a mirror image when the program commands G51. The program can restore the normal scale by commanding G50. When each program starts, the default mode is G50. The Absolute Position window and system variables indicate the actual position.

The G51 block defines the center of scaling with X and Y, and the scale factor with “P”, “I” or “J”. To command 1.0 scale (where the contoured shape is the same as the programmed shape), the G51 block uses P1000 (or I1000 or J1000). The G51 block can use I and

J to command separate scale factors for the X and Y axes (respectively). To contour a mirror image of the programmed shape, the block commands I or J with a negative value. The control does not scale the kerf compensation offset distance when the program commands scaling.

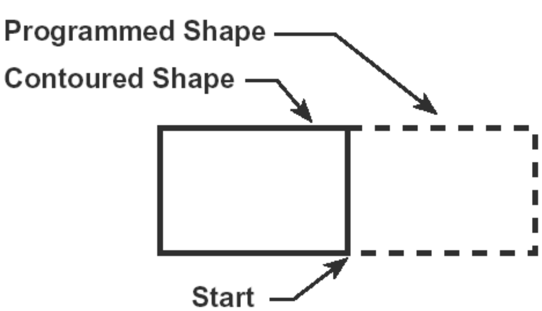
Example 1:

```
G91
G51 X0 Y0 P500
G01 X6
Y4
X-6
Y-4
G50
```



Example 2:

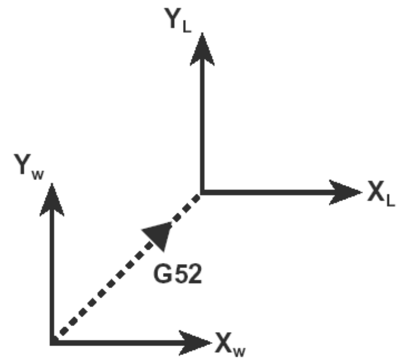
```
G91
G51 X0 Y0 I-1000
G01 X6
Y4
X-6
Y-4
G50
```



1.52 G52 LOCAL WORK COORDINATE SYSTEM

The G52 command temporarily defines a new work coordinate system while remembering the original. The zero position of the new (or "local") coordinate system is at the coordinates in the original coordinate system specified by X and Y in the G52 block. After the G52 block, the program makes contouring moves using the new coordinate system. To restore the original work coordinate system, the program commands "G52 X0 Y0".

G52 X__ Y__



The G52 block does not move the cutting nozzle. The Absolute Position window changes to indicate the nozzle position in the temporary coordinate system.

To demonstrate how a program could use G52, consider a program that uses a sub-program to contour the same shape several times, and both the main program and sub-program use G90 (absolute) mode. The main program would command a work coordinate system with G92 and the sub-program would command a local coordinate system with G52, then cancel it with G52 X0 Y0.

1.53 G53 RAPID MOVE TO MACHINE COORDINATES

G53 X_ Y_

The G53 command moves the cutting nozzle at the rapid traverse rate to a position specified by X and Y in the machine coordinate system. G53 is only active in one block and only in G90 absolute mode. No motion occurs if the program commands G53 in G91 (incremental) mode. The control does not change the machine coordinate system when the program commands kerf compensation, rotation, scaling, or mirror image, or if the program changes the work coordinate system.

1.54 G54 THROUGH G59

WORK COORDINATE SYSTEM SELECTION

A program can use G54 through G59 to command one of six different pre-defined work coordinate systems. The user can set the distance from Machine X0 Y0 to the Work X0 Y0 position of each coordinate system with the

"Position, Work Offset" window, or the program can assign the distance with system variables #2501 through #2506 (X) and #2601 through #2606 (Y).

G54 (OFFSET 1)
G55 (OFFSET 2)
G56 (OFFSET 3)
G57 (OFFSET 4)
G58 (OFFSET 5)
G59 (OFFSET 6)

A work coordinate system defined with G54 through G59 does not need G92 to define its X0 Y0 position. G54 through G59 override G92 by commanding a work coordinate system with its X0 Y0 position preset on the machine.

The G54 through G59 block does not move the cutting nozzle. The absolute position window changes to indicate the nozzle position in the new work coordinate system.

If the block contains a syntax error, the control will display the message "WORK COORDINATE SYNTAX ERROR".

1.61 G61 EXACT STOP MODE

1.64 G64 CANCEL EXACT STOP MODE

G61 commands the CNC to use exact stop mode. In this mode, the axes decelerate to a stop at the end of every G00, G01, G02 or G03 block. The CNC remains in G61 mode until the program commands G64 or the program ends.

The G64 command cancels exact stop mode. The default mode when each program starts is G64. In G64 mode, the control proceeds to the next block when each axis position is within a specified distance of the commanded position. The specified distance is a system parameter.

1.65 G65 SUB-PROGRAM CALL

(WITH OPTIONAL ARGUMENTS)

The G65 block specifies the sub-program name after "P", and may use other arguments to set local variables in the subprogram.

G65 P_ (A_ B_ C_ D_ etc.)

The G65 block must include "P" followed by the name of the sub-program. If the sub-program is in the same file as the CNC program, then the sub-program name does not need an extension or path. However, if the sub-program is in a separate file then the G65 block must command "P" followed by the sub-program filename including its extension (if any) and its path if different from the calling program.

If the G65 command includes arguments, the command must have a space between the last character of the program name and the first argument. This is required because program names can contain both numerals and alphabetic characters.

Note: *Revised CNC software (installed July 2001 or later) does not require a space between the program number and the first argument if a G65 command specifies P9800 or P9900.*

For instructions on calling sub-programs with G65, see SECTION 5. If the G65 block contains a syntax error, the control displays the message "G65 SYNTAX ERROR".

1.68 G68 WORK COORDINATE SYSTEM ROTATION

1.69 G69 CANCEL ROTATION

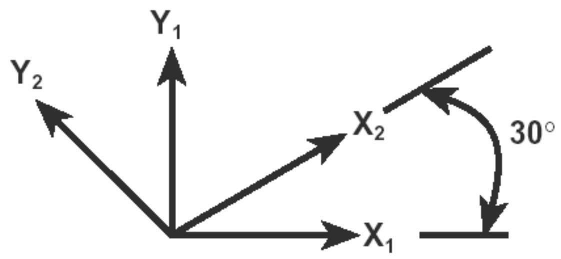
A program can use the G68 command to rotate the work coordinate system relative to the machine axes. The command specifies the center of rotation with X and Y work coordinates (or incremental distances). The command specifies the amount of rotation with "R" in degrees, with counterclockwise positive. In G90 mode, R is the absolute angle of rotation. In G91 mode, R is the incremental rotation angle that the control adds to any previous rotation.

G68 X_ Y_ R_

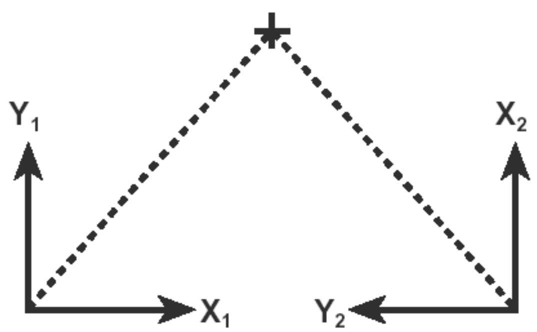
The work coordinate system remains rotated until the program commands G69 or the program is reset. G69 cancels all coordinate rotation. To cancel only the last incremental rotation, command G68 in G91 mode with the opposite amount for "R".

The G68 or G69 block does not move the cutting nozzle. The Absolute Position window and System Variables indicate the nozzle position in the un-rotated work coordinate system.

Example 1: G68 X0 Y0 R30



Example 2: G68 X5 Y5 R90



1.90 G90 ABSOLUTE MODE

1.91 G91 INCREMENTAL MODE

In G90 absolute mode, the nozzle moves to the coordinate location specified by the arguments in a G00, G01, G02, G03 or G53 command. G90 mode is active until the program commands G91 mode. When each program starts, the default mode is G90.

In G90 mode, X and Y coordinate values are modal. In other words, if a block does not specify X or Y, the control uses the last commanded value for X or Y.

In G91 incremental mode, the cutting nozzle moves a distance from its starting location specified by X and Y in a G00, G01, G02 or G03 command. G91 mode is active until the program commands G90 or the program ends. The control ignores a G53 command while operating in G91 mode.

1.92 G92 WORK COORDINATE SYSTEM SETTING

This command sets the work coordinate system location. When the machine completes the Axes Home operation, the control establishes the work coordinate system with X0 Y0 at Machine X0, Y0. The G92 command can move the work coordinate system to any location.

G92 X_ Y_

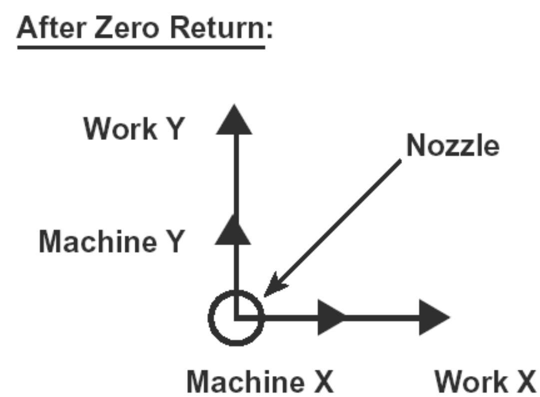
X and Y define the new work coordinates corresponding to the cutting nozzle position when the G92 block is executed.

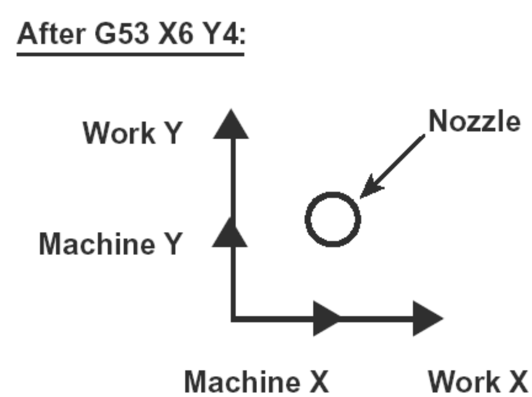
The G92 block does not move the cutting nozzle. The Absolute Position window changes to indicate the nozzle position in the new work coordinate system.

Example: G92 X0 Y0

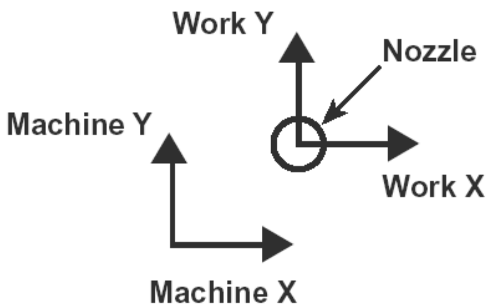
The G92 X0 Y0 command moves the work coordinate system X0 Y0 location to the current position of the cutting nozzle. Programmers often use this command to begin a sub-program written in G90 mode.

G92 Example:





After G92 X0 Y0:



The CINCINNATI control has built-in functions programmed with custom G-Codes.

CODE	DESCRIPTION	SEC.
G84	Pierce and Start Cut	2.84
G85	Start Cut without Pierce	2.85
G89	Process Parameters	2.89
G102	Additional Parameters	2.102
G103	Ramped Pierce Parameters	2.103
G120	Disable Non-Stop Cutting	2.120
G121	Enable Non-Stop Cutting	2.121
G123	Programmable Blend	2.123
G124	Default Blend	2.124
G125	Auto Blend	2.125

2.84 G84 PIERCE AND START CUT

2.85 G85 START CUT WITHOUT PIERCE

A program uses G84 or G85 to begin user-programmed cutting sequences. G84 and G85 command the Z-axis to move the nozzle down to the standoff position (if not already there), and then command the pierce and/or cut parameters. When the control finishes the G84 or G85 command, it returns to the program with the laser beam on, assist gas on, and shutter open, ready to proceed with contouring commands (G01, G02, G03). G84 and G85 also turn coolant on if the process parameters specify coolant.

A program uses G85 to begin a cut sequence when the application does not require the pierce cycle of G84. G85 duplicates all other functions of G84, including precut dwell and power burst time (see G102 description). After a program commands processing parameters with G89, any cut sequence can start with G84 or G85. Examples of G85 applications are: starting a cut inside an opening, off the edge of the sheet, or in a kerf.

AUTO RESTART

When a laser system has the CINCINNATI control, the CNC program does not require special codes or commands to activate Auto Restart. When an alarm condition interrupts a program, the operator can restart the program at any block. After correcting the condition that caused the interruption, the operator can select Tracing mode, press Cycle Start, then hold down the Tracing Function Forward or Reverse button to move in the forward or reverse direction to another program block. De-select Tracing mode and then press Cycle Start to resume the cut. If an alarm condition interrupts a program and the operator presses Cycle Start without selecting Tracing mode, the cutting nozzle moves to the start of the interrupted block and resumes cutting.

PIERCE OPTIONS (G84 T_)

Each process parameter library file has one set of cutting parameters and three pierce options. The G84 "T" argument selects the pierce option for each cutting path.

Normal Pierce G84 or G84 T1:

G84 T1 is the same command as G84. The program commands normal pierce parameters with a G89 library file, or explicitly with G89, G102 and G103 macro calls.

Rapid Pierce G84 T2:

The program commands G84 T2 to use rapid pierce. Rapid pierce has separate laser power, gas pressure, dwell time and standoff parameters. Laser pulse mode is always 5000 Hz and 100% duty cycle. G84 T2 uses the same assist gas (#1 or #2) and part coolant status as normal pierce.

Rapid pierce uses a single power level during the pierce, so ramped pierce is always OFF. Rapid pierce also has a cooling time parameter independent from G84 T1 and airblast time parameters.

G89 loads rapid pierce parameters from a library file. The NC program cannot set rapid pierce parameters explicitly with G89, G102 or G103. When the program commands normal pierce parameters explicitly, the default T2 parameters are the same as T1.

G84 T3:

G84 T3 operates the same as G85 (no pierce).

Note: *All G84 pierce options (T1, T2 or T3) command pre-cut dwell before returning to the program. For a description of pre-cut dwell, see G102 in this section.*

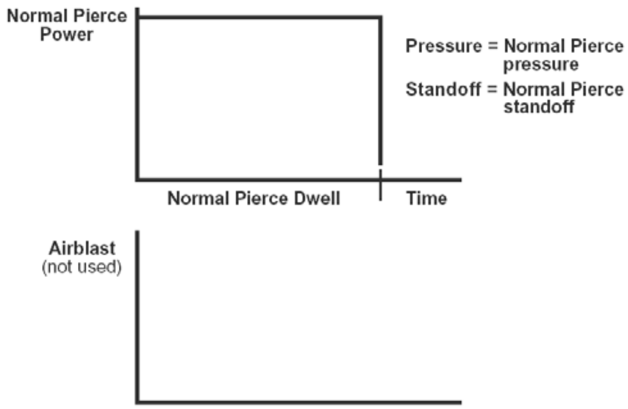
AIRBLAST

The rapid pierce process uses a separate blast of compressed air to help clear molten material from the pierce area. Two airblast parameters ("OFF time" and "ON time") control the opening of the airblast solenoid valve.

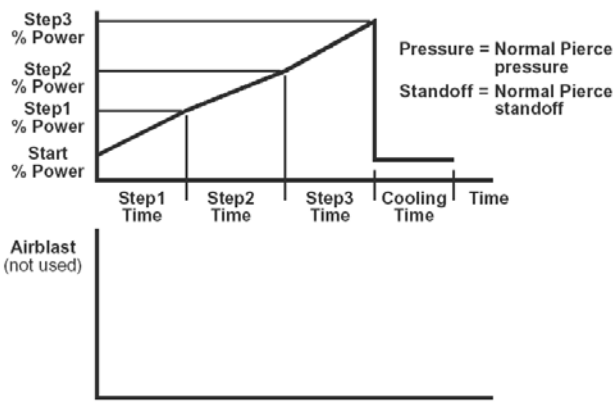
The OFF time is a delay that starts when the pierce begins. The air valve is closed during the OFF time. When the delay ends, the air solenoid valve opens. The valve then stays open for the ON time. To edit the airblast times, open the Process Library Window.

The following figures show the function of G84 T1 and T2 parameters:

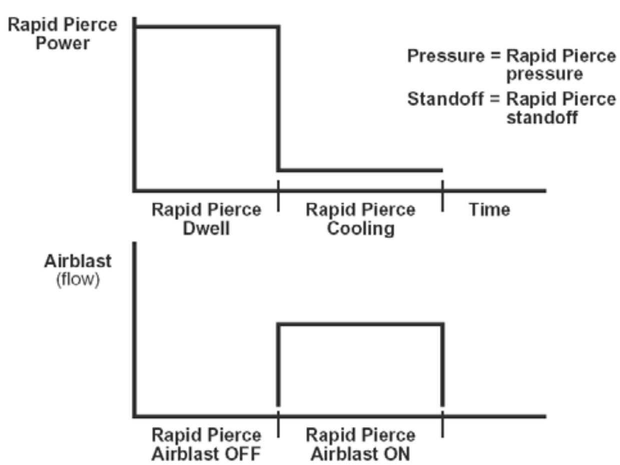
Normal Pierce (G84 T1) with Ramped Pierce OFF



Normal Pierce (G84 T1) with Ramped Pierce ON



Rapid Pierce (G84 T2)



2.89 G89 PROCESS PARAMETERS

The program sets processing parameters by commanding G89. When G89 loads processing parameters with a library file, the operator can edit the parameters while the program is running; however, changes will NOT take effect until the next G84 (or G85). To change parameters, open the library file, edit the parameter(s) then save the library file.

The CINCINNATI control will also accept G89, G102 and G103 commands programmed with explicit parameters.

G89 WITH LIBRARY FILE

G89 P*filename*.lib

The G89 command uses address “P” to specify a library file. The operator can edit library files in the Process Parameter window. The default path is:

D:\CNCLSR32\MATERIAL\

The filename must include the “.lib” extension.

If the library file is not in the MATERIAL folder, the G89 command must include the path. The user can create other library directories, in either the MATERIAL folder or elsewhere on the disk.

CINCINNATI INCORPORATED provides a set of read-only library files in this folder:

“D:\CNCLSR32\MATERIAL\ARCHIVE\”

The MATERIAL directory includes copies of the same library files, which the user can edit.

Library filenames provided by CINCINNATI INCORPORATED begin with an abbreviation for material:

AL . . . Aluminum
MS . . . Mild Steel
SS . . . Stainless Steel

After the material abbreviation, the library filename has a three-digit number representing the material thickness in mils.

Example: (For 10 gauge mild steel 0.135”): MS135

The filename may include other characters after the thickness number, to indicate a resonator type or processing application.

After the thickness number, the filename may have a chemical abbreviation for the cutting assist gas:

O2 . . . Oxygen
N2 . . . Nitrogen

For applications using coolant, the library filename ends with the word “wet”.

Examples: (10 gauge mild steel, oxygen cut)
Without coolant: MS135O2.lib
With coolant: MS135O2wet.lib

G89 CALL WITH ARGUMENTS:

G89 T_ A_ I_ M_ S_ C_ D_ Q_ B_ E_ H_ R_ J_ K_ U_ V_

T = Cut power level, watts.
A = Cut gas code. See Note 1.
I = Cut gas pressure. See Note 2.
M = Cut laser mode, see Note 3.
S = Cut pulse code, see Note 4.
C = Cut coolant code. See Note 5.
D = Pierce time, seconds.
Q = Pierce power level, watts.
B = Pierce gas code. See Note 1.
E = Pierce gas pressure. See Note 2.
H = Pierce laser mode, see Note 3.
R = Pierce pulse code, see Note 4.
J = Pierce coolant code. See Note 5.
K = Kerf width, see Note 2.
U = Maximum feedrate for Dynamic Power, see Note 2.
V = Minimum percent for Dynamic Power (% at zero feedrate)

Notes:

1. Assist gas codes (A & B):

   11 = Gas Port #1 (usually O2)
   12 = Gas Port #2 (usually N2)

2. G89 interprets pressures, kerf width, and dynamic power feedrate in the active units:

Parameter	G20 unit	G21 unit
I & E	PSI	kPa
K	inches	mm
U	IPM	mm/min

3. Laser mode (M & H):

   61 = Continuous Wave
   62 = Gated Pulse
   66 = Dynamic Power

4. Pulse codes (S and R):

   When pulsed laser output is used, frequency and duty cycle are specified with a 4-digit code in which the first two digits specify frequency (Hz/100) and the last two digits specify duty cycle (%).

   For DC (diffusion-cooled) resonator, maximum frequency is 5000 Hz and minimum duty cycle is the value necessary for a pulse ON time of 26 microseconds at the commanded frequency.

5. Coolant codes (C & J):

   8 = coolant ON
   9 = coolant OFF

When CINCINNATI laser systems with Fanuc control have the Macro Executor option, programs written for those laser systems can specify process parameters with G89 X_, where X is followed by a library code number from 1 to 100. The CINCINNATI control will accept a program with the “G89 X” command (instead of G89 P), if the Material folder has a library file with the same name as the number following “X”. For example, the CINCINNATI control will accept a program commanding “G89 X32” if the Material folder has a library file named “32.lib”.

When a program commands G89, G102 or G103 with explicit parameters, the CINCINNATI control checks the parameters for out-of-range values. If the control finds any, it displays an error message in a pop-up window indicating which parameter has the error. The window identifies parameters by the name used in the Process Parameter Library window, not by the G89, G102 or G103 argument. For example, “Pierce Gas Pressure out-of-range” instead of “G89 E out-of-range”.

2.102 G102 ADDITIONAL PARAMETER SETTINGS

The Parameter Library window includes settings for dynamic gas pressure, noncontact standoff, optional pressure, precut dwell and power burst time. In addition to commanding these parameters in a library file with G89, the program can also command these parameters explicitly with G102.

G102 A_ B_ S_ Z_ D_ I_ T_ Q_ R_ U_ V_

A = Dynamic gas pressure near field setting
B = Dynamic gas pressure far field setting
S = Pierce standoff
Z = Cut standoff
D = Precut dwell, seconds
I = Optional pressure
T = Power burst time, seconds
Q = Pierce Focus, Near Field

R = Pierce Focus, Far Field
U = Cut Focus, Near Field
V = Cut Focus, Far Field

G102 interprets pressure, standoff and focus settings in the active units:

Parameters	G20 unit	G21 unit
A, B & I	PSI	kPa
S & Z	inches	mm
Q, R, U & V	inches	mm

A & B: When a program commands dynamic gas pressure, the control regulates cutting assist gas pressure between the Near (A) and Far (B) field settings based on the machine position of the nozzle. Near field is where the laser beam length is shortest.

S & Z: The program uses these settings to command pierce and cut nozzle standoff distance for the noncontact head.

D: Before returning to the program, G84 and G85 command the cutting parameters and then command the pre-cut dwell.

I: The assist gas pressure controller uses the optional pressure setting when a program commands M67.

T: When the laser system starts a contouring move using dynamic power, the control maintains dynamic power at 100% for the time specified for Power Burst. After the Power Burst time, the control regulates dynamic power according to the actual feedrate.

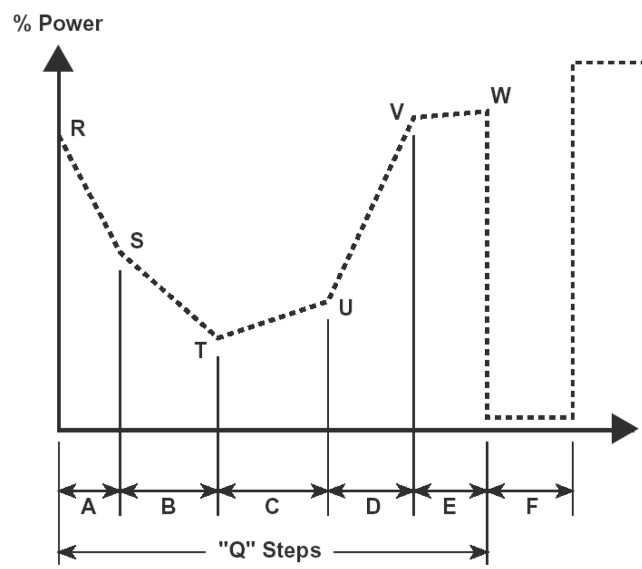
Q, R, U and V: When CL-707 lasers or CL-7A lasers with CINCINNATI control have the Auto Focus Cutting Head option, the G102 command has additional arguments to specify focus settings. The settings specify focus position relative to the nozzle tip. The Auto Focus drive uses the Near field settings when the cutting head is closest to the laser source, and changes focus between the Near and Far settings as X and Y-axis motion changes the optical path length. Q and R specify the Near and Far field pierce focus settings. U and V specify the Near and Far field cut focus settings.

2.103 G103 RAMPED PIERCE SETTINGS

To set parameters for ramped pierce power, the CNC program can either command G89 with a library file, or command G103 with explicit settings.

G103 A_ B_ C_ D_ E_ F_ Q_ R_ S_ T_ U_ V_ W_

A = Ramp 1 duration, seconds
B = Ramp 2 duration, seconds
C = Ramp 3 duration, seconds
D = Ramp 4 duration, seconds
E = Ramp 5 duration, seconds
F = Tip cooling time, seconds
Q = Number of ramp steps (1 to 5)
R = Percent power at start of first ramp
S = Percent power at start of second ramp
T = Percent power at start of third ramp
U = Percent power at start of fourth ramp
V = Percent power at start of fifth ramp
W = Percent power at end of fifth ramp



G103 Ramped Pierce Arguments

2.120 G120 DISABLE NON-STOP CUTTING

2.121 G121 ENABLE NON-STOP CUTTING

When a program commands Non-Stop Cutting (G121), the CNC replaces short G00 moves between cut sequences with “Smart Rapid” moves. A Smart Rapid move commands the laser beam off and on without stopping the axes. (See Smart Rapids description below.) During a Smart Rapid move, the control maintains assist gas flow, even when the laser beam is off.

Notes: *A program can only command Non-Stop cutting mode when the process parameters specify no pierce time and no precut dwell.*

If the operator edits and saves process parameters while running a program in Non-Stop mode, the CNC ignores the changes until the program ends.

The G120 command cancels Non-Stop cutting mode. When each program starts, the default mode is G120.

SMART RAPIDS

Programming software normally commands a single G00 linear rapid move between the end of one cut sequence and the beginning of the next. The G00 move commands the shortest distance between the two points (to minimize the time between cuts). However, in the default mode (G120), the nozzle must stop before and after the G00 move (to turn the beam off and on). In Non-Stop cutting mode (G121), the nozzle does not stop to turn the beam off or on.

When the operator loads a program, the control translates the program into commands for the CNC to execute. If the program specifies G121 mode, the control translates G00 moves into "Smart Rapid" moves. The control replaces the G00 move and the contouring moves before and after it with commands that maintain the original beam-on path without stopping the axes.

The CINCINNATI software performs these tasks to create a Smart Rapid:

1. Command M35.
2. Command a G01 move after M35 at the same feedrate and direction as the move before M35.
3. Command a G01 move at high feedrate between the G01 moves inserted in Steps 2 and 4.
4. Command a G01 move before G84 at the same feedrate and direction as the move after G84.
5. Command G84.
6. Lengthen the move after G84.

Although the CNC uses a "linear" G01 connecting move (instead of G00), the high G01 feedrate produces a curved path as the axes blend with the other G01 moves. The result is a smooth non-stop transition between cuts.

If the original program has anything other than a single G00 move between cuts (between M35 and G84), the control does not create a Smart Rapid.

During a Smart Rapid move, the nozzle may deviate from the original programmed path while the beam is off. If the nozzle must follow the original path, then the program should not command G00. The control will not replace G01 G02 or G03 moves with Smart Rapids.

Notes: *In this description, G85 can replace G84.*

When the move before M35 or after G84 is an arc, the control inserts a Smart Rapid G01 move tangent to the arc at the intersection point.

BEAM ON / OFF TIMING

The laser system can turn the beam on and off at a desired time within 3 to 5 milliseconds. This means that the cut actually starts or stops within a distance representing 3 to 5 milliseconds of travel on either side of the desired point.

Example:

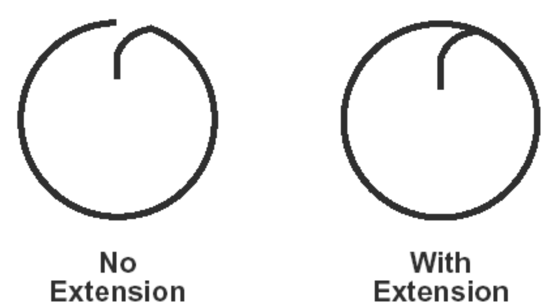
At 300 inches/min., the length of the tolerance band could be (300 in./min.) / (60 sec./min.) * .005 sec. = ± .025 inches (0.63 mm)

The size of the tolerance band depends on the control design. In the original control design, the tolerance is ± 5 milliseconds. For laser systems with the "Fast Pack" control design, the tolerance is ± 3 milliseconds.

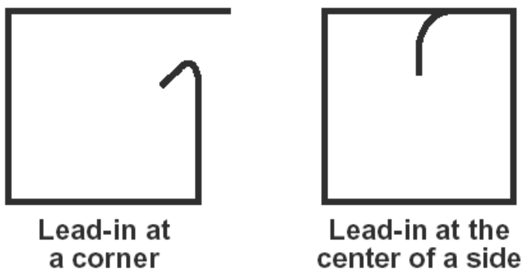
CUT EXTENSION

The CNC determines the G121 extension time to control how the laser system cuts shapes in G121 mode.

Consider a round hole with a radial lead-in. In conventional cutting, the programmer might end the lead-in with exact stop (G09). However, a lead-in programmed for Non-Stop mode may not command exact stop, and the end of the lead-in would blend into the start of the circular move. Also, beam On/Off positions are not as precise when using Smart Rapids (as described above). Thus, the slug from a 360-degree circular hole may not drop because the beam would not cut the entire perimeter of the hole (see figure below). Extending the circular move assures that the beam will cut the entire shape.

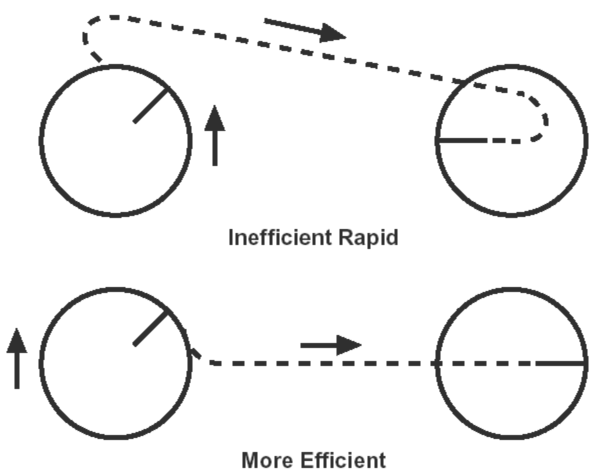


It is important to place the lead-in so the extension of its last entity will meet or overlap the beginning of the first contoured entity. If the program commands a lead-in at a corner, the extension of the last entity will cut past the desired perimeter of the feature, as shown in the following diagram.



EFFICIENCY

As described above, the path of a Smart Rapid is not a straight line. The ends are smoothed, the way a baseball player rounds the base paths on an extra base hit. Program efficiency is greatest when the path length is shorter. Path length between two features depends on cutting direction and placement of lead-ins, as shown in the following diagram.



The cases shown in the figure have two programming differences: the cutting direction of the left hole, and the location of the lead-in on the right hole. Note the "U-turns" required in the top path.

When selecting lead-in locations, the desire to minimize Smart Rapid path length may conflict with the desire to maximize "head down" operation (by avoiding tipped slugs). The conflicting requirements often require some compromise.

2.123 G123 PROGRAMMABLE BLEND

2.124 G124 DEFAULT BLEND

2.125 G125 AUTO BLEND

The process of ending one contouring move and beginning the next move usually requires changing the velocity of one or both axes. In this manual, this process is called a "blend".

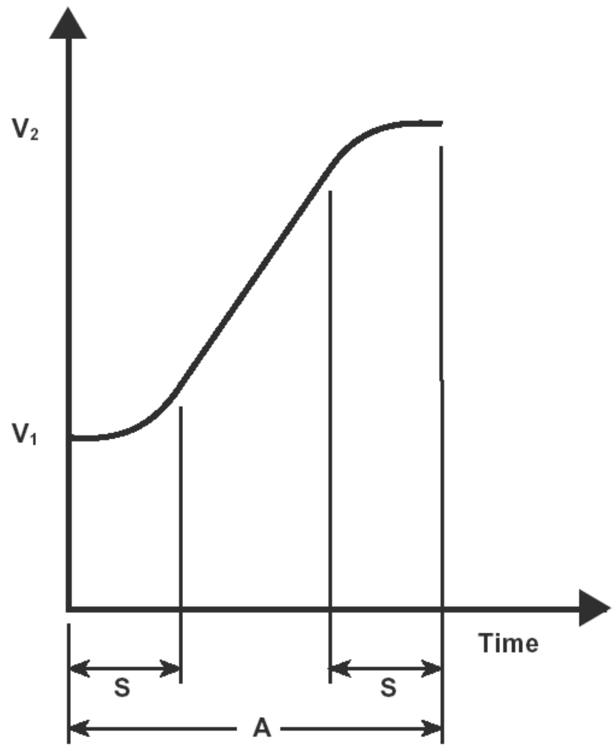
The objective of a blend is to change the axis from executing the preceding move at its constant velocity to executing the next move at a different constant velocity. To accomplish this transition, acceleration also changes during the blend. When velocity is constant, acceleration is zero. Therefore, during a blend (between linear moves, for example), acceleration begins at zero, increases or decreases to produce the velocity change, then returns to zero to complete the blend.

The CINCINNATI control executes a blend using two parameters: the overall time to complete the velocity change and the portion of that time which is used to change the acceleration. Just as a motion system has a maximum velocity and a maximum acceleration, it is also limited by how quickly it can change acceleration.

Short blend times improve contouring accuracy and increase productivity by using high acceleration, but can produce servo following errors if the machine attempts to exceed its acceleration capability. Long blend times avoid servo following errors but sacrifice contouring accuracy. Since blend time settings can affect processing results, users can program blend times with three different commands:

G123 specifies fixed time values for all blends:

G123 A_ S_
A = Total blend time, in milliseconds
S = Time for acceleration change, in milliseconds
The minimum value of "A" is S * 2.



G124 commands the control to use a set of default times for all blends. The default time values are set by CINCINNATI INCORPORATED.

G125 Auto Blend enables the control to determine the minimum X and Y blend times independently for each move.

When each program starts, the default mode is G125.

Different parts of a program can use different blend control modes. A program can change from G123, G124 or G125 to either of the other modes.

If either G123 or G124 setting is less than a minimum time, the control uses the minimum time without displaying an error message. The control determines the minimum blend time for each move using parameters set by CINCINNATI INCORPORATED.

Most M-codes command machine functions not directly related to CNC operation. If an M-code block does not have the proper syntax, program execution stops and a message window displays the message “INVALID M-CODE”.

CODE	DESCRIPTION	SEC.
M00	Cycle Stop	3.00
M01	Optional Stop	3.01
M02	Program End (No Rewind)	3.02
M30	Program End (and Rewind)	3.30
M35	Beam Off	3.35
M36	Noncontact Z-Axis Servo Hold	3.36
M37	Beam Off, Gas Off, Shutter Close	3.37
M38	Timed Z-Axis Servo Hold	3.38
M41	Lower Z –Axis	3.41
M42	Retract (Raise) Z Axis	3.42
M43	Enable Lower Pallet Special Function	3.43
M44	Disable Lower Pallet Special Function	3.44
M45	Apply Optional Standoff	3.45
M47	Raise Z-Axis, optionally by distance	3.47
M48	Disable Feedrate Override	3.48
M49	Enable Feedrate Override	3.49
M50	Switch Pallets	3.50
M51	Enable Timed Auxiliary Output	3.51
M67	Apply Optional Gas Pressure	3.67
M98	Call Sub-program	3.98
M99	Return from Sub-program	3.99
M130	Disable Anti-dive	3.130
M131	Enable Anti-dive	3.131
M135	Beam Off with Gas On	3.135

3.00 M00 PROGRAM STOP

When the CNC executes a program block commanding M00 (or M0), the program stops until the operator presses CYCLE START. If the laser beam and assist gas were on, the M00 command turns them off. M00 places the CNC in a Cycle Stop condition. Modal information does not change.

3.01 M01 OPTIONAL STOP

The M01 (or M1) command has the same function as the M00 command, except M01 is only active if the operator has selected the Optional Stop button on the CNC control window. The program resumes when the operator presses CYCLE START.

3.02 M02 END OF PROGRAM

A CNC program can use M02 (or M2) as the last block. This function disables all processing functions, resets all previously requested M-codes and prevents further execution. The program does not rewind automatically.

When the operator loads a program into the CNC, the control ignores any codes following M02.

3.30 M30 END OF PROGRAM WITH REWIND

Most CNC main programs use M30 as the last block. This function disables all processing functions, resets the CNC, cancels all previously requested M-codes, prevents further execution and rewinds the program.

3.35 M35 BEAM OFF

M35 turns the laser beam OFF at the end of a cut sequence. Laser discharge current stops. Assist gas flow also stops, unless the SPEED GAS option in the Variables menu is selected (see M135). Discharge current remains OFF until the next G84 or G85.

3.36 M36 SERVO HOLD FOR NONCONTACT Z-AXIS

The M36 command places the Z-axis control for the Non-contact head in a servo hold condition when the nozzle is in the cutting position and the laser beam is ON. The noncontact head does not follow the material after the program commands M36. To clear M36, the program commands M35, M37, M42, M47 or M30, or the operator presses RESET.

3.37 M37 BEAM OFF, GAS OFF AND SHUTTER CLOSE

M37 turns off the laser beam and assist gas and commands the shutter to close. M37 also resets any previously requested M-codes.

To avoid unnecessary cycling of the gas valves and shutter, most programs use M35 instead of M37. The control accepts M37 to support programs written for laser systems without M35.

3.38 M38 TIMED NONCONTACT SERVO HOLD

M38 places the Z-axis control for the Noncontact head in a servo hold condition for a period of time after the control establishes the cutting position. The M38 block specifies the time with the P argument (in milliseconds).

Example:

M38 P2000

(This example commands the Z-axis to maintain a fixed position for 2 seconds after reaching the nozzle standoff position.)

To start the M38 time, the Z-axis must be in the cutting position with the beam on, and the X or Y axis must be moving. The Z-axis maintains position until the time has elapsed. After the specified time, the Z-axis goes into tracking mode.

The program can command M38 before or after G84. To cancel M38, the program commands M36, M42, M47 or M30, or the operator presses RESET.

3.41 M41 COMMAND Z-AXIS DOWN TO CUT POSITION

M41 commands the Z-axis to move the cutting nozzle down to the commanded standoff position. Programs normally do not use M41 because G84 and G85 include that function. M41 allows a program to command the cutting nozzle down to the material without starting a cut.

If the cutting head does not find material, the Z-axis travels down until it reaches a minimum position or exceeds a time limit, causing motion to stop.

3.42 M42 RETRACT Z-AXIS

M42 commands the Z-axis to the full up position. The shutter is commanded to close. Programs use M42 to command the Z-axis to the required position for pallet motion.

3.43 M43 LOWER PALLET SPECIAL FUNCTION

In normal operation, the control only allows the beam on when the cutting nozzle is within 1.5 inches (38mm) of the material support height on either pallet. On laser systems equipped with the Lower Pallet Special Function option, users can extend the allowable beam-on range to 7.0 inches (178mm) on the lower pallet by commanding M43.

The M43 operating mode allows the machine to process square or rectangular tubing or other formed parts on the lower pallet. Operation above 1.5 inches (38mm) is not possible on the upper pallet because the Z-axis reaches its upper limit.

M43 changes these control functions:

1. The Z-axis travels down from the top position at a lower speed to find the material surface.
2. The pallets cannot be moved with M50 or with pallet JOG buttons.
3. The Y-axis cannot exceed the machine coordinate specified on the CONFIGURATION window for this option.

M43 disables pallet motion because material over 1 inch (25mm) high will not clear the upper pallet. Since the pallets cannot move and the X-axis beam delivery blocks access to the pallet from one side of the main frame, the operator must load and unload the pallet from the other side of the main frame (the operator side). To keep material within reach of the operator side, the control does not allow the Y-axis to exceed the machine position set on the CONFIGURATION screen.

Recommended procedure to use M43:

1. Move the upper pallet OUT and the lower pallet IN.
2. Jog the nozzle to Machine Y0.
3. Load and execute a program with only M43 and M30. (CINCINNATI provides such a program named “M43.cnc”.)
4. Verify M43 mode is active (FYI Message).
5. Load material on the lower pallet.
6. Load the CNC program for the loaded material.

Programs using M43 mode can begin with M43, but to avoid accidental damage, do not load material over 1 inch (25mm) high until M43 mode is already active. The control will execute the M43 command with Program Test on or off.

> **CAUTION**
>
> **Do not load material over 1 inch (25 mm) high on the lower pallet until M43 mode is active. When material over 1 inch high is on the lower pallet and M43 mode is not active, operation in JOG or AUTO mode can damage the machine and the workpiece.**

When a laser system has this option, the CNC is in M43 mode each time the operator turns on the machine control. M43 mode is NOT cancelled by M30, RESET or turning off the control. The only way to cancel M43 mode is to run a program commanding M44.

3.44 M44 CANCEL LOWER PALLET SPECIAL FUNCTION

M44 cancels the Lower Pallet Special Function mode commanded with M43. The M44 command restores normal Z-axis speed, pallet motion and Y-axis range. The control will execute the M44 command with Program Test on or off.

To cancel M43 mode:

1. Remove any material or fixture over 1 inch (25 mm) high.
2. Load and execute any program beginning with M44.

> **CAUTION**
>
> **Remove any workpiece or fixture over 1 inch (25 mm) high before executing a program with M44. When material over 1 inch high is on the lower pallet and M43 mode is not active, operation in JOG or AUTO mode can damage the machine and the workpiece.**

3.45 M45 APPLY OPTIONAL STANDOFF FOR CUTTING

When the CNC program commands M45 after starting a cut sequence, the non-contact head standoff changes to the “Optional Standoff” distance specified in the active parameter library file. The M45 command does not change pierce standoff.

3.47 M47 RAISE Z-AXIS, OPTIONALLY BY DISTANCE

M47 commands the Z-axis to raise a fixed distance, or a programmed distance if the command specifies a value with “P”. The Z-axis Maintenance Configuration window displays the fixed M47 distance as “Default Partial Z-Up Distance (M47)”.

The M47 command can specify a programmed distance with “P”. The distance units are thousandths of an inch for G20 mode and thousandths of a millimeter in G21 mode. Maximum command is M47 P3000 for 3 inches or M47 P76200 for 76.2 mm.

Programs raise the Z-axis with M47 to avoid interference with clamps or tipped parts during a non-cutting move.

3.48 M48 FEEDRATE OVERRIDE DISABLE

M48 disables the feedrate override dial on the operator control station and sets the feedrate to 100% of the value specified in the program. M48 is canceled by M49, M30 or RESET.

3.49 M49 FEEDRATE OVERRIDE ENABLE

M49 restores the function of the feedrate override dial on the operator control station. When a program starts, the default mode is M49.

3.50 M50 SWITCH PALLETS

M50 commands the upper and lower pallets to switch positions. The pallets will switch positions only if the Pallet Not Ready pushbutton/indicator is not illuminated. If necessary, the M50 command will also retract the cutting nozzle (like M42).

The control illuminates the PALLET NOT READY pushbutton when a program starts. The illuminated button indicates the pallets are “Not Ready” to switch positions. The operator can press the button to toggle the status ON or OFF before the program reaches the M50 block. If the button is not illuminated when the program executes the M50 block, the pallets will reverse positions.

If the PALLET NOT READY button is illuminated when the program reaches the M50 block, the program will stop. To resume the program, the operator can then press the PALLET NOT READY button and the pallets will reverse positions.

3.51 M51 AUXILIARY TIMED OUTPUT

M51 commands a set of isolated relay contacts to close for the time specified by the P argument. Data Range is 0 to 10000 milliseconds. The default time is zero.

Example:

M51 P1000

(This example commands the auxiliary contacts to close for one second.)

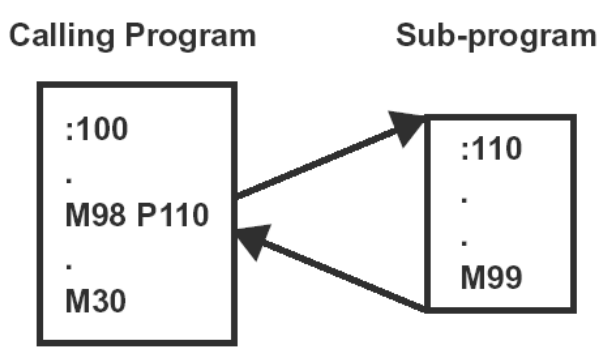
3.67 M67 APPLY OPTIONAL ASSIST GAS PRESSURE

M67 changes the cutting assist gas pressure command to the Optional Pressure setting in the parameter library file or specified by G102 I_. The pressure command is valid until replaced by the next G84 or G85.

3.98 M98 SUB-PROGRAM CALL WITH NO ARGUMENTS

The M98 command transfers control from the calling program to a sub-program. The M98 block specifies the sub-program name after "P". If the sub-program is in the same file as the calling program, only the program name is required. If the sub-program is in a separate file, "P" is followed by the filename including its extension (if any) and its path if different from the calling program.

Example:



The program can call the sub-program more than once by specifying the number of times with "L".

Example:

M98 P1200 L3

(This example calls a sub-program named "1200" three times.)

When a program calls a sub-program with M98, the two programs share the same set of local variables (See SECTION 5).

3.99 M99 END SUB-PROGRAM AND RETURN

The M99 command returns control to the program that called the sub-program. The block following the sub-program call is executed next. Sub-programs called with M98 or G65 can end with M99.

If a main program commands M99, the control restarts the main program from the beginning.

If the M99 command includes the optional "P" argument, the sub-program returns to the calling program at the sequence number specified after "P". If the M99 "P" command is in the main program, the control returns to the line number specified by "P" in the same program (same as GOTO).

Example:

M99 P500

(If commanded in a sub-program, this example returns to the calling program at line N500. If commanded in a main program, this example returns to line N500 in the main program.)

3.130 M130 Z-AXIS ANTI-DIVE DISABLE

3.131 M131 Z-AXIS ANTI-DIVE ENABLE

M130 and M131 disable and enable the Z-axis anti-dive function.

M131 enables the anti-dive function. The cutting head follows limited variation in the material surface but maintains Z-axis position when it does not detect material. M131 is the default mode. All programs start in M131 mode, with anti-dive enabled.

M130 disables the anti-dive function. If M130 is active, the Z-axis does not use anti-dive mode. The Z-axis lowers the cutting head until it detects material or an overtravel alarm occurs.

When a program commands M130, the control disables anti-dive until one of the following occurs:

1. The program commands M131.
2. The program commands M30.
3. The operator rewinds the program.
4. The operator loads a new program.

While M130 is active, the control displays the FYI message: “Z-axis anti-dive is disabled.”

Programs use M130 for applications with significant material vibration. M130 allows the head to follow moderately warped material or thin gauge material that flutters due to interaction with assist gas pressure.

CAUTION

Programmers are advised to carefully consider the application before using M130. If the program commands the cutting head to travel over a hole in M130 mode, the cutting head will most likely dive into the hole and breakaway at the magnet flange, possibly damaging the cutting head.

3.135 M135 DISCHARGE CURRENT OFF

M135 is similar to M35 except M135 leaves the assist gas ON. Discharge current remains OFF until the next G84 or G85. M35 acts like M135 when the SPEED GAS option in the Variables menu is selected. M135 leaves the gas ON independent of the SPEED GAS selection.

CINCINNATI macro programs simplify programming for common applications. The macros are in two groups: grid macros and cutting macros. Grid macros call a user's sub-program in a rectangular pattern of rows and columns. Cutting macros cut common shapes based on specified dimensions.

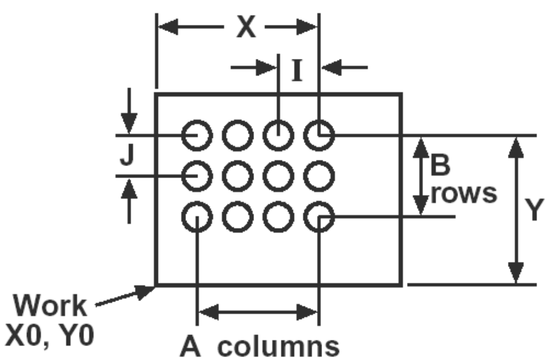
4.65 GRID MACROS

Programs use CINCINNATI grid macros to repeat a sub-program in a pattern of rows and columns. The sub-program can repeat a feature within a part, or repeat a part on a sheet.

PART SUB GRID MACRO G65 P9800

Programs can use the P9800 grid macro to repeat a part feature in a rectangular grid pattern. A typical application is a part with an array of holes or slots. The user must provide a separate sub-program to cut one feature. The part program calls the grid macro once, and the grid macro calls the sub-program several times to cut the features.

G65 P9800 A_ B_ I_ J_ X_ Y_ S_ (R_ K_)



(In this figure, the grid macro calls the sub-program at the center of each hole.)

A = Number of sub-program calls in local X direction (columns).

Note: The macro call should include a space between "9800" and "A" in order to separate "A" from the macro program name. However, revised CNC software (installed July 2001 or later) does not require a space between the program number and the first argument if a G65 command specifies P9800 or P9900.

B = Number of sub-program calls in local Y direction (rows).

I ,J = Local X and Y distances between sub-program calls.

X, Y = Local X and Y coordinates where the grid macro calls the sub-program farthest from local X0, Y0.

S = Sub-program name.

Since the macro call provides the sub-program name as a macro argument, the name must be an integer number with no extension or leading zeroes.

Note: *Revised software (installed July 2001 or later) will ignore leading zeroes in the sub-program name if necessary to find the specified sub-program number.*

The sub-program can be in the same file as the calling program or a separate file in the same directory. If the sub-program is a separate file and the filename has an extension, the grid macro will not find the sub-program. To rename the file without an extension, use Windows Explorer.

R = Rotation angle for the sub-program relative to the part coordinate system, in degrees. Default angle is zero with counterclockwise positive.

K = Quantity of sub-program calls for the grid macro to skip. Default is zero.

When restarting an interrupted program, the operator can use the "K" argument to make the macro skip some of the sub-program calls. The macro skips the number of calls specified by "K" and begins with the sub-program call at the next position.

G65 P9800 DESCRIPTION

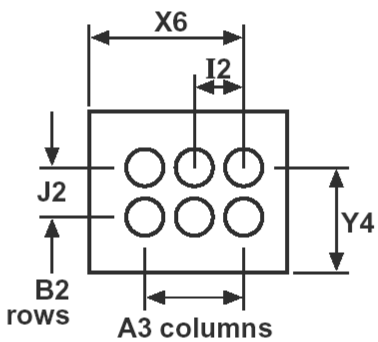
The 9800 grid macro moves the cutting head in rapid traverse to the locations defined by A, B, I, J, X and Y, and calls the sub-program from each location. When the macro call specifies "R", the macro commands coordinate rotation before calling the sub-program. If the sub-program is written is G91 mode, it must end at its starting point.

The 9800 grid macro does not raise or lower the cutting head. The macro maintains the Z-axis position at the end of the sub-program for the move between sub-program calls.

The grid macro calls the first sub-program from work coordinates X, Y when K is zero. To complete the first row, the macro proceeds in the local -X direction. The second row begins under the first part at a lower Y coordinate, and repeats in the local -X direction. This procedure continues until the macro completes all rows.

EXAMPLE PROGRAM USING P9800

This example part has six holes 1.5 inch diameter, on 2 inch centers in 3 columns and 2 rows in the center of an 8 x 6 rectangle.



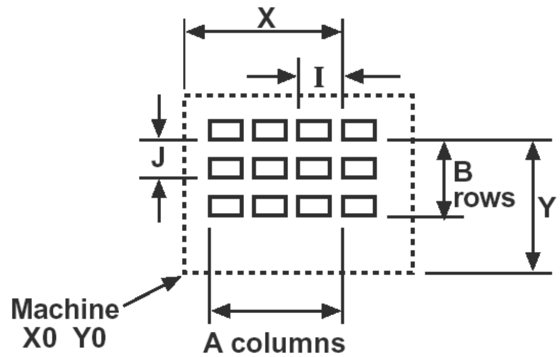

```
O2000  (PART PROGRAM WITH 9800)
G20 G90 F100
G89 PMS135.LIB
G92 X0 Y0
G65 P9800 A3 B2 I2 J2 X6 Y4 S2001 R0 K0
G86 X4 Y3 I8 J6
M30

O2001  (HOLE SUB-PROGRAM)
G91
G73 X0 Y0 D1.5
G90
M99
```

PART GRID MACRO: G65 P9900

Programs use the 9900 grid macro to repeat a part in a rectangular grid pattern on the sheet. The program calls the grid macro once and the grid macro repeatedly calls a sub-program for one part.

G65 P9900 A_ B_ I_ J_ X_ Y_ S_ (R_ K_ Z_)



(In this figure, the grid macro calls the sub-program from the lower left corner of each part.)

A = Number of parts in Machine X direction (columns).

Note: *The macro call should include a space between "9900" and "A" in order to separate "A" from the macro program name. Revised software (installed July 2001 or later) does not require a space between the program number and the first argument if a G65 command specifies P9800 or P9900.*

B = Number of parts in Machine Y direction (rows).

I, J = Distances between part calls in Machine X and Y directions.

X, Y = Machine coordinates where the grid macro calls the part farthest from machine X0, Y0.

S = Part program name.

Since the macro call provides the part sub-program name as a macro argument, the name must be an integer number with no extension or leading zeroes.

Note: *Revised software (installed July 2001 or later) will ignore leading zeroes in the sub-program name if necessary to find the specified sub-program number.*

The sub-program can be in the same file as the calling program or a separate file in the same directory. If the sub-program is a separate file and the filename has an extension, the grid macro will not find the sub-program. To rename the file without an extension, use Windows Explorer.

R = Rotation angle for the part program (relative to the machine coordinate system), in degrees.

Default angle is zero with counterclockwise positive.

K = Quantity of parts to be skipped before the first is cut. Default is zero.

When restarting an interrupted program, the operator can use the “K” argument to make the macro skip some of the sub-program calls. The macro skips the number of calls specified by “K” and begins with the sub-program call at the next position.

Z = Z-axis flag. Default is “Z1” (to raise head between parts).

When the macro call does not specify “Z”, or specifies “Z” with zero or a positive number, the grid macro commands M47 to raise the cutting head before moving into position to call the sub-program for each part.

When the macro call specifies “Z-1” (or any negative value), the macro still commands M47 before the move to the first part, but does not command M47 between parts in the same row or the same column. The macro commands M47 between the last part in one row and the first part in the next row when the grid has more than one part per row (A>1).

G65 P9900 DESCRIPTION

The 9900 grid macro moves the cutting head in rapid traverse to the locations defined by A, B, I, J, X and Y, and calls the part sub-program from each location.

The macro calls the first sub-program from machine coordinates X, Y when K is zero. To complete the first row, the macro proceeds in the -X machine direction. The second row begins under the first part at a lower Y coordinate, and repeats in the machine -X direction. The macro continues this procedure until it completes all rows.

When the macro returns to the calling program, it maintains the cutting head Z position commanded at the end of the last sub-program.

When the macro call specifies “R”, the macro commands coordinate rotation before calling the sub-program. The macro rotates the part about its starting point. Therefore, the programmer must consider the resulting position of the starting point when determining the I, J, X and Y values.

The grid macro commands G92 X0 Y0 before rotating the coordinate system. After a main program calls the grid macro with R > 0, the main program should use G53 to move between parts, or re-establish the work coordinate system with G92.

To avoid an extra move at the start of each part, the sub-program should begin with the cutting head already at the coordinates of the first pierce. If programmed in absolute (G90) mode, local X0 Y0 should be the location of the first pierce.

EXAMPLE PROGRAM

This program calls sub-program 1001 to cut a part 24 times in a pattern with 6 columns and 4 rows. The macro calls the first part first part from machine X48, Y24 and the part spacing is 6 inches in X and 4 inches in Y.

```
O1000  (MAIN PART GRID WITH 9900)
G20 G90 F100
G89 PMS135.LIB
G65 P9900 A6 B4 I6 J4 X48 Y24 S1001 Z1
M30
```

```
O1001  (PART SUB-PROGRAM)
G92 X0 Y0
G0 X_ Y_
G84
 .
 .
M35
M99
```

CUTTING MACROS

MACRO NAME	G-CODE	APPLICATION
Hole	G73	Part Feature
Slot	G76	Part Feature
Line	G79	Sheet Cutoff
Outside Circle	G83	Part Outline
Outside Rectangle	G86	Part Outline
Bolt Circle	G88	Part Features
Shape	G104	Feature or Outline
Lead-In	G105	Begin Cut Sequence

The cutting macros internally call G84 to begin each cut. G79 can also call G85. G84 or G85 then uses the cutting parameters specified by G89, G102 and G103.

The user’s program calls each macro with a series of arguments to specify programming options. This manual shows optional arguments in parentheses.

The “X” and “Y” arguments for G79 are machine coordinates. For the other cutting macros, “X” and “Y” are absolute work coordinates when called in G90 mode, and incremental distances when called in G91 mode.

The only cutting macro that raises the head before its first move is G79. The others rely on the cutting head position (established by the calling program before the macro call) to avoid interference from tipped slugs or clamps.

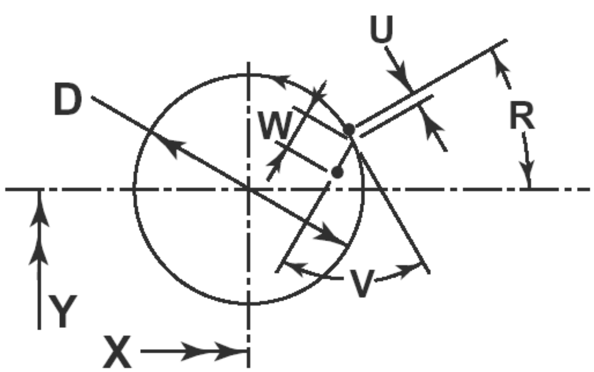
Except for G79, the cutting macros automatically apply kerf compensation (set by G89 K_ or the parameter library file).

G79 returns from the end of the cut and G105 returns from the end of the lead-in. The others return from the end of the cut when called in G90 mode, or from the center of the contoured shape when called in G91 mode.

Before returning to the calling program, G79 always raises the cutting head. G105 returns to the calling program with the head down and the beam ON. The user can program the other macros to either raise the head or leave it down when returning to the calling program.

4.73 G73 HOLE MACRO

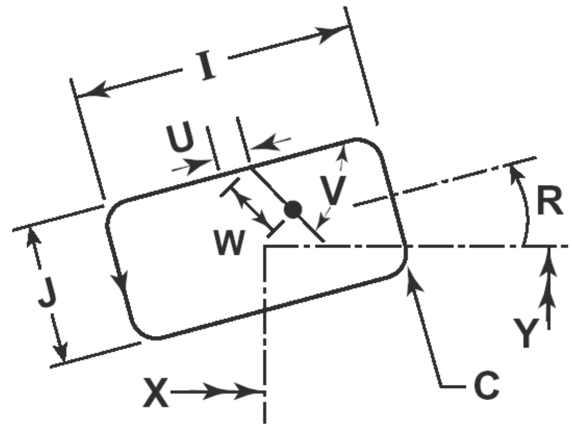
G73 X_ Y_ D_ (A_ B_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)



X, Y = Hole center X, Y coordinates
D = Hole diameter
Others = Optional Arguments, see G104.

4.76 G76 SLOT MACRO

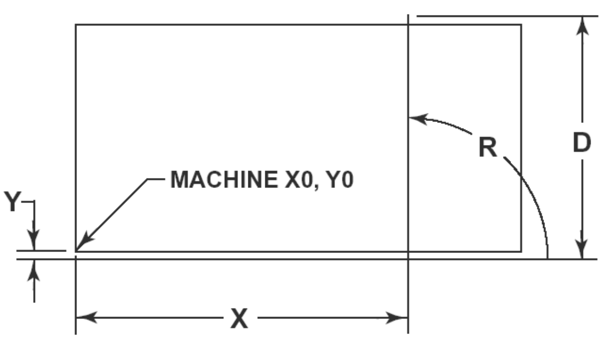
G76 X_ Y_ I_ J_ (A_ B_ C_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)



X, Y = Slot center X, Y coordinates
I, J = Slot overall dimensions in local X and Y directions (as if R =0)
C = Corner radius, default =0
Others = Optional Arguments, see G104.

4.79 G79 LINE MACRO

G79 X_ Y_ D_ R_ (E_ H_)



X, Y = Machine coordinates at start of cut
D = Length (distance) of cut
R = Rotation angle for cut direction, in degrees. Examples:
R0 = Machine +X direction
R90 = Machine +Y direction
R180 = Machine -X direction
R-90 = Machine -Y direction

E = Edge start flag:
E0 uses G84 (default).
E1 uses G85 (no pierce).
H = Optional pressure flag, see G104.

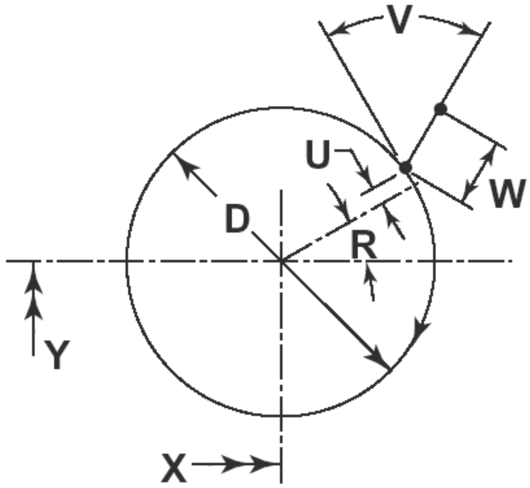
DESCRIPTION

A program can use G79 to cut a sheet, usually to separate the skeleton from a usable remnant or to cut the skeleton into smaller pieces for easier removal. The G79 macro raises the cutting head, moves to machine X, Y in rapid traverse and begins the cut with G84 or G85. G79 then commands M67 if H1 is set, completes the cut at the program feedrate, ends the cut with M35 and retracts the cutting head with M42.

Note: *G79 does not use a lead-in and does not check for interference with sheet clamps or material stops.*

4.83 G83 OUTSIDE CIRCLE MACRO

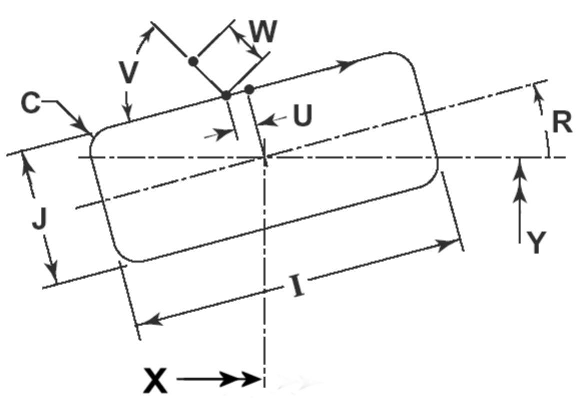
G83 X_ Y_ D_ (A_ B_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)



X, Y = Circle center X, Y coordinates
D = Circle diameter
Others = Optional Arguments, see G104.

4.86 G86 OUTSIDE RECTANGLE MACRO

G86 X_ Y_ I_ J_ (A_ B_ C_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)

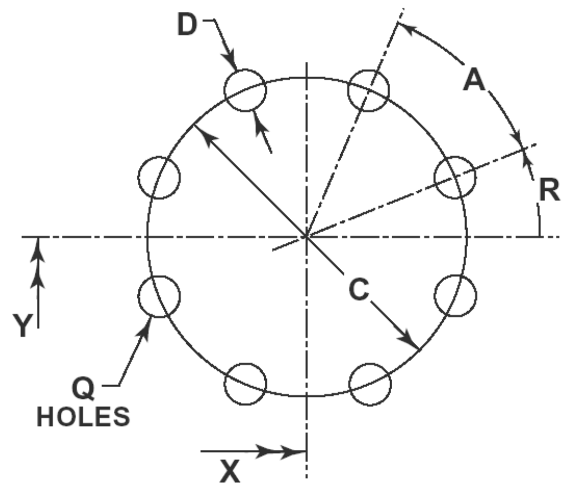


X, Y = Rectangle center X, Y coordinates

I, J = Rectangle overall dimensions in local X and Y directions (as if R =0)
C = Corner radius, default =0
Others = Optional Arguments, see G104.

4.88 G88 BOLT CIRCLE MACRO

G88 X_ Y_ C_ D_ Q_ (A_ B_ F_ H_ K _ M_ R_ T_ U_ V_ W_ Z_)



X, Y = Bolt circle center X, Y coordinates
C = Bolt circle diameter
D = Hole diameter
Q = Number of holes

Since G88 uses “Q” for the number of holes, the hole macro called by G88 cannot use “Q” for the number of segments after type “M2” lead-in (See G104). When a program calls G88 with “M2”, the hole macro uses the default number of segments.

A = Angle between holes, in degrees.
Default = [360 / Q].

The G88 macro call uses “A” when the application only requires part of a complete bolt circle. Since G88 uses “A” for this function, the hole macro called by G88 cannot use “A” for the first dwell of type “M2” lead-in. When a program calls G88 with “M2”, the hole macro uses the default first dwell.

R = Rotation angle from local +X axis to first hole, in degrees. The default angle is zero. Counterclockwise is positive.

Since G88 uses “R” for rotation angle, the hole macro called by G88 cannot use “R” for the lead-in position of each hole. The hole macro uses the default lead-in position for all holes.

Others = Optional Arguments, see G104.

DESCRIPTION

G88 moves to the specified X and Y coordinates in rapid traverse, and then calls G104 to cut each hole at its calculated coordinates. The macro raises the cutting head between holes if programmed by the Z argument (see G104). G88 returns from the center of the last hole when called in G90 mode, and from the center of the bolt circle when called in G91 mode.

4.104 G104 SHAPE MACRO

A program can use G104 to cut circular or rectangular shapes as internal cutouts or part outlines. Since G73, G76, G83 and G86 call G104 to produce their shapes, the programmer can avoid one level of sub-program nesting by using G104 instead of those macros.

The program calls G104 with different arguments depending on the desired shape and lead-in type:

Circles:

G104 X_ Y_ D_ (A_ B_ E_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)

Rectangles:

G104 X_ Y_ I_ J_ (C_) (A_ B_ E_ F_ H_ K_ M_ Q_ R_ T_ U_ V_ W_ Z_)

X, Y = Shape center coordinates
D = Circle diameter
I, J = Rectangle dimensions.
"I" = local X, "J" = local Y.
The macro interprets "I" and "J" as if "R" was zero.

C = Rectangle corner radius, default = 0

E = External cut flag:
E0 = internal (default),
E1 = external (for part outlines)

F = Feedrate for lead-in. Minimum is 30 IPM or the program feedrate if lower. Default and maximum depend on "M":

For M0: "F" is the lead-in feedrate. Default is 30 IPM. Maximum is contouring feedrate.

For M1: "F" is the feedrate of the first move after the lead-in. Default and maximum is the program feedrate. The macro commands the lead-in using three steps with feedrates of 20, 40 and 60 % of "F". The "U" move is at 80 % of "F".

For M2: "F" is the feedrate of the final lead-in move (see M2 description below). Default and maximum is the contouring feedrate.

H = Optional pressure flag:
H0 = Off (default), H1 = On. G104 commands M67 at the end of the lead-in when the macro call specifies "H1".

M = Lead-in type:
M0 = Single feedrate (default)
M1 = Multi-step at increasing feedrates
M2 = Cross lead-in followed by segments with increasing feedrates

R = Shape rotation angle in degrees. Default is zero. Counterclockwise is positive.

For circles: "R" defines lead-in position, where default is local +X intersection.

Note: *Since G88 uses "R" for the angle from +X to the first bolt hole, all G88 bolt holes have the default lead-in position.*

T = Contouring accuracy tolerance. Default is .001 inch (.025 mm).

G104 calculates the feedrate for arcs and circles based on radius and tolerance. The macro uses the calculated feedrate unless it exceeds the modal program feedrate. The programmer can use "T" to affect the calculated feedrate.

Since the CINCINNATI cutting macros use the "T" argument for radius tolerance, they cannot command pierce options with T1, T2 and T3 like G84. Instead, cutting macros can specify the pierce option with a decimal digit after the "M" argument. For example, to command a hole with a single entity lead-in (type M0) and pierce option 2, program: G73 X_ Y_ D_ M0.2

The default pierce option is 1. For example, (G73) M2 is the same as M2.0 or M2.1. Sheet cutoff macro G79 uses the decimal digit of "M" for pierce option even though G79 has no lead-in (G79 ignores the ones digit of M).

U = Length of last lead-in entity:

For M0 and M1: "U" is the length of the last lead-in move, which is commanded in line with the first contouring move. Default and minimum length is one kerf width.

For M2: "U" is the length of each contouring segment inserted after the lead-in. Default and minimum length is .080 inch.

For circles with M2: maximum U = [π * D] / [4*Q] (The arc segments are always completed in the first quadrant.)

For rectangles with M2: maximum U is the distance from the center of the longer side to the start of the corner radius, divided by Q.

V = Angle between lead-in line and first contouring move, in degrees. Default is 90 degrees (perpendicular lead-in).

The macro also uses the default angle when the call specifies "V0". Use "V360" to command the lead-in in the same direction as the first contouring move.

W = Lead-in length.

For M0 and M1: default "W" is the smaller of .25 inch or half the minimum width of the shape.

For M2: default "W" is one fifth of the minimum width of the shape, but not more than 0.2 inches. Minimum "W" is eight kerf widths.

Z = Z-axis retract flag.

When the macro call does not include "Z", or specifies "Z" with zero or any positive number, G104 commands M47 to raise the Z-axis after cutting the shape. To leave the cutting head on the material, call G104 (or G73 etc.) with "Z-1".

ARGUMENTS ASSOCIATED WITH M2 ONLY

A = Dwell (seconds) before lead-in move. Default = 0.250 seconds. The macro also uses the default dwell if the call specifies "A0".

Note: *G88 uses "A" for the angle between holes. When a program calls G88 with M2, this dwell is always 0.250 seconds.*

B = Dwell (seconds) after lead-in move, before the first segment. Default = 0.060 seconds. The macro also uses the default if the call specifies "B0".

K = Dwell (seconds) after contour. Default = 0.060 seconds. The macro also uses the default if the call specifies "K0".

Q = Number of segments after lead-in. Range = 1 to 5, Default = 3

Note: *G88 uses "Q" for the number of holes. When a program calls G88 with M2, the number of segments is always 3.*

G104 DESCRIPTION

G104 moves the cutting head to the pierce location in rapid traverse and calls G84. For rectangles, the lead-in ends at the center of the longer side. G104 completes the lead-in, commands M67 if the call specified "H1", contours the shape with kerf compensation, and ends the cut with M35. G104 then commands M47 (unless called with "Z-1") and returns.

M2 LEAD-IN DESCRIPTION

The "M2" lead-in was developed to improve cutting in heavy steel plate. This method uses a "cross" lead-in, cutting a plus sign shape before the actual lead-in move. The size of the cross is one fourth of the lead-in length. The macro commands the cross moves at 7 IPM (178 mm/min).

After completing the cross shape, the macro commands a dwell before beginning the lead-in move. The macro call can specify this dwell with the "A" argument (except for G88).

The macro call can set the lead-in length with "W" and the lead-in angle with "V". Lead-in feedrate is the contouring feedrate unless the macro call specifies a lower feedrate with "F". The macro commands another dwell at the end of the lead-in, with a duration set with "B".

Instead of beginning the shape at the contouring feedrate, the M2 method first divides a portion of the first contouring entity into segments and commands them at increasing feedrates. The macro call can set the number of segments with "Q" (except for G88), and the length of each segment with "U".

Each segment feedrate is a percentage of the contouring feedrate, based on the number of segments. The macro also maintains a minimum feedrate of 15 IPM.

The macro commands a third dwell at the end of the shape, to complete the return to the lead-in point. The macro call can set this dwell with "K".

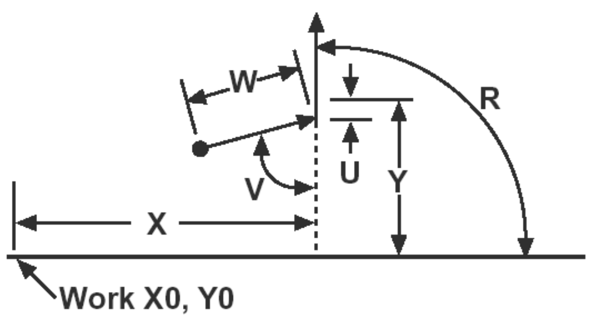
M2 Lead-In Segment Feedrates (% of contouring rate):

Q	Segment				
	1st	2nd	3rd	4th	5th
1	50%				
2	40%	80%			
3	30%	55%	80%		
4	20%	40%	60%	80%	
5	20%	35%	50%	65%	80%

4.105 G105 LEAD-IN MACRO

G105 provides the flexibility of a programmable lead-in to user-programmed contouring paths. The operator can change the length, angle, speed and type of lead-in by editing the G105 macro call arguments at the machine.

G105 X_ Y_ R_ (A_ B_ C_ F_ H_ M_ Q_ U_ V_ W_)



X, Y = Work coordinates for end of lead-in.

R = Direction angle (in degrees) of the first contouring move after the lead-in. For “R0”, the lead-in is parallel to the local +X direction. The macro interprets positive “R” as counter-clockwise.

C = Compensation direction. Default is “C41” (for G41 = left side). “C42” is for G42 = right side.

F = Feedrate for lead-in (see G104).

H = Optional pressure flag:
H0 = Off (default), H1 = On. G105 commands M67 at the end of the lead-in when called with “H1.

M = Lead-in type: (See G104 descriptions)
M0 = Single feedrate (default)
M1 = Multi-step at increasing feedrates
M2 = Cross lead-in followed by segments with increasing feedrates

U = Length of the last lead-in entity.

For M0 and M1: “U” is the length of the last lead-in move which G105 commands in line with the first contouring move. Default and minimum length is one kerf width.

For M2: “U” is the length of each contouring segment inserted after the lead-in. Default and minimum length is .080 inch. G105 commands the M2 segments in the same direction specified by “R”, with the last segment ending at the G105 X and Y coordinates.

V = Angle between lead-in line and first contouring move, in degrees.

Default “V” is 90 degrees (perpendicular lead-in). G105 also use the default if called with “V0”. Use “V360” to command the lead-in in the same direction as the first contouring move. Positive angle is toward the beam side.

W = Lead-in length. Default length is 0.25 inches. For M2, minimum “W” is eight kerfs.

ARGUMENTS ASSOCIATED WITH M2 ONLY

A = Dwell (seconds) before lead-in move. Default = 0.250 seconds.

B = Dwell (seconds) after lead-in move. Default = 0.060 seconds.

Q = Number of segments after lead-in. Range = 1 to 5; default = 3

G105 DESCRIPTION

G105 calculates the starting position based on R, U, V, W, X and Y. The macro moves the nozzle to that position and calls G84 to start the cut. G105 contours the lead-in with kerf compensation and returns with the beam ON.

SECTION 5 PROGRAM STRUCTURE

CNC programs are instructions for motion interpolation, machine functions and program control. CNC programs for the CINCINNATI INCORPORATED Laser System are ASCII text files written in the ISO format (International Standard Organization).

CNC programs execute from the program name down to the end of program statement. Each line is one CNC block that may contain statements, expressions, program control, G-code or M-code commands. Each block can have a line number (also called the sequence number) assigned to it.

A program contains the following items:

- ♦ Program Name
- ♦ Program Body
- ♦ Optional Comments
- ♦ Optional Line Numbers
- ♦ Optional Block Delete Character (per line)
- ♦ End of Program (M02 or M30), or End of Sub-program (M99)

5.1 PROGRAM NAME

Program names are required for sub-programs included in the same file with other (main or sub) programs. Otherwise, program names are optional. The block containing the program name can begin with either a colon character (:) or the letter O, followed by a string of up to 128 alphanumeric characters (see Notes). The end of the name is determined by a space.

Note: *There is one case where the program name cannot include alphabetic characters: The program name must be an integer number (maximum 9 digits) when the program is a sub-program called by a macro and the macro call specifies the sub-program name as a macro argument.*

Revised CNC software (installed July 2001 or later) ignores leading zeroes in the subprogram name if necessary to find the specified subprogram number.

If the program name does not have correct syntax, the control stops loading the program and displays the message: PROGRAM NUMBER SYNTAX ERROR.

5.2 PROGRAM BODY

The program body contains one or more CNC blocks. When the program is displayed as a text file, each block is one line of text. At the end of every block is the ISO command for line feed.

When the operator loads a program, the control checks the syntax of each block. If a block contains a syntax error, the control displays the message: INCORRECT SYNTAX FOR LASER APPLICATIONS.

A block of code can be a:

- ♦ CNC move command (G-code)
- ♦ CNC modal command (G-code or Feedrate)
- ♦ Math function (variable assignment)
- ♦ Logic statement (IF [...] THEN ... , GOTO ... etc.)
- ♦ Machine (M-code) function
- ♦ Macro call

Normally the control executes each block before proceeding to the next block. However, when a block assigns a math function to a variable, the control may look ahead and begin processing the math function during the execution of preceding blocks.

5.3 BEAM ON AND OFF COMMANDS

To begin a laser cutting sequence, move the cutting nozzle to the starting location and command G84 (or G85) in the next block. End the cut sequence with M35 (or M37 or M135).

Example:

```
G00 G90 X10.5 Y6.25
G84  (BEAM ON)
G41
G01 X10.75 F100
G03 X10.75 Y6.25 I-.5 J0 F300
M35  (BEAM OFF)
G00 X11.5
```

5.4 PROGRAM COMMENTS

Because CNC program statements are usually not direct readable text, it is sometimes convenient to add comments to the program. For example, the top of a program may contain set-up instructions for the operator.

Comments can be added to the program by enclosing the comment characters between parentheses ( ), or by starting the comment with an apostrophe (‘).

When the control reads a left parenthesis in a program block, it interprets all characters to the right as a comment, until it finds a right parenthesis. The block terminates with the end of block character (line feed). When the control reads an apostrophe in a program block, it interprets all characters to the right as a comment, until it finds the end of block character.

Examples:

G92 X0 Y0 (SET WORK COORD.)
M98 P2000 ‘ CALL PROGRAM 2000

If the comment contains a syntax error, the control displays the message: COMMENT SYNTAX ERROR.

5.5 PROGRAM LINE NUMBERS

When a block directs execution to another block with a GOTO or M99 P command, the destination block requires a line number. Otherwise, line numbers are optional. To assign a line number, begin the block with the character N followed by an integer. The line number has a useable range of 1 to 999999. The block may contain the block delete character before the line number.

Example: N2300 G01 X50

(This example assigns line number 2300.)

The CNC RUN WINDOW displays line numbers with the program text. If a line number contains a syntax error, the control displays the message: LINE NUMBERING SYNTAX ERROR.

5.6 BLOCK DELETE

The operator can control the execution of a block with the BLOCK DELETE function. When a block begins with the / (forward slash) character, the control does not execute the block if the BLOCK DELETE button is selected (highlighted) on the CNC LASER control window. The operator can toggle this function ON or OFF any time during the execution of the CNC program.

5.7 END OF PROGRAM

M02

M02 completes execution of a program. All previously requested M-codes are reset and local variables are set to zero. However, the CNC RUN WINDOW does not return to the top of the program.

M99 (P_)

The M99 block completes execution of a sub-program. The CNC RUN WINDOW returns to the program that called the subprogram and displays the next block.

M30

The M30 block completes execution of a program. All previously requested M-codes are reset, local variables are set to zero and the CNC RUN WINDOW returns to the top of the program.

5.8 SUB-PROGRAMS AND MACROS

Sub-programs are separate CNC programs that execute when called by another program. The sub-program returns to the calling program when finished. The control maintains modal conditions established in the calling program unless the sub-program changes the modal condition.

Sub-programs are useful for repeating a series of commands. For example, a main program can use a sub-program to repeat a part on a sheet.

There are two types of sub-program calls, M98 P_ and G65 P_. The call specifies the sub-program name after “P”. If the sub-program is in the same file as the calling program, then the call only needs the program name after “P”. If the sub-program is in a separate file, the call should command “P” followed by the filename including its extension (if any) and its path if different from the calling program.

Each type of sub-program call has different properties and applications.

M98 P__ (L_)

G65 P__ (L_) (A_ B_ C_ D_ etc.)

A single program block can call a sub-program more than once, by commanding M98 or G65 with “L” followed by the number of times to repeat the sub-program.

Example:

M98 P1000 L3

(This example calls a sub-program named “1000” three times before returning.)

LOCAL VARIABLES

The major difference between M98 and G65 is the treatment of local variables. When a program calls a sub-program with M98, the two programs share the same set of local variables (#1 through #99). In other words, “local” variables become “common” variables between a program and a sub-program called with M98. The control assigns undefined local variables a value of zero.

When a program calls a sub-program with G65, the sub-program has its own set of local variables. The calling program can assign values to most of the variables #1 through #26 in the sub-program by including arguments in the G65 call. This allows the calling program to pass data to the sub-program without assigning separate common variables.

Each argument is a letter followed by a numerical value for its corresponding local variable. The calling program cannot use arguments “G”, “L”, “N”, “O” or “P”. The sub-program can use local variables that would correspond to G, L, N, O and P, but the calling program cannot assign their values with arguments in the G65 call.

Since program names in the CINCINNATI control can have alphabetic characters, the G65 block requires a space between the last character of the program name and the first argument letter.

Examples:

G65PMYSUBA1

(This example calls a sub-program named “MYSUBA1”.)

G65PMYSUB A1

(This example calls sub-program “MYSUB” with argument A equal to 1.)

Note: *Revised software (installed July 2001 or later) does not require a space between the program number and the first argument if a G65 command specifies P9800 or P9900.*

To be consistent with local variable assignments used by other CNC controls, the CINCINNATI control assigns arguments D through K to local variables out of sequence with their alphabetical order. This table shows how the control assigns local variables to G65 arguments:

Argument	Local Variable
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Example:

G65 P2000 X12.5 Y3.5

(This example calls a sub-program named “2000”. When the program named “2000” starts, it has local variables #24 = 12.5 and #25 = 3.5.)

The format for sub-programs is the same as other programs, except sub-programs use M99 instead of M30 for the End of Program statement. If a sub-program does not end with M99, the control displays the message: SUB-PROGRAM CALL WITHOUT A RETURN STATEMENT.

NESTED SUB-PROGRAM CALLS

Sub-programs can call other sub-programs with M98 or G65, until the total calls are nested 10 deep.

Note: *CINCINNATI cutting macros and grid macros (See Section 4) are sub-programs and contain sub-program calls. The total nesting limit available to the user is reduced when a program calls these macros.*

This table shows how CINCINNATI macro calls affect sub-program nesting:

Macro	Equivalent Sub-program Calls
G73	3
G76	3
G83	3
G86	3
G88	3
G104	2
G65 P9800	2
G65 P9900	2

6.1 LOCAL AND COMMON VARIABLES

The operator can display and edit local and common variables by opening the "Local/Global" window from the CNC Laser "Variables" menu.

CAUTION

Editing variables while a program is running can cause improper execution and is strongly discouraged.

Note: *The "Local/Global" window will display variables with the current values they have in the control buffer. However, the buffer may be several blocks ahead of the currently executing CNC program block.*

LOCAL VARIABLES: #1 - #99

Each program has 99 local variables named #1 through #99. Each sub-program called with G65 also has its own set of local variables. Sub-programs called with M98 share the same set of local variables with the calling program. All local variables are zero by default and return to zero after M30 or Program Rewind.

COMMON VARIABLES: #100 - #999

All programs and sub-programs share a set of common variables named #100 through #999. Common variables are not cleared by M30, RESET or turning off control power.

One common variable can be reserved for use by CINCINNATI INCORPORATED. The Maintenance Configuration window has an option checkbox for "Process Library Feedrate". If that option is selected, G89 writes the material feedrate parameter from the library file into a common variable. Another Configuration setting specifies the variable number (148 by default). G89 sets the variable with a value consistent with the G20 / G21 (inch / metric) units mode in effect when the program calls G89. However, the variable does not change if the program changes G20 / G21 mode after calling G89.

6.2 SYSTEM VARIABLES

System variables give the programmer the ability to read and write information for special functions in the CNC.

There are four types of System Variables:

SYSTEM VARIABLES	DESCRIPTION
#2000 - #2999	Offset Data
#3000 - #3999	CNC Data
#4000 - #4999	Modal Data
#5000 - #5999	Coordinate Data

OFFSET DATA SYSTEM VARIABLES

KERF OFFSET: #2000

A CNC program statement can read or write the cutter radius value used for kerf width compensation with #2000. The variable value is consistent with the active units mode (G20 inch, G21 mm).

A CNC program can read the active compensation value by assigning the value of #2000 to a program variable or using #2000 in a math or logic statement. The program can change the active compensation value by calling G89, or by assigning a value to variable #2000.

WORK COORDINATE SYSTEM OFFSETS

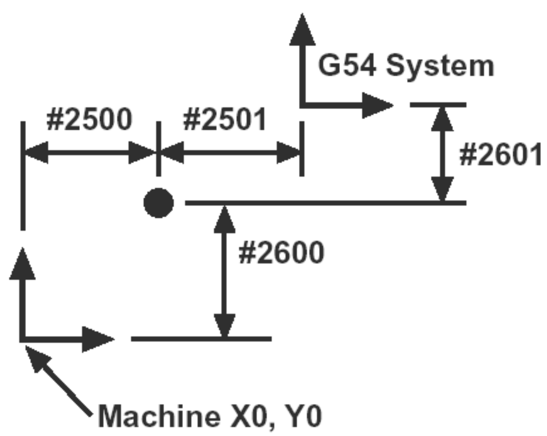
A CNC program can read the value of a Work Coordinate Offset by assigning the value of its system variable to a program variable, or by using the system variable in a math or logic statement. The program can also change the Work Offsets by assigning values to the system variables. The control interprets the variables in the active units (G20 inch, G21 mm).

SYSTEM VARIABLE	Work Offset Group	Axis
#2500	External	X
#2600	External	Y
#2501	1 (G54)	X
#2601	1 (G54)	Y
#2502	2 (G55)	X
#2602	2 (G55)	Y

SYSTEM VARIABLE	Work Offset Group	Axis
#2503	3 (G56)	X
#2603	3 (G56)	Y
#2504	4 (G57)	X
#2604	4 (G57)	Y
#2505	5 (G58)	X
#2605	5 (G58)	Y
#2506	6 (G59)	X
#2606	6 (G59)	Y

External work offsets (#2500 and #2600) are normally zero. Programs use #2500 and #2600 to change the distance from machine X0, Y0 to a common reference point for the G54 through G59 work coordinate systems.

Example:



CNC DATA SYSTEM VARIABLES

Clock Variables:

Date:	#3011	YYYYMMDD
Time:	#3012	HHMMSS
Day:	#3013	D

The control maintains these system variables for clock functions. A CNC program statement can assign their value to a variable or use the system variables in a math or logic statement. A program cannot change these system variables by assigning a value to the variable.

If the syntax of a variable assignment statement is incorrect then the control displays the message: VARIABLE ASSIGNMENT ERROR.

CNC DATE: SYSTEM VARIABLE #3011

The CNC sets system variable #3011 with the date and calendar year. The variable value is an eight-digit integer with the year, month and date separated as shown:

yyyymmdd

The year can be 0 to 9999, month is 01 to 12, and date is 01 to 31.

CNC CLOCK: SYSTEM VARIABLE #3012

The CNC maintains #3012 with the current time from the CNC clock. The variable value is a six-digit integer with the hour, minute and second separated as shown:

hhmmss

The data range for #3012 is 000000 to 235959. The hour is in 24-hour clock format (Military Time), where 000001 is one second after midnight.

WEEKDAY: SYSTEM VARIABLE #3013

Programs can use #3013 to read the Day of the Week. The variable value is a one-digit integer, 1 thorough 7 for Sunday through Saturday.

Sun	Mon	Tue	Wed	Thu	Fri	Sat
1	2	3	4	5	6	7

BEAM ON TIME: SYSTEM VARIABLE #3015

This variable accumulates the elapsed time between Beam ON (G84 or G85) and Beam OFF commands (M35, M37, M42, M47 or M30). The accumulated time includes any dwell times.

System Variable #3015 is a floating-point number in seconds. Resolution is .001 sec.

The control resets #3015 to zero each time the operator starts the CNC Laser Windows application. A CNC program cannot change variable #3015. The Maintenance Statistics window displays Total accumulated Beam ON Time.

To measure the beam-on time during a program, save #3015 in a common variable and subtract the saved value from the new #3015 value before the program ends.

Example:

```
#100 = #3015 (START)
.
.
#100 = #3015 – #100 (END)
```

MODAL DATA SYSTEM VARIABLES

Programs can use the #4000 series system variables in a math or logic statement. A program cannot change a #4000 series variable by assigning a value to the system variable.

SYSTEM VARIABLE	DESCRIPTION
#4001	0, 1, 2, or 3 for G0, G1, G2 or G3
#4003	90 or 91 for G90 or G91
#4006	20 or 21 for G20 or G21
#4007	40, 41 or 42 for G40, G41 or G42
#4008	54 through 59 for G54 through G59
#4011	50 or 51 for G50 or G51
#4015	61 or 64 for G61 or G64
#4016	68 or 69 for G68 or G69
#4109	Modal Feedrate for G1, G2 & G3
#4114	Value following address N in the last executed block. If the last block did not specify N, then #4114= - 1.

COORDINATE SYSTEM VARIABLES

A program can determine the machine or work coordinates of the last completed block by reading system variables. The control maintains the variables in the same units as the active units mode (G20 inch, G21 mm). The program cannot change these system variables by assigning a value to the variable.

Machine Coordinates:

#5021 Machine X position
#5022 Machine Y position
#5023 Machine Z position

#5061 Position Capture Machine X position
#5062 Position Capture Machine Y position

Work Coordinates:

#5041 Work X position
#5042 Work Y position
#5043 Work Z position

7.1 MATH FUNCTIONS

CNC programs can use math functions to assign a calculated value to a variable, or to substitute a calculation for a numerical value.

In the following examples, “a” represents a variable and “b” and “c” represent variables, constants or functions.

Function	Example
Assignment	a = b
Addition	a = b + c
Subtraction	a = b - c
Multiplication	a = b * c
Division	a = b / c
Binary Addition	a = b OR c
Binary Subtraction	a = b XOR c
Binary Multiplication	a = b AND c
Sine	a = SIN [b]
Cosine	a = COS [b]
Tangent	a = TAN [b]
Arc Tangent	a = ATAN [b]
Square Root	a = SQRT [b]
Rounding	a = ROUND [b]
Truncating	a = FIX [b]
Add 1 for fraction	a = FUP [b]
Absolute Value	a = ABS [b]
BCD to Binary	a = BIN [b]
Binary to BCD	a = BCD [b]

The CNC reads the math statement, checks for the correct syntax and evaluates the functions by order of precedence. The default order of precedence is: functions (SIN, SQRT, etc.), multiplication and division, addition and subtraction.

If the syntax is in error, or the statement attempts division by zero, the control displays the message: MATH STATEMENT ERROR.

SIN [ ] Sine of an angle.

COS [ ] Cosine of an angle.

TAN [ ] Tangent of an angle.

The SIN COS and TAN functions interpret the specified angle [ ] in degrees.

ATAN [ ] Arc-tangent of an expression.

The result of the ATAN function is in degrees, between -90 and +90.

SQRT [ ] Square root of a positive number.

Evaluating the square root of a negative number produces an error message.

ROUND [ ] Rounding off a number.

Expressions with fractions 0.5 and above are rounded to the next higher integer. Expressions with fractions below 0.5 are rounded to their integer value.

FIX [ ] Truncating a number.

Any fractional portion is ignored. The number is reduced to its integer value only.

FUP [ ] Add 1 for fraction.

If the number has any fractional portion, the FUP function removes the fraction and adds 1 to the integer portion. If the number is already an integer (no fractional part), it stays the same.

ABS [ ] Absolute Value

ABS [ ] returns the absolute value of a variable or function.

BIN [ ]

The BIN function converts the specified value from Binary Coded Decimal to Binary.

BCD [ ]

The BCD function converts the specified value from Binary to Binary Coded Decimal.

If a syntax error occurs the message FUNCTION ERROR is displayed in the CNC Message Window.

BRACES [ ]

Statements use braces to control precedence of math functions and identify conditional expressions. There is no limit on the number of nested braces.

Example 1: #1 = 3 *[2 + 3]

The braces set the priority of the math functions; addition is performed before multiplication in example 1.

Example 2: #2 = SIN[2 * [3 *[2 + 3]]]

In example 2, the statement completes addition first. The sum is multiplied by 3 and that product is multiplied by 2. The SIN function operates on the product of the multiplication.

Each left brace must have a right brace. If the statement contains an error in brace syntax, the control displays the message BRACE OPEN/CLOSE ERROR.

7.2 LOGIC FUNCTIONS

Logic functions include conditional expressions and program control commands.

CONDITIONAL EXPRESSIONS

A program uses a conditional expression to compare the value of a variable, constant or calculation with another variable, constant or calculation. The program block can use the result of the comparison to direct program flow.

Comparison	Example
Equal	[b EQ c]
Not Equal	[b NE c]
Greater Than	[b GT c]
Less Than	[b LT c]
Greater Than or Equal	[b GE c]
Less Than or Equal	[b LE c]

The CNC evaluates the conditional expression for proper syntax and a true or false condition. The evaluation of a conditional expression occurs from left to right.

If the statement has a syntax error, the control displays the message: CONDITIONAL EXPRESSION ERROR.

PROGRAM CONTROL COMMANDS

A program can determine which blocks the control executes by using the Program Control Commands:

GOTO
IF [ ] GOTO
IF [ ] THEN . . ELSE . . ENDIF
WHILE [ ] . . END

GOTO Statement

The control normally executes program blocks in sequential order (top to bottom through the program). A program can direct the control to execute any numbered block with the "GOTO nnn" command, where "nnn" is the sequence number of the destination block.

A program block specifies its sequence number with "N" followed by an integer number. To direct the control to a block with a sequence number, command "GOTO" followed by the numerical value of the sequence number. The GOTO command does not include the "N" address.

Example:

```
N020 GOTO 50
N030 . . .
N040 . . .
N050 G01 X10 Y5
```

(In this example, the control executes the block with sequence number N050 immediately after the block with sequence number N020.)

If the GOTO block has a syntax error or the program has more than one line with the destination sequence number, then the control displays the message: ILLEGAL GOTO STATEMENT.

IF [ ] GOTO Statement

IF [<conditional expression>] GOTO <line>

If the expression is true, the program jumps to the specified line number. If false, the program continues with the next block.

IF [ ] THEN ENDIF Statement

```
IF [<conditional expression>]
THEN < expression >
ENDIF
```

The CNC evaluates the conditional expression. If true, the control executes the block containing THEN. If the expression is false, the program jumps to the ENDIF block.

The THEN block must be a separate block immediately following the IF block. Each IF THEN pair must be followed by a separate ENDIF block.

Example:

```
IF [#1 EQ #2]
THEN #3 = #4 / #5
ENDIF
```

IF THEN statements can be nested four deep.

If the statement has a syntax error, the control displays the message: IF THEN SYNTAX ERROR.

IF [ ] THEN ELSE ENDIF Statement

```
IF [<conditional expression>]
THEN <expression>
ELSE <expression>
ENDIF
```

If the conditional expression is true, the control executes the block containing THEN and the program jumps to the ENDIF block. If the expression is false, the control executes the block containing ELSE.

The THEN block must be a separate block immediately following the IF block. The ELSE and ENDIF blocks must also be separate blocks.

IF..THEN..ELSE..ENDIF statements can be nested 4 deep.

WHILE Statements

The control accepts two types of WHILE statements:

WHILE . . END Statement:

```
WHILE [<conditional expression>] DO m
.
.
ENDm
```

(In the WHILE . . END structure, "m" is an integer from 1 to 3, used to identify nested loops.)

WHILE . . ENDWHILE Statement:

```
WHILE [<conditional expression>]
.
.
ENDWHILE   (one word)
```

The control evaluates the conditional expression. If true, the control executes the block(s) between WHILE and END (or ENDWHILE), then the program returns to the WHILE block and repeats the process. When the expression becomes false, the program jumps to the block after END (or ENDWHILE).

WHILE loops can be nested up to five deep.

To avoid the possibility of an infinite loop, a block between WHILE and END normally changes the status of the conditional expression.

Example:

```
#1=0
WHILE [#1 LE 50] DO2
.
.
#1=#1+1
END2
```

The blocks between WHILE and END may include a GOTO command to exit the loop.

If the WHILE or END statement has incorrect syntax, the control displays the message: WHILE LOOP SYNTAX ERROR.

7.3 AUXILIARY COMMANDS

DPRNT COMMAND (OPTION)

The program can transmit a text string to the RS-232 port using the DPRNT command.

DPRNT *textstring*

The text string can begin immediately after DPRNT, or the block may include spaces between DPRNT and the first string character. The control reads the text string until it finds a space or end-of-block character. The text string cannot include braces [ ]. If the txt string includes a variable name, the control will only transmit the variable name (the control will not transmit the value of the variable).

When the control processes the DPRNT statement, it adds the DC2, nulls, and DC4 commands, and transmits the string. The control does not use "OPEN" or "CLOSE" commands for DPRNT.

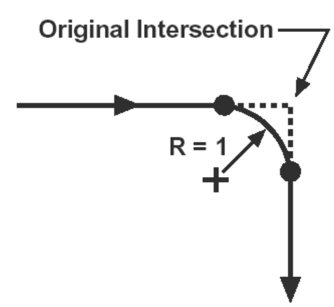
Com4 is the RS-232 port. Users can set the communications parameters (baud rate, bits per character, etc.) with the "Ports" item under Windows NT Control Panel functions. Com4 is wired to a convenience outlet on the side of the control console.

AUTOMATIC CORNER ROUNDING

A program can command a rounded corner by specifying the intersection point of the two side elements and the corner radius. The control inserts a circular arc tangent to any two contouring moves when the first move ends with a comma followed by R and the arc radius. The first and second contouring moves can be G01, G02 or G03.

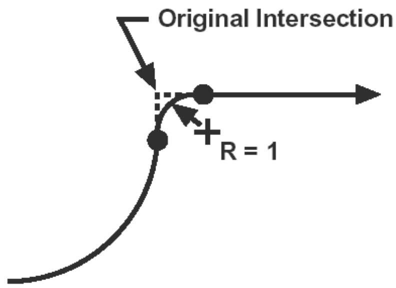
Example:

(G90) G01 X4 Y2, **R1**
G01 Y0



Example:

(G90) G03 X5 Y5 R5, **R1**
G01 X10 Y5

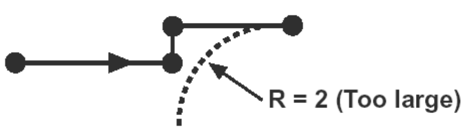


If a contouring block (G01, G02 or G03) does not follow the block specifying corner rounding, the control displays the message: ILLEGAL CORNER ROUNDING COMMAND.

If the control cannot insert the programmed arc tangent to both original lines (or arcs), the control displays the message: CORNER RADIUS TOO LARGE.

Example:

(G90) G01 X5 Y0
G01 X5 Y1, **R2**
G01 X8 Y1



If the angle between the tangent lines is 1 degree or less, the control ignores the corner rounding request and contours the original intersection.

7.4 PROGRAMMING FOR MATERIAL HANDLING OPTION

To use the material handling system (MHS), the operator creates and runs a batch program file in the laser system control. The Batch Program Window displays a list of CNC program files that the control will load and execute in succession. The operator can configure the batch program for up to one hundred different CNC programs and repeat any CNC program up to one hundred times. When creating the batch program, the operator also assigns a material shelf number for each CNC program in the list. Each CNC program can process a different material (limited by the number of available shelves).

CNC programs in the batch must meet the following requirements:

1. Each CNC program processes one sheet (one pallet).
2. The CNC programs do not command M50 to switch pallets.
3. Each CNC program ends with M30 (not M99).

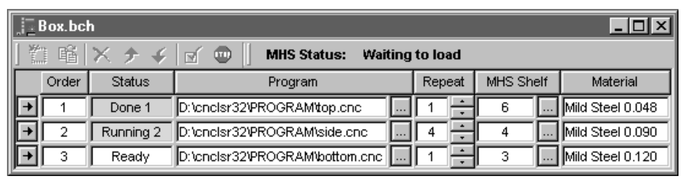
The operator loads the completed batch program. The loaded program starts when the batch document window is the active window and the operator presses CYCLE START. The batch program commands the MHS to load the first sheet on the pallet in the load frame. The batch program does not switch the pallets until the operator presses the illuminated PALLET NOT READY button to approve pallet motion. After the pallets switch positions, the control runs the first CNC program.

While the first CNC program runs, the MHS loads the next sheet on the other pallet. When the next pallet is ready, the operator can clear the "Pallet Not Ready" status so the batch program can switch the pallets when the CNC program finishes.

After finishing the first CNC program, the batch program switches the pallets (if approved by the operator) and then loads the second CNC program. The operator presses CYCLE START to run the second CNC program. While the second CNC program runs, the MHS unloads the first pallet and then loads the next sheet. When the laser system finishes the last CNC program, the batch program switches the pallets (if approved) and the last completed pallet moves to the load/unload position.

Creating and editing a batch file in the laser system control:

1. Select File, New and then select "Batch" for the new file type.



Batch Document Window

2. In the Batch document window, select the New Entry button once for every program added to the list. The New Entry button is at the left end of the tool bar. The list will have one row for each New Entry selection.
3. In each row, enter the program filename in the Program column.
4. In the Repeat column, enter the number of times to run each program.
5. In the Shelf column, enter the MHS shelf number for the intended material.

6. To edit an entire row, use the ROW SELECT button in the left column, then use the toolbar buttons to move up, move down, copy, (insert) New Entry or delete a row.

The batch document window continuously displays the status of each program in the list. The Status column indicates if each program is Ready, Running (with repeat number), or Done.

7.5 WORKPIECE EDGE DETECTION

Programs use this option to find the location and orientation of a sheet of material that was not placed exactly at the machine origin (X0, Y0) or parallel with the machine axes. It uses the noncontact head to detect the coordinates of the material edges. The CNC control uses the edge coordinates to automatically shift and rotate the work coordinate system, effectively aligning the part program with the workpiece. This method allows full utilization of the sheet, regardless of minor variations in material position on the pallet. Users can apply Workpiece Edge Detection with an automated pallet loading system or in cases where the material is not clamped. The noncontact head is the only hardware required to use this option.

HOW IT WORKS

The control finds the material position and orientation by detecting the edge of the sheet at three points. To find the location of an edge, the control first commands the nozzle to a low standoff (called the scan standoff) and locks the Z-axis servomotor. The control then changes the target standoff to a higher value called the detection threshold. The commanded servo lock prevents the Z-axis from moving up to the higher standoff. The control moves the cutting head horizontally in the direction of the expected edge until the measured standoff appears to increase to the detection threshold. This change in the apparent standoff occurs when the nozzle reaches the edge. The control records the X and Y-axis positions when it detects the edge. After adjusting for slight offsets between the detection point and the actual edge, the scan is complete.

SETUP

If the laser system has never used Workpiece Edge Detection, complete the following steps:

1. Open the Maintenance Configuration window, select the Options tab and verify that the checkbox for Workpiece Edge Detection is checked.

2. Set the following common variables:

   #520 = 0.5
   #521 = 7.0
   #529 = 25.4

CALIBRATION

1. Position a 9 x 9 inch (230 x 230mm) or larger scrap workpiece on the worktable. Mild steel in any thickness from 16 gauge to 10 gauge works well.

2. Perform Standoff Calibration (see Section 7 of the laser system Operation, Safety and Maintenance manual).

3. Jog the cutting head to the approximate center of the workpiece and run CNC program cal_edge_det.cnc

   (This program file is in folder D:\ CNCLSR32 \ PROGRAMS \ UTILITIES)

```
(CALIBRATION FUNCTION FOR EDGE DETECT)
(CALCULATES SCANNING OFFSETS FOR)
(NONCONTACT HEAD)
(NOTE:  USE AT LEAST A 9” x 9” PIECE)
(OF MATERIAL, USE CALIPERS TO)
(VERIFY CUT SQUARE IS)
(3.00 x 3.00 ± .003 INCHES)

G90  G20
G89  P  MS060N2.LIB
    (Edit G89 block for appropriate library filename)
M47
G69
#11=#5021
#12=#5022
G92  X#5021  Y#5022
    (G76 cuts a 3” x 3” square)
G76  X#5021  Y#5022  I3  J3  C.1  R0
M0  (Remove square, press CYCLE START)
G65 P9700 A0.020 B0.070 C3 D3 F60 X#11 Y#12
M30
```

   This program cuts a 3” x 3” (76.2 x 76.2mm) square hole in the workpiece. Be sure that the G89 block specifies the correct library filename for the material.

4. When the hole is cut and the program stops, remove the square slug from the hole. Be sure not to bump the workpiece in which the hole was cut. Any movement of the test hole before the calibration scan (Step 5) will cause an equal size error in the edge detection function. (If the workpiece is bumped, the program must be restarted from the beginning.)

5. After removing the square slug, press CYCLE START. The calibration program will then scan the square hole to set up its internal calibration values.

6. When the calibration is complete, measure the dimensions of the 3” x 3” (76.2 x 76.2mm) square hole (not the slug). If the hole dimensions differ from the programmed value by more than 0.002” (0.05mm), correct the kerf value in the material library and/or adjust the focus, then repeat the calibration. Increase the kerf parameter if the hole is too large. Any error in the size of the test hole will cause an equal size error in the edge detection function.

OPERATION

1. Position the workpiece near the “home” corner of the worktable with the longer side of the workpiece along the X-axis, that is, lengthwise on the worktable. The edges of the workpiece can be anywhere from 0 to +2.0 inches (50.8mm) from the “zero” position of the worktable along the X-axis and along the Y-axis.

   Note: *All programmed moves must be within the maximum cutting area of the worktable. If the program attempts a move beyond the machine limits, the program will stop and generate an overtravel alarm.*

2. Insert the following command in the CNC program, after a G89 command and before any move commands:

   G65 P9712 Xn

   In this command, “n” is the length of the workpiece in the X-axis direction.

When the control executes this command, the cutting head will scan one workpiece edge at two points and a perpendicular edge at one point to determine the position of the two edges. The control will then shift and rotate the work coordinate system to make the X and Y axes coincide with the measured edges. The edge detection function ends when the control moves the cutting head to the X0, Y0 corner of the workpiece, in the partial Z-up (M47) position.

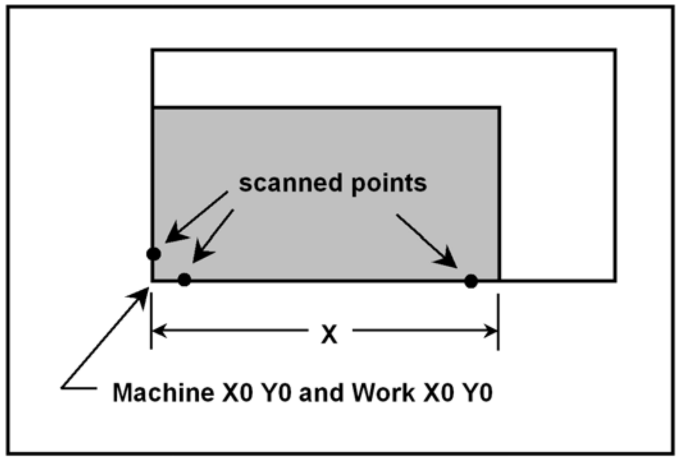


Figure 7-1 Edge Detection using default U0, V0, R0

Optional Functions

When the NC program calls the edge detection macro with only the X argument, the macro finds a workpiece with one corner near the machine X0 Y0 location. The NC program can also use the edge detection macro when the workpiece does not have a corner near machine X0 Y0. To locate a sheet with a corner near machine coordinates other than X0 Y0, the macro call includes the optional arguments U and V. The macro interprets U and V as machine coordinates of the sheet corner where the scanned edges intersect.

When the macro call does not specify the optional R180 argument, the macro locates a sheet with the X and Y work coordinate axes in the same general direction as the machine axes. If the macro call specifies R180, the macro locates a sheet with the X and Y work coordinate axes in the opposite directions from the machine axes. Figures 7-2 and 7-3 show workpiece orientations that would use the optional R, U and V arguments:

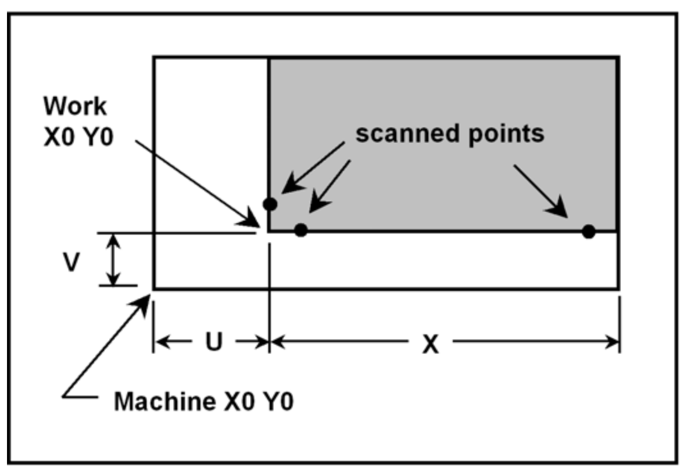


Figure 7-2 Edge Detection using U, V and default R0

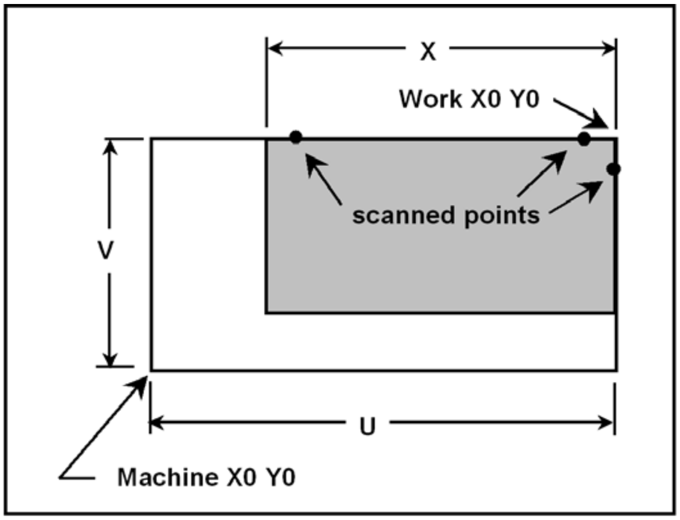


Figure 7-3 Edge Detection using U, V and R180

COMMENTS

1. The edge detection function works in a program using either English units (G20) or metric units (G21).

2. The edge detection function changes the commanded feedrate. After completing each scan, the control resets the commanded feedrate to the value of variable #148. If variable #148 was not defined before the program called the 9712 macro, the macro will return with the commanded feedrate equal to the scan feedrate. If an error occurs, the 9712 macro might not properly reset the original feedrate.

3. The edge detection function changes the pierce and cut standoff setting. To ensure that the program uses the proper standoffs for cutting, always include a G89 or G102 command after the edge detection macro call.

   Example:

   ```
   G89 PMS060N2.lib
   G65 P9712 X120.0
   G89 PMS060N2.lib
   ```

4. To maintain the shifted coordinate system, the user program may not contain any of the following commands after calling the edge detection function: G52, G53, G54 through G59, G68, G69 or G92.

5. After the initial setup and calibration, be sure not to change common variable #529. This variable indicates the current system of units (inch or metric) for the stored calibration values. If this variable changes, repeat the initial setup steps and the calibration procedure.

6. To change the feedrate used when scanning an edge, modify the feedrate used in the calibration routine. The “F” argument in the G65 P9700 macro call (calibration) sets the scan feedrate.

   Note: *High scan feedrate will reduce accuracy.*

SPECIFICATIONS

EDGE DETECTION SPECIFICATIONS	
Position Accuracy	±0.100 inch (2.54 mm)
Sheet Position Tolerance	0 to +2.0 inches (50.8 mm) from machine coordinate zero at the scan points
Scan Feedrate	Maximum programmable = maximum contouring feedrate. Maximum practical = 100 IPM (2540 mm/min).
Scan Standoff	Minimum programmable = 0.010 inch (.254 mm). Minimum practical = 0.020 inch (0.508 mm). Maximum programmable = 0.400 inch (10.2 mm). Maximum practical = 0.100 inch (2.54 mm).
Detection Threshold	Minimum = scan standoff. Maximum= 0.400 inch (10.2 mm). Maximum practical = 0.250 inch (6.35 mm).

TROUBLESHOOTING

Note: *Error messages shown in quotes in this table will appear in the CNC Run Window, at the line where the macro program stopped after finding the error.*

Problem	Possible Causes	Solution
“Standoff out-of-range”	The macro 9700 (Calibrate Sensor) scan standoff argument (A) is outside the acceptable range of 0.010 inches to 0.400 inches.	Correct the “A” argument in the macro call.
“Threshold out-of-range”	The macro 9700 (Calibrate Sensor) threshold argument is less than the scan standoff argument or greater than 0.400”. • The "A argument, the "B" argument, or both are incorrect when calling macro 9700.	Correct the “A” and/or “B” argument in the macro call.
“Move distance too small”	Macro 9701 (Scan) move distance is less than the minimum value set by common variable #520. • Common variable #520 is incorrect.	Set common variable #520 to 0.500 inches (12.7 mm).
“Move distance too large”	The macro 9701 (Scan) move distance is greater than the maximum value set by common variable #521. • Common variable #521 is incorrect.	Set common variable #521 to 7.000 inches (177.8 mm).
“Started too close to edge”	Macro 9701 (Scan) detected the edge within 0.250 inches (6.35 mm) of the scan starting point. If the start point is too close to the edge, the macro does not maintain detection accuracy. • The edge of the workpiece is too close to the start point.	Move the workpiece.
“Edge not detected”	Macro 9701 completed the scan, but did not detect an edge. • The edge of the workpiece is too far from the start point.	Move the workpiece.
“X-axis length too small”	The “X” argument in the 9712 (Align to Sheet) macro call is less than 12 inches (304.8 mm). It is not practical to scan a workpiece less than 12 inches long.	Correct the “X” argument in the macro call.
“Arguments missing”	A block calling macro 9700 (Calibrate Sensor) or macro 9712 (Align to Sheet) does not have the required arguments.	Make sure the arguments in the macro call match the required arguments. See the macro description for the argument list.
Laser-cut shapes are not accurately positioned relative to the workpiece.	• The program commanded G69 after calling the Align to Sheet macro. • The program commanded G92 after calling the Align to Sheet macro. • The nozzle tip was replaced or damaged since the last calibration. • The cutting head was removed and replaced since the last calibration. • The scan feedrate is too fast.	• Modify the program. • Modify the program. • Repeat the calibration. • Repeat the calibration. • Repeat the calibration with a lower scan feedrate.
The cutting head missed the workpiece during a scan.	A scanned workpiece edge is more than 2.0 inches from the Machine X=0 or Y=0 position.	Move the workpiece.

TECHNICAL INFORMATION

This section includes additional technical information not needed for normal operation of the workpiece edge detection function. This information is provided as a service aid to help debug problems that might occur in special applications. This section includes listings of the common variables and system variables used by the edge detection macros. In addition, this section includes the following information for each macro: a brief description of the macro, a list of arguments needed when calling the macro, a list of local variables used by the macro, the common variables returned by the macro, and a list of error messages generated by the macro.

Common variables used by the workpiece edge detection macros:

#131 = Xde1 = X-work coordinate of detection point adjusted for probe offset, but not detection shift
#132 = Yde1 = Y-work coordinate of detection point adjusted for probe offset, but not detection shift
#519 = Qr = calculated coordinate system rotation angle (deg)
#520 = Rprtg,min = minimum scan distance from start position to target position [0.5 in]
#521 = Rprtg,max = maximum scan distance from start position to target position [7.0 in]
#522 = Fcal = feedrate at last calibration
#523 = Rds = detection shift at last calibration (positive indicates detection point is past actual edge)
#524 = Xof = X-axis probe offset at last calibration
#525 = Yof = Y-axis probe offset at last calibration

Note: *The probe offset is the distance from the effective probe center to the effective laser beam center.*

#526 = Zsc = scan standoff at last calibration
#527 = Zth = detection threshold at last calibration
#529 = Units constant (1.0 for mm, 25.4 for inch)
#530 = Xsh = calculated X-axis translation
#531 = Ysh = calculated Y-axis translation

System variables used by the workpiece edge detection macros:

#5021 = present X-machine coordinate
#5022 = present Y-machine coordinate
#5041 = present X-work coordinate
#5042 = present Y-work coordinate
#5061 = X-work coordinate when SKIP signal received (that is, edge detected)
#5062 = Y-work coordinate when SKIP signal received (that is, edge detected)

MACRO DESCRIPTIONS

MACRO 9700 (CALIBRATE SENSOR)

Description: The macro call must specify the scan standoff and detection threshold values, Zsc and Zth, to be used for scanning with the noncontact head. A square calibration hole (at least 3” x 3”) must be cut prior to calling this macro. The macro call must specify the programmed coordinates Xcn and Ycn of the center of the hole and the measured dimensions Lx and Ly of the hole. The macro scans the calibration hole to determine the detection shift Rds and probe offsets Xof and Yof.

If an argument is missing or out of range, the macro preserves the results from the last calibration. If an error occurs after the calibration scan has started, the detection shift and probe offsets are set to zero.

Arguments:

A (#1) = Zsc = scan standoff
B (#2) = Zth = detection threshold
C (#3) = Dx = measured edge-to-edge distance in X-direction

D (#7) = Dy = measured edge-to-edge distance in Y-direction

F (#9) = Fr = feedrate for calibration
X (#24) = Xcn = nominal X-work coordinate of calibration hole center

Y (#25) = Ycn = nominal Y-work coordinate of calibration hole center

Local Variables:

#17 = Xds = X axis detection shift
#18 = Yds = Y axis detection shift
#19 = Xlm = measured X-coordinate of left edge
#20 = Xrm = measured X-coordinate of right edge
#21 = Ybm = measured Y-coordinate of bottom edge
#22 = Ytm = measured Y-coordinate of top edge

Returns:

#522 = Fcal = feedrate at last calibration
#523 = Rds = detection shift at last calibration
#524 = Xof = x-axis probe offset at last calibration
#525 = Yof = y-axis probe offset at last calibration
#526 = Zsc = scan standoff at last calibration
#527 = Zth = detection threshold at last calibration

Calls: 9701 (Scan), 9709 (Convert Units)

Called by: User program

Error messages:

"Standoff out of range"
"Threshold out of range"
"Arguments missing"

MACRO 9701 (SCAN SUBPROGRAM)

Description: This subprogram lowers the Z-axis to the scan standoff, puts the Z-axis in servo-hold and commands standoff equal to the detection threshold. It starts at the present position (Xpr, Ypr) and moves on a straight line path toward the specified target position (Xtg, Ytg) at the feedrate used in the last calibration. The macro uses G31 to determine the position where the noncontact head detected an edge. It stops just beyond the detect position (or at the target position if it does not detect an edge) with a partial Z-up command. The subprogram does not return to the start position.

The subprogram modifies the pierce standoff and cut standoff from the values set by the most recent G89 or G102 macro call. The calling program must command a new G89 or G102 block immediately after calling the 9700 or 9712 macro. The subprogram returns with the commanded feedrate equal to variable #148. If variable #148 was not defined before the macro call, the subprogram returns with the commanded feedrate equal to the scan feedrate.

Arguments:

X (#24) = Xtg = x-work coordinate of target
Y (#25) = Ytg = y-work coordinate of target

Local Variables:

#1 = Present X-work coordinate = starting X-coordinate
#2 = Present Y-work coordinate = starting Y-coordinate
#18 = Xprde = X-coordinate distance from start to detect
#19 = Yprde = Y-coordinate distance from start to detect
#20 = Rprde = Radial distance from start to detect
#21 = Xdetg = X-coordinate distance from detect location to target
#22 = Ydetg = Y-coordinate distance from detect location to target
#23 = Rdetg = Radial distance from detect location to target
#26 = Xprtg = Xpr - Xtg = X-coordinate distance from start to target
#27 = Yprtg = Ypr - Ytg = Y-coordinate distance from start to target
#28 = Rprtg = Radial distance from start to target

Returns:

#131 = Xde1 = X-work coordinate of detection point adjusted for probe offset, but not detection shift
#132 = Yde1 = Y-work coordinate of detection point adjusted for probe offset, but not detection shift

Calls: None

Called by: 9700 (Calibrate Sensor)
9702 (Find Hole)
9712 (Align to Sheet)

Error messages:

"Move distance too small"
"Move distance too large"
"Started too close to edge"
"Edge not detected"

MACRO 9709 (CONVERT UNITS)

Description: If the active CNC units (inch or mm) are not the same as the edge detection parameter units, the 9709 macro converts the edge detection parameters to the active units.

Arguments: None

Local Variables: None

Returns: Parameter units consistent with CNC units.

Calls: None

Called by: 9700 (Calibrate Sensor)
9710 (Align to Holes)
9712 (Align to Sheet)

Error messages: None

MACRO 9712 (ALIGN TO SHEET)

Description: This macro shifts and rotates the work coordinate system to align it with the sheet position.

Arguments:

X (#24) = Lx = length of sheet along X axis

Local Variables:

#10 = Lx, min = minimum sheet length [12 in]

#11 = Lb = backup distance from corner to test points [2.250 in]

#12 = Rprtg = distance from start to target when scanning an edge [2.500 in]

#13 = #507 = save value of pierce time so it can be restored

#14 = Xa = X-work coordinate of sheet edge at position A

#15 = Yb = Y-work coordinate of sheet edge at position B

#16 = Yc = Y-work coordinate of sheet edge at position C

#17 = Qr = angle of sheet edge relative to X axis based on Yb and Yc

#18 = Xe = X-coordinate of home corner of sheet based on Xa, Yb, and Yc

#19 = Ye = Y-coordinate of home corner of sheet based on Xa, Yb, and Yc

Returns:

#519 = Qr = calculated coordinate system rotation angle
#530 = Xsh = calculated x-axis translation
#531 = Ysh = calculated y-axis translation
Coordinate system shifted and rotated.

Calls: 9701 (Scan)
9709 (Convert Units)

Called by: User program

Error messages:

"X-axis length too small"
"Arguments missing"

7.6 OPTICAL PROBE

The Optical Probe option includes a fiber optic sensor, amplifier, actuator and macro software. The CNC program can use this option to find a pre-processed hole pattern, or to find the orientation of a sheet.

When used to find a pre-processed hole pattern, the optical probe scans across two holes to find their center coordinates. The macro program modifies the work coordinate system so the measured holes are at their expected work coordinates.

When used to find sheet orientation, the probe scans across two sheet edges and the macro program modifies the work coordinate system to represent the sheet edges. With the modified coordinate system, the CNC program can cut shapes at the correct location relative to the pre-processed hole pattern or sheet edges.

CINCINNATI INCORPORATED supplemental manual EM-467 includes instructions for optical probe programming, set-up, troubleshooting and maintenance.

7.7 LASER OPTICAL PROBE

The laser optical probe locates features on the material surface by sensing the change in reflected energy from a diode laser source. The CNC program uses a macro program to align the work coordinate system with the material. The macro program uses the probe to find the edge or other features on the material. For the Airbag System option, the laser optical probe detects the location of dark threads woven into the airbag fabric. The option includes a diode laser, mounting hardware, sensing electronics and software. The laser sensor head moves with the Z-axis assembly, and does not need an actuator to move the sensor closer to the material. For more information, see the Laser Optical Probe manual supplement (EM-487).

SECTION 8 FILE TRANSFER & NETWORKING

8.1 FILE TRANSFER

Users can transfer program files to and from the CINCINNATI control by these methods:

- Ethernet network
- Floppy disk
- RS-232 interface

NETWORK:

For downloading programs from a remote computer, the CINCINNATI control has built-in support for Ethernet networking via the Windows NT operating system. See Section 8.2.

FLOPPY DISK:

A standard PC compatible 3.5 inch 1.44 MB floppy disk drive is built into the control. This drive can be selected from the Windows "File, Open" and "File, Save-As" dialog boxes.

RS-232 INTERFACE:

The only support for file transfer via RS-232 is the TERMINAL.EXE program included with Windows NT operating system.

CINCINNATI does not recommend file transfer via RS-232. It requires operator set-up at both ends and is 1000 times slower than network transfer. The user must properly configure the RS-232 port and the operator must enter the filename.

Users cannot transfer files via RS-232 while the laser system is executing a program.

8.2 NETWORKING

The CINCINNATI control can network to other computers. Networking allows users to transfer part program (.cnc) files and material library (.lib) files between a remote computer and the CINCINNATI control.

Cabling:

The CINCINNATI control includes a network interface card with connectors for both Thin-net (BNC) and unshielded twisted pair (UTP) cable connectors.

CABLE SPECIFICATIONS		
Cable Type:	Thin Ethernet (Thin-net) RG-58	Unshielded Twisted Pair (UTP)
Connector:	Coaxial (BNC)	RJ-45
Maximum Distance:	607 ft. (185 m) with repeater	328 ft. (100 m) to hub
Minimum Distance:	19.7 in. (0.5 m)	N / A

Cabling Notes:

RG-58 cable must conform to 10BASE-2 specifications and must have 50 ohm termination at each end.

Unshielded twisted pair (UTP) cable can be 22, 24 or 26 gauge wire and must comply with IEEE 802.3 10BASE-T standard. Possible sources of UTP cable:

NETWORK CABLES

Pairs	Manufacturer	Cable Number
2	Belden	9562
4	Data Set	2404
6	Belden	9566

RJ-45 (8 pin) connector layout:

8-PIN CONNECTOR LAYOUT

Function	Female Pin No.	Male Pin No.
TX+	1	1
TX-	2	2
RX+	3	3
	4	4
	5	5
RX-	6	6
	7	7
	8	8

Pins 1 and 2 must be connected to a twisted pair, and pins 3 and 6 must be connected to a twisted pair.

NETWORKING OPTIONS

Windows Network:

File transfer via Windows network requires a remote PC running Windows for Workgroups, Windows 95, Windows 98, Windows 2000 or Windows NT, a network adapter card, and cabling from the PC to the laser system.

The remote PC can view and/or transfer files at any time (even while the laser system is running a program). The CINCINNATI control can access files at the remote PC in directories that are visible (shared) to the network.

Network with FTP on TCP/IP protocol:

File transfer via FTP (File Transfer Protocol) requires a UNIX (or comparable) workstation that can communicate via TCP/IP protocol, a network adapter card and cabling from the workstation to the CINCINNATI control.

The workstation can view and/or transfer files at any time. However, the laser operator cannot initiate file transfer.

Network with Novell NetWare and IPX/SPX protocol:

File transfer requires a PC running Novell NetWare, a network adapter card and cabling from the PC to the CINCINNATI control.

The remote PC can view and/or transfer files at any time. The CINCINNATI control can access files at the remote PC in directories that are visible (shared) to the network.

Users may experience significant difficulty when setting-up to use Novell NetWare for the first time.

Network with Dial-Up Access:

File transfer requires a remote PC running Windows for Workgroups, Windows 95, Windows 98, Windows 2000 or Windows NT, and Remote Access client software. The PC must have a modem and a dedicated phone line to the CINCINNATI control. This phone line cannot be used for operator phone calls.

The remote PC can view and/or transfer files at any time (even while the laser system is running a program). The CINCINNATI control can access files at the remote PC in directories that are visible (shared) to the network.

File transfer is slower than Ethernet, 28800 bps maximum. The user must initiate the network connection at the remote computer, not at the CINCINNATI control.

If the phone system is digital, this method of file transfer may not work reliably.

INDEX