ACE-G3

USB Motion Controller
3-Axis / G-Code

CE RoHS 2002/95/EC

First edition, February 2015

Table of Contents

1. Introduction

ACE-G3 is a 3-axis motion controller with USB communication that supports the commonly used G-code and M-code language.

ACE-G3 uses HID Device Class for USB communication and does not require a special USB driver installation. HID is supported by all major operating systems such as Windows, Linux, and Mac, and enables immediate use of ACE-G3.

ACE-G3 supports streaming of the G-code and M-code commands from host PC through USB communication. The streamed commands are buffered in 1K lines of buffer space so that G-code can be continuously streamed from the host PC through USB to ACE-G3 for continuous motion.

Linear interpolation motion is supported on XYZ axes. Arc/circular interpolation is supported on XY axes. Maximum raw pulse rate supported is 1MHZ.

Plus and negative limit switch inputs are available for hard limit sensing. Soft limits are also available for controlling the motion range. Home input is available on each axis for homing.

4 digital outputs are available for general digital output control and 3 digital inputs are available general digital input sensing.

For Spindle control, additional pulse/dir output signals are available.

3 sets of encoder inputs are available for XYZ axes in case position verification is desired using the encoders.

2. Electrical and Thermal Specifications

Parameter	Min	Max	Units
Main Power Input from Host PC USB (V_{DD})	+5	+5	V
	-	500	mA
Operating Temperature [1]	-20	+80	°C
Storage Temperature [1]	-30	+120	°C

Table 1.0

Notes:

- VDD (+5V) power from Host PC USB is used to power the ACE-G3.
- Power rating determined from component rating.
- Temperature rating determined from component rating.

3. Dimensions

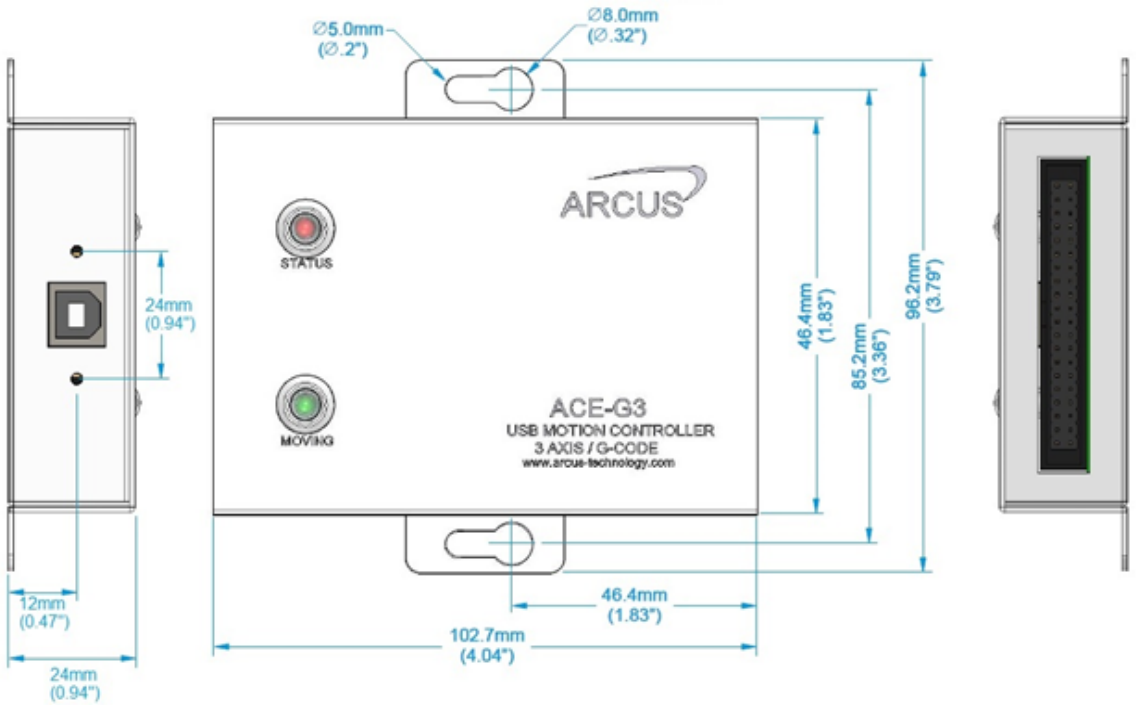

Figure 1.0

4. Connectors

4.1 USB Connector

ACE-G3 accepts standard USB male B connector.

ACE-G3 also accepts USB cable with two thumb screws to securely lock the connector to the controller. Picture of USB B connector with two locking screws is shown below.

Figure 2.0

Two thumb screws are #4-40 thread and 24 millimeters apart. This cable is available from Arcus Technology.

Important note on USB connector:
In a high noise environment, a USB cable with chokes is recommended for noise suppression and improved data integrity.

4.2 40-pin Connector

ACE-G3 interfaces with motion system through the standard 40 pin 0.1” dual row connector as shown below.

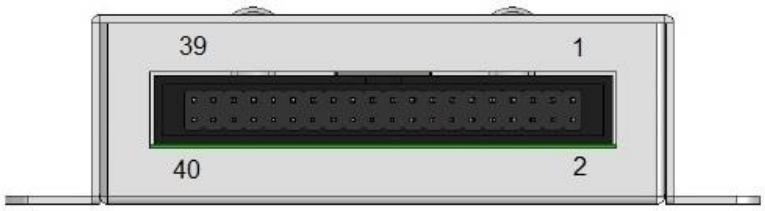

Figure 3.0

An example picture of 40 pin female connector/cable is shown below.

Figure 4.0

4.3 Connector Pin Assignment for 40 pin connector

	Name	Pin #	Pin #	Name	
+5V supply from USB	**VCC**	1	2	**GND**	Ground from USB
Pulse output X axis	**PULSEx**	3	4	**DIRx**	Direction output X axis
Pulse output Y axis	**PULSEy**	5	6	**DIRy**	Direction output Y axis
Pulse output Z axis	**PULSEz**	7	8	**DIRz**	Direction output Z axis
Pulse output Spindle	**PULSEs**	9	10	**DIRs**	Direction output Spindle
Digital Output 1	**DO1**	11	12	**DO2**	Digital Output 2
Digital Output 3	**DO3**	13	14	**DO4**	Digital Output 4
No Connection	**NC**	15	16	**NC**	No Connection
No Connection	**NC**	17	18	**GND**	Ground from USB
Neg Limit Input X Axis	**NLIMx**	19	20	**HOMEx**	Home Input X Axis
Pos Limit Input X Axis	**PLIMx**	21	22	**DI1**	Digital Input 1
Neg Limit Input Y Axis	**NLIMy**	23	24	**HOMEy**	Home Input Y Axis
Pos Limit Input Y Axis	**PLIMy**	25	26	**DI2**	Digital Input 2
Neg Limit Input Z Axis	**NLIMz**	27	28	**HOMEz**	Home Input Z Axis
Pos Limit Input Z Axis	**PLIMz**	29	30	**DI3**	Digital Input 3
No Connection	**NC**	31	32	**GND**	Ground from USB
Encoder A - X Axis	**EAx**	33	34	**EBx**	Encoder B - X Axis
Encoder A - Y Axis	**EAy**	35	36	**EBy**	Encoder B - Y Axis
Encoder A - Z Axis	**EAz**	37	38	**EBz**	Encoder B - Z Axis
No Connection	**NC**	39	40	**GND**	Ground from USB

Table 2.0

4.4 Junction Boards

The following junction boards allow easy interface of ACE-G3 to external systems. Junction boards may be purchased from Arcus Technology.

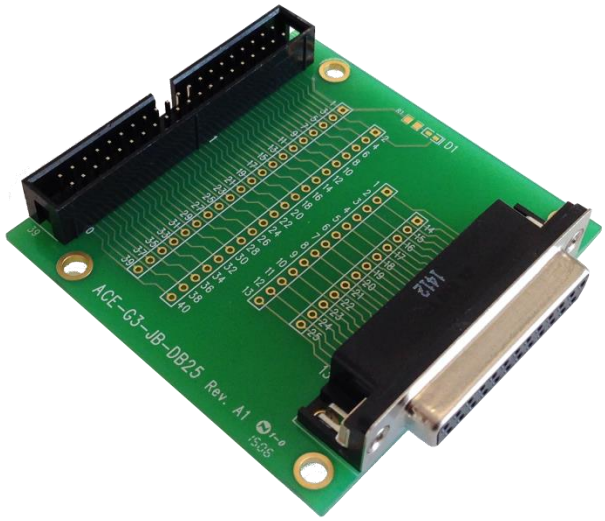

Figure 5.0 ACE-G3-JB40-DB25
40 pin to DB25 pin interface board.
(Requires custom soldering for connecting 40 to 25 pin)

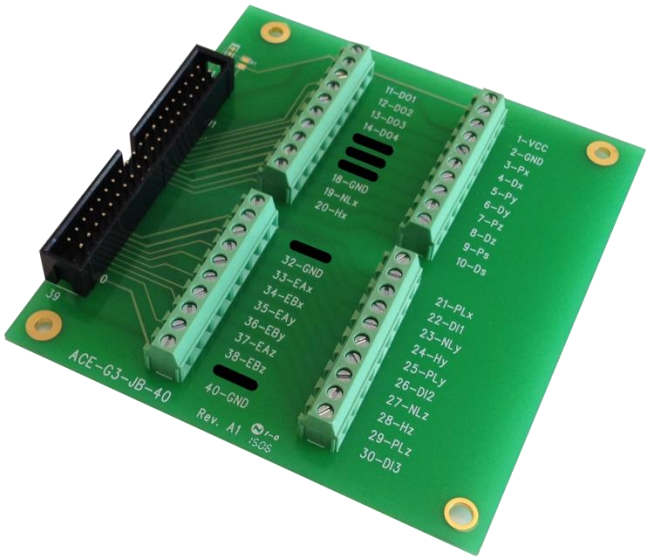

Figure 6.0 ACE-G3-JB40-S
40 pin to individual screw terminals.

4.5 LED Lights

ACE-G3 has two LED lights to indicate the status of the controller.

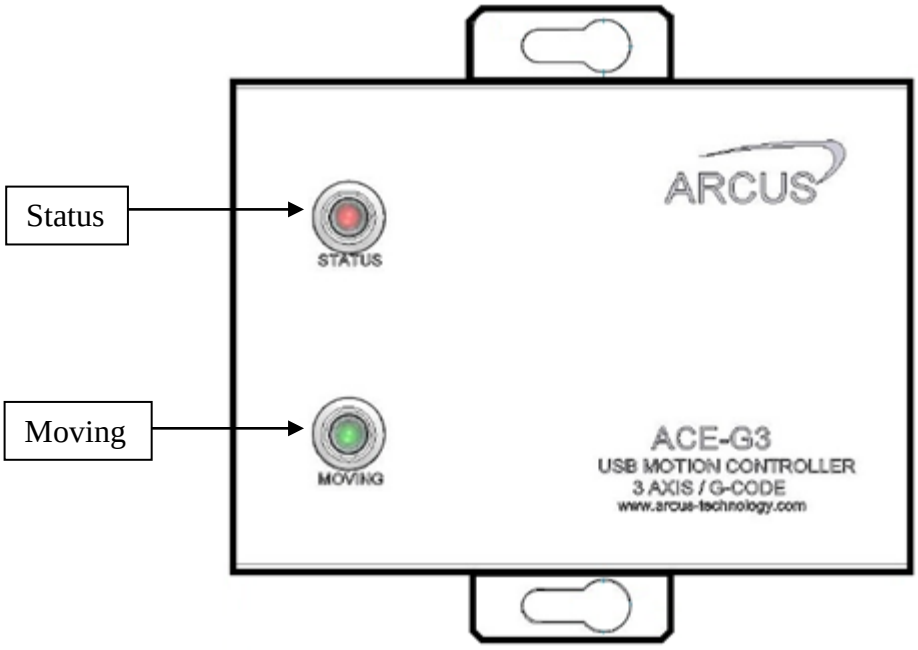

Figure 7.0

5. Input / Output Signal Description

5.1 VCC/GND

The ACE-G3 controller uses the VBUS power from the Host PC to power the Motion Control Circuit as can be seen on the diagram below. VBUS is also connected to the VCC signal on the 40-pin connector through the 350mA resettable fuse to limit the current use on the VCC signal. A diode is connected between the VCC and GND signal for reverse protection.

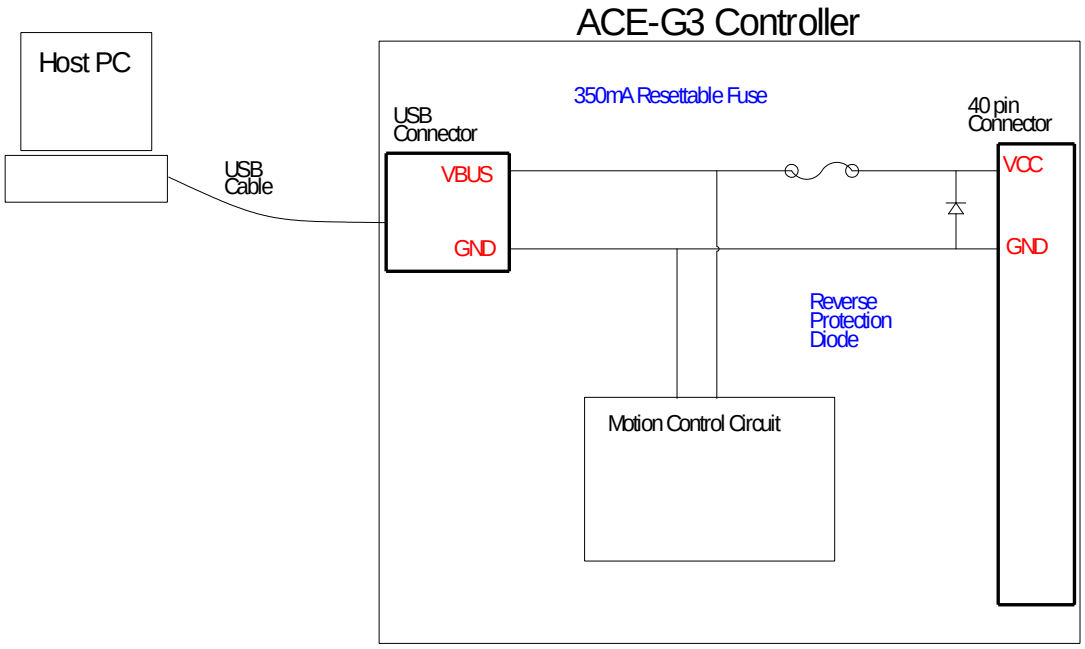

Figure 8.0

Pin Name	Pin Number on 40 pin connector
VCC	1
GND	2, 18, 32, 40

Table 3.0

Care should be taken when using the VCC (+5V) supply from the host USB VBUS. Current limit and usage should be carefully considered.

Important Note on using VCC and GND (VBUS and GND):
We recommend that you limit the use of the VCC signal from the 40 pin connector to power any external circuit. Our recommendation is to use the external +5V supply for external circuits.

Improper use and connection of the VCC and GND (VBUS and GND) may result in damage to host PC. Extreme care should be taken to ensure proper protection of the host PC. Arcus is not liable for damage caused by improper usage.

5.2 PULSE/DIR/DO Digital Outputs

Pulse, Direction, and DO - General Purpose digital outputs use TTL 74LS07 open collector output chips as shown below.

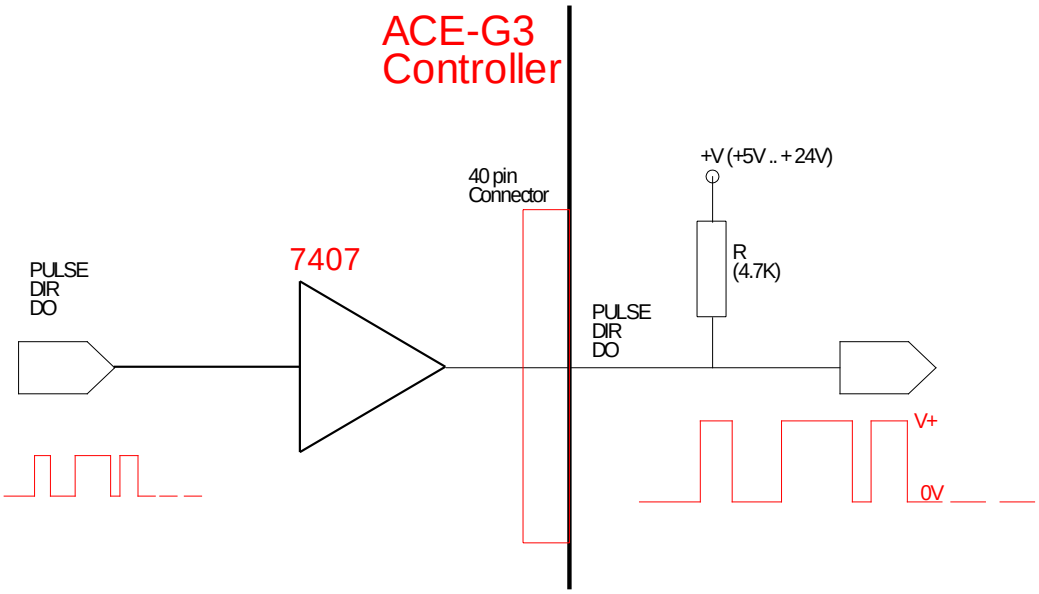

Figure 9.0

Note: For Open-collector output, a pull up resistor is needed as shown on the example.

Pin Name	Pin Number on 40 pin Connector
PULSEx	3
PULSEy	5
PULSEz	7
PULSEs	9
DIRx	4
DIRy	6
DIRz	8
DIRs	10
DO1	11
DO2	12
DO3	13
DO4	14

Table 4.0

Maximum raw pulse output rate of PULSEx, PULSEy, and PULSEz is 1M pulses per second.

5.3 LIMIT/HOME/DI/EA/EB Digital Inputs

Limits, Home, General Purpose Digital Inputs, and Encoder inputs use TTL 7414 Schmitt Trigger chip.

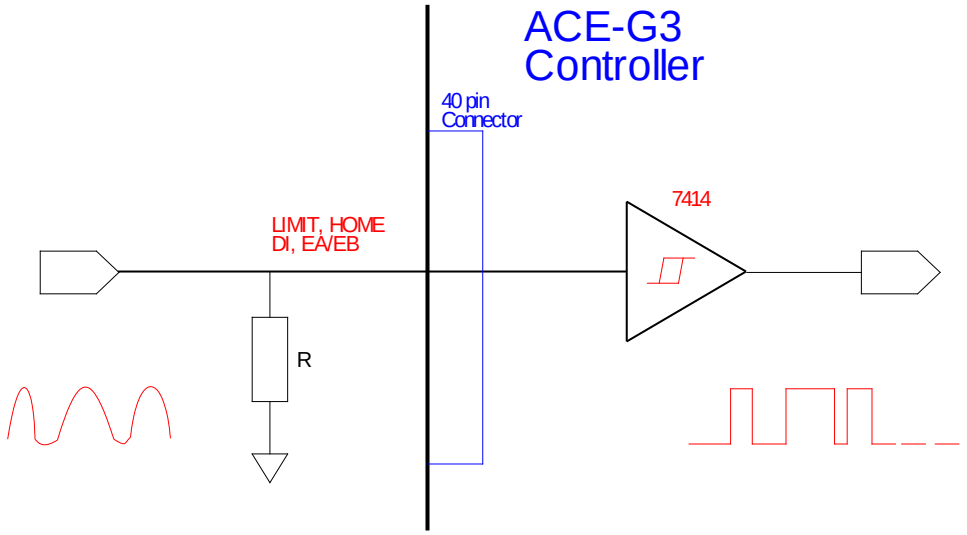

Figure 10.0

The benefit of the Schmitt trigger is the squaring off of edges to make up for the slow rise and fall time of the input signal.

Pin Name	Pin Number On 40 pin Connector
NLIMx	19
PLIMx	21
NLIMy	23
PLIMy	25
NLIMz	27
PLIMz	29
HOMEx	20
HOMEy	24
HOMEz	28
DI1	22
DI2	26
DI3	30
EAx	33
EBx	34
EAy	35
EBy	36
EAz	37
EBz	38

Table 5.0

6. USB Communication

ACE-G3 is a 3 axis motion controller that supports USB 2.0 communication using HID (Human Interface Driver) that is available on all operating systems including Windows 7/8, Win CE, Embedded Win, Mac, and Linux.

ACE-G3 can be plugged into any host PC's USB port and immediately used without any USB driver installation.

For Windows 7/8 OS, DLL file called HIDApiDll.dll is provided for use with Windows programs. Sample executable program is available for test of the controller. Source code is also provided for customization by the user for special applications.

For the Windows program, refer to the ACE-G3 Windows program and manual for details.

Sample screen capture is shown below of the Windows program.

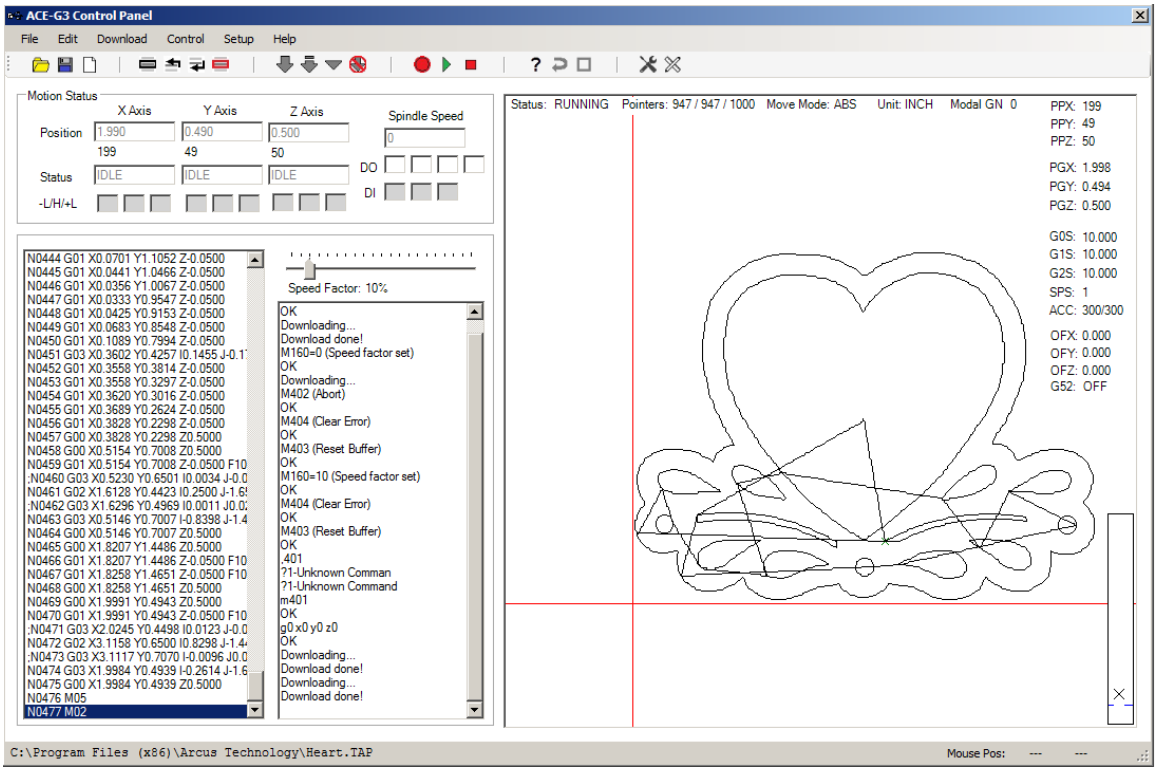

Figure 11.0

Programs for other operating system such as Linux and Mac OS will be available in the near future.

7. Command Structure and Nomenclature

Commands to ACE-G3 controller are sent as character strings from Host PC through USB HID string packet.

Maximum size of the command and reply string is 64 characters. Care should be taken when programming to allocate correct size for the command and reply strings.

7.1 Valid Characters

Following is a list of valid characters used in the commands from Host PC to ACE-G3.

Chars	Description	Example
N	Used to indicate line number. Accepted but not used or stored by the controller.	N1000 G2 X2Y3 N2000 G90
G	Used in G-code to indicate G-code number	G90 G01 X200 Y300
M	Used in M-code to indicate M-code number	M2 M130=120
XYZ	Used to indicate XYZ axis for G-code 0/1/2/3 commands.	G1 X2Y3Z0.2 G2 X3.5Y2.0 I1
IJ	Used to indicate relative center location for G-code 2/3	G2 X2Y1 I1J2 G3 Y3X1 I1.5
F	Used for G0/1/2/3 command to indicate feedrate (speed) of move	G1 F2.5 G2 X1 I1 F0.5
S	Used for Spindle command G97 to indicate RPM speed of Spindle	G97 S1000
A	Used for acceleration for G0 rapid move and G97 Spindle speed	G0 F12 A200 G97 S1000 A200
P	Used for dwell duration amount in milliseconds for command G4	G4 P120
=	Used in M-code to assign a value	M102=1 M204=3.5

Table 6.0

Following characters are valid to use for numbers: 0,1,2,3,4,5,6,7,8,9,+,-,.(period).

Note that some numbers are float numbers and some are whole numbers, depending on the type of number supported in the G-code and M-code command.

Examples:

```
G0 X1.52 Y2.69      ;***Sets the move target positions to 1.52 and 2.69 (floats)
M204=130.87         ;***Sets the current position to 130.87 (float)
M101=1              ;***Sets the digital output 1 (whole number)
```

7.2 Case Sensitivity

Commands are not case sensitive and upper or lower case commands can be sent.

Examples:
Host PC: m2
ACE-G3: OK

Host PC: g1 x2.0 Y3.0 z 2
ACE-G3: OK

7.3 Reply

All commands are acknowledged with a reply. There are two types of replies, one that does not require a return value(s) and one that does require return value(s).

Command that does not require a return value is acknowledged with an "OK" string.

Command that requires a return value is replied with a value string.

If the command string is invalid, an error message is sent as a reply. Error messages start with "?" character plus error code and error message, if any.

Examples:
Host PC: G0 X1Y1
ACE-G3: OK

Host PC: G18
ACE-G3: ?49-Unknown G-code number

Host PC: M204
ACE-G3: 2.000

7.4 Line Number

Even though line number is not used by the ACE-G3 controller, line number can be sent along with the command for compatibility with standard G-code command structure.

Examples:
N001 G90
N002 G0 X1Y1Z1 F5
N003 G1 F1
N004 X1
N005 Y2 Z3

7.5 Comments

Use of comments is supported in command structure by ACE-G3. Comments are not used or stored by ACE-G3 controller.

Character of semicolon ";" indicates that all following characters as comment.

Character of "(" indicates start of comment and ")" indicates end of comment. This type of comment is useful for inserting comments in middle of a command.

Use of "(" without ")" will be considered as open ended comment and treated same as using ";" comment.

Examples:

```
G0 ;*** Set to G0
G1 (Set to G1) X2 Y4
N23 G90 (Set to Absolute mode)
M3 (Open ended comment
```

7.6 G/M-Code Command Restrictions

Only single G-code or M-code per command per line is allowed.

Multiple G-code or M-code per command line is not allowed. Following are examples of invalid commands.

Example of invalid commands:

```
G90 G21              ;***Two G-code command in single line
M2 G1 X2 Y3          ;***M and G-code command in single line
N20 G1 X5 Y0 Z0 M3;***G and M-code command in single line
```

8. Buffering

All ACE-G3 communication is done using G-code and M-code commands.

There are two types of commands: ***Buffered*** and ***Immediate***.

Buffered commands are loaded into the buffer list and are executed in a sequential order when the buffer state is in RUN state.

Immediate commands are executed as soon as the command is sent to the ACE-G3 controller. Example ***immediate*** commands are position query commands, parameter read/write commands, and digital IO query commands.

All G-code commands are ***buffered*** commands. Buffer commands are loaded to the buffer list and are executed only when the buffer state is in RUN state.

M-code commands between 1 and 99 are ***buffered*** commands and executed only when the buffer state is in RUN state.

All M-code commands above 99 are ***immediate*** commands which are executed as soon as the command is received by the controller.

8.1 Buffer List

There are a total of 1000 buffer command spaces available for download. There are two pointers: head and tail pointers as shown below.

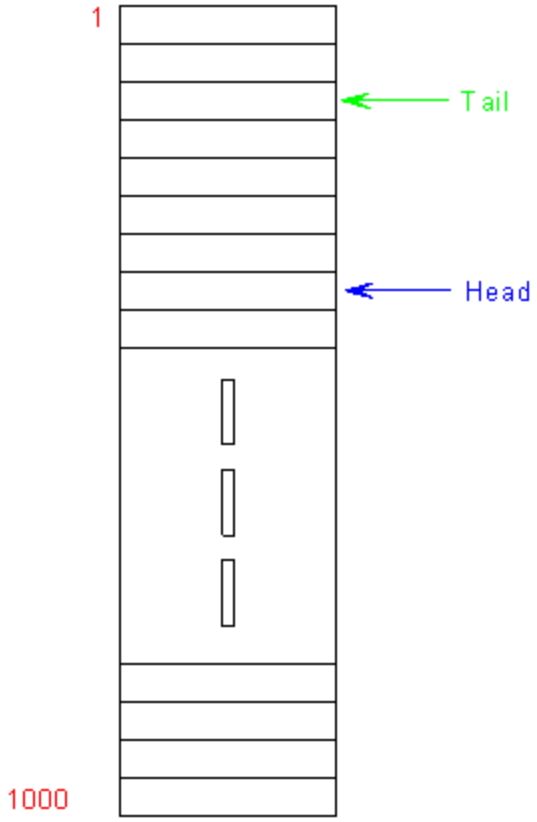

Figure 12.0

8.2 Head Buffer Pointer

Head pointer points to the latest command that has been loaded into the buffer list. As the new command is loaded, the head pointer is incremented to the next buffer position. When the head pointer reaches the 1000th space, the next head pointer goes to position 1. Head buffer pointer does not surpass the tail pointer. When downloading a buffered command from host PC, verify the availability of buffer space before downloading.

Valid and accepted buffered command is replied with "OK"

If the buffer list is full when the buffer command is received by the controller, buffered command is rejected with an error message "?Buffer Full"

8.3 Tail Buffer Pointer

Tail pointer points to the next buffer command to be executed when the buffer state is in RUN state. When the command at tail pointer is successfully executed, tail pointer is incremented to the next buffer position. When tail pointer reaches the 1000th location, next tail pointer goes to 1st location. Tail pointer does not surpass the head pointer.

8.4 Buffer Management Commands

Following are buffer management commands.

M-Code	Name	Description
400	Stop Buffer	Sets the buffer state to IDLE from RUN state. When buffer state is in RUN state, issuing stop command will change the buffer state to IDLE but ***any motion in progress and in queue will be completed.*** Issue run command from IDLE to continue the buffer list execution.
401	Run Buffer	Sets the buffer state to RUN. Any buffered command in the queue will be executed sequentially. If the buffer state is in ERROR, error must be cleared first to make buffer state IDLE before issuing the run command.
402	Abort Buffer and Motion	Immediately stops all motion in progress and in queue and sets the buffer state to IDLE. ABORT command discards all buffered commands in buffer queue and sets the tail pointer equal to the head pointer. ABORT command stops the Spindle. All digital outputs will remain at current state. ***Note: Use ABORT command only for emergency situations since all motions will come to an abrupt stop and all buffered commands will be lost.*** For normal situations, use STOP command (M320) for graceful way to change to IDLE state.
403	Reset Buffer Head/Tail	Resets the head and tail pointers to 1.
404	Clear Error	Clears the buffer state from ERROR state to IDLE state. Abort command will also clear the buffer state from IDLE to ERROR, but abort command will also clear the buffer list and reset the buffer list pointers.
405	Get Error Message	Returns the current error message. When buffer state is in ERROR, use this command to see the cause of the error from the error message.
406	Get Buffer Head Pointer	Returns buffer head pointer value. Range is 1 to 1000.
407	Get Buffer Tail Pointer	Returns buffer tail pointer value. Range is 1 to 1000
408	Get Available Buffer Size	Returns total available buffer size for download. Range is 0 to 1000.
409	Get Buffer State	Returns the buffer state: 0 – IDLE 1 – RUNNING 3 – ERRORED
410	Get All Buffer Info	Returns buffer head, tail, size, and state in a single reply

Table 7.0

9. Configuration Parameters

ACE-G3 is a 3 axis motion controller ideal for controlling XYZ mechanical systems. In order to properly use the controller, ACE-G3 must be configured to match the physical setup of the XYZ system.

Flash Stored Parameters
Important configuration values are stored in the non-volatile flash memory so that at power-up, configuration values are read from flash memory and loaded automatically to the controller. Following parameters are stored on the flash memory.

M-Code	Name
301	System Unit (mm or inch)
302	X Axis Ratio
303	Y Axis Ratio
304	Z Axis Ratio
305	Limit Polarity
306	Home Polarity
307	Direction Polarity
308	Digital Input Polarity
309	Device ID
310	Spindle Ratio

Table 8.0

Store to flash memory is done with the ***M300*** command. When this command is received, the parameters are written to the flash memory. Flash memory retains the value of parameter even with no power. When ACE-G3 is powered the next time, stored parameters are loaded and used by the controller. Other parameters that are not stored on the flash memory are set to default values.

Non-Flash Stored Parameters
Following parameters are NOT stored in the flash memory. At power-up these parameters values are set to default values.

M-Code	Name	Default Value At Startup
311	Enable Soft Limit (0-disable, 1-enable)	0 – disabled
312	Neg Soft Limit – X axis	-1
313	Pos Soft Limit – X axis	1
314	Neg Soft Limit – Y axis	-1
315	Pos Soft Limit – Y axis	1
316	Neg Soft Limit – Z axis	-1
317	Pos Soft Limit – Z axis	1
318	Homing Mode – X axis (0-Limit Only, 1-Home/Limit)	0
319	Homing Mode – X axis	0
320	Homing Mode – X axis	0

Table 9.0

Reading and Writing Configuration Parameters
Configuration parameters are read or written using M commands.

When reading a parameter, send the M-code command and a reply with value will be returned. For example, to read the system unit type, send command "M301" and the reply will be either "0" (mm) or "1" (inch).

When writing a parameter, send the M-code command with equal sign "=" and the desired value. Reply will be either "OK" or question mark "?" and error code/message. For example, to set the unit as inch, send command "M301=1" and reply will be "OK". Remember to store the updated flash stored parameters with ***M300*** command to retain the values for next power cycle.

9.1 System Unit Setup

User can select the working ***System Unit*** in inch or millimeter. Once ***System Unit*** is set, all queried position values will be given in this unit. For example, if the ***System Unit*** is set as inch and X position is queried using ***M204*** command, the return value of 10 means 10 inch position value. If the system unit is setup as mm, the value of 10 will represent 10 mm position value.

M-Code	Description	Range
301	***System Unit*** setup value	0 – mm 1 – inch

Table 10.0

ACE-G3 also supports ***G-code Unit*** which can also be inch or mm. ***G-code Unit*** is an on-the-fly unit that can be changed from G-code anytime, whereas ***System Unit*** is configured during the initial system setup and remains permanent in the system until

system change. Use ***G20*** and ***G21*** to ***set*** the G-code Unit. Use ***M221*** to ***read*** the current G-code unit value.

9.2 XYZ-Ratio Setup

Once the ***System Unit*** (mm or inch) is set, ***ratio*** of each axis movement must be configured so that the number of pulses generated from ACE-G3 matches one unit of movement. For example, if ***System Unit*** is set to inch, and one inch of X movement equals 1800 pulses, ***X-Ratio*** value is set to 1800.

Each axis can have its own individual ***ratio***. For example, X and Y ***ratio*** values can be 1200 and Z ***ratio*** value can be 800. Even with different ***ratio***s, ACE-G3 will generate the coordinated pulse amounts for each axis at proper feedrate so that the linear coordinated motion will be achieved for all axes.

Restriction Note:
For Arc/Circular motion, X and Y ratio values must be identical. If the ratios are different when Arc/Circle command is executed, buffer state will go into error.

M-Code	Description
302	X Axis Ratio – number of pulses per unit
303	Y Axis Ratio – number of pulses per unit
304	Z Axis Ratio – number of pulses per unit

Table 11.0

Example:

```
;***Unit is setup as inch M288=0
M302 = 2000  ;*** 2000 pulses represent 1 inch of X motion
M303 = 1000  ;*** 1000 pulses represent 1 inch of Y motion
M304 = 500   ;*** 500 pulses represent 1 inch of Z motion.
```

9.3 Direction Polarity Setup

ACE-G3 generates PUL (pulse) and DIR (direction) signals for each axis. Number of PUL output represents the amount of movement and DIR output represents the direction of the movement. DIR signal can be configured to match the desired physical direction of mechanical movement.

M-Code	Description	Range
307	Direction Polarity	Bit 0 – X axis Bit 1 – Y axis Bit 2 – Z axis

Table 12.0

9.4 Limit Input Polarity Setup

Each XYZ axis has plus and minus limit inputs. Polarity of each axis limit inputs can be configured.

M-Code	Description	Range
305	Limit Input Polarity	Bit 0 – X axis Bit 1 – Y Axis Bit 2 – Z Axis

Table 13.0

Note: Use M120, M121, M122 and M123 to read status of motion inputs.

9.5 Home Input Polarity Setup

Each XYZ axis has a home input. Polarity of each axis home input can be configured.

M-Code	Description	Range
306	Home Input Polarity	Bit 0 – X axis Bit 1 – Y axis Bit 2 – Z axis

Table 14.0

Note: Use M120, M121, M122 and M123 to read status of motion inputs.

9.6 Digital Input Polarity Setup

ACE-G3 controller has 3 general purpose digital inputs. Polarity of each digital input can be configured.

M-Code	Description	Range
308	Digital Input Polarity	Bit 0 – DI1 Bit 1 – DI2 Bit 2 – DI3

Table 15.0

Note: Use M110, M111, M112 and M113 to read status of digital inputs.

9.7 Device ID Setup

ACE-G3 can be assigned a unique device ID so that multiple ACE-G3, each with unique ID, can be connected to a single host PC.

M-Code	Description	Range	Default
309	Device ID	00 to 99	00

Table 16.0

9.8 Spindle Ratio

ACE-G3 has pulse and direction output for Spindle speed control. For proper speed control of the Spindle, Ratio must be set to match the number of pulses for 1 revolution of the Spindle.

M-Code	Description
310	Spindle Ratio – number of pulses per revolution

Table 17.0

Note: Use M3, M4, and M5 for Spindle control.
Use G97 for setting Spindle Speed and Acceleration.
Use M214 for reading actual Spindle Speed.
Use M220 and M223 for reading Spindle Speed and Acceleration settings.

9.9 Soft Limit

ACE-G3 has built-in soft limit function to limit the movement of XYZ within the preset range of motion.

Following are soft limit parameters.

M-Code	Description	Default Value at Startup
311	Enable Soft Limit (0 – disable, 1-enable)	0
312	X negative soft limit	-1
313	X positive soft limit	1
314	Y negative soft limit	-1
315	Y positive soft limit	1
316	Z negative soft limit	-1
317	Z positive soft limit	1

Table 18.0

Note: Soft limit parameters are NOT stored on flash memory

At controller power-up, soft limit is disabled by default and default values of the limits are -1 for negative and 1 for positive soft limit values.

Soft limit values are checked before the executing motion related G-code commands: G0, G1, G2, and G3. For Arc/Circle G-code commands (G2 and G3), full circumference is checked for soft limit even though the motion might be an arc.

If soft limit is detected at RUN state, buffer state goes from RUN to ERROR state.

9.10 Homing Mode

ACE-G3 has built-in homing routines. Homing can be done using the limit and/or home inputs. The following are homing mode parameters.

M-Code	Description	Range	Default Value at Startup
318	X Axis Homing Mode	0 – limit only homing 1 – home/limit homing	0
319	Y Axis Homing Mode	0 – limit only homing 1 – home/limit homing	0
320	Z Axis Homing Mode	0 – limit only homing 1 – home/limit homing	0

Table 19.0

Note: Homing mode parameters are NOT stored on flash memory

10. G-Codes

10.1 G0/G1 - Linear Motion

ACE-G3 supports the following linear interpolation G-code commands

G-Code	Description	Format
0	Rapid Linear Move with Acceleration/Deceleration	G0 X[value] Y[value] Z[value] F[value] A[value]
1	Linear Move with no Acceleration/Deceleration	G1 X[value] Y[value] Z[value] F[value]

Table 20.0

XYZ values are target positions. When in relative mode (G91), XYZ values are relative values from previous target positions. When in absolute mode (G90), XYZ values are absolute values from the system zero positions.

F (feedrate) value is the speed of movement in unit/second. Use **M223** to read modal G0 and G1 speed settings.

A (acceleration) value is valid for G0 only and represents the acceleration/deceleration value in milliseconds. Range is 0 to 1000 milliseconds. Use **M220** to read modal G0 acceleration setting.

XYZ position values and F feedrate use ***G-code unit***. For example, if G20 (inch) is previously executed, the XYZ and F values are in inch. If G21 (mm) is previously executed, XYZ and F values are in mm.
Feedrate is multiplied by the Global Feedrate Factor. Global Feedrate Factor range is from 1 to 100 representing 1% to 100% of the feedrate. Use Global Feedrate Factor to reduce all the speeds of the movements.

M-Code	Description	Format
160	Global Feedrate Factor	1 to 100

Table 21.0

Note: Default value of Global Feedrate Factor at power-up is 100.

G0 – Rapid Linear Movement
G0 is a linear interpolated motion with acceleration and deceleration to support rapid (high speed) motion.

Acceleration/deceleration for G0 is set with A[value].

Default value of acceleration/deceleration at power-up is 300 msec.

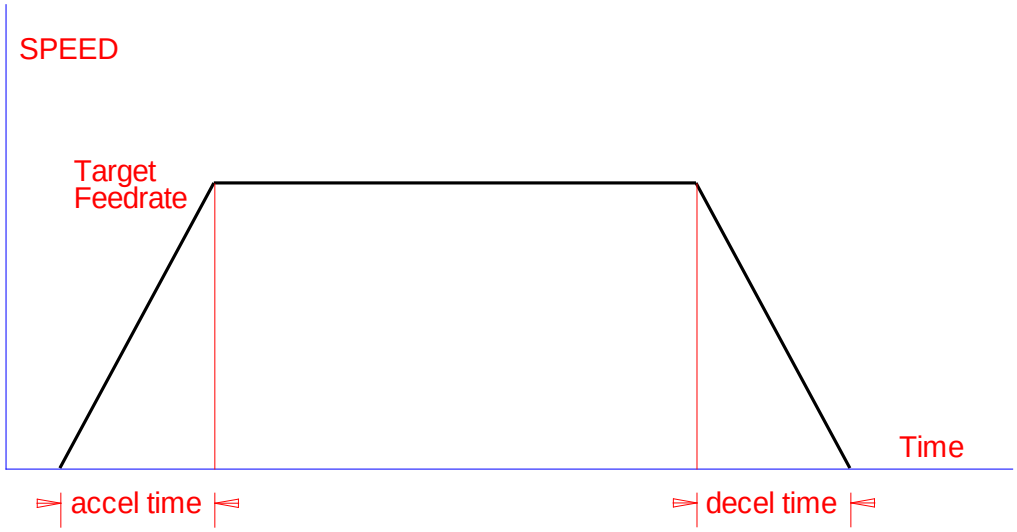

Figure 13.0

For G0 command, acceleration and deceleration values are the same. This results in a symmetrical trapezoidal speed profile as shown above.

G1 – Linear Movement

G1 is a linear motion with no acceleration/deceleration that requires constant linear speed throughout all the motion. For example, glue dispensing requires constant motion to dispense an equal amount of glue along the motion path. Another example is a laser cutting application where constant laser power is required along the motion path.

Since acceleration and deceleration are not used for G1, use low feedrate to reduce the jerk to the system.

Modality

Modality is supported on G0 and G1 with some restrictions.

Once G0 or G1 command is executed, any following command without G command will retain the previous G command number.

Example:

```
G0
F10
X1Y1        ;***G0 move from current position to (1,1) at feedrate 10
G1
F2
X0Y0        ;***G1 move to (0,0) at feedrate 2
```

G0 and G1 support XYZ modal positions. When in absolute mode (G90), XYZ positions that are not defined in the command are set to previous target position. When in relative mode (G91), positions that are not defined are set to zero value.

Modal positions are shared among G0, G1, G2, and G3. This means G0 target modal XYZ positions are the same as G1, G2, and G3 target modal position and vice versa. Note that G2 and G3 use only XY positions.

Modal feedrates for G0 and G1 are not shared. G0 has its own modal feedrate and G1 has its own modal feedrate.

Example: The following G-code commands draw a box from position (1,1):

```
G90         ;***Set to absolute coordinate
G0 F10      ;***Set to G0 move and set the feedrate to 10 units / sec
X1 Y1       ;***Move to X=1 and Y=1 at feedrate of 10
G1          ;***Change to G1 move.
X2 F3       ;***Move to X=2 at feedrate of 3
Y2          ;***Move to X=2, Y=2 at feedrate of 3
X1          ;***Move to X=1, Y=2 at feedrate of 3
Y1          ;***Move to X=1, Y=1 at feedrate of 3
F1          ;***Change the G1 feedrate to 1
X2Y2        ;***Move diagonally to position X=2, Y=2 at feed rate 1
G0 X1Y2     ;***Move to X=1, Y=2 at speed 10 (G0 modal feedrate)
G1 X2Y1     ;***Move diagonally to X=2, Y=1 at speed 1 (G1 modal feedrate)
G0 X0Y0     ;***Move to (0,0) at feedrate of 10
```

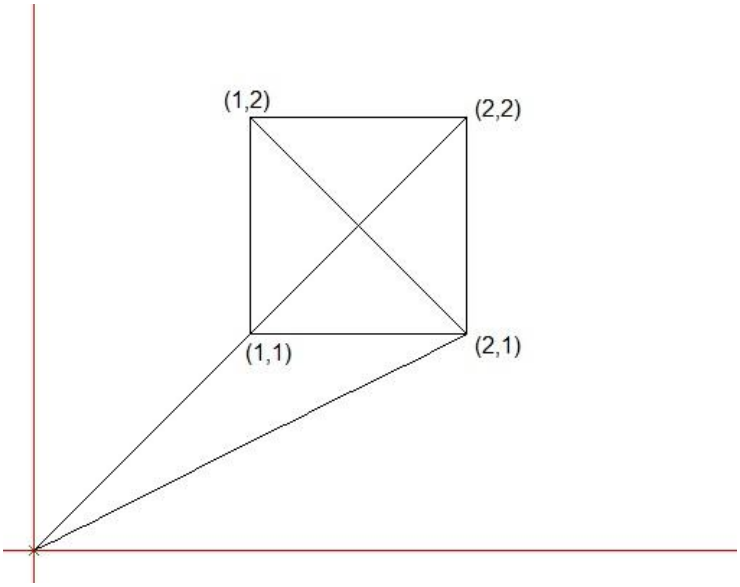

Figure 14.0

Example: Following G-code commands shows the use of relative mode (G91) and modal moves by draw 3 steps.

```
G91                 ;Relative mode
G0 X0 Y0 Z0 F20     ;G0 move to 0,0,0 at speed 20.
G1 Y1 F1            ;G1 move to 0,1,0 at speed 1
X1                  ;G1 move to 1,1,0 at speed 1
Y1                  ;G1 move to 1,2,0 at speed 1
X1                  ;G1 move to 2,2,0 at speed 1
Y1                  ;G1 move to 2,3,0 at speed 1
X1                  ;G1 move to 3,3,0 at speed 1
G90                 ;Absolute mode
G0 X0Y0             ;G0 move to 0,0,0 at speed 20
```

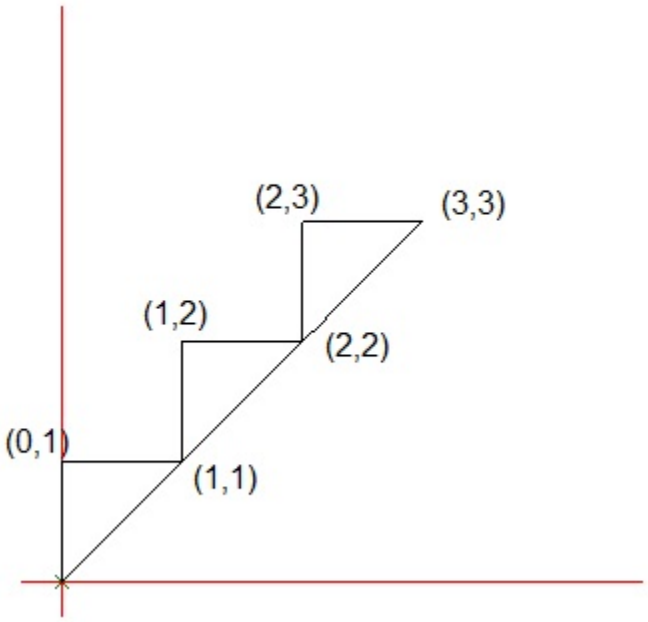

Figure 15.0

10.2 G2/G3 - Arc/Circular Motion

ACE-G3 supports arc/circular interpolation G commands.

G-Code	Description	Format
2	Clock wise arc/circular move	G2 X[value] Y[value] I[value] J[value] F[value]
3	Counter-clock wise arc/circular move	G3 X[value] Y[value] I[value] J[value] F[value]

Table 22.0

The following are restrictions on the arc/circular moves.

- Only X and Y axes are used for arc/circular interpolation moves.
- Z axis value is accepted but not used in the arc/circular move.
- X and Y ratio values (M302 and M303) must be same value. If this value is not same and arc/circle command is executed, the buffer state will error.
- Similar to G1, acceleration and deceleration are not used for G2 and G3 arc/circular moves.

In G2 and G3 commands, start position is the current position. XY values are target end positions. When in relative mode (G91), XY values are relative values from the current positions. When in absolute mode (G90), XY values are absolute position values.

I and J values designate the center locations relative from the current positions. I and J are always ***relative*** values regardless of G90 and G91. If either I or J is not defined, value of zero is used. Both I and J cannot be zero which would mean zero radius value.

F (feedrate) value is the speed of movement in unit/second. Use **M223** to read modal G2/G1 speed setting.

Feedrate is multiplied by the Global Feedrate Factor. Global Feedrate Factor ranges from 1 to 100, representing 1% to 100% of the feedrate. Use Global Feedrate Factor to reduce all the speeds of the movements.

M-Code	Description	Format
160	Global Feedrate Factor	1 to 100

Table 23.0

Note: Default value of Global Feedrate Factor at power-up is 100.

If XY values are not defined, ***full circle*** motion will be performed.

Zero value for radius is not allowed, therefore both I and J cannot be zero.

Modality

Modality is supported on G2 and G3. Once G2 or G3 command is executed, any following commands without G command will retain the previous G command number.

X and Y values are modal and are shared among G0, G1, G2, and G3. When in relative mode (G91), X and Y value not defined will be zero. When in absolute mode (G90) X or Y value not defined will have previous modal value. When both X and Y values are not defined, G2 and G3 will be perform a full circle motion.

Modal feedrate for G2 and G3 is same.

Target Position Range
Target end positions XY are checked so that they are within the allowable range of circle motion. If they are within the allowable range but not on the circumference of the circle, an additional linear move will be performed so that the end positions are XY positions.

Allowable target positions (XY) is specified below and must be between outer and inner limits of the arc/circle move.

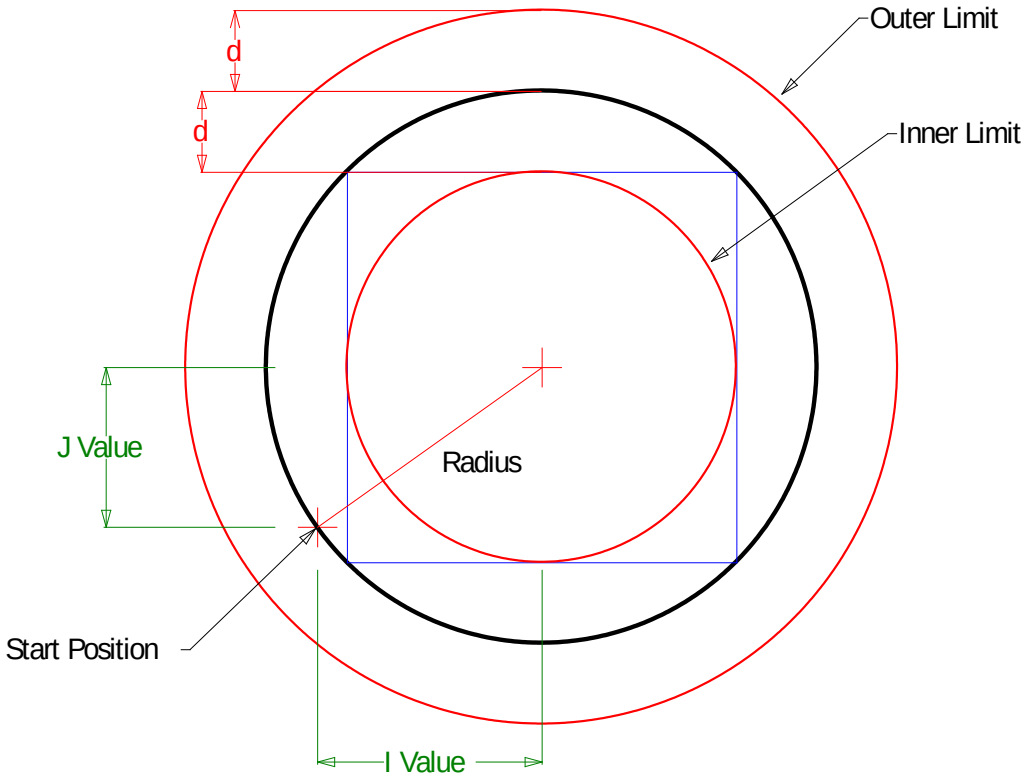

Figure 16.0

Tolerance value **d** is specified using following equation:

d = **Radius** * (1–1/Sqrt(2)) = **Radius** * 0.293

Inner limit radius = **Radius** – **d**
Outer limit radius = **Radius** + **d**

In RUN buffer state, if the XY target values of G2 or G3 are outside of the allowable range, buffer state will go into ERROR state. If the XY target values are within the allowable range but not on the circumference of the arc/circle, additional linear motion will occur to ensure that the end position will be the target XY positions.

Examples below show the end XY positions that are not on the circumference path but within the inner and outer allowable range.

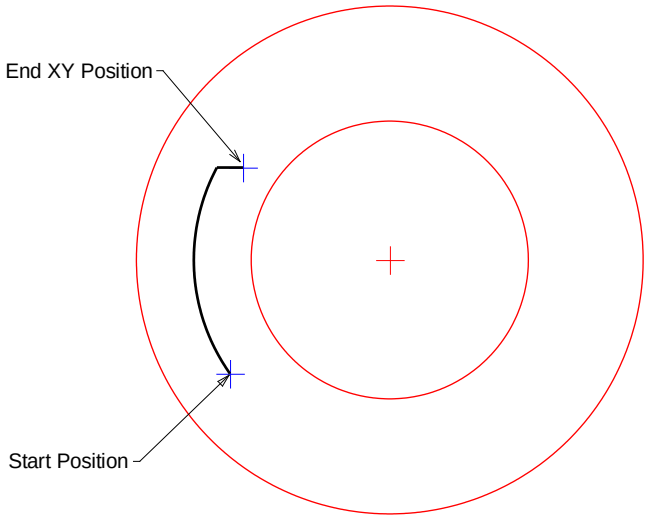

Figure 17.0

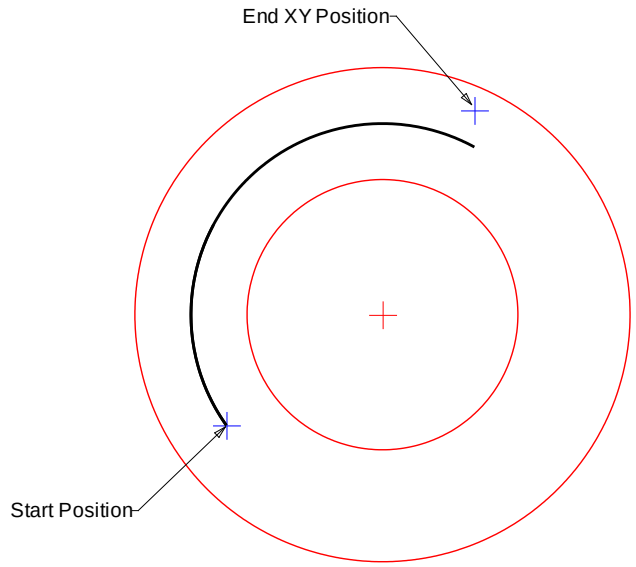

Figure 18.0

Example: The following example draws a cone.

```
G90                 ;***Set to Absolute Mode
G0 X0 Y0 Z0 F25     ;***Move all the axes back to zero position
G2 X2 I1 F10        ;***Make half circle with radius 1 with end point at (2,0)
G1 X0Y0 F10         ;***Move back horizontally back to (0,0)
G1 X1Y-3            ;***Move to bottom corner of cone
G1 X2Y0             ;***Move up to upper right corner of cone.
```

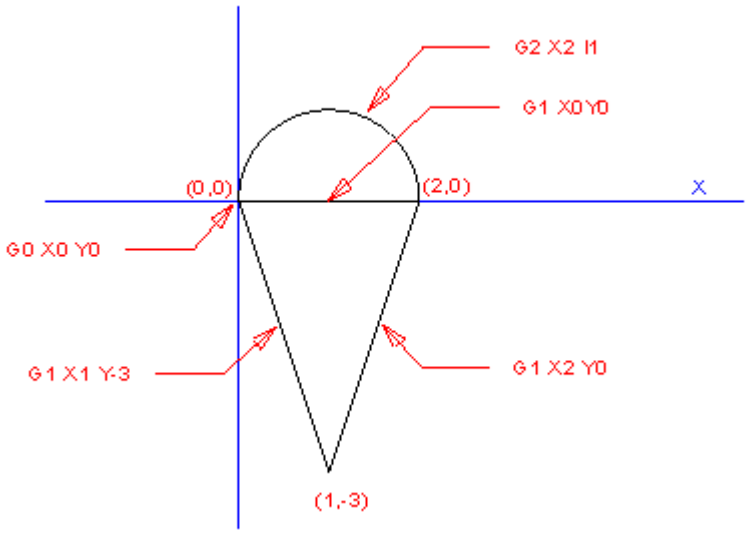

Figure 19.0

Example: The following example draws a smiling face

```
G90                          ;***Set to absolute mode
G0 X0Y0Z0 F10                ;***Move to zero position for all axis
X1Y1                         ;***Move to (1,1)
G2 I2 J2                     ;***Draw full outer circle, come back to same loc
G0 X1.5 Y1.5                 ;***Move to (1.5, 1.5)
G3 X4.5 Y1.5 I1.5 J1.5 ;***Draw smile arc
G0 X4.5 Y3.5                 ;***Move to bottom of right eye
G2 J0.5                      ;***Draw right eye
G0 X1.5 Y3.5                 ;***Move to bottom of left eye
G2 J0.5                      ;***Draw left eye
```

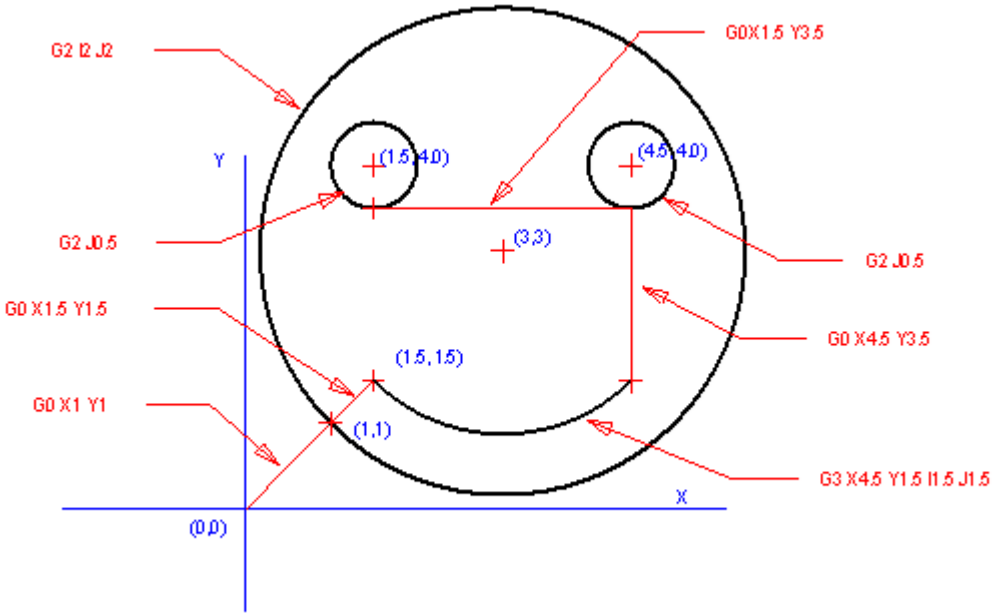

Figure 20.0

10.3 G4 - Dwell

ACE-G3 supports the following dwell G commands

G-Code	Description	Format
4	Dwell	G4 P[value]

Table 24.0

Before dwell is executed, all axes (XYZ) must be idle.

Dwell time is designated with character P with value of dwell time in milliseconds. Minimum value of P is 1 and maximum is 10,000. If dwell time P is not specified, default value of 1 is used.

Example:

```
G0 X1 Y2 Z3
G4 P100      ;***After moving to (1,2,3) wait 100 msec
X2
G4 P200      ;***After moving to (2,2,3) wait 200 msec
Y5
Z6
G4 P300      ;***After move to (2,5,6) complete, wait 300 msec
```

10.4 G20/G21 - Unit

ACE-G3 supports following unit setup commands.

G-Code	Description	Format
20	XYZ and Feedrate Unit in Inches	None
21	XYZ and Feedrate Unit in Millimeters	None

Table 25.0

G20 and G21 commands set the **G-code Unit,** which sets the XYZ and feedrate values used in G-code motion commands as inch or mm.

If the **System Unit** is set to inch (M301=1) and G21 command (mm) is issued, all the position values (XYZ and IJ) and feedrate values will be divided by 25.4 to convert to inch.

If the **System Unit** is set to mm (M301=0) and G20 command (inch) is issued, all the position values (XYZ and IJ) and feedrate values will be multiplied by 25.4.

Refer to the **System Unit** parameter setup in section 9.1 for detailed explanation of **System Unit** and **G-code Unit**.

Modality
G20 and G21 are modal and remain in this mode until changed.

Important Note:
Caution is advised when using G20 and G21. Incorrect use of G20/G21 may result in movement that is multiplied by 25.4!

10.5 G28 - Homing

ACE-G3 supports following G-code command for homing.

G-Code	Description	Format
28	Homing	G28 [X value][Y value][Z value]

Table 26.0

Homing Axis
When using G28 command, include the axes that need homing. ***If no axis is specified, all axes will perform home search in negative direction***.

Homing Direction
The value next to the axis indicates the direction of homing. A value greater than zero will perform home search in positive direction. Any value less than or equal to zero will perform home search in negative direction. If no value is specified, homing will be done in negative direction.

Homing speed
Homing routine shares the feedrate with G1 command. To set the homing feedrate, first set the G1 feedrate.

When homing, soft limit monitoring is not applied. At completion of homing, positions are reset to zero.

As with other motion G-codes, feedrate for homing is multiplied by the global feedrate factor using M228.

A useful command for homing is the **G68** command which set the positions to specified value. The example below shows a homing completion which moves out to location and then reset the position to zero.

Example:

```
G1 F1.5        ;***Sets the homing speed to 1.5
G28 X1 Y-1     ;***Home X axis in positive direction and Y in negative
G1 F0.8        ;***Sets the homing speed to 0.8
G28 Z-1        ;***Home Z axis in negative direction
G4P100         ;***Wait for all axis to complete and dwell 100 msec
G1 X10Y10Z2;***Move to location (10,10,2)
G68 X0Y0Z0  ;***Reset all the positions to zero.
```

Homing Mode Selection

Homing requires the limit or home switch be installed on the system and connected to the ACE-G3 controller. If only limit switch is to be used, set the homing mode to 0 which will use only the limit input for homing. To use the home and/or limit input, set the homing mode 1.

M-Code	Description	Range	Default value at power-up
318	X Axis Homing Mode	0 – limit only homing 1 – home/limit homing	0
319	Y Axis Homing Mode	0 – limit only homing 1 – home/limit homing	0
320	Z Axis Homing Mode	0 – limit only homing 1 – home/limit homing	0

Table 27.0

Homing Mode=0

The sequence below shows the homing mode 0 using limit switch only.

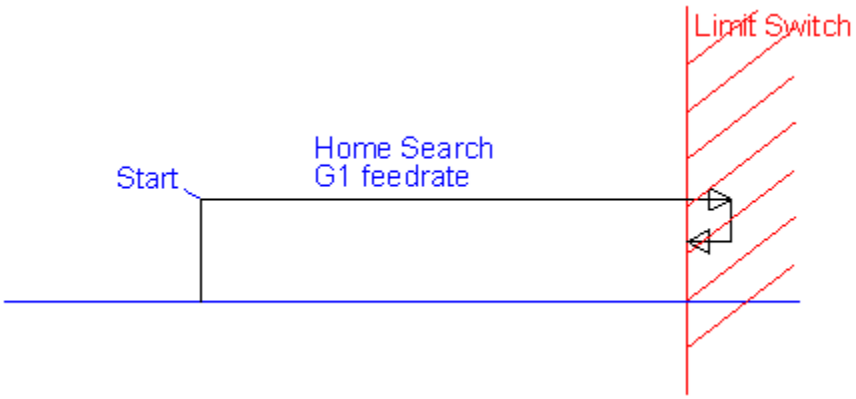

Figure 21.0

Homing Mode=1
Sequences below show various homing routines using homing mode 1. This homing mode has various possible start positions but regardless of the start position, a consistent home position will be found as shown below.

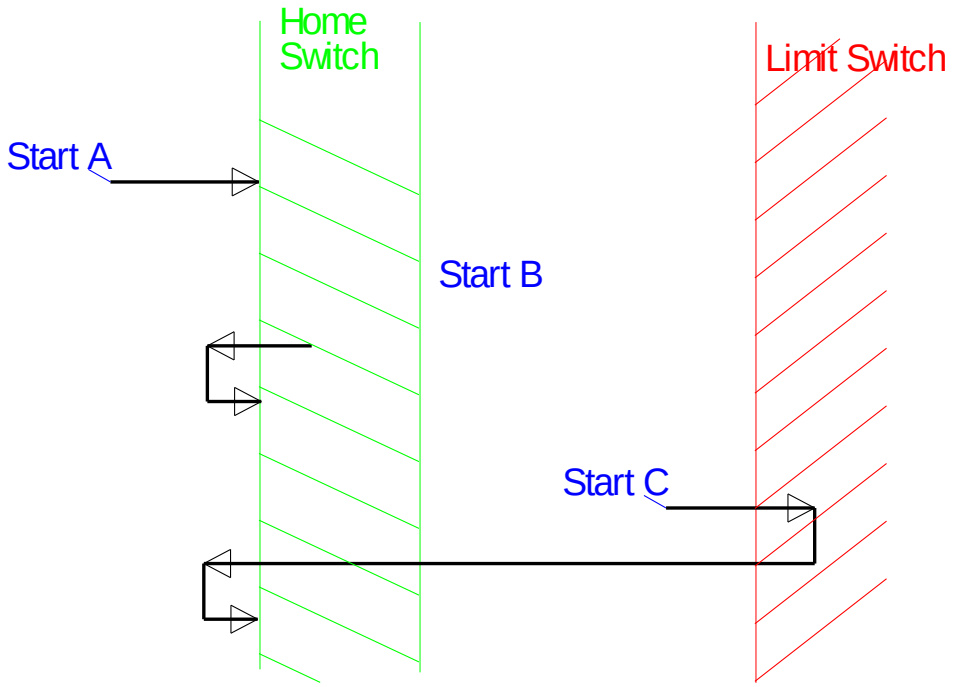

Figure 22.0

10.6 G52 - Temporary Coordinate System Offset

G52 command is used to offset the local coordinate system's origin by a specified amount. Multiple G52 codes are not cumulative.

G-Code	Description	Format
52	Temporary Coordinate System Offset	G52 X[value] Y[value] Z[value]; G52 (to disable)

Table 28.0

G52 shares with G0 on modal speed and acceleration.

Enable the **Temporary Coordinate System Offset** by issuing the G52 command with offset X, Y, and Z values. This will move the axes to the specified positions and consider this location as the zero positions for the subsequent G-codes.

Disable the **Coordinate System Offset** by issuing the G52 by issuing empty version of G52 command by itself.

Temporary Coordinate System Offset status can be read by following M-code commands.

M-Code	Description	Return Values
190	Get All Temporary Offset Status and Values	[X offset],[Y offset],[Z offset],[Offset Status]
191	X Axis Offset	[X offset]
192	Y Axis Offset	[Y offset]
193	Z Axis Offset	[Z offset]
194	Offset Status	0 – offset disabled 1 – offset enabled

Table 29.0

10.7 G68 - Set Position

G68 command is used to set the current position to a desired value. This command can be used in conjunction with the homing routine to set the current position to a desired value.

G-Code	Description	Format
68	Reset Positions to value	G68 [X value][Y value][Z value]

Table 30.0

Note that G68 is a buffered command and executed when buffer state is in RUN state and when all axes are in idle status. For immediate position command, use M204, M205, and M206 commands to set the position at desired value.
Before position values can be set to desired values using G68, all axes XYZ must be idle and not moving. Positions cannot be set while in motion.

10.8 G90/91 - Absolute/Relative Mode

ACE-G3 supports absolute or relative position values.

G-Code	Description
90	Absolute Positions used
91	Relative Positions used

Table 31.0

Important Notes:

- ***I and J values used in G2 and G3 are always in relative mode regardless of G90 or G91.***
- ***Caution is advised when using G90/G91. If motion is in absolute position value, but the controller is in relative mode, motion may result in motion outside of system range which may result in mechanical crash!***

10.9 Miscellaneous G-Codes

ACE-G3 supports following miscellaneous G commands.

G-Code	Description	Format
97	Spindle Speed Acc	G97 S[value in RPM] A[accel]

Table 32.0

S is the speed of the Spindle in RPM (rev/min).
A is the acceleration in milliseconds. Range of A is 0 to 1000.
Spindle can be controlled using M3, M4, M5.

11. Buffered M-Codes

M-codes less than 100 are loaded into the buffer list and executed sequentially when the buffer state in RUN state. M-codes above 100 are executed immediately.

11.1 M2 - Stop program

M2 command is used to change the buffer state to IDLE. Note that M2 is a buffer command which is loaded into the buffer list and executed sequentially with other commands in the buffer list when the buffer state is in RUN state. M30 is equivalent to M2.

11.2 M3/M4/M5 - Spindle Control

M3/M4/M5 commands are used for Spindle control.

M-Code	Description
3	Spindle On CW
4	Spindle On CCW
5	Spindle Off

Table 33.0

Above Spindle control M commands are loaded to the buffer list and executed sequentially with other buffered commands when the buffer state is in RUN state.

Spindle control M commands are executed when the motion status of XYZ is idle.

M3 and M4 commands are executed only when the Spindle is idle.

For example, if the Spindle is moving, and another Spindle move command is issued, buffer state goes to ERROR. If the Spindle is moving, issue the Spindle off command. Only when the Spindle is idle, issue Spindle move command.

When Spindle is turned on, pulse rate will be generated at a rate that corresponds to the RPM setting done by setup parameter below.

M-Code	Description
310	Spindle Ratio – number of pulses per revolution

Table 34.0

Spindle Speed and accelerations are set using the following G-codes.

G-Code	Description	Format
97	Spindle Speed and acceleration	G97 S[value in RPM] A[accel]

Table 35.0

11.3 M-Code for Digital Output Control

ACE-G3 has four digital outputs that can be controlled using digital output M-codes as a part of buffer commands.

Note that the commands are loaded into the buffer list and executed sequentially with other buffer commands when the buffer state is in RUN state.

Digital control buffer commands are executed only when all the XYZ axes status are idle.

The following are standard M-Codes supported for digital output control.

M-Code	Description	Equivalent DO
7	Coolant Mist On	DO1=1
8	Coolant Flood On	DO2=1
9	Coolant Mist/Flood Off	DO1=0 / DO2=0
10	Pallet On	DO3=1
11	Pallet Off	DO3=0
15	Dispense On	DO4=1
16	Dispense Off	DO4=0

Table 36.0

12. ACE-G3 specific M-Codes

M-codes above 100 and above are executed immediately whereas M-codes below 100 are executed as a part of buffer list, which are executed sequentially only when the buffer state is in RUN state. M-codes above 100 and above are commands specific to ACE-G3 controller.

12.1 Digital Inputs and Outputs

ACE-G3 has three general purpose digital inputs and 4 digital outputs are read or written using the following commands.

M-Code	Description	Details	R/W
100	Read All Digital Output Status	Returns digital output status. Bit 0 – DO1 Bit 1 – DO2 Bit 2 – DO3 Bit 3 – DO4	R
101	DO1	To set DO1 value, use = character and the value. Example: M101=1. To read send command and reply is DO1 value of 0 or 1.	RW
102	DO2	To set DO2 value, use = character and the value. Example: M102=0. To read send command and reply is DO2 value of 0 or 1.	RW
103	DO3	To set DO3 value, use = character and the value. Example: M103=1. To read send command and reply is DO3 value of 0 or 1.	RW
104	DO4	To set DO4 value, use = character and the value. Example: M104=0. To read send command and reply is DO4 value of 0 or 1.	RW
110	Read All Digital Input Status	Returns digital input status. Bit 0 – DI1 Bit 1 – DI2 Bit 2 – DI3	R
111	DI1 Read	Returns 0 or 1	R
112	DI2 Read	Returns 0 or 1	R
113	DI3 Read	Returns 0 or 1	R

Table 37.0

Important Note:
Digital Output commands, M101-M104, are Immediate commands and will update the digital output regardless of the XYZ motion status or Buffer state. If the Digital Outputs are connected to external system, extreme caution is advised when issuing the DO commands.

12.2 Motion Inputs

ACE-G3 has home and limit inputs for each axis which can be rad using the following M-code commands.

M-Code	Description	Details	R/W
120	Get all motion input status.	Returns [X Axis motion DI],[Y Axis motion DI],[Z Axis motion DI].	R
121	X Motion IO Status – Limits and Home	Bit 0 – Minus Limit Bit 1 – Home Bit 2 – Plus Limit	R
122	Y Motion IO Status – Limits and Home	Bit 0 – Minus Limit Bit 1 – Home Bit 2 – Plus Limit	R
123	Z Motion IO Status – Limits and Home	Bit 0 – Minus Limit Bit 1 – Home Bit 2 – Plus Limit	R

Table 38.0

12.3 Motion Positions

From ACE-G3, current positions are monitored with the following commands.

M-Code	Description	Details	R/W
200	Get All Positions	Returns unit and raw pulse position values [Xpos],[Ypos],[Zpos],[Xpulse],[Ypulse],[Zpulse]	R
201	X Raw Pulse Position Value	Value returned is a raw pulse counter value. Value type is whole number.	R
202	Y Raw Pulse Position Value	Value returned is a raw pulse counter value. Value type is whole number.	R
203	Z Raw Pulse Position Value	Value returned is a raw pulse counter value. Value type is whole number	R
204	X Position Value	Value returned is unit based position value that has been updated with unit/pulse ratio. Value type is decimal float value.	R
205	Y Position Value	Value returned is unit based position value that has been updated with unit/pulse ratio. Value type is decimal float value.	R
206	Z Position Value	Value returned is unit based position value that has been updated with unit/pulse ratio. Value type is decimal float value.	R
207	X Raw Encoder Position Value	Value returned is a raw encoder counter value from EA/EB encoder inputs. 4X quadrature encoding is used. Value type is whole number.	RW
208	Y Raw Encoder Position Value	Value returned is a raw encoder counter value from EA/EB encoder inputs. 4X quadrature encoding is used. Value type is whole number.	RW

209	Z Raw Encoder Position Value	Value returned is a raw encoder counter value from EA/EB encoder inputs. 4X quadrature encoding is used. Value type is whole number.	RW

Table 39.0

Important Notes:
XYZ positions and pulse positions are read only. For setting these positions to desired values, use G68 command.

12.4 Motion Statuses

From ACE-G3, current motion statuses are monitored with the following commands.

M-Code	Description	Details	R/W
210	Get all axes status	Returns all XYZ axes status [X Axis Status],[Y Axis Status],[Z Axis Status]	R
211	Get X Axis Status	0 – Idle 1 –accelerating 2- constant speed 3-decelerating	R
212	Get Y Axis Status	0 – Idle 1 –accelerating 2- constant speed 3-decelerating	R
213	Get Z Axis Status	0 – Idle 1 –accelerating 2- constant speed 3-decelerating	R

Table 40.0

12.5 Modal Status Reading

From ACE-G3, current modal values are monitored with the following commands.

M-Code	Description	Details	R/W
220	Get Modal Values	Returns following modal values: 1) G-code Number (0,1,2,3) 2) Abs/Relative (1-Abs, 2-Relative) 3) G0 Acceleration 4) Spindle Acceleration Values are separated by commas.	R
221	Read Modal Position Unit	Read the current modal unit value as set by G20/21. Return value: 0 – mm 1 – inch	R
222	Read Modal Position Values	Read the current modal XYZ position values. [X modal position],[Y modal position],[Z modal position],[X modal pulse position],[Y	R

		modal pulse position],[Z modal pulse position],[mm or inch]	
223	Read Modal Speed	Returns following modal speed values. 1) G0 modal speed 2) G1 modal speed 3) G2/G3 modal speed 4) Spindle Speed Values are separated by commas	R

Table 41.0

12.6 Miscellaneous M-codes

M-Code	Description	Details	R/W
160	Global Feedrate percent factor	Range is 1 (1%) to 100 (100%) for global feedrate factor.	RW
190	Get All Temporary Offset Values and Status	Returns X offset, Y offset, Z offset, and Offset status	R
191	Get X Offset	Returns X offset	R
192	Get Y Offset	Returns Y offset	R
193	Get Z Offset	Returns Z offset	R
194	Get Offset Status	Returns Offset Status	R
214	Read Spindle Speed Value	Returns current RPM speed of the Spindle.	R
291	Software Version	Firmware version	R

Table 42.0

Appendix

G-Code List

#	Name	Format
G0	Rapid Linear Move	G0 X[value] Y[value] Z[value] F[value] A[value]
G1	Linear Move	G1 X[value] Y[value] Z[value] F[value]
G2	Arc/Circle CW	G2 X[value] Y[value] I[value] J[value] F[value]
G3	Arc/Circle CCW	G3 X[value] Y[value] I[value] J[value] F[value]
G4	Dwell	G4 P[value in milliseconds]
G20	Unit - Inch	G20
G21	Unit - Millimeter	G21
G28	Home Search	G28 X[dir value] Y[dir value] Z[dir value]
G52	Temporary Coordinate System Offset	G52 X[value] Y[value] Z[value]; G52 (to disable)
G68	Set Position	G68 X[value] Y[value] Z[value]
G90	Absolute Mode	G90
G91	Relative Mode	G91
G97	Spindle Speed	G97 S[value in RPM] A[value in msec]

M-Code List

#	Name	Buffered/ Immediate	Read/ Write
M2	Stop Buffer	B	W
M3	Spindle On CW	B	W
M4	Spindle On CCW	B	W
M5	Spindle Off	B	W
M7	Coolant Mist On	B	W
M8	Coolant Flood On	B	W
M9	Coolant Mist Flood Off	B	W
M10	Pallet On	B	W
M11	Pallet Off	B	W
M15	Dispense On	B	W
M16	Dispense Off	B	W
M30	Stop Buffer	B	W
M100	Get All DO	I	R
M101	DO1	I	RW
M102	DO2	I	RW
M103	DO3	I	RW
M104	DO4	I	RW
M110	Get All DI	I	R
M111	DI1	I	R
M112	DI2	I	R
M113	DI3	I	R
M120	Get All Motion DI	I	R
M121	X Motion DI	I	R
M122	Y Motion DI	I	R
M123	Z Motion DI	I	R
M160	Global Speed Percent	I	RW
M190	Get All Temporary Offset Values	I	R
M191	Get X Offset Value	I	R
M192	Get Y Offset Value	I	R
M193	Get Z Offset Value	I	R
M194	Get Offset Status	I	R
M200	Get All Positions	I	R
M201	X Pulse Position	I	R
M202	Y Pulse Position	I	R
M203	Z Pulse Position	I	R
M204	X Position	I	R
M205	Y Position	I	R
M206	Z Position	I	R
M207	X Encoder Position	I	RW
M208	Y Encoder Position	I	RW
M209	Z Encoder Position	I	RW
M210	Get All Motion Status	I	R
M211	Get X Status	I	R
M212	Get Y Status	I	R
M213	Get Z Status	I	R
M214	Get Spindle Speed	I	R
M220	Get All Modal Info	I	R
M221	Get Modal G-code Unit	I	R
M222	Get Modal Positions	I	R

M223	Get Modal Feedrate	I	R
M291	Get Software Version	I	R
M300	Store to Flash	I	W
M301	System Unit	I	RW
M302	Ratio X	I	RW
M303	Ratio Y	I	RW
M304	Ratio Z	I	RW
M305	Limit Polarity	I	RW
M306	Home Polarity	I	RW
M307	Dir Polarity	I	RW
M308	DI Polarity	I	RW
M309	Device ID	I	RW
M310	Spindle Ratio	I	RW
M311	Soft Limit Enable	I	RW
M312	X Neg Soft Limit	I	RW
M313	X Pos Soft Limit	I	RW
M314	Y Neg Soft Limit	I	RW
M315	Y Pos Soft Limit	I	RW
M316	Z Neg Soft Limit	I	RW
M317	Z Pos Soft Limit	I	RW
M318	X Homing Mode	I	RW
M319	Y Homing Mode	I	RW
M320	Z Homing Mode	I	RW
M400	Stop Buffer	I	W
M401	Run Buffer	I	W
M402	Abort Buffer	I	W
M403	Reset Buffer	I	W
M404	Clear Buffer Error	I	W
M405	Get Buffer Error Msg	I	R
M406	Get Buffer Head	I	R
M407	Get Buffer Tail	I	R
M408	Get Buffer Size	I	R
M409	Get Buffer State	I	R
M410	Get All Buffer Status	I	R