CHAPTER 2

SPECIFICATIONS

2.1 SPECIFICATION

1. Controlled Axes 3 axes (X, Y and Z)

2. Simultaneous controllable axes

   3 axes (linear interpolation)
   2 axes in any plane (circular interpolation)
   X,Y axis circular, Z linear (helical interpolation)

3. Least input increment

Inch system	.0001
Metric system	.001
Degrees	.001

4. Least output increment

   Inch .0001 (.00254 mm)

5. Maximum programmable dimensions

Inch	+/-838.8607
Metric	+/-8388.607
Degrees	+/-720.000

6. Part program code, EIA RS-358-A, 7 bit with even parity

7. Part program code format

   Variable block, word address format

   Inch system
   :5 N4 G3 a+34 A+33 P+34 F31 S4 T2 M2;

   Metric system
   :5 N4 G3 a+43 A+33 P+43 F4 S4 T2 M2;

NOTES

a represents X,Y,Z,U,V,W,I,J,K,R
; represents end of block

Decimal point programming is required in all a, F, A, and P fields. Leading zeroes to the left of the decimal place and trailing zeroes to the right of the least significant digit may be omitted.

8. Table Travel

X Axis	17.5 inches	444 mm
Y Axis	11.1 inches	282 mm

9. Rapid Traverse rate

250 ipm X, Y (6350 mmpm)
200 ipm Z

10. Feedrate

Feedrate is programmable directly between .1 and 100.0 ipm (2 to 2540 mmpm).

11. Automatic acceleration and deceleration

Deceleration normally occurs at the end of each part programmed block. A G99 code can be used to override the deceleration.

12. Feedrate Override Enable

Rapid Traverse Override. In Setup mode and in Block Run mode, the feedrate override knob can be used to modify the rapid traverse rate.

Feedrate Override Enable. When this option is turned on, programmed feedrates can be modified between 10 and 125% in 1% increments. Feedrate Override Enable is turned off either manually or by the next programmed tool change.

13. Preparatory functions (G functions)

G codes consist of address G plus up to 3-digits and specify various control modes.

The following G codes are in effect on system powerup:

G0 (rapid traverse), G17 (XY plane circular interpolation)
G30 (mirror image off), G40 (diameter compensation off)
G70 (inch data input), G72 (transformation off)
G75 (multi-quadrant input), G90 (absolute programming)

14. Absolute/Incremental Programming (G90/G91)

G codes select either absolute or incremental programming.

G90: Absolute programming
G91: Incremental programming

In addition U, V, and W are always incremental X, Y, and Z moves respectively, B is an incremental A move.

15. Programming absolute zero point (G92)

A G92 command with the address X, Y and Z establishes a local coordinate system with the current tool position having the specified coordinate values.

16. Fixture offsets (G97)

A G97 command with X, Y data sets up a work coordinate system which is offset from the base coordinate system by the designated distance. G96 re-establishes the base coordinate system.

17. Positioning (G00) G0X___Y___Z___;

A G0 modal command moves the machine at rapid traverse rate. If a Z move has been programmed and is above the current Z position, Z motion will occur, then XY motion; if the Z move is minus, XY motion will occur then Z motion.

18. Linear Interpolation G1X___Y___Z___F___;

With this command, 3 axis linear interpolation occurs at a feedrate specified by the F code.

19. Buffer storage

During normal operation one block of program information is read into the buffer register while the previous block is being performed. This is to avoid interruptions between blocks during transfer of data to the active registers. During tool radius compensation 2 data blocks are buffered.

20. Circular interpolation G2 (G3)X___Y___I___J___F___;

These commands move the tool along the specified circular path at a feedrate specified by the F code. X and Y specify the end point of circular motion, I and J specify the center of the circular path in the XY plane. With the proper selection of addresses similar circular interpolation is programmed also in the YZ and ZX planes. G2 is for clockwise motion, and G3 is for counterclockwise motion.

G18 G2 (G3)X___Z___I___K___F___; ZX Plane
G19 G2 (G3)Y___Z___J___K___F___; YZ Plane

NOTE

Circular interpolation is possible over 1 or more quadrants.

21. Circular interpolation by radius programming
G2 (G3)X___Y___R___F___;

In circular interpolation in the XY plane, the radius of an arc can be specified directly instead of the I, J center point data. The circle must be less than 180 degrees.

Circular interpolation by fillet programming
G22(G23)X___Y___R___F___;

In Circular interpolation in the XY plane, the radius of a fillet can be specified together with the intersection of the extended line segments. The fillet arc will be blended tangent to the two lines.

22. Dwell (G04) G04/___;G04;

These commands specify and initiate a control dwell in which the tool remains motionless. The minimum and maximum programmable dwell times are .01 and 163.84 seconds.

23. Helical interpolation G12 (G13)A___Z___F___;

Circular Interpolation in the XY plane can be combined with linear interpolation in the Z plane. It is assumed the tool is positioned over the start point with a polar coordinate move, therefore, R, A (start angle), I, J (center coordinates) have been previously defined. In the helical interpolation command A is the total

incremental angular distance, Z is the helical depth.

24. Polar coordinate programming R___I___J___A___;

As an alternate to defining dimensions using rectilinear coordinates (X,Y) it is possible to use polar coordinates where R is the polar radius, I and J define the pole center and A is the angle measured from the polar axis (the positive X axis).

25. Mirror Image (G30, G31, G32)

These codes act to invert the direction of X, Y axis input commands and thus enable mirror images of the programmed part.

G30: Mirror Image Cancel
G31: Mirror X-Axis
G32: Mirror Y-Axis

26. Tool diameter compensation (G40, G41, G42)

This function is for automatically compensating the tool path to the right or left of the programmed path by a distance equal to the radius of the tool used.

G40: Tool Diameter Compensation, Cancel
G41: Tool Diameter Compensation, Left
G42: Tool diameter Compensation, Right

Tool diameter values can be specified either via manual data input or by inserting them in the part program in the form:

T//Tool Dia

The maximum tool diameter that can be entered is +/- 3.2768 inch (83.231 mm).

27. Constant Surface Feed (G44, G45)

When a linear feed is programmed in conjunction with work surface programming, the value of cutter diameter compensation entered will cause this feedrate to be modified around a radius. Constant surface feed is imposed by G45 to reduce the feed around an inside radius and increase the feed around an outside radius as a function of the cutter diameter compensation value in effect. Cancel with G44.

28. Inch/Metric (G70, G71)

These codes designate the inch input system (G70) or the

metric input system (G71).

Usually these codes are written at the beginning of the program in a single block. If the input system is switched in the middle of the part program it is necessary to enter the current X, Y, Z position in the new dimension system coordinates.

29. Transformation Function: G73X___Y___Z___A___;

With this function, geometrical shapes specified by part programs can be enlarged or reduced in any desired ratio or may be rotated.

The range of reduction and enlarging is from .001 to 99.999 times.

G72: Transformation off

30. Drilling Canned Cycles: (G81-G87, G89)

These canned cycles perform various drilling operations:

SINGLE HOLE	MULTI-HOLE ROW	MULTI-HOLE FRAME	OPERATION	MOVEMENT IN	DWELL	SPINDLE	MOVEMENT OUT
G81	G181	G191	DRILL	FEED	NO		RAPID
G82	G182	G192	G'SINK OR S'FACE	FEED	YES		RAPID
G83	G183	G193	DEEP HOLE	PECK	NO		RAPID
G84	G184	G194	TAP	FEED	NO		FEED
G85	G185	G195	BORE REAM	FEED	NO		FEED
G86	G186	G196	BORE	FEED	STOP	STOP	RAPID
G87	G187	G197	CHIPBREAK	PECK	NO		RAPID
G89	G189	G199	BORE	FEED	YES		FEED

FORMAT:

G81(G___)X___Y___Z___F___ Single hole location
G83(G87) X___Y___Z___Z___Z___F___ Peck
G81(G___)X___X___Z___F___ Row parallel to X
G81(G___)Y___Y___Z___F___ Row parallel to Y
G81(G___)A___A___Z___F___ Bolt circle

For a row at any angle, determined by the initial hole location, clearance plane, and the signed incremental distance to the last hold in X and Y, and the number of holes.

G181(G___)X___Y___Z___X___Y___Z___P___F___ or polar
G181(G___)A___Z___A___Z___P___F___

For a frame of holes starting XYZ at either one of the four corners and signed incremental XY spacings.

G191(G___)X___Y___Z___X___Y___Z___P___P___F___

Parameter P shown above defines the number of holes.

31. Mill cycles

These functions enable machining frequently used shapes with one part program block.

The following mill cycles are available:

G170: Outside rect. frame with corner radius
G171: Inside rect. frame with corner radius
G172: Pocket mill ends with G171 automatically for finish pass
G173: Face mill rect. shape (zigzag path)
G174: Pocket mill rect. shape (zigzag path)
G175: Circle mill outside a boss
G176: Circle mill inside a circular shape
G179: Slot mill

The mill cycles include an approach and departure move tangential to the shape programmed, roughing and finish cuts and a Z axis step capability for deep shapes. In addition, the cycles as defined use the tool data table to generate a diameter compensation tool path.

The parameters for these cycles are:

	170 171	172	173 174	175 176	179
Center Point	X	X	X	X	X
Center Point	Y	Y	Y	Y	Y
Start Point	Z	Z	Z	Z	Z
Length	X	X	X		P
Width	Y	Y	Y		P - DIA
Fillet Radius	R	R			
Radius				R	
Angle of Rotation from X + axis					P
Full Depth	Z	Z	Z	Z	Z
Step Depth	Z	Z	Z	Z	Z
Stepover,overlap		P	P		
Clearance	P	P	P	P	
Mill feed	F	F		F	
Finish allowance	P	P	F	P	F

Finish feed	F	F		F	
Plunge feed	F	F	F	F	F

32. Spindle speed function (S functions)

The codes for spindle speed are designated directly in rpm. The range is from 60 rpm to 4200 rpm.

33. Tool function (T function)

The range of programmable tool numbers is from 1 to 24.

34. Tool length compensation

Tool length compensation can be entered into the system using any of the following methods:

1. Manual data input
2. Touch-off
3. Inserting in a part program

The format is T___/___ where the tool number is designated and after the </> is the tool length offset unsigned value.

The maximum tool length offset that can be entered is 6.5536 inches (166.461 mm).

24 sets of offsets are allowed.

The tool length offset is automatically set into the system when a tool change (M6) command is executed.

35. Miscellaneous function (M functions)

The conventional series of functions will be accepted and executed by the control. Additional functions are shown in the table below. Motion data in the block containing these functions takes place before the action.

STANDARD	ACTION	MOVE TO CLEAR POINT THEN ACTION
M00	Program Stop	M20
M01	Optional Stop	M21
M02	Rewind	M22
M06	Tool Change	M26
M07,08,09	Mist,Flood,Off	
M25	Quill Up	
M51	Index	

36. Expression evaluator

This enables the use of algebraic and trigonometric functions to define variables that can be inserted in a part program command block for execution later.

The algebraic operators are:

Operator	Meaning	Example
*	Multiplication	V1=1.5*X1
/	Divide	V1=V1/X1
+	Addition	V1=A1+1.5
-	Subtraction	V1=A1-A2

The following functions are available:

SIN___	Sine, ___ is expressed in degrees
COS___	Cosine, ___ is expressed in degrees
SQR___	Square root
TAN___	Tangent, ___ is expressed in degrees
ATN___	Arc Tangent, will return expression in degrees
ABS___	Absolute value of an expression
DTF___,___	Evaluates SQR (A1*A1+B1*B1)
#PI	Equal to 3.1415927

37. Conditions

"IF-THEN, END IF" type statements allow conditional execution of part program blocks. The allowable conditionals are:

<GE>	greater or equal to
<GT>	greater than
<LE>	less or equal to
<LT>	less than
<EQ>	equal to
<NE>	not equal to

The format of the conditional statement is:

```
?variable 1 <conditional> variable 2 or constant
___         ;  part program code
___!        ;  where !  is the END IF terminator
```

38. Labels (L function per program)

Up to 9 labels may be defined. This enables jumping back to a labelled part program block previously defined. This provides a powerful programming tool when combined with conditionals and expression evaluation.

39. Macro

A macro command is a special character (#n) referring to a group of part programming text statements terminating with another special character ($). A number (n) is

assigned to the group of statements (macro). The macro is stored, until called for later in the program by a call statement which will then permit it to be executed. The macro may contain unassigned variables (*) which must be assigned in the macro call statement. A total per program of 40 different macros are permitted.

40. Repetitive programming

Looping may be used to repeat a set of part program blocks a designated number of times.

The loop call is of the format:

=N loop end sequence number/number of repeats

When the part program block with the loop end sequence number is reached, an internal system counter tracking the number of repeats is decremented by one. If the counter is greater than zero, part program execution will loop back to the part program block following the loop call; otherwise it will exit the loop and execute the next part program data block.

Loops may be nested up to 4 deep.

The maximum number of repeats is 16,384.

41. Program number

Up to 5-digits can be written as program numbers immediately after the address: for example :123;

42. Block delete

While the OPTION/DEL is ON, a command block starting with / is ignored.

43. Part program search

A search can be made of the part program for a value designated MDI in any one of the following word address fields:

a. N___, sequence number

b. T___, tool number

c. :___, program number

44. Operator part program messages

Alphanumeric codes embedded in the part program text between ` ' will be displayed on the LCD when an M0, M1, M6, M20, M21 and M26 command occurs. This can be used to provide messages containing operator instructions.

45. Definition blocks

During SEARCH, part program blocks starting with "." are executed. This enables setting up part program modal information.

46. Dry run

The following dry run options are available:

a. NO Z. All Z axis motion is inhibited

b. No XYZ. All axis motion is inhibited

c. MAX F. All programmed feedrates are set at 100.9 ipm.

d. BRKPT. This enables specifying a sequence number at which program execution will halt.

47. Operator options

a. /DEL: Optional block skip

b. OPSTOP: Optional program stop

c. METRIC: Metric operation

d. BOSS 8 I, BOSS 4-7: This provides compatibility mode operation of part programs written for previous Bridgeport CNCs.

e. BAUD RATE: This enables setting the Port A, Port B serial line interface baud rates. The allowable settings are 110, 300, 1200, 4800, 9600 baud.

48. Feed hold

The feed hold function interrupts tool feed temporarily during automatic operations.

49. Emergency Stop

This function makes all commands inoperative. The axis drives are turned off, the spindle is stopped by braking. Emergency Stop causes loss of axis position. The axis drives must be re-"homed".

50. Automatic relocation

Automatic relocation of machine zero reference point on a start up after loss of power or at the beginning of a new day the control will automatically locate the machine zero reference point for all 3 axes.

51. MDI and LCD display

A 2 row by 20 alphanumeric character display can be used to show system status and to aid the operator in manually entering data into the system. The following data can be entered or displayed:

a. X,Y,Z axis position from part program zero

b. X,Y,Z axis position from mechanical reference zero

c. Tool length offsets and diameters

d. Setup points: TLO touchoff, part program zero, clear point, reference point

e. Moves to position: X, Y point, Z point, clear point, reference point

f. Part program data blocks

g. Part program messages

h. Selecting load part program source

i. Clear: Run, part program, tool data

j. Run status: sequence number, tool number, feedrate, speed

k. System error messages

l. MDI input data dialogue

52. Part program editing via Local data terminal

Part programs entered into the system can be edited using a data terminal connected to the Port A serial

interface line. The following functions are available:

a. APPEND string to current line.

b. Set line pointer to BOTTOM.

c. CHANGE old string to new string.

d. DELETE a number of lines.

e. FIND text.

f. INSERT part program text.

g. KILL the entire text buffer.

h. LIST the entire text.

i. Move line pointer forward or backward to NEXT desired line.

j. ABORT printout.

k. PRINT a designated number of lines.

l. PUNCH paper tape.

m. SUSPEND and RESUME printout.

n. Mass MODIFY all the old strings in the part program text buffer to a new string.

o. REPLACE line with new text.

p. Move line pointer to TOP.

q. DELETE characters in input text line.

r. DELETE input text line.

s. REPEAT input text line entered.

53. Part program editing via Front Panel

The Front Panel Keyboard can be used to FIND part program text, to DELETE a part program block and to INSERT data blocks using MDI.

54. Part program capacity

The capacity of part program memory is 12000 characters. This is equivalent to 100 feet of paper tape.

The part program text is retained when power is removed from the system. Duration of battery backup power is approximately 1 year.

55. Operation modes

The operational modes of the system are:

a. SETUP

The following SETUP functions are available:

SET: TLO offset via touchdown, part program zero, clear point, reference point.

MOVE: X, Y, Z, clear point, reference point

QUILL UP: Retracts quill and Z absolute position is set to the current TLO value.

JOG,JOG KNOB,JOG INCR: JOG continuously moves a designated axis while the axis direction key is pressed. JOG KNOB uses a rotary pulse generator to move the axes. JOG INCR moves the axes .0001 inch every time the MOVE key is pressed.

LOAD,CLEAR,EDIT: LOAD selects either the paper tape reader, remote serial interface or DNC link as a means of loading data to or from the system. CLEAR resets system parameters, the part program text or the tool tables. EDIT enables editing via a data terminal connected to the serial interface line at Port A.

b. MDI

In MDI the control communicates with the operator so that the proper input sequence and format is entered by asking for the required data. The control asks and the operator responds. In conjunction with the canned milling and drilling cycles which enable programming frequently used shapes and patterns in one data block, the dialogue type MDI simplifies shop floor programmable parts.

c. MDI STORE

This MDI mode stores the part program as it is input by MDI and executed enabling the operator to store the data while the first part is being machined.

d. RUN

There are two RUN modes:

AUTO: The control automatically executes the part program until a programmed STOP occurs.

BLOCK: The part program is executed one block at a time.

56. Spindle interlock

If the spindle is not turned on, all axis feed is inhibited. If the spindle contactor drops out in the middle of a feed move, an EMERGENCY STOP condition will be initiated.

57. Overtravel

Power to the axis drive motors is removed when the overtravel limit switch is actuated. When the machine is stopped by the overtravel limit, the axes must be jogged back off the limit and the system rehomed.

58. Software overtravel limit

This function checks the programmed axis move against a stored stroke limit. If the move has been programmed outside of the allowable move dimension, operation is suspended and an error message is displayed.

59. Error diagnostics

During operation, the following checks are made for error conditions:

a. Part program data input

b. Cutter compensation calculations

c. Illegal interpolation command

d. Communications diagnosis

e. System diagnosis

f. Front panel diagnosis

g. Auxiliary function controller diagnosis

h. Axis drive fault

i. Tool table or part program memory fault

In addition, on power-up the system executive program embedded in PROM is validated.

60. Data input/output interface

a. Paper tape reader.

A REMEX paper tape reader can be connected via a parallel port to input part programs.

b. Local RS232 or current loop (20ma) serial interface designated as Port A.

A data terminal connected to this interface can input part program data via the Editor. It also can be used in conjunction with the on-line debugging monitor to check system operation. Additionally, data entered via the Port A serial interface can be sent out to a remote computer linked via the serial interface line Port B.

c. Remote RS232 or RS422 serial interface designated as Port B.

This provides means for loading part programs via an optional EZ-FILE or via a remote EZ-CAM~ part programming terminal or mass storage device. The desired part program number can be entered via the Front Panel Keyboard and transmitted to the remote device.

NOTE

Baud rates for both Ports A and B serial interfaces are setable via the Front Panel Keyboard.

61. DNC link

A communications protocol using data verification and having retry capability is available for use with part programs over 12000 characters. The DNC link is established via serial interface line labelled Port B for use with the EZ-CAM~ or mass storage device.