

## **Copyright Warning & Restrictions**

The copyright law of the United States (Title 17, United States Code) governs the making of photocopies or other reproductions of copyrighted material.

Under certain conditions specified in the law, libraries and archives are authorized to furnish a photocopy or other reproduction. One of these specified conditions is that the photocopy or reproduction is not to be “used for any purpose other than private study, scholarship, or research.” If a user makes a request for, or later uses, a photocopy or reproduction for purposes in excess of “fair use” that user may be liable for copyright infringement,

This institution reserves the right to refuse to accept a copying order if, in its judgment, fulfillment of the order would involve violation of copyright law.

**Please Note: The author retains the copyright while the New Jersey Institute of Technology reserves the right to distribute this thesis or dissertation**

Printing note: If you do not wish to print this page, then select “Pages from: first page # to: last page #” on the print dialog screen

The Van Houten library has removed some of the personal information and all signatures from the approval page and biographical sketches of theses and dissertations in order to protect the identity of NJIT graduates and faculty.

INTRODUCTORY GUIDE TO  
BRIDGEPORT SERIES I CNC  
MILLING, DRILLING AND BORING MACHINE

by *V. Doshi*  
Apurva V. Doshi

Thesis submitted to the Faculty of the Graduate School of  
the New Jersey Institute of Technology in partial fulfillment of  
the requirements for the degree of  
Master of Science in Mechanical Engineering  
1986

APPROVAL SHEET

Title of Thesis: Introductory Guide to Bridgeport Series I  
CNC Milling, Drilling and Boring Machine

Name of Candidate: Apurva V. Doshi  
Master of Science in Mechanical  
Engineering, 1986

Thesis and Abstract Approved:

\_\_\_\_\_  
Dr. Sachio Nakamura      Date  
Assistant Professor  
Mechanical Engineering Department

\_\_\_\_\_  
Date

\_\_\_\_\_  
Date

VITA

Name: Apurva Vinodchandra Doshi.

Permanent Address:

Degree and date to be conferred: M.S.M.E., 1986

Date of birth:

Place of birth:

Secondary education: Himmat High School, 1979.

Collegiate institutions attended	Dates	Degree	Date of Degree
Saurashtra University .....	June,79 -May,83 .....	B.E.(Mech.) .....	June,83 .....
N. J. Institute of Technology .....	Sept.,83 -Dec.,85 .....	M.S.M.E. .....	May,86 .....

Major: Mechanical Engineering

## ABSTRACT

Title of Thesis: Introductory Guide to Bridgeport Series I  
CNC Milling, Drilling and Boring Machine

Apurva V. Doshi, Master of Science in Mechanical  
Engineering, 1986

Thesis directed by: Dr. Sachio Nakamura

Assistant Professor

Mechanical Engineering Department

This thesis is intended to serve as an easy introductory guide on how to work with Bridgeport Series I CNC Milling, Drilling and Boring machine. An attempt has been made to make the subject matter simpler to give the reader a good start in programming and operating this machine.

The spectrum of technological implementation of CNC machine tools is extremely broad. Companies utilizing advanced technologies such as computer graphics, DNC and Group Technology are in this field for some time. Even small shops are getting equipped with CNC machine tools. The present CNC systems, which are less costly, more reliable and far more capable than conventional automatic equipments, have brought CNC within reach of even the smallest shop. Considering their evergrowing demand and acceptance, almost every mechanical engineer will have to deal with them at one point.

The work presented in this thesis is a concise introductory guide to Series I CNC Milling machine. Various functions of NC machine have been linked with the main text to provide a reasonably understandable introductory manual for the particular CNC machine.

Blank Page



To my Mother,  
without whom this would  
not have been possible.

## TABLE OF CONTENTS

Chapter	Page
LIST OF FIGURES.....	xi
I. INTRODUCTION.....	1
1.1. Introductory Comments.....	1
1.2. Numerical Control (N/C) Machine.....	3
1.3. Computer Numerical Control (CNC).....	5
1.4. Direct Numerical Control (DNC).....	5
1.5. Scope of Thesis.....	7
II. OPERATION OF A NUMERICAL CONTROL INSTALLATION...	8
2.1. Introductory Comments.....	8
2.2. Operation of an NC Machine.....	8
III. PART PROGRAMMING FOR BRIDGEPORT SERIES I CNC MACHINE.....	19
3.1. Features and Functions of The Machine...	19
3.2. Coordinate System.....	21
3.3. Program Entries.....	23
IV. PROGRAMMING EXAMPLES.....	39
4.1. Introductory Comments.....	39
4.2. Programming Examples.....	39
4.3. Looping.....	64
4.4. Macro Subroutines.....	65
V. OPERATION OF BRIDGEPORT SERIES I CNC MACHINE...	68
5.1. Introductory Comments.....	68
5.2. Operator's Main Control Panel.....	68
5.3. Spindle Control Panel.....	76

Chapter	Page
5.4. Special Operations Panel.....	78
5.5. Operation In Set up.....	82
5.5.1 Positioning Along An Axis.....	82
5.5.2 Tool Length Offset.....	82
5.5.3 Tool Diameter.....	87
5.6.1 Normal Operation Procedure.....	88
5.6.2 Special Conditions.....	90
VI. CONCLUSION.....	92
6.1. Concluding Remarks.....	92
6.2. Future Updating.....	92
BIBLIOGRAPHY.....	95

## LIST OF FIGURES

Figure	Page
I. Fig. 1-1 Bridgeport Series I CNC Machine.....	6
II. Fig. 2-1 Standard Tape.....	11
III. Fig. 2-2 Standard BCD Formats.....	12
IV. Fig. 2-3 A CNC System Layout.....	16
V. Fig. 3-1 Part Coordinate System.....	22
VI. Fig. 3-2 Table 3-1, Input Tape and Program Codes.	24
VII. Fig. 3-3 CW or CCW Motion in Switchable Planes...	30
VIII. Fig. 4-1 Manuscript Layout.....	40
IX. Fig. 4-2 Example 1.....	43
X. Fig. 4-3 Example 2.....	43
XI. Fig. 4-4 Arc Center Offsets.....	45
XII. Fig. 4-5 Example 3.....	47
XIII. Fig. 4-6 Example 4.....	47
XIV. Fig. 4-7 Example 5.....	49
XV. Fig. 4-8 Example 6.....	49
XVI. Fig. 4-9 Example 7.....	53
XVII. Fig. 4-10 Example 8.....	53
XVIII. Fig. 4-11 G81 - Drilling Cycle.....	56
XIX. Fig. 4-12 G82 - Spotfacing Cycle.....	56
XX. Fig. 4-13 Deep Hole Drilling Cycles.....	58
XXI. Fig. 4-14 G84 - Tapping Cycle.....	58
XXII. Fig. 4-15 G85 - Boring Cycle.....	60
XXIII. Fig. 4-16 G86 - Boring Cycle.....	60

Figure	Page
XXIV. Fig. 4-17 G89 - Boring Cycle.....	60
XXV. Fig. 4-18 Example 9.....	63
XXVI. Fig. 5-1 Operator's Main Control Panel.....	69
XXVII. Fig. 5-2 Spindle Control Panel.....	77
XXVIII. Fig. 5-3 Special Operations Panel.....	79
XXIX. Fig. 5-4 Spindle Speed Changer and Brake.....	81
XXX. Fig. 5-5 Machine Coordinate System.....	83
XXXI. Fig. 5-6 Tool Length Offset.....	85

## CHAPTER I

### INTRODUCTION

#### 1.1. INTRODUCTORY COMMENTS

The first numerically controlled machine was born shortly after World War II in 1952 at MIT. The machine used for the Air Force was a three axis numerically controlled profile milling machine. Since then, the NC technology has advanced rapidly. By 1979, the percentage of numerical control shipments as compared to the total dollar shipments of all metal-cutting machine tools of a comparable type like mills, drills, lathes, boring mills, machining centers has reached the mark of 74% (1).

The definitions of Numerical Control are as varied as its applications. The most simple definition is, ' a process controlled automatically by coded instructions expressed as numbers.' The most universally accepted definition is the one offered by the Electronic Industries Association which defines it as: " A system in which actions are controlled by the direct insertion of numerical data at some point. The system must interpret at least some portion of this data." (1) Here the word "interpret" means translating numerical data to distance, speed, feed, angle etc. which can be expressed numerically.

The advantages of numerical control system are listed in the following:

(1) Increased productivity

All machine functions are controlled automatically so delays due to human element are reduced or eliminated. Thus, it enables the machine to perform more work by keeping it effectively useful for greater percentage of time.

(2) Better accuracy and uniformity of parts

As manual handling is virtually eliminated, the parts manufactured using the same tape are practically identical. Close tolerances are maintained. So, there are less parts to be rejected and less inspection is required. Thus, it improves quality control.

(3) Reduced tool and fixture cost

As jigs, fixtures and templates are eliminated and only coded tape is required which is a lot cheaper and easier to store, it reduces tooling and storage cost.

(4) Greater capability

It can produce parts which are very complex and almost impossible or very difficult and time consuming as well to be produced manually using conventional machine tools. It allows the engineers to have more freedom to design parts.

Use of computer facilities speeds up the program preparation and makes the job for the design engineer or the programmer a lot easier.

The rapid climb of numerical control is speeded up by availability and use of low cost mini/micro-computer, use of automatic programming systems, development of softwares for particular applications and computer graphics.

Using automatic programming systems like APT, COMPACT, EXAPT etc., a set of instructions is prepared and then passed through the computer. The computer handles the required calculations and prepares the tape ready for input in to the NC control system. Little or no calculation is required. On the other hand, CNC systems accept a tape with simple codes and the built in mini/micro computer handles most of the calculations. Computer Aided-Design (CAD) which involves computer graphics is one of the latest developments. The programmer can interact with a computer by developing part pictures on a CRT and moving a circle along it representing a cutter. This illustrates the partnership that can occur between the design engineer and the manufacturing engineer and can be considered as a part of Computer Aided Manufacturing (CAM) or Computer Integrated Manufacturing (CIM).

A Bridgeport Series I CNC machine is installed at the Mechanical Engineering Department of the New Jersey Institute of Technology. It is used as a part of laboratory work for undergraduate students. The author worked as a Teaching Assistant for that laboratory course. However, due to the lack of well documented beginner's guide, he had difficulty in teaching the course. It is hoped that this thesis would serve as an introductory manual the beginners.

## 1.2 NUMERICAL CONTROL (N/C) MACHINE

The main components of a conventional Numerical Control system are as follows:



(1) Data input, (2) Control unit, (3) Feed-back devices, (4) NC machine tool.

Data input is supplied manually or automatically. Usually, the data is supplied automatically to the control unit and the machine tool through a punched tape. The sections of punched holes on a tape are called blocks and contain coded instructions. The finalised tape is used over and over again. The tape is punched on a typewriter with a tape punching unit attachment or punching unit directly connected to a computer.

Either mechanical or photo-electric type of tape readers are used. Photo-electric type has silicon photo diodes located in the head which detect the light passing through the holes in the moving tape. The light beams are then converted to electrical signals and amplified. These signals are then sent to registers in the control unit and thus, the data is received and stored. The control then decodes and relays actuation signals to the machine tool drives which direct the machine tool through the required motions.

In a closed loop system, feed back devices are used to eliminate the errors between the commanded position and the actual location of the moving slides of NC machine tool. A sensor records the position of the slides and relays it back to the control unit. Signals thus received are compared to input signals on the tape and any discrepancies between them are rectified.

### 1.3 COMPUTER NUMERICAL CONTROL (CNC)

Computer Numerical Control is more sophisticated than straight NC and more common now a days. The control unit has a built in mini/micro-computer which has subprograms of its own. Because of this computer, the tape needs to be read only once for unlimited number of runs. It also relieves the programmer of many calculations and provides for factors like cutter compensation.

### 1.4 DIRECT NUMERICAL CONTROL (DNC)

Direct Numerical Control is becoming more and more popular. It does not use a punched tape as in NC or CNC. A computer interfaced with the control unit provides the information flow. The computer receives the equivalent of punched tape information from a compiling facility and directs its storage in a disk or a magnetic tape storage unit. The computer may be a mini-computer, several mini-computers linked together, a mini-computer linked to a large computer or a large computer. All controls originate from a remotely located control data processing room. Connected to the data processing room is a small console at the machine which operates as an input-output device. A programmer or a production supervisor working at the machine tool is thus able to make on the spot program changes which are relayed to and stored in computer's library. This arrangement of the control unit permits program tryout, debugging and modification in less time than is required with a tape system.

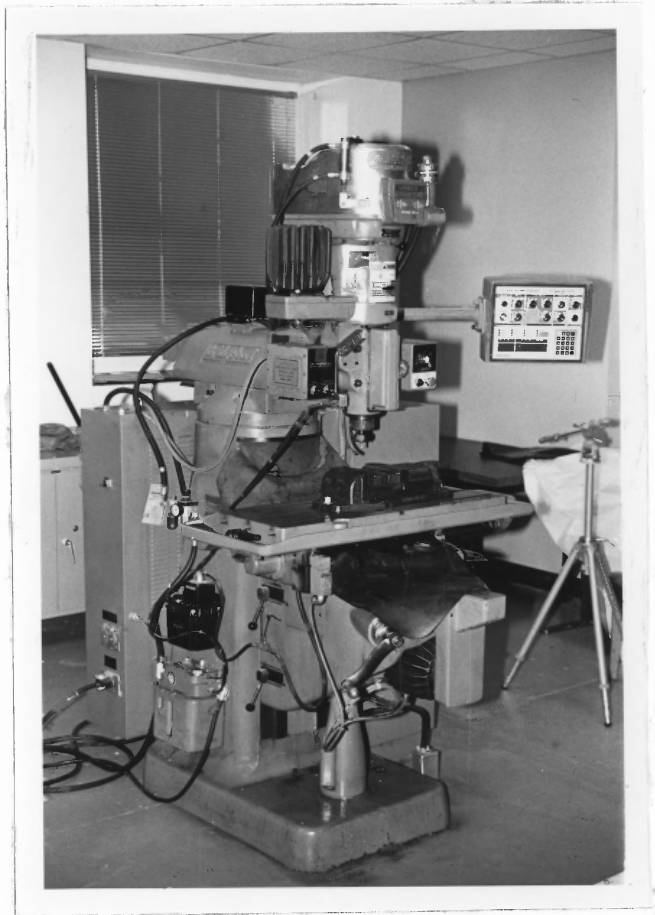


Figure 1-1 Bridgeport Series I CNC Machine

## 1.5 SCOPE OF THESIS

Owing to faster production, reduced production cost, capability to produce almost any part and low cost NC machines, every big industry has NC machines installed and even small shops are now getting equipped with them. Looking at the growth and wide acceptance of NC machines it is very likely that an engineer involved in design, manufacturing or maintenance will have to deal with them at one point. Therefore, it is essential for a mechanical engineer to be familiar with NC or CNC operations.

This thesis gives an introduction and a good start to a beginner on how to work with CNC machines. It is intended to serve as a beginner's guide to Bridgeport Series I CNC Milling, Drilling and Boring machine installed at the Mechanical Engineering Department at NJIT. Chapter 2 gives general description of CNC equipment while, chapter 3, 4 and 5 describe how to program and work with Bridgeport Series I CNC machine.

## CHAPTER II

### OPERATION OF A NUMERICAL CONTROL INSTALLATION

#### 2.1 INTRODUCTORY COMMENTS

The general operation of Numerical Control was briefly described in the previous chapter. In this chapter, all the components of an NC installation are discussed with emphasis on CNC system operation.

#### 2.2 OPERATION OF AN NC MACHINE

Generally, a numerical control installation comprises of following six elements:

(1) The control system, (2) The feedback unit, (3) The drive unit, (4) The electrically operated control equipments such as starters, relays etc., (5) The manual controls like switches, dials, buttons, (6) The machine tool.

In the following, these elements are discussed.

##### (1) The Control System

So far there has been four generations of control systems.

First generation systems used vacuum and gaseous tubes for control and amplification of electronic data. Second generation systems were almost entirely built of solid state components like transistors. Third generation systems are known as integrated circuitry in which, one relatively small device combines the functions that would be performed by a number of separate components like solid state diodes or

amplifiers.

Fourth generation uses the concept called Computer Numerical Control in which , instead of wired or specially designed circuits developed to perform NC functions and to meet the electronic requirements of a particular machine tool, a general purpose mini-computer is used. The particular machine tool and system requirements are programmed in to the computer. It offers advantages like, ability to edit a program while in the control system and the storage of a number of programs, canned cycles, macros, programmable subroutines, maintenance diagnostics, cutter compensation or lead screw error compensation. The machining instructions punched on a tape are read only once by the tape reader and then they are stored in the mini/micro-computer. Thus, the program can be repeated unlimited number of times without reading the tape every time.

Mini-computers are being replaced by micro-computres which are more compact, less costly, easier to maintain and better suited to shop environment.

#### 1a. Circuit Boards

A circuit or modular board consists of electronic circuits and components arranged with respect to a printed circuit pattern on the board. A faulty board can easily be detected and replaced reducing down-time. Most control systems contain duplicate module boards which may be interchanged between control systems of same manufacturer or within the same control system.

## 1b. Tape

Tape is used as a permanent medium to store the machine instructions. It is usually made of paper or Mylar film coated. Mylar tape is more expensive and used where repetitive runs are expected. The paper tape is more popular with CNC. The standard tape as shown in fig. 2-1 is one inch wide having eight rows of holes running along the length of the tape.

The following terms are used to describe the numerical control tape:

Channel or level or track:- A path parallel to the edge of the tape.

Row:- A path perpendicular to the edge of the tape.

Bit:- Presence or absence of a punched hole along a row.

Character:- A group of bits in a row.

Word:- An ordered string of characters.

Block:- A group of words that provides one complete instruction to the control system.

As shown in fig. 2-2, two standard binary coded decimal (BCD) formats RS-244-B and RS-358-B are used for character coding on the tape, the latter being popular with CNC systems. But, most CNC systems can handle either code. The number of holes across the tape are even and more for most of the characters for RS-358-B compared to RS-244-B with odd number of holes across the tape. The 8-level binary coded decimal tape can express numerical values, address characters like X, Y, Z and alphabetic and symbolic

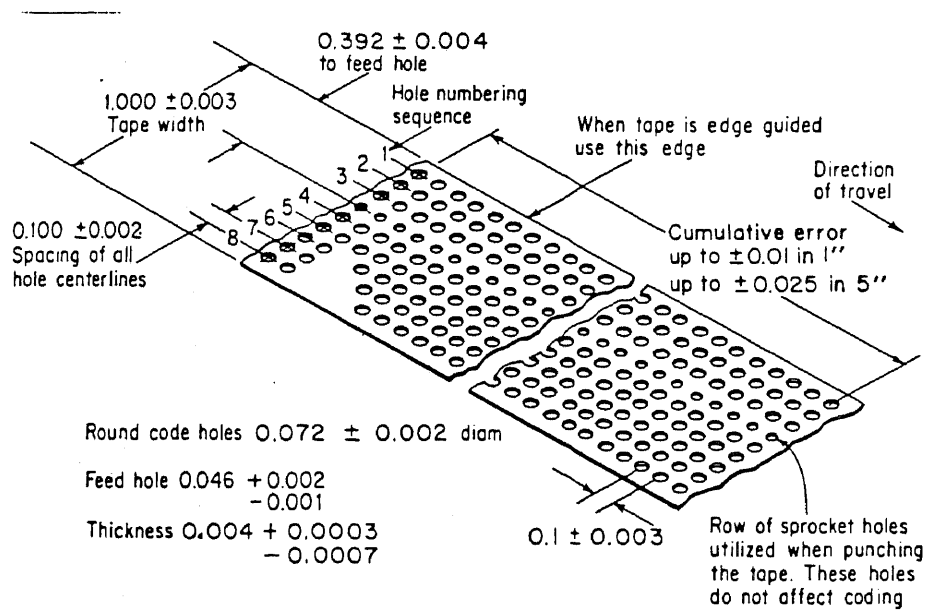
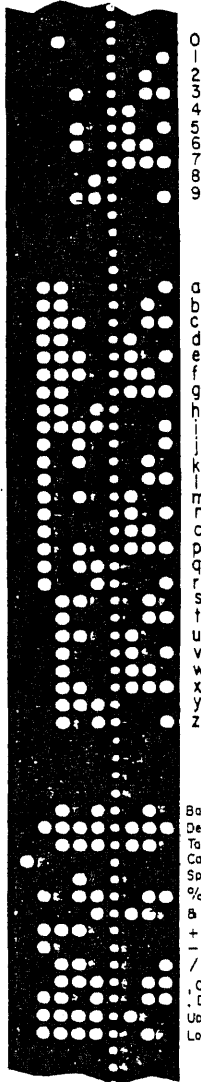


Figure 2-1 Standard Tape



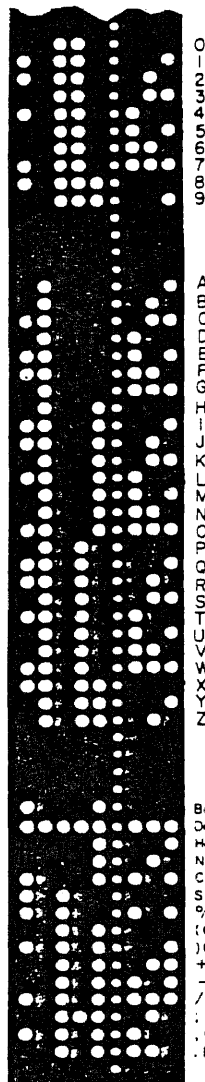
RS-244-B  
Level no.  
8 7 6 5 4 3 2 1



0  
1  
2  
3  
4  
5  
6  
7  
8  
9  
  
a  
b  
c  
d  
e  
f  
g  
h  
i  
j  
k  
l  
m  
n  
o  
p  
q  
r  
s  
t  
u  
v  
w  
x  
y  
z

Back space  
Delete  
Tab  
Carriage return and EOB  
Space  
%  
&  
+  
/  
, Comma  
. Decimal point  
Upper case  
Lower case

RS-358-B  
Level no.  
8 7 6 5 4 3 2 1



0  
1  
2  
3  
4  
5  
6  
7  
8  
9  
  
A  
B  
C  
D  
E  
F  
G  
H  
I  
J  
K  
L  
M  
N  
O  
P  
Q  
R  
S  
T  
U  
V  
W  
X  
Y  
Z

Back space  
Delete  
Horizontal tab  
New line  
Carriage return  
Space  
%  
( Open parenthesis  
) Closing parenthesis  
+  
/  
: Colon  
, Comma  
. Decimal point

Ignored by control  
Ignored by control  
Ignored by control  
Used for end of block  
Ignored by control  
Ignored by control  
Used for rewind stop  
Used for control out  
Used for control in  
  
Used for black delete (optional)  
Reference rewind stop

Figure 2-2 Standard BCD Formats

characters as well.

#### 1c. Tape Reader

Tape reader is the bridge between the coded punched information and the electronic signals of the control system. Usually, there are two types of tape readers, mechanical and photoelectric type. Mechanical type readers are used for slow speed reading and insert feelers or metal prongs in to the holes. But with development of CNC, photoelectric type has become more popular which are much faster and more accurate. Holes are detected by light passing through to a photoelectric cell. Readers with spoolers to handle the tape have a feature to rewind the tape automatically from a coded auxiliary instruction on the tape.

#### 1d. Data Storage

The coded data read is stored in electronic storage sections. Sometimes, there are two storage sections; one called 'Buffer storage' and the other called 'Active storage'. Buffer storage may not be present with point-to-point operation systems as there are fewer blocks to be read. With CNC systems, it may or may not be required depending on the speed of data transmission of the mini/micro computer. Buffer storage accepts data from the reader while the machine is operating under the instructions stored in active storage. This arrangement is very useful with contouring because the time required to

read the contouring information is relatively long. A contour is usually expressed as a series of very short segmented straight lines and each straight line needs a block of data. So, the buffer enables the control to operate on instructions from the active storage while the data for the next movement are being read in to the buffer. The transfer of data from buffer to active storage takes millionths of a second and occurs at a precise moment that the previous movement is completed for the block of information in the active storage.

#### 1e. Pulse Generation

The number of pulses produced and their rate of distribution depends on the distance to be moved in a particular axis and the feed rate. Suppose, each pulse represents a distance of 0.001 inch and it is desired to move 3.456 inches at a rate of 10 ipm then, the pulses produced will be 3456 and will be metered out in 0.3456 minutes. While contouring, the movements in different axes are very closely coordinated. At that time, the pulse train is measured such that the required number of pulses for each axis is completed precisely at the same time. If the result of metering the pulses is a straight line, it is called straight line interpolation and if it is an arc, it is called circular interpolation. If it were desired to move at the same rate along two axes such that the resultant is a 45 degree straight line, the number of pulses per a period of time for each axis will be same. But, if one axis component

were twice that of the other, the number of pulses also doubles for that axis.

Usually, pulses are generated and coordinated by a pulse rate multiplier or a digital differential analyzer (DDA). For pulse rate multiplier, pulses for all the axes are generated by one common digital clock which keeps the relationship of the pulses for different axes constant even if the clock changes its rate. With DDA, the pulses are fed to a register which allows an 'overflow' at a reduced rate. This rate is determined by the relationship of the movement of the axes. They are sometimes called overflow counters.

## (2) The Feedback Unit

The feedback unit is a device which 'feeds back' the information about the actual position of the moving slides of the NC tool to the control unit. The commanded position is then compared with the actual position and if necessary, an adjustment is made for the correction of the difference.

The feedback signals are generated by a 'resolver' or an 'encoder' which is attached to the drive mechanism. When there is a difference, a voltage is fed to the drive motor such that the motor does not run too fast or the table does not overshoot. For this purpose, a voltage generating tachometer is attached to the drive mechanism which counteracts the voltage to the drive motor and keeps the required feed rate in line. When the commanded position is reached, voltage to the drive motor is cut down. For commands like, to select a tool from automatic tool changer

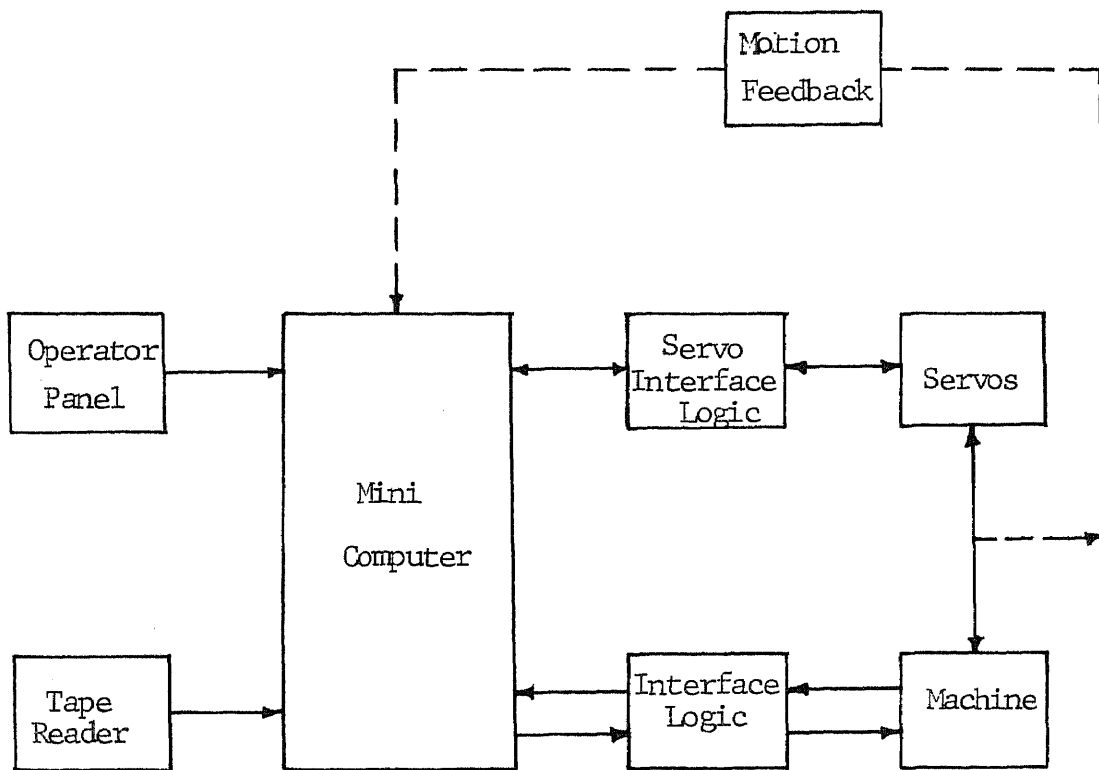


Figure 2-3 A CNC Design with Motion Control Provided by Software Within the Minicomputer

or turning the spindle motor on, they are directly fed to electrical controls where power is amplified and fed to the control mechanism.

Considering this in terms of pulses, the metered pulse train is fed to a 'modulator' where it is superimposed on a constant rate pulse train. If the motion has to be positive, these pulses are added but subtracted if the motion is negative. These pulses are then passed through a 'frequency divider' which reduces the number of the pulses and converts the output to a square wave. This square wave is fed to a 'demodulator' whose function is to compare them with the input from a rotary resolver connected to the driving mechanism. The output from the demodulator, a d-c voltage proportional to the phase difference between two inputs is then amplified and fed to the drive motor.

With CNC systems digital encoder is more popular, which feeds back the pulses generated by a rotary photo arrangement to the control system. Here, quite often the resolver's analog signal is converted to a digital feed back form as a CNC system incorporates a digital computer.

### (3) The Drive Unit

The function of this unit is to move a machine tool table, head or other member. Most of the time, a precision recirculating ball-nut lead screw is used as it offers accuracy and low friction which allows for rapid feed back response.

The drive could be hydraulic or electric power

depending on the size of the load to be moved. For less than 10 horsepower electric motor is preferred.

## CHAPTER III

### PART PROGRAMMING FOR BRIDGEPORT SERIES I CNC MACHINE

#### 3.1 FEATURES AND FUNCTIONS OF THE MACHINE

This chapter is an introduction to the coordinate system and function codes required for programming for Bridgeport Series I CNC milling, drilling and boring machine. All the general function codes are described with emphasis on the codes for this particular machine.

Bridgeport Series I CNC machine incorporates micro-computer and its Bridgeport Operating System Software (BOSS) which provides the following features and functions:

- 3 axis continuous path contouring.
- 2 axis circular interpolation in switchable planes.
- Word address variable block format to EIA RS274C.
- Absolute/Incremental data input.
- Plus or minus programming. Zero reference can be at any point.
- Feedrate programmed directly in IPM (inch per minute) at constant vector velocity.
- Manual feedrate override 1-120%, infinitely variable.
- Acceleration/Deceleration override.
- 8 canned Z axis cycles.
- Feed hold and restart without loss of position.
- Full jog control of any axis in any direction at 120 ipm (inch per minute).
- Jog increment control in steps of 1.0, .1, .01, .001



inch.

- Programmable mirror image capability.
- Block delete.
- Optional stop.
- Tool length offset (up to 24 values).
- Operating controls (like a 3 digit readout display of sequence number, feedrate, spindle speed, tool number and 5 digit readout for XYZ position, stored tool length offset value, reference tool diameter and status lamps to indicate control mode and operating conditions).

The machine in the Mechanical Engineering Department at NJIT has Bridgeport Operating System Software 4.0 (BOSS 4.0) which provides following extended features:

- Part program storage:- A part program equivalent of approximately 80 feet of paper tape can be stored by the system.
- Macro subroutine:- Up to 16 subroutines can be defined, each having variables that can be modified in the subroutine call statement.
- Repetitive subprogram capability:- Looping techniques can be used.
- Special canned cycles:- Three special canned cycles are provided for face milling, pocket milling and an internal bore.
- On line part program editing:- A part program editor in the micro-computer memory enables text input,

manipulation and modification via a local data input device. A 20ma serial line interface is provided to handle the device.

- Inch/Metric programming (optional):- The normal state is inch output from stored data in inch dimensions (G70). The modal preparatory function G71 denotes that the system is to output metric data from metric stored data.

### 3.2 COORDINATE SYSTEM

Cartesian coordinate system is the basis for NC part programming. Here two or three fixed lines perpendicular to each other described as axes are used to locate a point. The common point where all the axes intersect, is called origin. The Bridgeport Series I CNC machine has three axes, X, Y and Z. X dimensions to the right of the Y axis, Y dimension above the X axis and Z dimensions above the plane formed by the X and Y axes are considered positive. The positive motions are shown in Fig. 3-1.

There are two modes of describing the coordinates of a specific point. In absolute mode, which is more preferable because of less chances of error, all coordinates of a point are given in reference to the origin which would be located anywhere as desired. In incremental mode, the coordinates of a point are given as distance to be moved in direction of each axis from previous operating point.

For Bridgeport Series I CNC machine, the three axes movement is achieved by combination of table and saddle

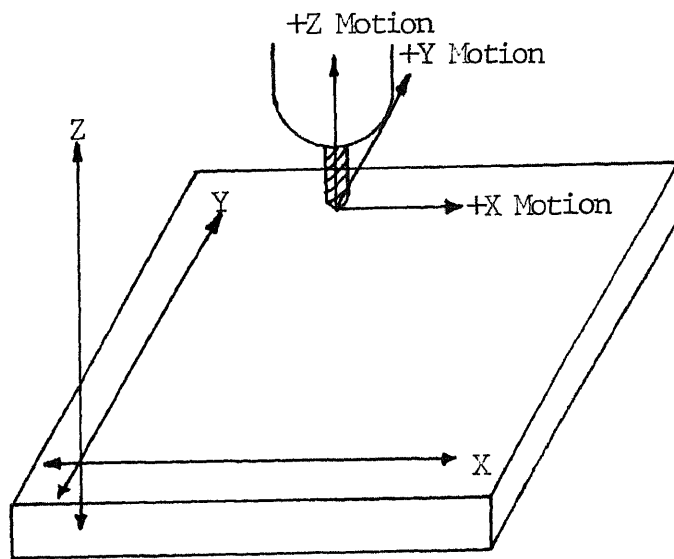


Figure 3-1 Part Coordinate System

movement (X and Y axes) and quill movement (tool holder and spindle assembly movement in up and down direction, Z axis). The machine axes travel limits under numerical control are:

X axis (Table) - 18"  
Y axis (Saddle) - 12"  
Z axis (Quill) - 5"

The dimension on the part can refer to a zero reference some distance from the part or on the part itself.

The tape codes shown in Table 3-1 are the functional characters recognized by the control of Bridgeport Series I CNC. Any character not listed is illegal. The first character of each word must be an address letter. The letter address may be followed by maximum of six digits. Plus sign is not required to be entered and is ignored by the control and so are the nonfunctional codes from the Table 3-1. The last character of every block is the end of block code.

### 3.3 PROGRAM ENTRIES

The following describes all the program entries being used with this machine.

% - Rewind stop

Stops rewinding of tape from that point.

\_ - Definition block

If this is the first character of the block, the information in that block will be executed during the search

SUBSET OF U.S.A. STANDARD CODE FOR INFORMATION  
INTERCHANGE, EIA STANDARD RS 358

EVEN PARITY								CHARACTER	FUNCTION
I S O									
8	7	6	5	4	3	2	1		
		6	5	0				0	NOTE: Any character not listed is illegal.
8		6	5	0			1	1	
8		6	5	0			2	2	
		6	5	0			2	3	
8		6	5	0	3			4	
		6	5	0	3		1	5	
		6	5	0	3	2		6	
8		6	5	0	3	2	1	7	
8		6	5	4	0			8	
		6	5	4	0		1	9	
	7			0			1	A	ANGLE (BOSS 5 or 6 only)
8	7				3		1	E	TAPE REWIND (TAPE READER ONLY)
8	7			0	3	2		F	FEED CODE
	7			0	3	2	1	G	PREPARATORY FUNCTION
8	7		4	0			1	I	X COORDINATE OF ARC CENTER
8	7		4	0		2		J	Y COORDINATE OF ARC CENTER
	7		4	0		2	1	K	IGNORED BY CONTROL
	7		4	0	3		1	M	MISCELLANEOUS FUNCTION
	7		4	0	3	2		N	SEQUENCE NUMBER
8	7		5				2	R	RADIUS (BOSS 5 or 6 only)
	7		5	0			2	S	IGNORED BY CONTROL
8	7		5	0	3			T	TOOL WORD
									IGNORED BY CONTROL
8	7		5	4	0			X	X COORDINATE
	7		5	4	0		1	Y	Y COORDINATE
	7		5	4	0		2	Z	Z COORDINATE
		6		4	0	3	2	.	DECIMAL POINT
8		6		4	0	3		,	IGNORED BY CONTROL
8		6		4	0	3	2	/	BLOCK DELETE & TLO/-DWELL
		6		4	0	2	1	+	IGNORED BY CONTROL
		6		4	0	3	1	-	MINUS
8		6		0				SPACE	IGNORED BY CONTROL
8	7	6	5	4	0	3	2	DELETE	IGNORED BY CONTROL
8				4	0	3	1	CARRIAGE RETURN	END OF BLOCK
8		6		0		3	1	%	REWIND STOP (TAPE READER ONLY)
				4	0		1	TAB	IGNORED BY CONTROL
				4	0		2	LINE FEED	IGNORED BY CONTROL
				0				BLANK TAPE	IGNORED BY CONTROL
				0				VIRGIN TAPE	IGNORED BY CONTROL
		6		0	3			\$	MACRO END
8		6		4	0		2	*	MACRO VARIABLE
8		6					2	#	MACRO START
8		6	5	4	3		1	=	MACRO OR LOOP CALL STATEMENT

NOTE: Channel 8 is stripped when outputting to a terminal in the EDIT mode.

Table 3-1

Figure 3-2 Input Tape and Program Codes

mode for a particular block as well as during program execution.

WARNING:- Motion words should not be programmed as the data is executed during search mode.

∟ - Block delete and value separation

If it is entered at the beginning of the block and the DELETE ON-OFF switch on control console is ON, that block is ignored by the tape reader while reading the tape. It is also used for value separation with G 04/n and Ta/b commands described later.

N - Sequence number letter address

It is used to identify blocks and is followed up to 5 digits ranging from 1 to 19999.

F - Feedrate command

Feedrate in inches per minute can be directly programmed using letter F and a three digit number ranging from 2 to 320 which indicates feedrate from .2 to 32.0 ipm. Higher feedrates programmed are set at 32.0 ipm. It is used with G01, G02 and G03 commands and remains effective until a G00 command or another feedrate is encountered. For circular cuts, feedrate is modified as follows:

For cutting around the outside of a circle, the feedrate should be increased by multiplying original feedrate by  $(PR+CR)/PR$  and should be decreased by multiplying by  $(PR-CR)/PR$  while cutting around the inside of a circle. Here, PR

denotes part surface radius and CR denotes the cutter radius.

S - Spindle speed command

It is ignored by the system but can be programmed with a four digit number to instruct the operator to set the spindle speed manually.

I - Tool select command

It is followed by a two digit number ranging from 1 to 24. It is also used to store tool length offset values. The format is Ta/b, where a is tool number (1 to 24) and b is Z axis offset value. (NOTE:- Tool length offset - TLO must be entered up to third decimal point only. e.g. T3/1.2345 will be stored as 12.345). Also tool diameter can be stored using the format Tn//d, where n is tool number and d is tool diameter.

M - Miscellaneous function commands

The following notations are used in describing the function codes:

B - Activated before axis motion occurs.

W - Activated at the same time axis motion occurs.

A - Activated after axis motion is complete.

M00 - (A) Program stop

Used to stop program cycle for manual inspection or adjustments like tool change. To continue, depress START/CONTINUE button.

M01 - (A) Optional program stop

To execute this command, the OP STOP ON-OFF switch on control console must be ON sometime prior to this command is encountered.

M02 - (B,A) End of program

It stops the program, retracts the quill to the 'Home' position before axis motion occurs, sets the sequence number to 0 and resets the part program counter to 0. To repeat the part program depress START/CONTINUE.

M06 - (B,A) Tool change

It denotes that a tool change is requested. The effects are, (1) Z axis retracts to its UP position and then X and Y motion occurs if programmed. (2) Z absolute register is set to the value indicated by the tool select number. Operator stops the spindle and changes the tool and restarts by depressing START/CONTINUE.

M25 - (B) Z axis to 'Home'

The quill retracts to UP position and then X and Y motion occurs if programmed.

CR (Carriage Return) - End of block command

Carriage Return is essential at end of each block because it initiates the control function within that block.

E - Tape rewind

It is put at the end of the program for tape rewinding after loading if the tape length is less than 80 feet.



## G - Preparatory functions

The letter address G followed by two digits indicates the mode of operation. More than one G function can be programmed in one block. A list of G functions and their description is given below.

### G00 - Rapid traverse, point-to-point positioning

This function causes the machine to operate in rapid traverse positioning mode. Movement in XY axes towards programmed point occurs first at 45 degrees followed by continued motion of the major axis. If X, Y and Z moves are programmed concurrently, the logic in control console splits the XY motion and Z move. If Z move requires the quill to go up, Z motion occurs first followed by X and/or Y move. If Z move requires the quill to go down, Z motion occurs after X and/or Y move. G00 cancels G01, G02 and G03 and will also set the system in G80 mode. The programmed feedrate remains in the feedrate register.

### G01 - Linear interpolation, Feed

This function causes the machine to operate in feed range. The axes travel in a straight line. Starting point is defined by the X, Y, Z coordinates of previous block and end point coordinates are given in the block. G01 cancels G00, G02 and G03.

### G02 - Circular interpolation, clockwise

This function defines that the arc is to be generated in clockwise direction. The arc radius is defined by the

start point of the arc (from previous block), the end point and arc center (I,J,K) coordinates. The arc motion could be in any two of the three axes. G02 cancels G00, G01 and G03.

G03 - Circular interpolation, counterclockwise

This function is same as G02 but defines the arc to be generated in counterclockwise direction. G03 cancels G00, G01 and G02.

G04 - Dwell

The format is G04/n where, n is an integer from 1 to 9999 giving a dwell time of .01 seconds to approximately 151 seconds. If dwell value is programmed earlier in the program some Z cycles can execute this time. The programmed value is counted as follows:

$$\text{Programmed value, } n = 6.6 \times \text{revolutions required} \\ \times \text{rpm}/60$$

For G17, G18 and G19 see Fig. 3-2.

G17 - XY plane selection (Plan view)

This function sets up XY plane for cutting arc in XY plane. Unless specified, this is the plane of operation with machine start up. It cancels G18 or G19.

G18 - ZX plane selection (Front elevation)

This function sets up ZX plane for cutting arc in ZX plane. It is programmed after G02 or G03. The input must be incremental input. It is cancelled by G00, G01, G17 and G19.

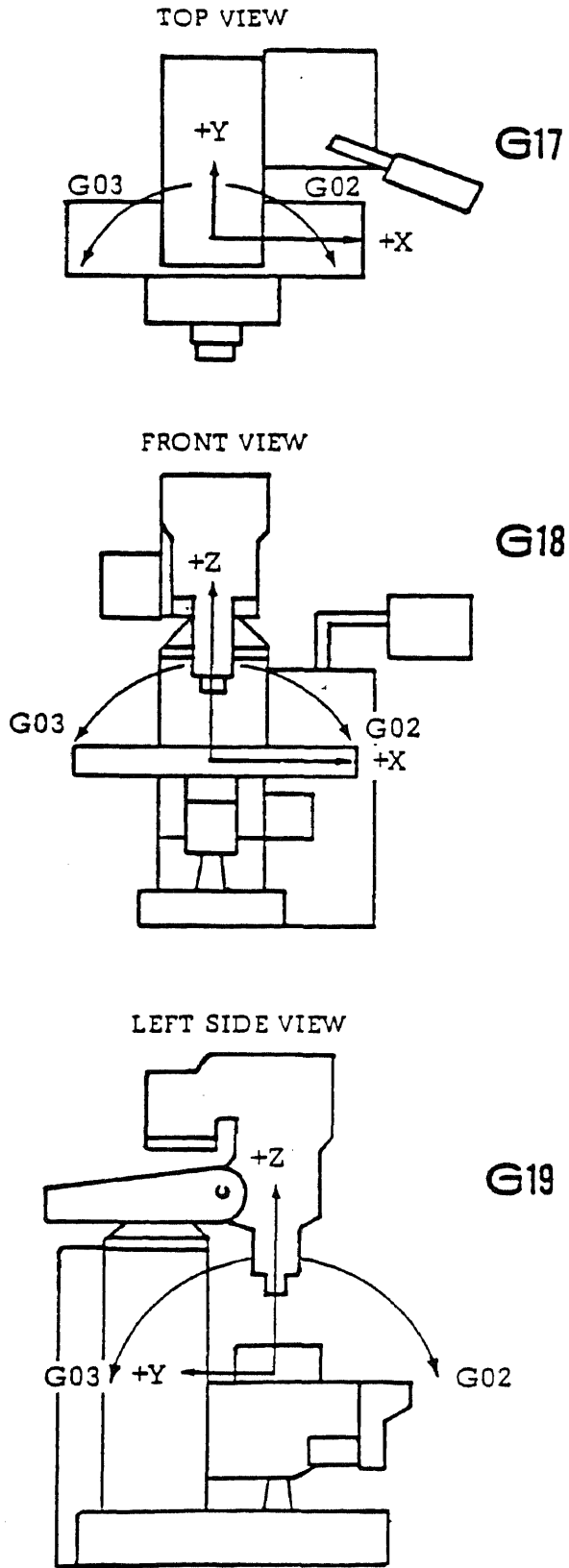


Figure 3-3 CW or CCW Motion in Switchable Planes

G19 - YZ plane selection (L.H. side elevation)

This function is same as G18, except it sets up YZ plane for cutting arc in YZ plane. It is cancelled by G00, G01, G17 or G18.

G30 - Cancel sign reversal

This function sets plus and minus directions for X and Y axes according to EIA standard RS-267. It is a normal POWER ON state for the control. G30 is cancelled by G31 or G32.

G31 and G32 are used for what is called mirror image programming. They must be programmed at axis of symmetry.

G31 - Reverses the direction signs for the X axis.

G32 - Reverses the direction signs for the Y axis.

G31 or G32 are cancelled by G30.

G70 - Inch dimension system input

Unless specified, this is the normal POWER ON state for control. It is cancelled by G71.

G71 - Metric dimension system input (Optional with BOSS 4.0)

This function instructs the control unit to interpret the following data as being in millimeter. It is cancelled by G70.

Both G70 and G71 have resolution of seven digits. (000.0000" or 0000.000 mm.)

G77, G78 and G79 are three special mill cycles.

G77 - Facing cycle

The format of the cycle is:

G77 Xn1Yn2Zn3 Fv        where,

n1 = incremental distance to be milled along X axis.

n2 = incremental distance to be milled along Y axis.

n3 = Z axis stepover value.

v = feedrate value.

G78 - Pocket milling cycle

The format of the cycle is:

G78 Xn1Xn2Xn5Yn3Yn4 Fv1 Fv2        where,

n1 = Distance from the center of the pocket to the wall  
along X axis minus cutter radius.

n2 = X axis stepover value.

n3 = Distance from the center of the pocket to the wall  
along Y axis minus cutter radius.

n4 = Y axis stepover value. If not programmed, it is  
set equal to n2.

n5 = Distance for the final boundary cut. If not  
programmed, it is set at the default value of  
0.02".

v1 = Feedrate for all roughing passes.

v2 = Feedrate for boundary cut. If not programmed, it  
is set equal to 1.5V1.

G79 - Internal hole mill cycle

It is used to mill a hole to size. The format of the  
cycle is:

G79 JnFv                    where,

n = Radius of the hole to be milled minus cutter radius.

v = Feedrate value.

G80 - G89 are canned Z axis cycles. They reduce programming by storing the following repetitive information:

- a. The depth of Z motion to be under cycle control. This must be incremental unsigned value.
- b. The desired feedrate for Z motion.
- c. The Z axis peck increments for G83-G87. The sum of peck increments need not to be equal to Z axis depth to be drilled.

All of these cycles work as follows:

- a. First rapid traverse to X-Y coordinate as designated which can be either incremental (G91) or absolute (G90).
- b. Then the Z axis cycle occurs.

After the fixed cycle is programmed, the system must have an X and/or Y coordinate word. In absolute coordinates, it could be a repeat of the existing position and in incremental mode, it could be axis zero. When a change in Z depth or a rapid traverse Z move is required, the fixed cycle has to be terminated, the change in depth or rapid traverse inserted and then the cycle is reinstated with fixed cycle code and Z increment value.

G80 - Fixed cycle cancel

This function cancels all fixed cycle codes.

G81 - Drilling cycle

This function provides for a feed in - rapid out sequence for drilling a series of holes with same diameter and depth.

G82 - Spotfacing cycle

This function provides for a feed in - delay - rapid out sequence for spotfacing, counterboring etc. . The length of delay is set by DWELL command G04/n, previously programmed.

G83 - Deep hole drilling cycle

This function is programmed with Z1 - the total Z depth, Z2 - incremental distance for first peck increment, Z3 - peck distance for remaining increments. If Z3 is not specified, Z2 becomes peck distance for all increments. This function provides a feed in, rapid out, rapid in to bottom of hole, feed in, rapid out etc. sequence until the specified Z depth is reached.

G84 - Tapping cycle

This function provides a feed in - feed out sequence for tapping with a tapping attachment for non-reversing spindles. The required spindle speed is calculated as follows:

$$\text{Spindle rpm} = \text{feed in ipm} \times \text{thread/inch}$$

A chart of feed and speed values for tapping various pitches is given in operating manual for the Series I CNC machine.

G85 - Boring cycle

This function provides a feed in - feed out sequence suitable for boring or reaming.

G86 - Boring cycle

This function provides a feed in, feed stop, wait for operator command, rapid out, operator restart sequence for boring. After the programmed Z depth is reached, the feed is commanded to stop. Then the operator stops the spindle and orients it. After pressing CONTINUE, the Z axis is rapid traversed up and the programmed spindle direction is reinitiated by the operator. This feature allows to position the tool a suitable distance from the finished bore before withdrawing the tool from the hole.

G87 - Chip breaking cycle

This function is programmed exactly like G83 - deep hole drilling cycle, and performs the same function. But, instead of rapid traverse out of the hole and back after a feed move as in G83, the quill rapid traverses up and down by 0.05" to break the chip.

G89 - Boring cycle

This function provides for a feed in - dwell - feed out sequence for boring. The dwell time is set by previously programmed G04/n command.



The following table gives a summary of all the fixed cycles.

Code	Movement in to Z depth	Dwell	Movement out
G81 - Drilling	Feed	No	Rapid
G82 - Spotfacing	Feed	Previous G04/n	Rapid
G83 - Deep hole drilling	Feed(peck)	No	Rapid
G84 - Tapping	Feed	No	Feed
G85 - Boring	Feed	No	Feed
G86 - Boring	Feed	Wait for operator	Rapid
G87 - Chip breaking	Feed	No	Rapid
G89 - Boring	Feed	Previous G04/n	Feed

For all fixed cycles, Z value is incremental unsigned distance. A G80 or G00 will cancel any of them.

#### G90 - Absolute input

This function causes the control to accept all X, Y, Z entries as absolute coordinates (with reference to an origin). The control automatically compares the present absolute coordinate position with the new programmed absolute coordinate position and uses the difference between them as a motion command for appropriate axis. This is normal power ON state for the control. It is cancelled by G91.

### G91 - Incremental distance input

This function causes the control to accept all X, Y, Z entries as incremental distances (with reference to present position). So, all motion entries are interpreted as commands to move the programmed distance from the present position and not an absolute position. It is cancelled by G90.

### G92 - Preset absolute registers

This function provides the ability to preset absolute position registers to desired dimension. It allows programming from a starting point on a part which is defined by absolute coordinates specified in G92 command. It also allows to translate an original coordinate system to a new part coordinate system which is very useful for making multiple parts on a fixture. As G92 command destroys the previous contents of the absolute registers, a new G92 command and a positioning move will be necessary to return to original coordinate system.

### G99 - Deceleration override

If the feedrate is higher than 2.8 ipm, the table accelerates from and decelerates to the programmed value during each block. Deceleration in a particular block can be reduced by programming a G99 code in that block. If the feedrate is over 8 ipm, some deceleration will still occur.

NOTE:- Do not use G99 when in rapid traverse (G00) or when in feed if the next block is rapid traverse (G00).

The examples on use of the above described function codes are given in the next chapter.

## CHAPTER IV

### PROGRAMMING EXAMPLES

#### 4.1 INTRODUCTORY COMMENTS

This chapter contains programming examples on most of the preparatory function codes (G commands). To a beginner, they should give an easy understanding about how to use these commands in a program. The explanation of these function codes should be sufficient to write simple programs. However, the operating manual for the Bridgeport Series I CNC machine should be referred to when writing programs for more complicated parts. This chapter is intended to give a good start at programming this machine.

First, the blueprint or the part sketch should be studied and a sequence of operations to be performed should be laid out. Then, the coordinates of all the operating points should be calculated. Following this, a manuscript should be prepared. Fig. 4-1 shows a convenient manuscript layout. A manuscript is a planning chart or list of instructions which describes the detailed and precise step-by-step operation of the machine. One should be able to punch the tape directly from the manuscript.

#### 4.2 PROGRAMMING EXAMPLES

With the existing software BOSS 4.0, the axis motion (X, Y, Z) can take one of the following forms:

- (a) Rapid traverse, point to point positioning mode.



(b) Linear interpolation, feed mode (cutting in a straight line).

(c) Circular interpolation, feed mode (arc generation).

Any of the above mode can be programmed in either

(a) Absolute coordinates or (b) Incremental coordinates.

(a) Absolute coordinates

When a G90 command is used, the distance to be moved for a particular axis is with respect to a zero reference point. The zero reference can be set so that the axis motion may occur in either plus or minus direction. The motion has to be within the machine travel limits. Full four quadrant plus and minus absolute programming in cartesian coordinate system is standard. The X, Y or Z coordinate can be described using up to a seven digit number with resolution of 000.0001". The non-significant zeros, both to the left and to the right of the decimal may be dropped. For example, 000.9000 can equally be written as .9 .

(b) Incremental coordinates

When a G91 command is used, the distance to be moved for a particular axis is from the present position and not an absolute coordinate. Each axis command consists of a letter address X, Y or Z, algebraic sign to indicate direction and up to a six digit number with a resolution of 00.0001".

## Programming Rapid Traverse (point to point positioning)

### Example 1

Move from (1, -1, 2.5) to point P1 (1, 1, -0.5) and from point P1 to point P2 (3, 3, 0.5) as shown in Fig. 4-2.

	X	Y	Z
Initial absolute coordinates	1.0	-1.0	2.5
Move to point P1	1.0	1.0	-0.5
Move to point P2	3.0	3.0	0.5

The above moves can be programmed as follows:

(a) In absolute coordinates,

```
N1G90G00Y1.0Z-0.5
```

```
N5X3.0Y3.0Z0.5
```

(b) In incremental coordinates,

```
N1G91G00Z-3.0
```

```
N5X2.0Y4.0Z1.0
```

In rapid traverse, when a move has to occur in X, Y and Z axes (as in moving from point P1 to point P2), it is split in to two rapid traverse moves. If the Z motion is minus (as in moving from initial point to point P1), the XY rapid occurs first followed by negative Z move. But, if the Z motion is positive (as in moving from point P1 to point P2), the Z move occurs first followed by the XY move.

## Programming Linear Interpolation, Feed Mode

For cutting in a straight line, G01 command is used. Drilling a hole or cutting a slot is linear interpolation.

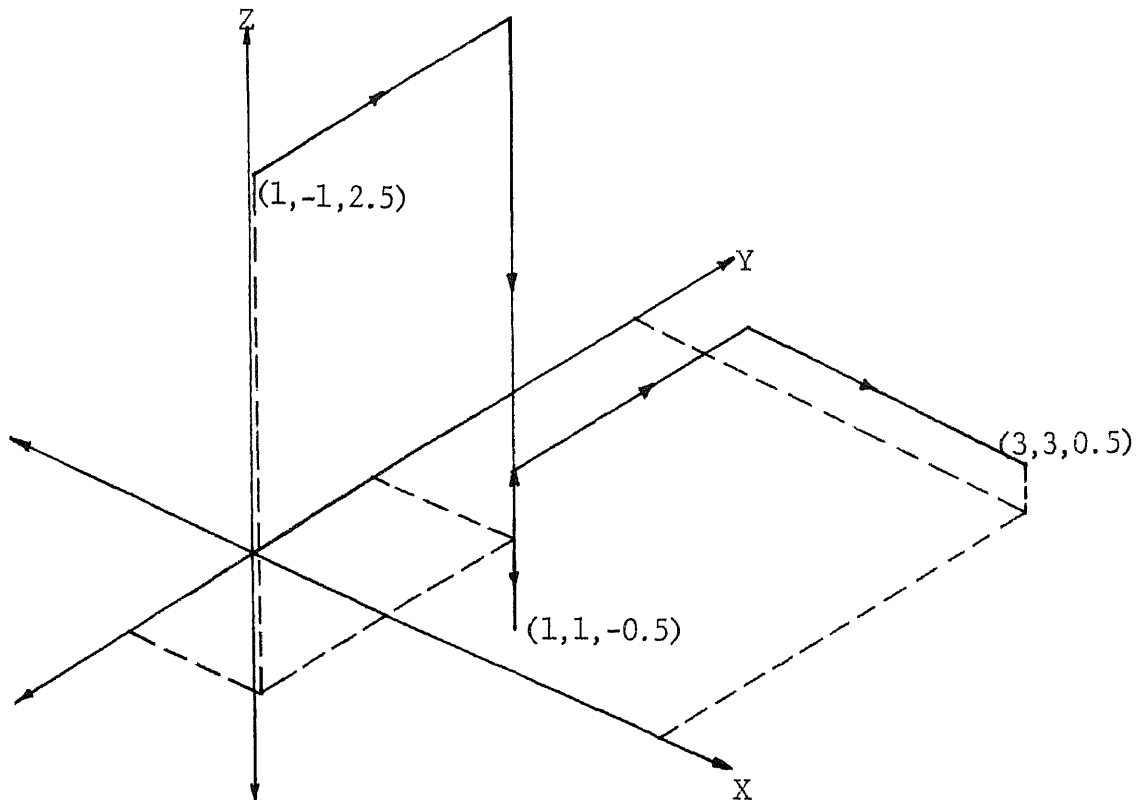


Figure 4-2 Example 1

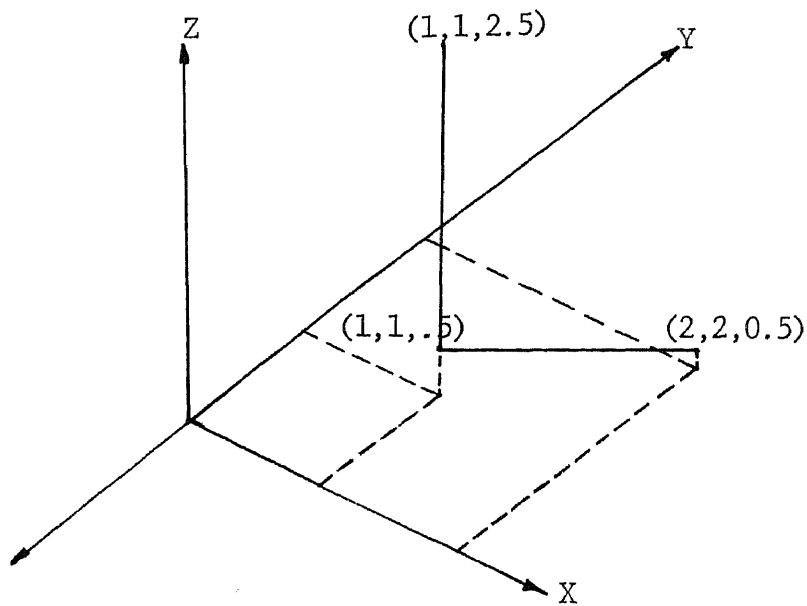


Figure 4-3 Example 2



### Example 2

Cut a slot of 2" depth starting at (1, 1, 2.5) and ending at (2, 2, 0.5) as shown in Fig. 4-3.

	X	Y	Z	Feedrate
Initial abs. coordinates	1.0	1.0	2.5	
Drill a hole of 2" depth	1.0	1.0	0.5	8 ipm.
Mill the slot	2.0	2.0	0.5	5 ipm.

(a) In absolute coordinates,

```
N1G90G01Z0.5F80
```

```
N5X2.0Y2.0F50
```

(b) In incremental coordinates,

```
N1G91G01Z-2.0F80
```

```
N5X1.0Y1.0F50
```

### Programming Circular Interpolation, Feed Mode

G02 and G03 commands are used to generate an arc. The following arc center offset entries are used with these commands. See Fig. 4-4.

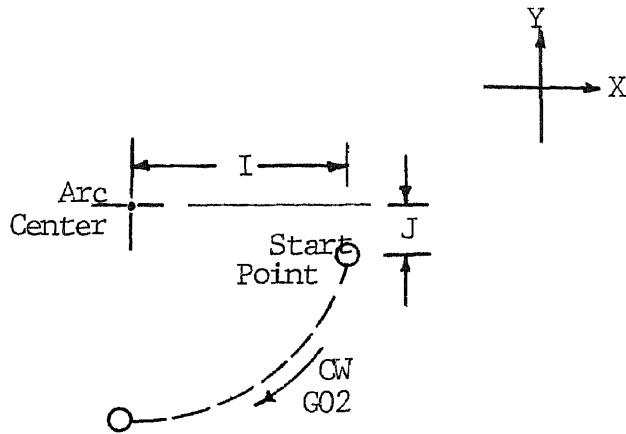
I - The distance parallel to the X axis from the starting point of the arc to the arc center.

J - The distance parallel to the Y axis from the starting point of the arc to the arc center.

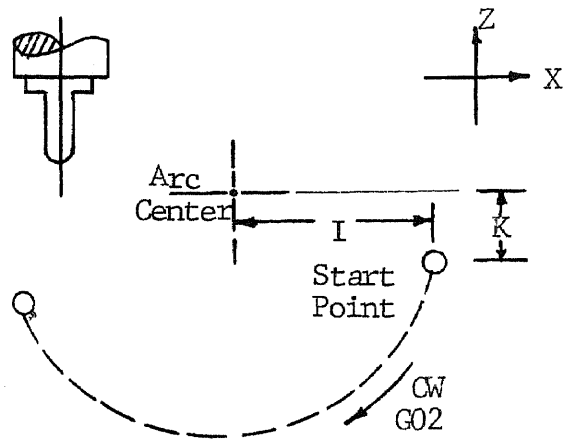
K - The distance parallel to the Z axis from the starting point of the arc to the arc center.

Signs must not be used with arc center offset entries and I0.0, J0.0 or K0.0 need not be programmed.

G17  
XY Plane



G18  
ZX Plane



G19  
YZ Plane

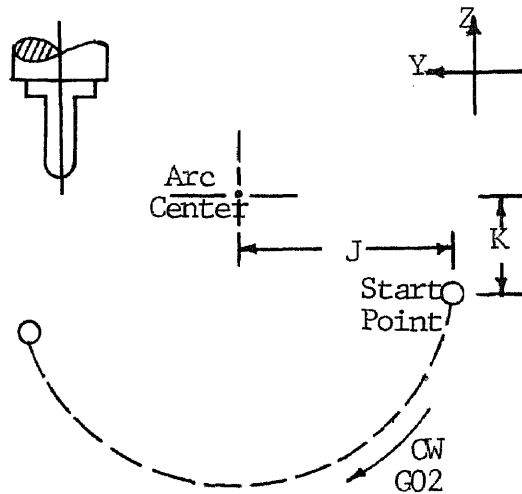


Figure 4-4 Arc Center Offsets  
45

### Example 3

Generate the arc as shown in Fig. 4-5.

	X	Y	Z	Feedrate
Initial abs. coordinates	1.0	2.5	-0.5	
Final abs. coordinates	2.4	1.0	-0.5	10 ipm

The program block will be,

In absolute mode,

```
G90G02X2.4Y1.0I0.5J1.9F100
```

In incremental mode,

```
G91G02X1.4Y-1.5I0.5J1.9F100
```

### Example 4

Write a program to cut the part as shown in Fig. 4-6. The Z absolute coordinate = -1.0 is constant through out.

	X	Y	I	J	Feedrate
Initial abs. coord.	0.5	2.0			
P1	5.5	2.0			10 ipm
P2	11.5	3.75			10 ipm
P3	14.0	7.5	1.5	3.75	10 ipm
P4	14.0	8.5			10 ipm

The program will be,

```
N5G90G00X0.5Y2.0Z0.05T1M6
```

```
N10G01Z-1.0F100
```

```
N15X5.5
```

```
N20X11.5Y3.75
```

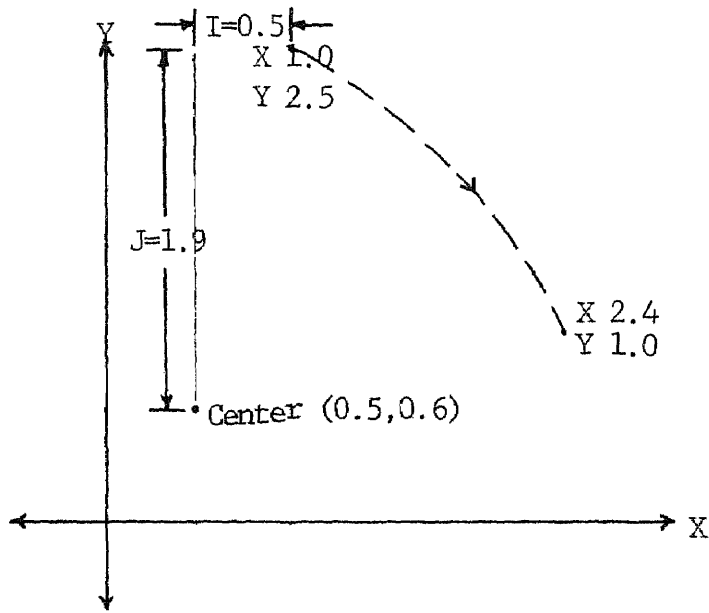


Figure 4-5 Example 3

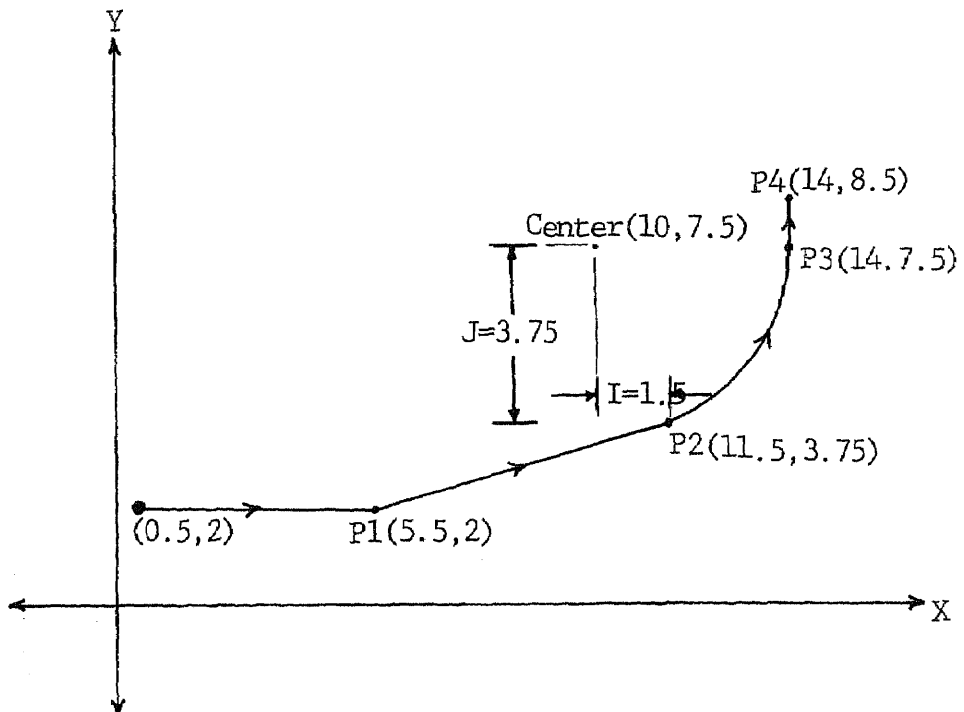


Figure 4-6 Example 4

N25G03X14.0Y7.5I1.5J3.75

N30G01Y8.5

N35G00X0Y0M2

Sequence no. 5 is rapid traverse to starting point. Sequence no. 10 will feed move the tool down to the required Z depth. Sequence no. 15 and 20 will generate the slots from initial point to P1 and from P1 to P2. Sequence no. 25 will generate the arc P2P3 in counter clockwise direction. Sequence no. 30 will generate the slot from P3 to P4. Sequence no. 35 will move the tool back to the origin.

#### Programming Mirror Image

A G31 command reverses the direction signs of X axis and G32 command reverses the direction signs of Y axis. G30 command cancels both, G31 and G32.

#### Example 5

As shown in Fig. 4-7, an L shape in the upper right hand side of the axis of symmetry and its three mirror images are to be produced. To program this, concept of macro is required. Refer to section 4.4 for details. The program will be:

```
#1                - Start of macro
N5G00G90X1.0Y1.0Z.05 - Tool at axis of symmetry
N10X2.0Y1.25      - Tool at P1
N15G01Z-0.5F100  - Tool 0.5" in the piece at P1
N20G91X-0.5      - Tool at P2
N25Y0.75         - Tool at P3
```

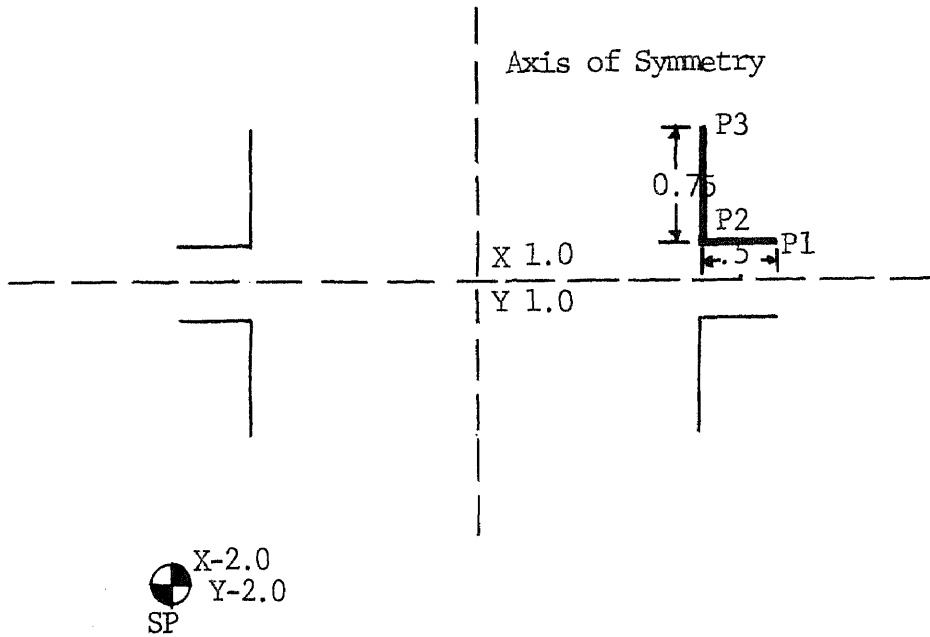


Figure 4-7 Example 5

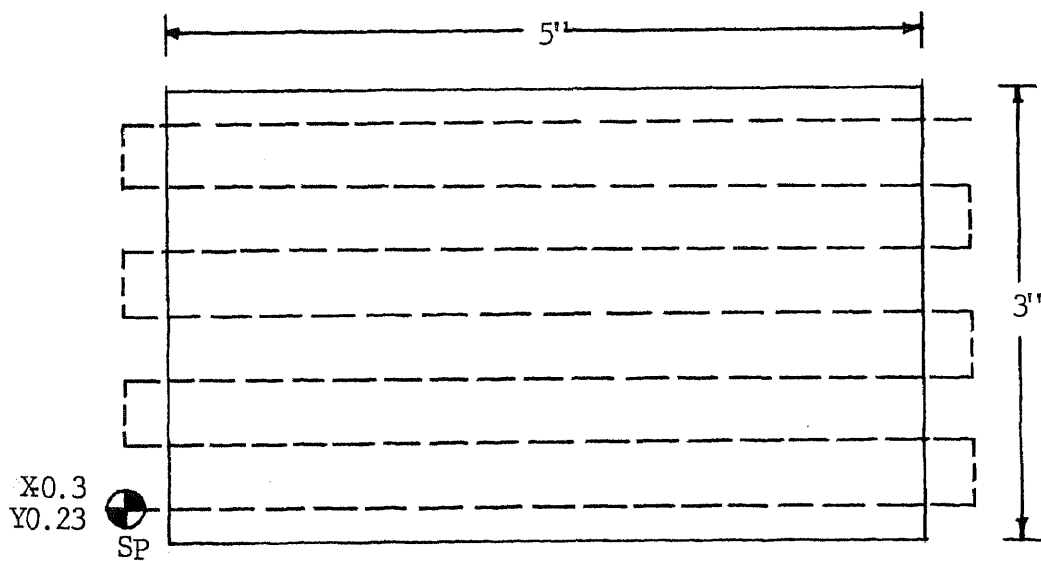


Figure 4-8 Example 6

```

N30G00G90X2.0Y2.0Z.05 - Return to axis of symmetry
$                - End of macro

N100G00G90X-2.0Y-2.0T1M6 - Generate the shape in upper
=#1                right corner
N200G31           - Generate the image in upper
=#1                left corner
N300G32           - Generate the image in lower
=#1                right corner
N400G31G32        - Generate the image in lower
=#1                left corner
N500G00G30G90X-2.0Y-2.0M2 - Cancel mirror image codes.
                        Go back to starting point.

```

#### Programming Special Mill Cycles

Commands G77, G78 and G79 give three special mill cycles. The examples are given below.

#### G77 - Facing cycle

The format is :

G77Xn1Yn2Yn3Fv      where,

n1 = incremental distance to be milled along the X axis

n2 = incremental distance to be milled along the Y axis

n3 = Y axis step over value

v = feedrate

### Example 6

A block of 5.0" x 3.0" has to be facemilled with .5" diameter end mill as shown in Fig. 4-8. The Z axis coordinate will remain constant which is set according to the desired depth. The program block will be:

```
G77X5.6Y2.58Y0.43F100
```

Note:- Absolute coordinates must not be used in this block but, the following block must be programmed in absolute coordinates.

The effect of the above command will be as follows:

From the starting point at the bottom left hand corner, the first move will be a +X axis move equal to 5.6" (input value). The next move will be a +Y axis move equal to 0.43" (step over value) followed by a -X axis move of 5.6". The next move again will be +Y axis move of 0.43" followed by a +X axis move of 5.6". This zigzag pattern will continue until the sum of step over values is equal to the input Y distance of 2.58". Then the last X axis move will be made.

### G78 - Pocket Milling cycle

The format is:

```
G78Xn1Xn2Xn5Yn3Yn4Fv1Fv2      where,
```

n1 = Distance from the center of the pocket to the wall along X axis minus cutter radius

n2 = X axis step over value

n3 = Distance from the center of the pocket to the wall along Y axis minus cutter radius



- n4 = Y axis step over value. If not programmed, it is set equal to n2
- n5 = Distance for the final boundary cut. If not programmed, it is set at the default value of 0.02"
- v1 = Feedrate for all roughing passes
- v2 = Feedrate for boundary cut. If not programmed, it is set equal to 1.5v1

#### Example 7

Mill a pocket of dimensions 5" x 3" x 0.5" using a 0.5" diameter cutter as shown in Fig. 4-9.

Here, X axis is major axis and Y axis is minor axis. The maximum step over value for X axis can be 0.25" (radius of the cutter). The step over value for Y axis for the most efficient result should be  $0.25" \times (n3/n1)$ .

$$n1 = (5.0/2) - 0.25 = 2.25"$$

$$n2 = 0.25"$$

$$n3 = (3.0/2) - 0.25 = 1.25"$$

$$n4 = 0.25" \times (1.25/2.25) = 0.139"$$

$$n5 = 0.02" \text{ (Default value)}$$

$$v1 = 100 \text{ (10 ipm.)}$$

$$v2 = 120 \text{ (12 ipm.)}$$

The program will be:

N5G00G90X-1.0Y-1.0T1M6

N10X2.5Y1.5Z0.05

N15G91G01Z-0.55F120

N20G78X2.25X0.25X0.02Y1.25Y0.139F100F120

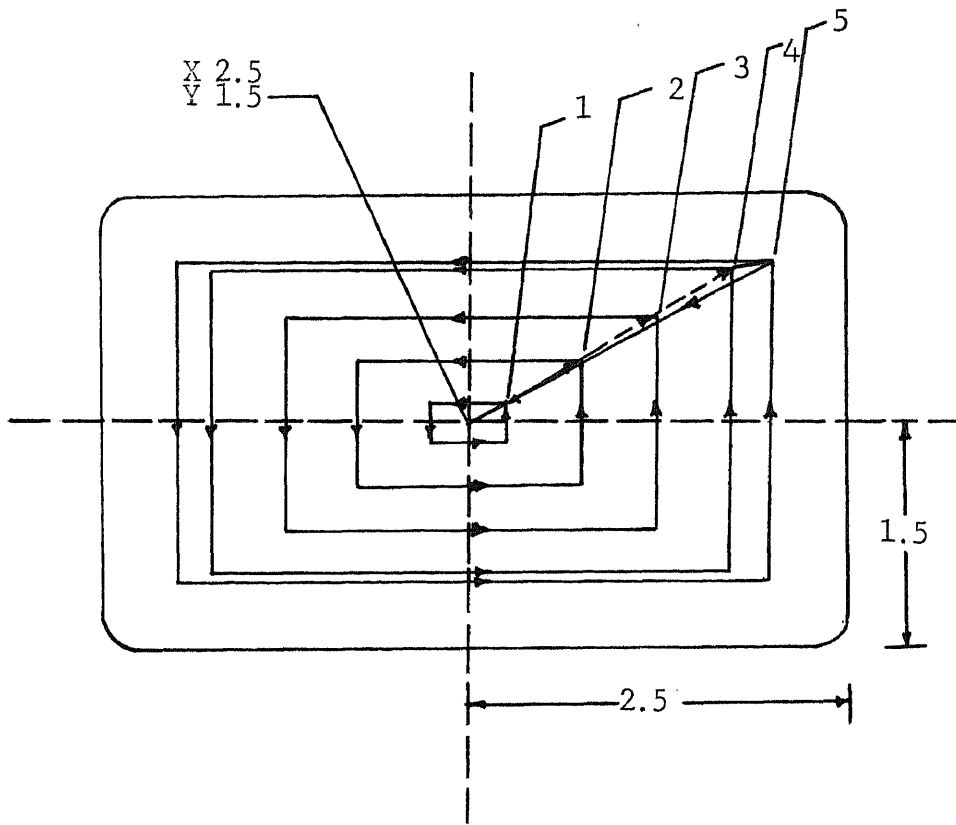


Figure 4-9 Example 7

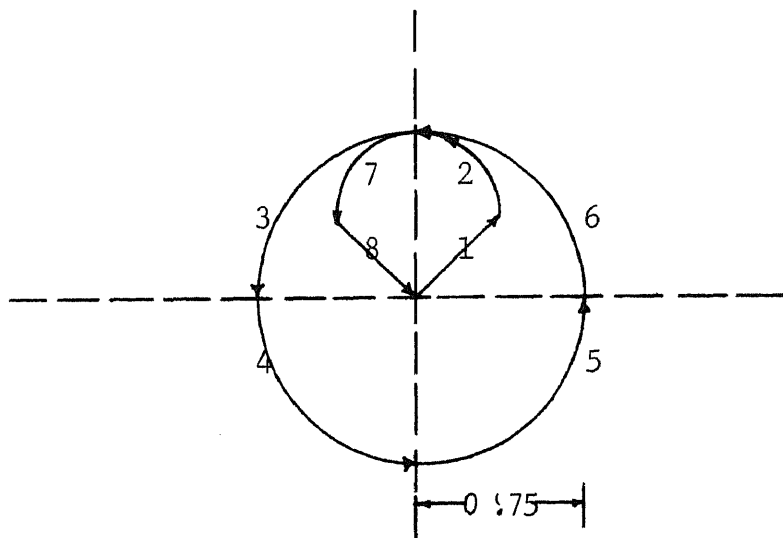


Figure 4-10 Example 8

N25G00G90X-1.0Y-1.0M2

Sequence no. 5 will move the tool to starting point (-1,-1). Sequence no. 10 will move the tool to the center of the pocket to be milled. Sequence no. 15 will cut the hole of the required depth (the tool will be in the workpiece to that depth). The effect of the sequence no. 20 will be as follows:

From the center of the pocket, the first move will be to point 1 (X and Y axis step over). Then the cutter center will move -X, -Y, +X, +Y at a value equal to twice the accumulated step over distance (X axis = 0.5", Y axis = 0.278"). The cutter then will move to point 2. (Another X and Y axis step over). This pattern will continue until the cutter surface is within 0.02" of the pocket wall. Once an axis reaches 0.02" from the pocket wall, it is no longer incremented during step over move. The last step over move will be 0.02" in both X and Y axes at a feedrate 12 ipm. All the moves before, will be at a feedrate 10 ipm. Finally, the cutter will feed at 12 ipm from point 6 to the center of the pocket to end the cycle.

G79 - Internal Hole Mill cycle

This cycle is useful to mill an existing hole to size i.e. required final diameter.

The format is:

G79 J (Value) F (Value) where,

J = Radius of hole to be milled minus cutter radius

### Example 8

An existing hole has to be milled to size of 1" radius with 0.5" diameter cutter as shown in Fig. 4-10. The command will be:

```
G79J0.75F100
```

The numbers on the figure represent the sequence in which the cutter will move. The entering and leaving tangent small circle radius will be equal to  $J/2 = 0.75/2 = 0.375$ ".

### Programming Canned Cycles

Commands G80-89 give different fixed cycles. The examples are given below.

#### G81 - Drilling cycle

This command will drill a series of holes of same diameter and depth in a feed in - rapid out sequence. See Fig. 4-11. The program will be:

```
N5G00G90Z0.05  
N10G81X2.0Y1.0Z1.05F100  
N15X3.0Y2.0  
N20G00X0Y0
```

This will drill two holes of 1.0" depth at desired locations as follows.

Sequence no. 5 will move the tool to 0.05" above zero reference plane. Sequence no. 10 will move the X and Y axes to the location of the first hole (X = 2.0", Y = 1.0") at rapid traverse and the Z axis will feed 1.05" down at 10 ipm giving 1" deep hole if the zero reference plane is at part

### DRILLING CYCLE G81

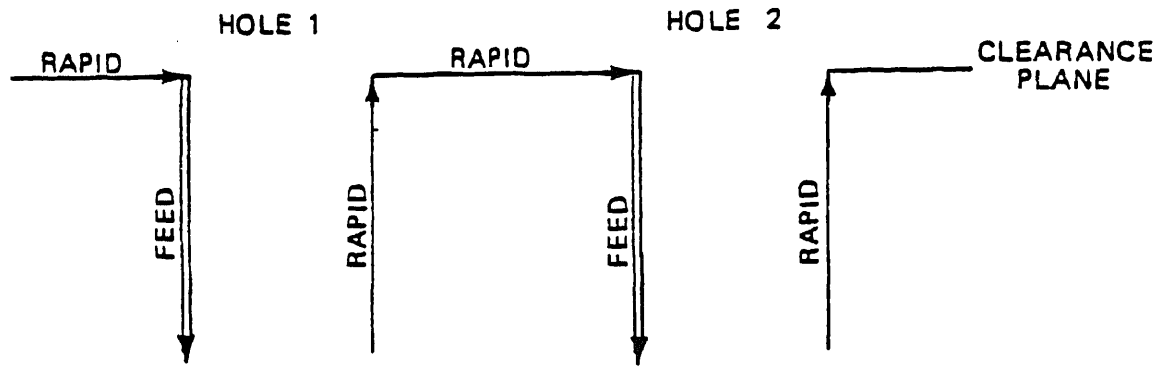


Figure 4-11 Drilling Cycle

### SPOTFACING CYCLE G82

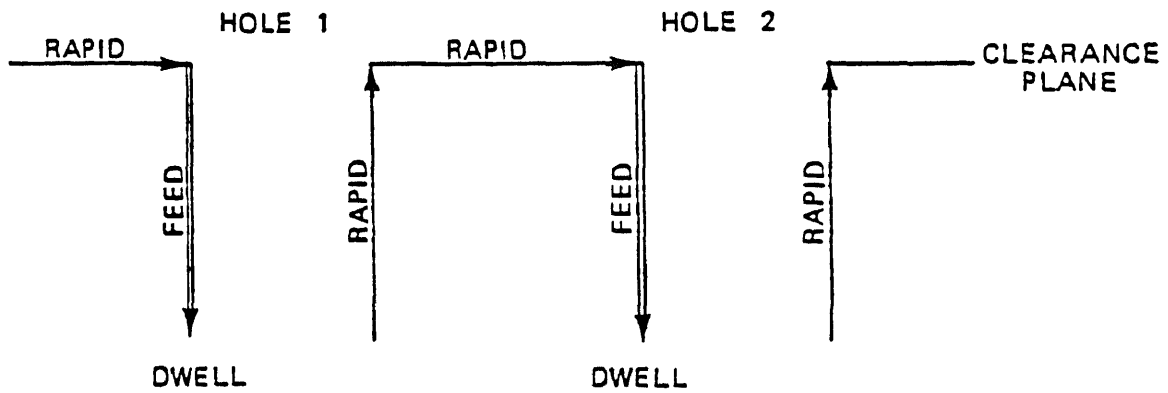


Figure 4-12 Spotfacing Cycle

upper surface. (1" depth of hole + 0.05" clearance = 1.05"). After the depth has been reached, the tool traverses out 1.05". Sequence no. 15 will have X and Y axes motion to the location of the second hole (X = 3.0", Y = 2.0"). Z axis will feed in 1.05" and rapid traverse out 1.05". Sequence no. 20 denotes the end of fixed cycle (G00) and rapid traverse to X = 0, Y = 0.

#### G82 - Spotfacing cycle

The sequence is same as G81. See Fig. 4-12. The only difference is that a delay is forced when the Z axis reaches the required depth. The length of the delay is set by Dwell command previously programmed as G04/n.

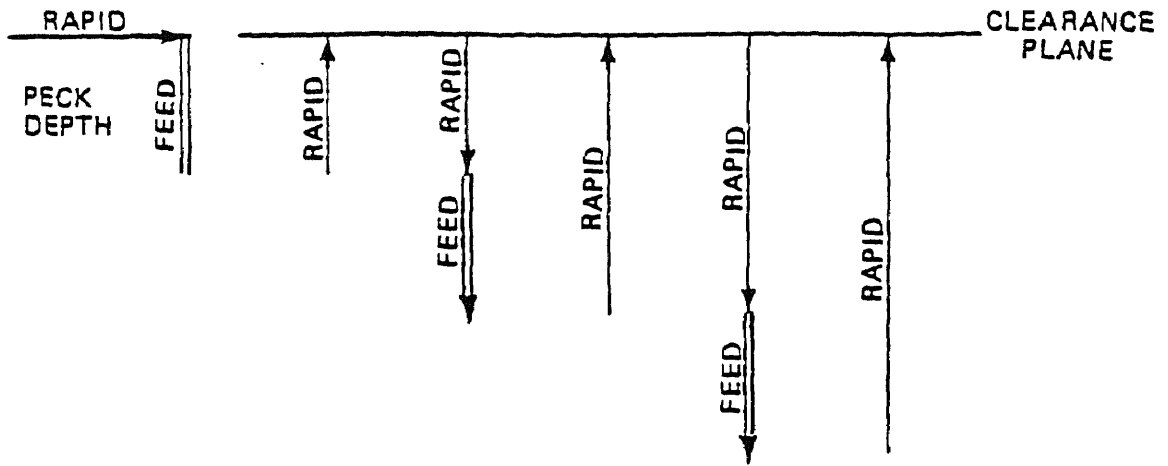
#### G83 - Deep Hole Drilling cycle

When a deep hole is to be drilled, this command is used which gives feed in, rapid out, rapid in to bottom of hole, feed in, rapid out sequence until the required depth is reached. See Fig. 4-13. The example is:

```
N5G83X2.0Y1.0Z2.5Z0.7Z0.3F100
```

This will drill a hole 2.5" deep at X = 2.0, Y = 1.0. The first Z axis feed in will be 0.7". Then it will rapid traverse out 0.7", rapid traverse in 0.7". The Z axis will then feed in 0.3", rapid traverse out 1.0" and rapid traverse in 1.0". Again the feed in of additional 0.3" and the sequence is continued until the required depth of 2.5" is reached. The final feed in will be the remaining distance to be drilled which may or may not be equal to 0.3".

DEEP HOLE CYCLE G83



DEEP HOLE CYCLE G87

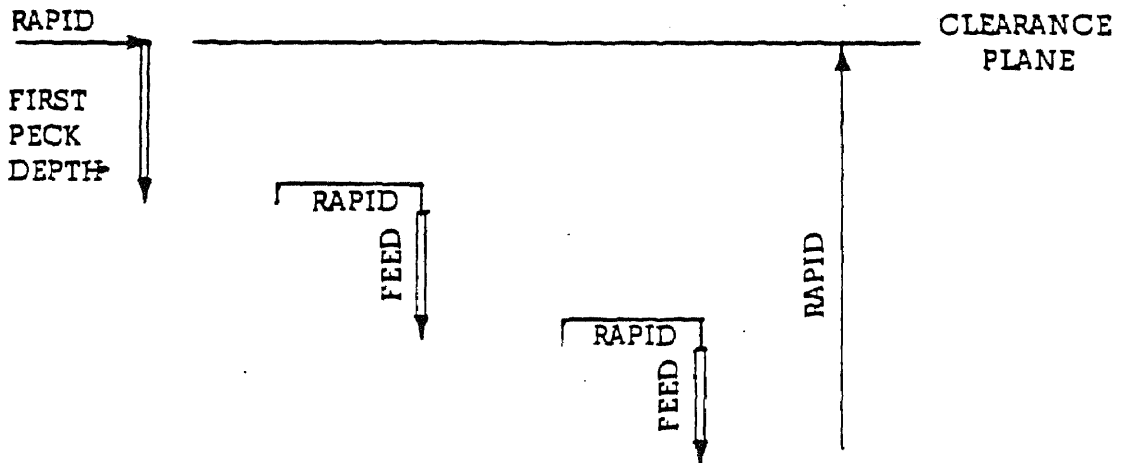


Figure 4-13 Deep Hole Drilling Cycles

TAPPING CYCLE G84

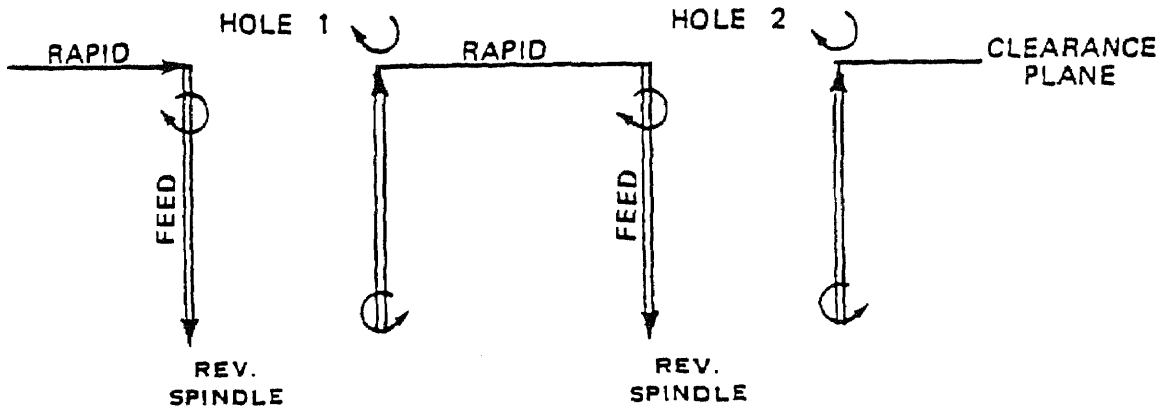


Figure 4-14 Tapping Cycle

#### G84 - Tapping cycle

According to the number of threads per inch and required feedrate, the spindle speed is calculated. To program 1/4-18 tap at 10.0 ipm, the required rpm will be  $10 \times 18 = 180$  rpm. The rpm is set manually using the spindle speed control panel. A hand held stroboscope is used to set exact rpm. See Fig. 4-14.

#### G85 - Boring cycle

This cycle has same format as G81 with a difference that instead of rapid traverse out of the hole as in G81, the Z axis feeds out of the hole at the programmed feedrate. See Fig. 4-15.

#### G86 - Boring cycle

This cycle has same format as G81. It provides feed in, feed stop, wait for operator command, rapid out, operator restrat cycle. See Fig. 4-16. The example is:

```
N5G86X1.0Y2.0Z0.6F25
```

After rapid traverse to  $X = 1.0$ ,  $Y = 2.0$ , the Z axis will feed in 0.6" at 2.5 ipm. After the required depth is reached, the feed will stop. The operator stops the spindle and orients it if necessary. He then presses CONTINUE. The Z axis rapid traverses up and the programmed spindle direction is reinitiated by the operator.

#### G87 - Chip Breaking cycle

This cycle is programmed in exactly the same manner as G83 with only difference that instead of rapid traverse out



BORING CYCLE G85

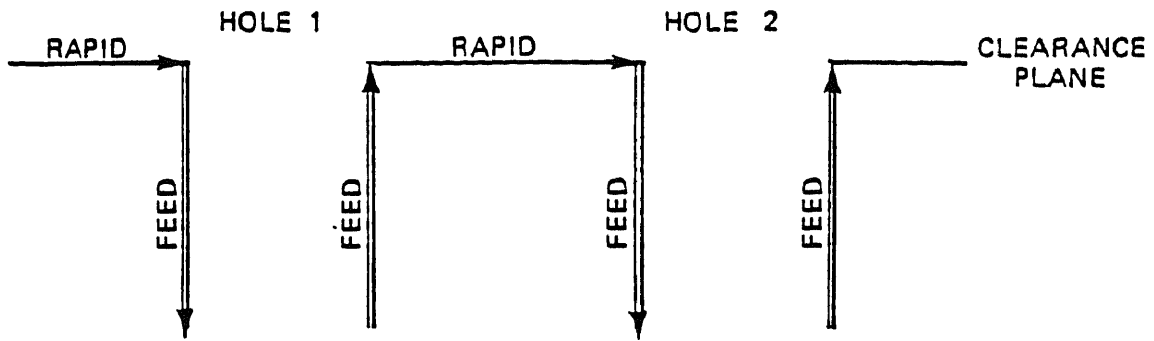


Figure 4-15 Boring Cycle

BORING CYCLE G86

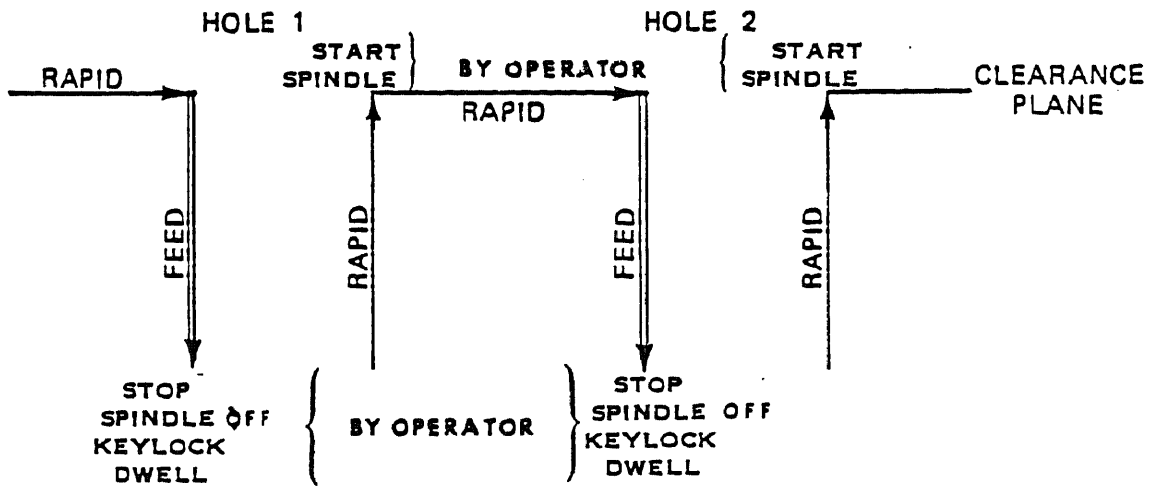


Figure 4-16 Boring Cycle

BORING CYCLE G89

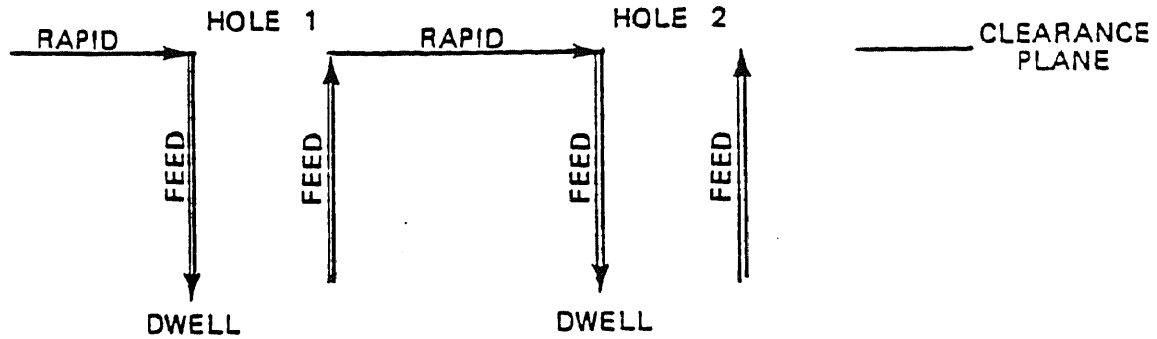


Figure 4-17 Boring Cycle

of the hole and back in the hole after a feed move, the tool rapid traverses up and down by 0.05" to break the chip. See Fig. 4-13.

#### G89 - Boring cycle

This cycle has same format as G81 and has same feed in - feed out sequence like G85 except that a dwell occurs between the feed in and feed out moves. The dwell length is set by G04/n previously programmed. See Fig. 4-17.

#### Programming Multihole Row Fixed cycle

When programmed in this format, the result is a row of holes that may be programmed using looping technique described in section 4.3. Any G81-89 fixed cycle can be used. The format is:

(G81-89)Xn1Xn2(Zn3Fv - according to the cycle  
being used)  
or (G81-89)Yn1Yn2(Zn3Fv - according to the cycle  
being used)

where,

n1 = Total distance to be traveled along the X or  
Y axis (maximum 16.0")

n2 = Incremental distance between holes (must be  
in multiple of 0.0005")

n3 = Z axis incremental depth

v = Feedrate

Whenever initiated, the first hole will be drilled at the existing position. Successive holes will be drilled at

the incremental distance n2 until the last hole is drilled after the last move. n1 may be programmed in absolute or incremental mode.

#### Example 9

Five holes are to be drilled at a distance of 0.2" between them starting at an absolute coordinate of X = 2.0, Y = 1.0 as shown in Fig. 4-18. The depth of a hole is 1.5". First Z peck increment of 0.5" and the successive increments of 0.2" are to be used. The cycle used is G87 - Chip breaking cycle. The program will be:

In absolute mode,

```
N5G00G90X2.0Y1.0Z0.05
```

```
N10G87X1.8X0.2Z1.55Z0.5Z0.2F100
```

or In incremental mode,

```
N5G00G90X2.0Y1.0Z0.05
```

```
N10G91G87X0.8X0.2Z1.55Z0.5Z0.2F100
```

In absolute mode, X1.8 denotes the absolute X coordinate of the last hole. While in incremental mode, X0.8 denotes the distance between the first and the last hole. Both give the same result.

Also, if the above example were programmed as,

```
N5G00G90X2.0Y1.0Z0.05
```

```
N10G87X1.8X0.22Z1.55Z0.5Z0.2F100
```

the spacing between first four holes will be 0.22" as specified but the distance between the fourth and the last will be 0.14" in order to make the total span of 0.8" as

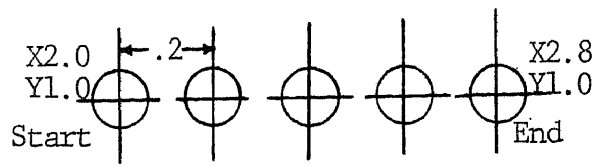


Figure 4-18 Example 9

programmed.

#### 4.3 LOOPING

Looping is a technique which instructs the computer to execute a set of blocks in the part program a number of times specified by the programmer. It relieves the programmer of writing a sequence of statements which has to be repeated. The form is:

$=Na/b$         where,

a = Loop end block sequence number

b = Number of times the loop has to be repeated

The range of the loop includes blocks following the looping statement up to and including the block with the loop end sequence number. All the blocks in the range are executed following a loop statement. After executing the sequence for b number of times, the system falls through to the next program block following the loop end block. Rules for the looping technique are as follows:

- (1) The loop end block sequence number must be in later part of the program.
- (2) All blocks following a loop statement must have a sequence number.
- (3) Nesting of the loops is possible up to 4 levels. The range of an inner loop must be within the range of next outer loop but they may have the same loop end block sequence number.
- (4) The loop end block must not be the last block of a program.

- (5) The maximum number of repeats is 19999.
- (6) Metric data should not be programmed incrementally as cumulative error could be high.

As an example of looping technique, the following statements will cause 5 holes to be drilled, each 0.4" deep and at a distance of 0.2" between them. The first hole will be drilled at 0.2" away from the Y coordinate the system was at before running the loop.

```
=N10/2  
N10G91G81Y0.2Z0.4F100
```

#### 4.4 MACRO SUBROUTINES

This feature of Bridgeport CNC provides the programmer with the ability to program an addressed parameter as a variable parameter and execute a subprogram consisting of a group of blocks with variable parameters. Here, the variable parameters are assigned their values by the calling statement. Thus, a subprogram with variable parameters can be executed with different values of them. The form of a macro is like:

```
#1  
N5G00G90X*Y*  
N10G01Z-0.5F*  
$
```

Here, #n (range from 1 to 16) denotes the beginning and \$ denotes the end of the macro subprogram. The variable parameters are followed by \*. The macro call statement is of the form:

=#nX\*2.0Y\*1.0F\*100

Where, n is the macro subprogram to be executed and X\*2.0, Y\*1.0, F\*100 are the parameter values to be assigned for the unspecified macro variables. Rules for macro subprogramming are as follows:

- (1) A macro subprogram must not be defined within another macro subprogram.
- (2) The number of unspecified parameters within the macro and the number of specified parameters in the call statement must be the same.
- (3) A macro may include a loop and macro call may be included within a loop. If the macro includes a loop, the \$ character must not be on the loop end block number.
- (4) A macro may call another macro. They can be nested up to 4 levels.
- (5) The macro call can contain maximum of 47 characters. Also, the total number of active macro variables (\*) can not be more than 12. This is the limit for one macro or a group of nested macros.
- (6) The largest value a variable parameter can have is  $\pm 16.0000$ .
- (7) If two macros with the same tag are defined within a program, the call statement will execute the last macro defined.
- (8) If a macro variable has the same value in the next call as was previously defined, it need not be

reentered. But, this call must be the next call.

Example:        =#1X\*1.0F\*100  
                  =#1X\*F\*

will call the same values of X = 1.0 and F = 100 but  
in the following example,

                  =#1X\*1.0F\*100  
  
                  =#2X\*2.0F\*120  
  
                  =#1X\*F\*

the last call statement will call macro #1 but with  
the values X = 2.0 and F = 120.

Use of looping and macro subprograms is a powerful  
tool for an imaginative programmer.



## CHAPTER V

### OPERATION OF BRIDGEPORT SERIES I CNC MACHINE

#### 5.1 INTRODUCTORY COMMENTS

This chapter gives familiarity with operator's controls and indicators. The main components of the machine control are as follows:

(1) Logic control cabinet, (2) Operator's main control panel, (3) Power control cabinet, (4) Spindle control panel, (5) Special operations panel.

#### 5.2 OPERATOR'S MAIN CONTROL PANEL

Of all the components above, operator's main control panel is the most important as far as operating the machine is concerned. Familiarity with all controls available is advised before attempting to operate the machine. The function of each switch available on the panel is described below. See Fig. 5-1 for the location of each of the switches.

##### MODE

It is a 5 position selector switch with  
MDI STORE/MDI/AUTO/BLOCK/SET UP.

##### SET UP

It allows to use AXIS MOTION (X, Y, Z) set up controls and ABS/TLO register controls. This is the mode type while setting up the origin. Also, while loading paper tape and



Figure 5-1 Operator's Main Control Panel

using EDIT feature, switch has to be in SET UP.

#### BLOCK

While in this mode, everytime the START/CONTINUE pushbutton is depressed, one instruction block (one line) from the program is executed. It is useful while checking the part program.

#### AUTO

While in this mode, when the START/CONTINUE button is depressed, the part program is automatically and continuously executed.

#### MDI

It is used to input and execute a block of data manually using the MDI keyboard.

#### MDI STORE

It allows the operator to store the block of data inputted using the MDI keyboard and a part program can be built up from successive blocks.

#### FUNCTION

There are two pushbutton switches, each with two positions.

#### TOP SWITCH

#### RESTART

When depressed in this mode, the top switch resets various system registers and flags and puts the system in a

mode where part program can be executed from the beginning.

#### START/CONTINUE

When depressed, it begins the program execution if at the beginning of the program or after an interruption, such as a system HOLD or M0, M1, M6. Also, while using BLOCK mode, to execute next block depress the button in this position.

#### BOTTOM SWITCH

#### EDIT

While depressed in this position, it transfers the control to the system if the MODE switch is in SET UP position. To use this feature, a local data input device is required.

#### RDI

It transfers the control to a remote data link if the MODE switch is in SET UP position. To use this feature, the system must have this option.

#### AXIS MOTION

There are two selector switches and one pushbutton under this title. By combining proper selection modes, a motion in the selected axis, in a particular direction for a particular distance can be obtained in rapid traverse. The MODE switch has to be in SET UP position.

X/Y/Z

Controls the axis selected for motion.

MOVE +/-

This two position pushbutton controls the direction of motion in the selected axis.

JOG/1/.1/.01/STEP

This selector switch controls the incremental distance to be moved in rapid traverse. When it is in JOG position, a continuous rapid traverse is obtained in the selected axis as long as the MOVE +/- pushbutton is depressed. While, depressing the MOVE +/- button once, 1/.1/.01/STEP positions would give motion of 1", .1", .01", .001" respectively in the selected axis.

ABSolute/Tool Length Offset (ABS/TLO)

This section has a five position selector switch which works in conjunction with a selector pushbutton below it. They are effective in SET UP mode only. The following chart describes the result of different combination of the above two, when the pushbutton is depressed.

## XY/T1

## Z/TNO

ZERO	Zeros the XY ABS registers. After setting required X0, Y0, store that position using this.	Retracts the quill all the way up("Home" position) and zeroes Z ABS register.
GO TO	Moves the table to X0, Y0 position.	If the quill is "home", moves it down the distance stored as TLO. At this downpoint, ZABS=0
TNO		Sets the tool number=1. It advances the tool number by 1 everytime it is depressed.
STORE		Stores the current contents of ZABS register as a tool length offset (TLO).
GET		Fetches the value stored as tool length offset and sets ZABS register to that value.
HOLD		

When depressed, this pushbutton provides with an interruption of program execution without loss of position and the unused portion of the block of the data. To continue, depress START/CONTINUE.

## LIMIT OVERRIDE

This is a lighted indicator pushbutton. When the power is switched ON the first time, this light is illuminated.

Depress this pushbutton to get power to axis drive and to operate the machine. When a motion in a particular axis exceeds the limit for that axis, this light is illuminated indicating power interruption. It clears active and buffer storage but does not clear part program and X, Y, Z absolute position and tool length offset values. For details on how to resume the operation, see section 5.6.2.

#### FEEDRATE OVERRIDE

This is a variable potentiometer which allows adjustment from 1 to 120% of the programmed feedrate.

#### STATUS

This section contains a bank of LED's (Light Emitting Diodes) and two display windows which indicate the internal set status of the control. The following describes the meaning of each condition when that lamp is ON:

- RUN: System ready for input or part program ready to be executed.
- GO TO: Information in active storage (arithmetic) registers.
- HOLD: A condition when HOLD button is depressed or spindle is interlocked. Output of data to the drive system is temporarily stopped.
- WAIT: Special case of HOLD. Occurs when an M6 is encountered or computer is waiting for the quill up switch to be closed extrnally.
- ERR: An error found in start-up diagnostic test.

TOOL: An M6 (Tool change) instruction is encountered.  
CYCLE: A G81-89 cycle in process.  
ABS: System is in ABS mode (G90).  
METRIC: System is in METRIC mode (G71).  
EDIT: System is in EDIT mode.  
RDI: System is in RDI mode.

#### Display Windows

The three digit window displays sequence number (last 3 digits), feedrate, spindle speed (first 3 digits) or tool number.

The five digit window displays X, Y, Z absolute (+ or -) coordinates, tool length offset and tool diameter (for operator's information).

MDI keyboard has different keys to display desired functions. Until MDI is used, the display shows sequence number and X absolute coordinate. When the MDI keyboard is used to enter the data by depressing a letter key, the 5 digit display becomes blank and displays the numerals entered starting from the right and shifted to the left. When another letter key is depressed, the previous word is stored and the display becomes blank again.

#### MDI keyboard

This is a 20 button keyboard which allows the operator to enter a block of data and store it. Depressing the black key indicates end of block (EOB), execute data. The CE key will erase an incorrectly inputted numeric character.



(a). In MDI mode, all keys as labeled in their center are effective. Data should be entered in the order: N, G, X, Y, Z, F, M, EOB(black key). The functions not necessary in the block should be eliminated from the above order.

(b). In MDI store mode, the block entered by (EOB) is automatically stored and executed. To delete the stored block, depress the CE key before entering new block.

(c). In SET UP mode, the three keys in green field allow to enter tool number T, tool length offset TLO and tool diameter DIA (for operator's reference). To store them, depress the black key.

(d). In AUTO mode, depressing the N key followed by a number up to 5 digits causes an automatic rewind of stored data and then a forward search for the punched in sequence number. The search will stop at the end of the block before the punched in sequence number.

For details on how to use MDI keyboard, refer to Section 4.1 of Operating Manual for Bridgeport Series I CNC machine.

### 5.3 SPINDLE CONTROL PANEL

The following describes the functions of the selector switch and the pushbutton on the panel. Fig. 5-2 shows their locations on the panel.

#### LOW RANGE/OFF/HIGH RANGE

This selector switch is used to turn the spindle ON and determine the direction of spindle rotation. If this



Figure 5-2 Spindle Control Panel

switch is on LOW position and the gear selector lever in LOW or the switch on HIGH and the lever in HIGH, the spindle rotates in clockwise direction. Any other combination of the two gives anticlockwise direction of spindle rotation. If the program contains a block of data with feedrate motion but if the spindle is not ON, the operation is interrupted. To resume, switch the spindle ON and depress START/CONTINUE button.

#### EMERGENCY

This mushroom head pushbutton should not be used unless it is a real emergency because it results in a non-recoverable condition i.e. the control loses XY slide position if the XY slides were moving. But the part program and TLO are not lost. It deenergizes spindle motor and sets the system in RESTART mode. The limit override lamp will be illuminated. To resume, pull the button out.

#### SPINDLE STOP Lamp

This lamp is illuminated when any of the command, M0, M1, M2, M6 is executed. It stops the spindle even if the drum switch is showing spindle ON. To resume, switch the spindle OFF and switch it ON again.

#### 5.4 SPECIAL OPERATIONS PANEL

This is mounted inside the logic control cabinet. The following is a description of the functions of all the switches on this panel. Fig. 5-3 shows the panel with the tape reader mounted on it.

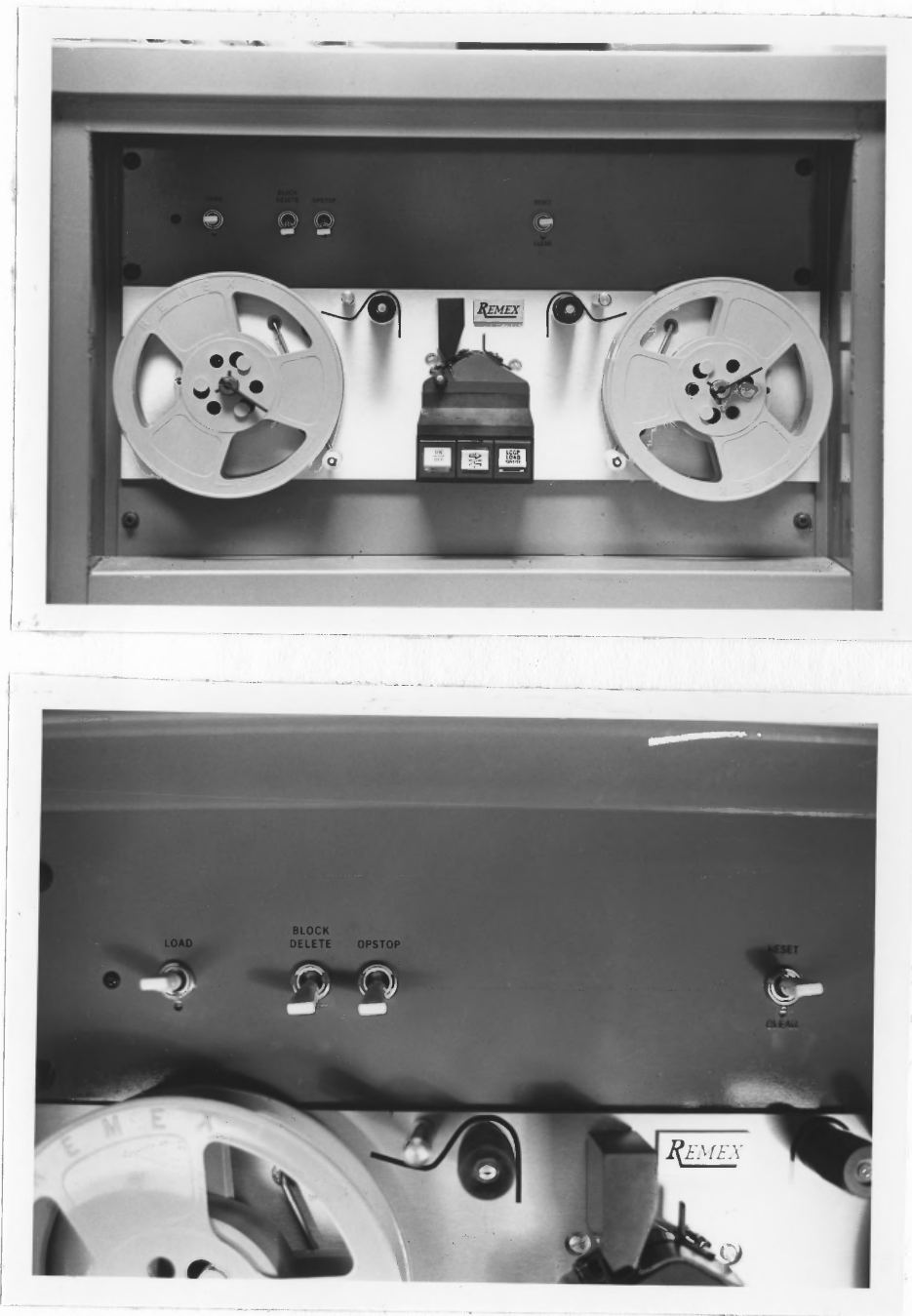


Figure 5-3 Special Operations Panel

## RESET/CLEAR

RESET destroys all program text stored, zeroes the XYZ absolute registers and TLD registers.

CLEAR - It clears contents of buffer and active system registers but does not clear XYZ absolute registers, TLD and the part program storage.

## LOAD

If the system is in SET UP mode, operating this switch will load the paper tape. If a new tape is to be entered, operate CLEAR. If a program is to be added to the existing program, operate only the LOAD switch.

## OP STOP ON/OFF

When it is on ON, the M01 code in the program is recognised otherwise it is ignored.

## BLOCK DELETE ON/OFF

When this switch is on ON, it deletes all data from the slash (/) to the end of the block character.

## Spindle Speed Changer and Brake

When the spindle is rotating, by selecting INCREASE or DECREASE and depressing that button the spindle speed can be increased or decreased to desired value. See Fig. 5-4. Because this is an air-turbine operating system from a pneumatic solenoid valve, the air supply has to be between 60-80 psi. .

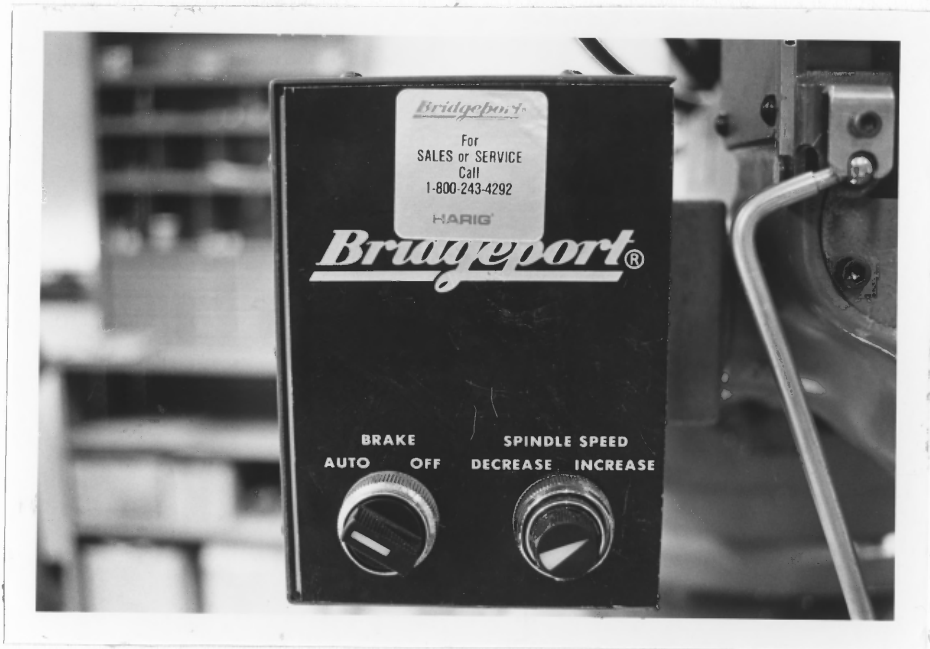


Figure 5-4 Spindle Speed Changer and Brake

## Spindle Brake

In AUTO position, the brake is applied when the spindle is switched OFF and is released when it is ON. In OFF position, it releases the brake for checking or for set up operations.

## 5.5 OPERATION IN SET UP

After becoming familiar with the different controls, one can use the following procedure to operate the machine.

### 5.5.1 POSITIONING ALONG AN AXIS

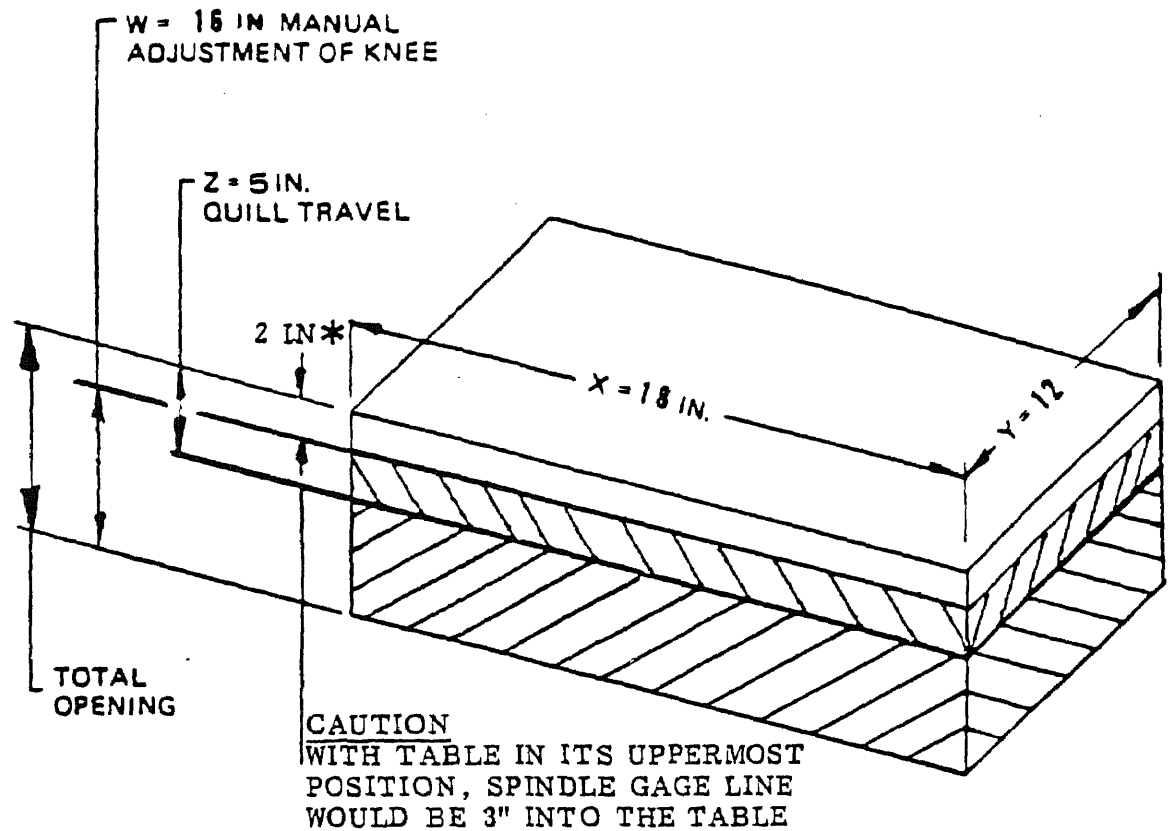
Refer to Fig. 5-5. Follow the sequence given below.

- (1) Select the desired axis (X, Y, Z).
- (2) Select the desired direction (+, -).
- (3) Select JOG, 1.0, .1, .01 or STEP.
- (4) Depress the axis direction pushbutton. 1.0, .1, .01, STEP would give axis motion of 1", .1", .01" and .001" respectively each time the button is depressed. While in JOG, axis motion will continue until the button is depressed. Fig. 5-5 shows total travel envelope.

NOTE:- If you exceed the limit in a particular axis, the power interruption will occur and the LIMIT OVERRIDE lamp will be illuminated. To resume, refer to LIMIT OVERRIDE in section 5.6.2.

### 5.5.2 TOOL LENGTH OFFSET

The following definition explains the meaning of tool length offset. Refer to Fig. 5-6.



TOTAL MACHINE TRAVEL ENVELOPE

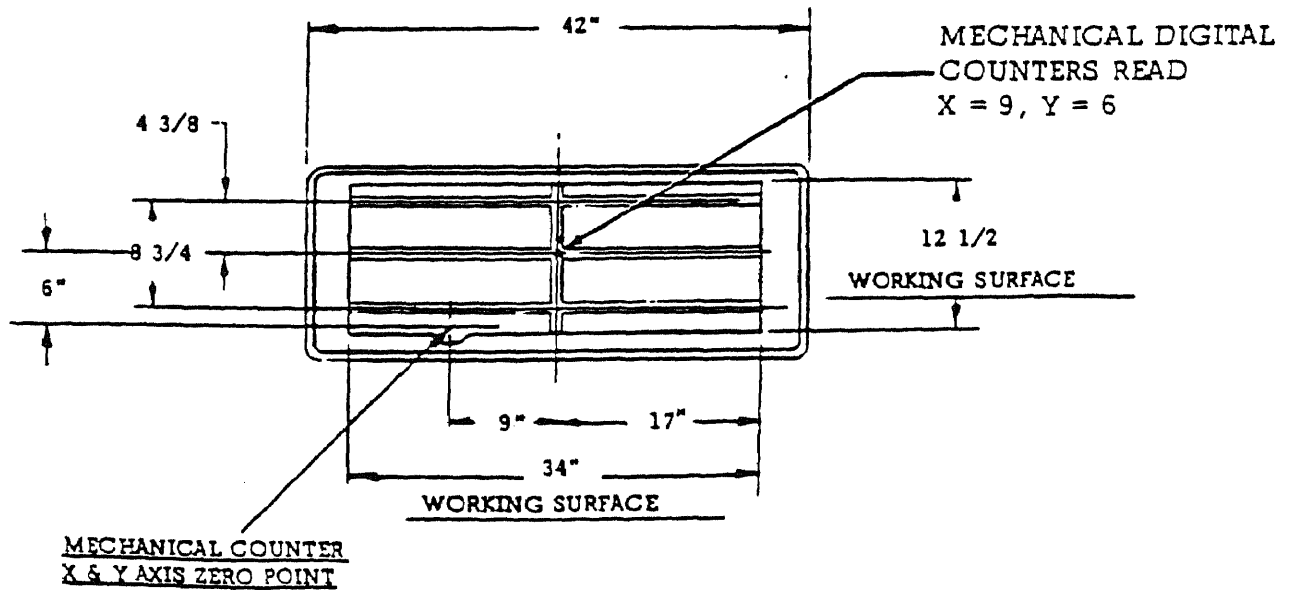


Figure 5-5 Machine Coordinate System



The tool length offset (TLO) is the distance from the bottom of the tool (tool tip) in the quill (spindle) "home" position (all the way up) to a point where ZABS is zero (the origin on the part coordinate system). This logic is used to accomodate different lengths of tools and allow the operator to correct for any difference between programmed length and actual length of the tool being used. When more than one tools are used, the tool length offset for each tool is stored in the memory. This TLO is automatically called out with coded tool number at the end of a tool change command code (M06) and the Z axis absolute register is set to that TLO value. The TLO value can be stored in three ways; in SET UP mode, (a) Using ABS/TLO switch, (b) Using MDI keyboard and (c) Using Tn/z command in the program.

While entering the TLO values, the following should be kept in mind.

The TLO value is a volatile memory i.e. when the power is switched OFF from the system, it is destroyed. When the power is ON, operating the RESET switch on special operations panel will set the TLO values to zero and new values can be entered by any of the three methods above. The TLO value entered by using the ABS/TLO switch or the MDI keyboard will override the TLO value entered by the part program. To enter the TLO value from part program, the stored value of TLO must be zero. This can be achieved by operating the RESET switch or by inputting a value of Z0 through (a) or (b) below in SET UP mode.

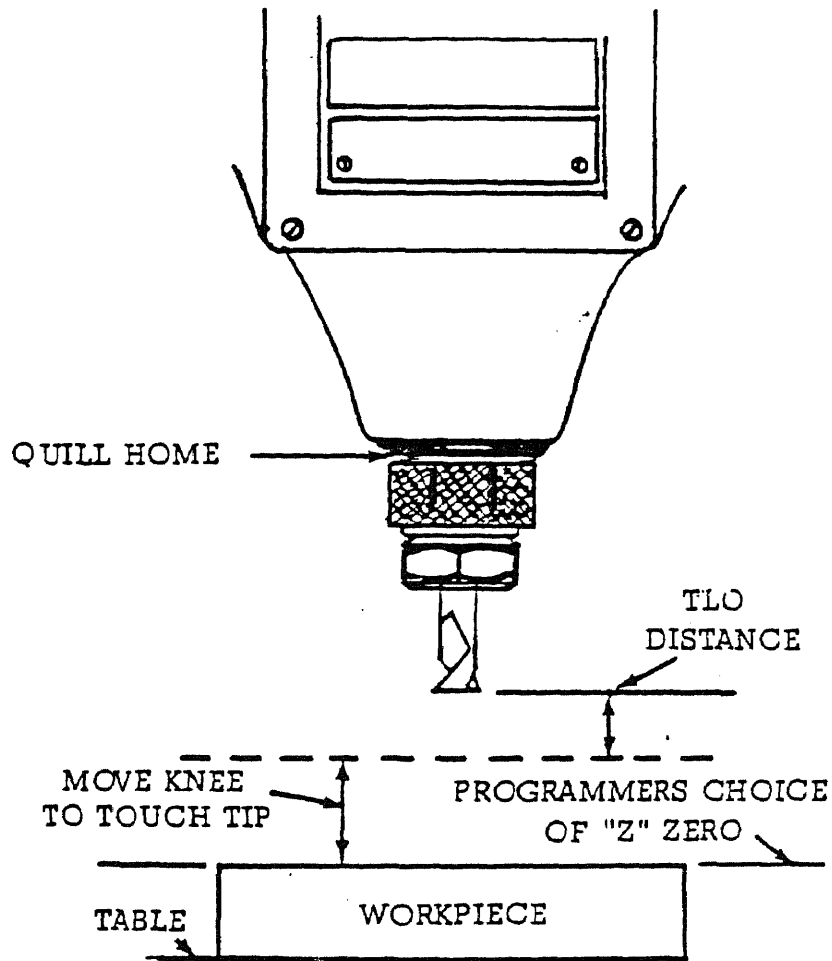


Figure 5-6 Tool Length Offset

NOTE:- The minimum TLO has to be 0.5".

- (a) Using the ABS/TLO switch
  - (1) Mount the longest tool in the spindle.
  - (2) Set the MODE switch to SET UP.
  - (3) Set the selector switch to TNO and the pushbutton below it to Z/TNO. By depressing the pushbutton advance the tool number to desired value.
  - (4) Set the selector switch to ZERO and depress the button. This will retract the quill (spindle) to "home" position.
  - (5) Using the AXIS MOTION switches and pushbutton, move the tool to the reference surface (origin, where ZABS is zero). Use the combination of X/Y/Z selector switch on Z axis, +/- pushbutton on - and the JOG/1/.1/.01/STEP switch on the desired distance.

CAUTION:- While selecting this distance, check for the distance between tool tip and part surface so that you do not damage the tool or the part surface.

- (6) Set the selector switch to STORE and depress the pushbutton. This will store this TLO value for this tool number.
- (7) To recheck the TLO value, set the selector switch to ZERO and depress the pushbutton to move the quill home. Select GO TO and depress the pushbutton. This will move the tool down to the stored TLO point.
- (8) Repeat steps 3-7 for each tool.

(b) Using the MDI keyboard

If the TLO values are known, they can be entered by using the MDI keyboard as follows:

- (1) Set the MODE switch to SET UP.
- (2) Depress the N key (T).
- (3) Using the numeric keys, enter the tool number.
- (4) Depress the G key (TLO).
- (5) Enter the tool length offset value using the numeric keys.
- (6) Depress the black key (EOB) to store this value.

(c) Using the Tn/Z command in the program

For details, refer to chapter-3 under T - Tool Select Command.

### 5.5.3 TOOL DIAMETER

The tool diameter can be stored in the memory as a reference for the operator. This memory is also volatile. It can be entered by (a) Using the MDI keyboard or (b) Using the Tn//d command in the program.

The diameter values are set to zero when the RESET switch is operated. They can be coded in the program at the beginning or they may be embedded in the program. The values entered by the MDI keyboard will override the values from the program if they are coded at the beginning of the program but will not override them if they are embedded in the program text.

(a) Using the MDI keyboard

- (1) Set the MODE switch to SET UP.
- (2) Depress the N key (T).
- (3) Enter the tool number using the numeric keys.
- (4) Depress the F key (DIA).
- (5) Enter the tool diameter.
- (6) Depress the black key (EOB) to store this value.

(b) Using the Tn//d command

Here, n is tool number and d is diameter.

For example: T1//.25

T2//.5

#### 5.6.1 NORMAL OPERATION PROCEDURE

The reader is advised to go through complete chapter before executing this sequence.

- (1) Switch the power on. Before operating the machine make sure that the air supply is between 60-80 psi. .
- (2) Depress the LIMIT OVERRIDE pushbutton on the operator's panel (illuminated in red).
- (3) Set the MODE switch to SET UP.
- (4) Switch the tape reader on which is mounted in the logic control cabinet on the side of the machine.
- (5) Operate the RESET switch if this is a new program or the CLEAR switch if this is a program to be added to the previously stored program.
- (6) Mount the tape on the spool and operate the LOAD switch to put the program in the memory. The

LOOP/LOAD/SPOOL switch on the tape reader should be on LOOP.

- (7) Mount the part on the table and mark the origin of the part coordinate system.
- (8) Use the AXIS MOTION selector switches and the pushbutton to move the table to the XY zero point of the part coordinate system (i.e. move the table to match the zero point with tool center). For details, refer to 5.5.1.
- (9) Set ABS/TLO selector switch to ZERO and the pushbutton below it to XY/T1 and depress it to store this value of XY position as X0Y0. (When you are at a point other than X0Y0, to return to the stored X0Y0, set the MODE switch to SET UP, ABS/TLO switch to GO TO and the pushbutton below it to XY/T1 and depress it.)

NOTE:- If the set up point has value other than X0Y0, use the MDI keyboard keys in this order to store that value: G92 X(value) Y(value) black key.

- (10) Store the tool length offset value (TLO) by any one of the methods described in 5.5.2. If the tool length offset value is not known beforehand, use method (a).
- (11) After storing X0Y0 and TLO as above, set the MODE switch to either BLOCK or AUTO position. If you are executing the program for the first time, it is advisable to use the BLOCK mode.
- (12) In BLOCK mode, operation will be interrupted at the

end of every block (i.e. every line) i.e. everytime the START/CONTINUE button depressed, only one line from the program will be executed. This mode is useful for checking the part program and debug it.

NOTE:- If the operator forgets to turn the spindle on, rapid traverse (positioning) moves will be executed but the operation will be interrupted at the first FEED instruction. To resume the operation, switch the spindle ON and depress the START/CONTINUE button.

It is a good practice to note the readings on the mechanical digital counters and the X, Y, Z absolute registers at the beginning and the end of the run and compare them.

#### 5.6.2 SPECIAL CONDITIONS

There are some special conditions which the operator may come across while operating the machine. The following is a description of these special conditions.

##### LIMIT OVERRIDE

When a motion causes a travel limit to be exceeded for that axis, the program will STOP and the LIMIT OVERRIDE lamp will be illuminated. Because this is an abrupt stop, the axes that were travelling could have "lost" up to .1" with reference to the origin. To resume the operation:

- (1) Set the mode switch to SET UP.
- (2) Depress the LIMIT OVERRIDE button simultaneously with the AXIS MOTION +/- button (distance selector switch

on JOG) to move in a direction opposite to the one which caused the limit override.

- (3) Once the LIMIT OVERRIDE lamp goes off, set ABS/TLO switch to GO TO and the button below it to XY/T1 and depress it. This will return the system to near X0Y0. Compare that position with original X0Y0 (origin) and make necessary moves using AXIS MOTION switches to place the tool to original X0Y0.
- (4) If the part program is not very long, set the FUNCTION top switch to RESTART and depress it to resume from the beginning of the program.

#### EMERGENCY STOP

If this pushbutton is depressed, the program operation will STOP and the system will be cleared as above. To resume, pull this button out and perform steps 3-4 as above in LIMIT OVERRIDE.

#### HOLD

When this button is depressed, the program operation will STOP without any memory loss. To resume, depress the START/ CONTINUE button.



## CHAPTER VI

### CONCLUSION

#### 6.1 CONCLUDING REMARKS

This thesis described basic operations and programming technique for the Bridgeport Series I CNC machine installed at the Department of Mechanical Engineering of New Jersey Institute of Technology. It is intended to serve as a beginner's guide for that particular machine as well as introductory guide to CNC machines in general.

Looking at the growth and increasing acceptance of CNC machines in the past few years, their demand should go up rapidly. Also, microprocessors have reduced the size and lowered the cost of the CNC machines. Thus, CNC has opened up whole new horizons for the N/C machines. The author believes that an engineering school offering a mechanical engineering program should include this subject in their curriculum to keep up with today's industry.

#### 6.2 FUTURE UPDATING

The CNC machine at the Mechanical Engineering Department at NJIT incorporates Bridgeport Operating System Software (BOSS) 4.0. The ERS card containing the BOSS 4.0 can be replaced by an ERS card containing BOSS 5.0. The additional chip required for floating point arithmetic on the LSI-11 microcomputer is already there. This updating would provide with the following additional features.

Multi-quadrant circular interpolation:- A G75 command can be used which allows the interpolation parameters I, J become the absolute coordinates of the circle center and any arc of up to 360 degrees can be programmed in a single block.

Polar coordinate commands:- This will allow the use of polar coordinates. Milling around an arc or milling an arc with increasing or decreasing Z depth (The Helix) could be done with looping.

Cutter diameter compensation:- A G41 or G42 command will form a new tool path equidistant from the programmed path at all times. This separating distance is the compensation dimension defined as half the difference between the actual tool diameter and the programmed tool diameter.

Cutter path transformation:- An entire cutter path and thus, a part shape with a symmetry can be rotated through an angle. The initial angle may be set and the programmed part shape could be rotated an absolute number of degrees. Also, using looping technique, the rotating angle could be incremented a certain amount. Also, the cutter path transformed could be scaled relative to its programmed dimensions.

Editor:- A data block can be executed from the terminal.

Macro subroutines:- The number of macros increase from 16 to 36.

Manual data input:- Allows circular interpolation data block to be inputted using MDI keyboard.

Remote computer data link:- A second 20ma serial line interface can be provided to link the machine with remote mass storage for down line part program loading.

The updating of the present system should be considered seriously so that, the students can learn updated techniques of programming.

## BIBLIOGRAPHY

- (1) Childs, James J., Principles of Numerical Control. New York: Industrial Press, Inc., 1982, pp. 1-100.
- (2) Operating Manual for the Series I CNC Milling, Drilling and Boring Machine. Bridgeport Machines Division of Textron Inc., April 1979.
- (3) Ramamirtham Narayanaswamy, Role of NC Machines in Metal Forming. Master's Thesis, Newark: New Jersey Institute of Technology, 1982.