SECTION I

PART PROGRAMMING

1.1 INTRODUCTION

Numerical control is the method of using coded, numeric instructions to execute automatically the steps involved in machining a part. For example, in the Bridgeport Numerical Control system, numeric codes control the following functions:

* Sequence number display
* Feedrate
* Table and saddle motion, also the third axis
* Quill position/up, down or drill cycle x
* Coolant / on-off x
* Spindle / on-off
* Tape reader rewind
* Turret stop position x
* Index table position x

x denotes optional external equipment

Numerical control represents a complete departure from the conventional machine tool operation whereby an operator or machinist studies an engineering drawing and then achieves dimensional accuracy by advancing the table and/or cutting tool through the use of the micrometer dial which is geared with a precision lead screw that translates the rotation into accurate linear movement.

With N/C, the drawings are studied and the information contained on them is converted to the series of numeric codes which direct the machine tool to perform the desired sequence of operations. The codes represent every motion or action the machine tool must take to machine the workpiece described in the Engineering Drawing. The complete set of codes necessary to machine a complete part is known as a part program.

Thus, the successful operation of the N/C machine rests not with the skill of the operator or machinist but the part programmer who envisions and then lists in logical sequence, the information which is required by the control to guide and direct the machining of the workpiece. The control system will do precisely and exactly what it is told to do. Anything other than correct instruction input results in faulty operation.

1.2 THE PART PROGRAMMER

It is the part programmer's job to translate the information contained in an engineering drawing onto a program manuscript which can then be converted into a media that will be used as input to the machine tool control system. Any function not directly executed by the machine tool but part of the machining process is transmitted as a separate set of instructions to the machine operator. Such instructions may include:

* The location of the set-up point
* The identification of tools and tool change points
* Settings of console control switches such as symmetry switches, dwell timer, peck cycle
* Spindle speed

The four basic prerequisites for a potential part programmer are:

1. He must be able to read and understand engineering drawings.
2. He should understand, and be capable of applying "shop mathematics" - arithmetic, algebra, plane and solid geometry, and trigonometry.
3. He must have a knowledge of good machining practices - selection of feeds and speeds, depth of cuts, chiploads, etc.
4. He must know thoroughly the abilities of the particular machine tool and control system for which he is programming.

The part programmer's functions may include the following:

1. Make, or at least be involved with, the decision as to what parts and what operations are to be made using N/C.
2. Establish the holding fixture requirements - locating and clamping surfaces.
3. Determine the cutting tools required, their speeds, feeds, and sequence.
4. Plan the maching sequence - how the part is to be run.
5. Write the program manuscript.
   Manual Programming. Using the PART PROGRAM sheets, the information required to machine a piece is written in the language of the N/C system and manually punched into tape. The tape and its printout must be checked against the manuseript for punching errors.
   Computer-Assisted Programming. The program is written in the language of the computer program used - for example, APT.
6. Write the N/C operator instructions, along with sketches, if required.
7. Check out the first run of the tape at the machine.

1.3 PROGRAMMING FOR N/C

The object of N/C is to maximize the time that the machine tool is cutting useful chips from a workpiece. The two important advantages that N/C has over conventional machining methods are:

1. The ability to machine complicated shapes accurately. N/C is often the only method of machining highly complex parts to a precise mathematical definition. Conventional machining may be so difficult that human error is very likely, and/or maintaining the extreme accuracy required by the design takes an inordinate amount of time or skill.

   Generally, N/C machining falls into two categories: Positioning (also known as Point to Point) and Contouring (or Continuous Path). In Positioning, the spindle is instructed to move to a specific point, perform an operation such as drilling, tapping, boring, counterboring, countersinking, etc., and move on to the next point to perform another task. While moving from point to point, the cutter tool is NOT in contact with the workpiece. The path that the table takes in moving to the specified point is not too important as long as it gets there quickly and does not touch the workpiece or clamping devices during its travel. In Contouring, the cutting tool is in constant contact with the workpiece as the machine elements are guided along a precisely controlled path. An example of Contouring is profile milling where the path the cutter follows to get from one point to the next determines the ultimate shape of the workpiece. Figure 1-1 shows a typical example of contouring work. Often machining a part involves both operations.

   The Bridgeport N/C Miller is capable of doing both three-axis Positioning and Contouring. When doing point-to-point work, the machine elements travel at a rapid traverse rate along a one, two, or three-dimensional linear path, which is the shortest distance between the programmed points. When doing contouring work, the machine elements travel at a controlled feedrate along either a one, two, or three-dimensional linear path or a two-dimensional arcuate path.

   Programming for Positioning is usually simple enough so that a program can be prepared manually with the help of trigonometric tables. Contour programming, on the other hand, can either be relatively simple, for example, profile milling where only a few slopes and arcs are involved, or very complex, as in machining aircraft parts that require several hundred thousand coordinate positions to sculpture the desired shape. For some parts, manual programming is impractical; too much time and effort are required to calculate and list accurately the several hundred coordinate positions needed. Computer-assisted programming should be considered for these difficult parts. In much the same way that N/C extends the abilities of its operator by enabling him to machine complex parts, the computer extends the abilities of the part programmer by enabling him to program complex parts. Just as N/C is a tool for machining, the computer is a tool for programming. Even though people not familiar with the use of a computer tend to shrug it off as being too difficult, in actuality, the technology jump from conventional machining to N/C is greater than the step required to go from manual programming to computer-assisted programming.

With one week of training, a good manual programmer can become proficient in computer-assisted programming. The Bridgeport N/C Miller can machine extremely complicated parts. To take full advantage of all its capability, computer-assisted programming is necessary.

2. The ability to monitor and thus control the overall machining process. While it is difficult to estimate how long it would take a skilled operator to machine a complex part, with N/C the machining time can be precisely determined. For example, the machine is not down while the operator reads a blueprint or an operation sheet between cuts to decide which tool to use, which cut to take, and how to make the cut.

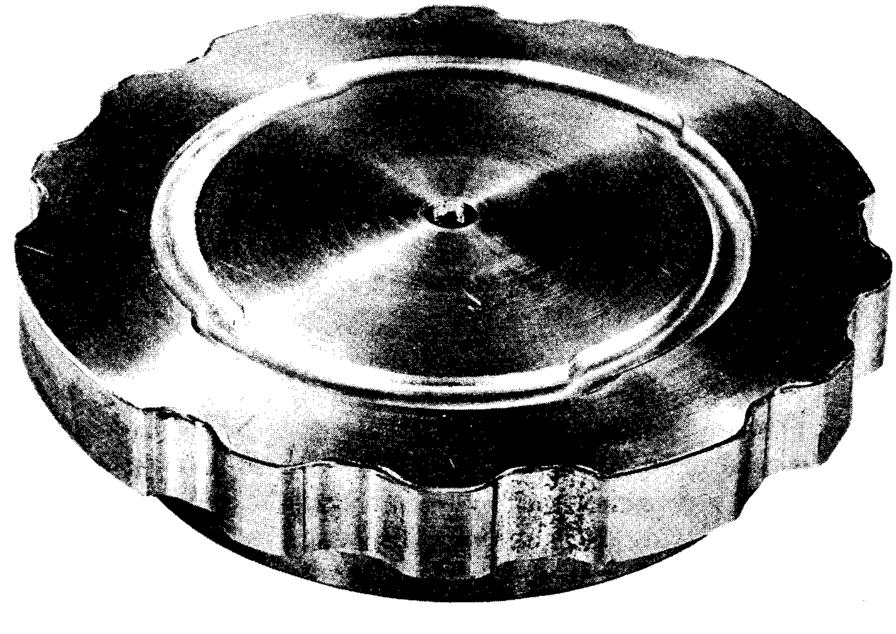


Figure 1-1. Typical Example of Contouring Work

Additionally, N/C and effective management techniques reduce the non-productive portions of the machining process. In a typical facility, a conventional machine tool may be operating 20 to 30 percent of the time, whereas it is not unusual to find N/C machines in operation 75 percent of the time. The use of N/C usually means that old shop practices have to be discarded - vaguely worded process sheets that leave many machining decisions to the machinist will not do. Every detail of producing a part by N/C must be fully worked out before a tape is cut. This means a new man-machine interface considerably more disciplined than heretofore. All functions must be analyzed, planned, and organized in greater detail. Alternate ways to do a job can be more readily explored, inefficient operations can be improved.

In one facility, analyzing the causes for N/C machine down time and taking corrective action increased the chip cutting utilization from 40 to 73 percent. Some remedial steps taken were:

* The table of each machine was self-milled with precision slots. Each fixture was provided with locating pins. Fixture installation was thereby reduced to setting the fixture with its pins in the proper slots. The tape programmer used his knowledge of slot and pin locations to program the machining tape from this pre-determined relationship.
* Two identical holding fixtures were provided. While tape directed operations were performed on one, the other was unloaded and reloaded.
* Tools were adjusted on a special fixture to pre-determined standardized lengths. The preset values were utilized by the tape programmer.
* Conventional length standard drills were replaced with short length, high precision types which could accurately locate without center drilling. Endmills were purchased from tool drawings to diameter tolerances less than commercial standards. As a result, piecepart tolerances were not seriously changed whenever a new tool was installed during process.
* Maintenance schedules were developed. A computerized daily list of service needs for each machine was supplied to the maintenance department to be finished the same day.
* Where possible, the operator was given secondary work such as deburring, countersinking, serial numbering, and inspection.
* Paper tapes were replaced with aluminized-mylar or paper-mylar. Since error-free control tapes of any great length are extremely hard to produce by the method of longhand programming followed by manual keyboard to punched tape conversion of data, the manual method of making tapes was replaced by computer generated tapes. This eliminated the mathematical, procedural, typing and proof-reading errors inherent in making tapes manually.

* An error-free tape is not necessarily an efficient tape. During production, each tape was reviewed for optimum feeds, speeds, and tool motions. Initially, it was found most jobs were running 25 to 40 percent of optimum feeds and speeds. The amount of material removed per pass was reconsidered. Tapes were identified with a manufacturing instruction issue number, part drawing issue number and using machine identification number.

Thus, N/C does not assure optimum profits. To achieve the correct product with maximum efficiency requires careful coordination of the many contributing professional and technical skills involved in the total production process. Only then can N/C fulfill its promise of higher productivity, consistent quality, less scrap, and increased profits.

1.4 THE N/C COORDINATE SYSTEM

1.4.1 Axis and Direction nomenclature

The Bridgeport Series II miller has three perpendicular axes of motion. Programming the machine involves the use of the Cartesian coordinate system. This system offers a means of expressing positions (or points) in space, and thus is ideally suited for expressing the location of a machine element, such as the knee, saddle or table, and for directing its movements.

Consider the two perpendicular lines shown in Figure 1-2. The horizontal line is designated as the X Axis while the vertical line is called the Y Axis. The intersection of the two axes is called the ORIGIN. The X coordinate of a point is the distance it is away from the X axis, the y coordinate of a point is the distance it is away from the Y axis. Thus, the coordinates of the ORIGIN (PXY) are X=O, y=O. By convention, that portion of the X axis which lies to the right of the Y axis has a positive value (+X), that portion which lies to the left of the Y axis has a negative value (-X).

Similarly, the portion of the Y axis above the X axis is positive (+Y), that portion below the X axis is negative (-Y). The X-axis and Y-axis divide the space of the enclosed two dimensional area into four sections, or quadrants. Any point falling in Quadrant I has a positive X value and a positive Y value. If the X distance P1 is 1.25" and the Y distance is .75", then the coordinates of the point are(1.25, .75). Similarly, the coordinates of P3 are (-1.0,-.5).

In addition to XY coordinates, a point in three-dimensional space has a Z coordinate. The Z axis passes through the XY origin point, perpendicular to the X and Y axes. Figure 1-3 shows the XYZ coordinate system concept as it applies to the Bridgeport N/C Miller. The table and saddle provide X and Y motions and the knee provides Z motion.

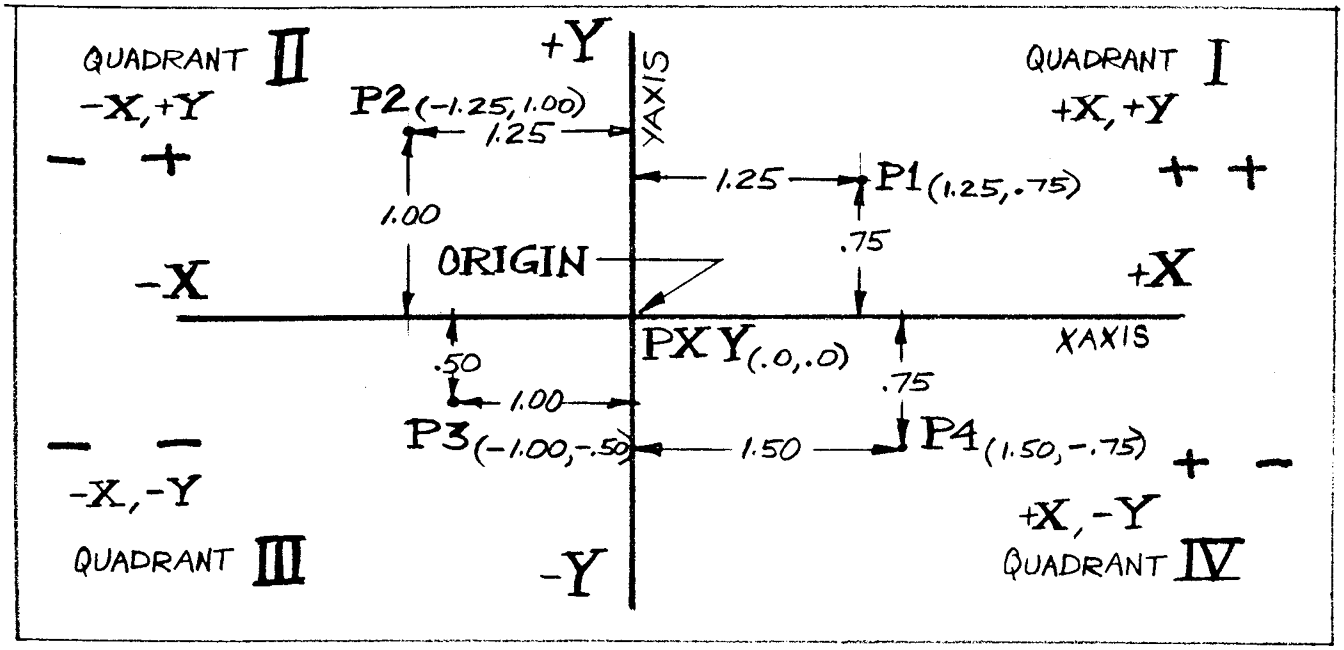


Figure 1-2. The N/C Coordinate System

1.4.2. Absolute and Incremental Dimensioning

There are two basic methods of listing coordinate positions: Absolute and Incremental. If a programmer is preparing instructions for an absolute control system, the positions will be stated as measurements from an origin point. The origin point may be fixed as part of the machine itself, such as a corner of the work table, or it may be established at some convenient point on the workpiece. For example, a simple program considering the four points P1, P2, P3 and P4 (Figure 1-2) with the origin as a start and end point would be as follows, using absolute dimensioning:

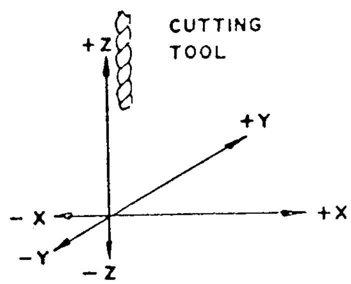
n	X	Y
1	1.25	.75
2	-1.25	1.0
3	-1.0	-.5
4	1.5	-.75
5	.0	.0

(The "n" column specifies the sequence number of the instruction)

In programming for an incremental control system, the positions are given in terms of distance and direction from the immediately preceding point. The use of plus and minus designations takes on a new meaning; a "+X" command refers not to a specific rectangular quadrant, but rather tells the tool to move "right" along the X axis from the point where it is now located. Similarly, "-X" directs the tool to the left, "+Y" designates a move "up", "-Y" is a command to move "down". A "+Z" is a command to move "away" from the workpiece, a "-Z" directs the tool toward or into the workpiece. A program coded for an incremental system using the same points as the above example would be:

n	x	y
1	1.25	.75
2	-2.5	.25
3	.25	-1.5
4	2.5	-.25
5	-1.5	.75

Note that in incremental dimensioning, if the startpoint and endpoint are the same, the sum of X axis and Y axis moves should each equal, zero. This self-checking feature is an important advantage of incremental programming.



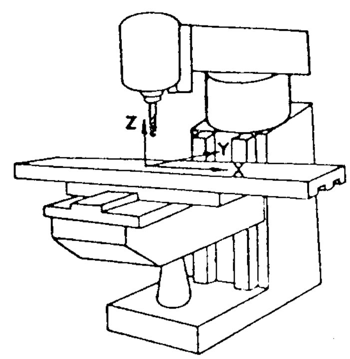


Figure 1-3. The Bridgeport N/C Miller Coordinate System