DOCUMENT RESUME

ED 300 589 CE 051 209

AUTHOR Mullen, Frank M.
TITLE Computer-Numerical-Control and the EMCO Compact 5 Lathe.
PUB DATE 88
NOTE 19p.
PUB TYPE Guides - Classroom Use - Materials (For Learner) (051)

EDRS PRICE MF01/PC01 Plus Postage.
DESCRIPTORS *Computer Software; *Industrial Education; Machine Tool Operators; *Machine Tools; *Numerical Control; Postsecondary Education; *Programing

ABSTRACT

This laboratory manual is intended for use in teaching computer-numerical-control (CNC) programming using the Emco Maier Compact 5 Lathe. Developed for use at the postsecondary level, this material contains a short introduction to CNC machine tools. This section covers CNC programs, CNC machine axes, and CNC coordinate systems. The following section is a specific treatment of the major programming functions of the lathe. This section discusses the Emco keyboard, program entry, G codes and related function of each code, tape deck operations (loading a program from tape and saving a program on tape), and error codes. (YLB)

ED300589

Computer-Numerical-Control

and the

EMCO Compact 5 Lathe

Dr. Frank M. Mullen
Dept of Industry & Technology
College of Engineering
Texas A&I Universtiy

CE051209

Introduction

Automatic machines are not a new idea. Various devices that moved by themselves (automatons) have been built for years. Such machines have ranged from automatic temple door openers in Greece to the very lifelike movements of some 18th century dolls. However, most of these early machines seemed designed as much for their entertainment value as for their ability to do useful work. It wasn't until the coming of the first industrial revolution that serious industrial applications of automatic control began to appear. During this period, automatic operation was added to several types of power equipment to increase its speed, consistency, and flexibility. The programmable Jacquard loom (1789) is a prime example of such an application.

Jacquard loom

The first automatic machines were controlled either by complex mechanical systems or clock mechanisms. Well into the 20th century, many automatic production machines (machine tools) continue to be controlled by mechanical or hydraulic systems which physically follow a prototype part, template, or cam. The automatic screw machine is a good example of such a machine. While these machine tools are well suited to long runs of identical parts, usually 10,000 or more, they are generally difficult to modify quickly and cheaply to produce different parts. Such machines are commonly referred to as "**hard automation**".

hard automation

The application of electronic control to machine tools was not developed until the period following WWII. During the war, electronic feedback (servo) control had been applied to a number of industrial and military engineering problems, such as the movement of aircraft control surfaces, gun control, and hazardous materials handling. While the resulting machines were the forerunners of modern automated machine tools and robots, they depended on constant control by a human operator. Only the simplest actions could be put under total automatic control.

servo control

Numerical Control

In the early 1950's, a group at the Massachusetts Institute of Technology (MIT), working under a grant from the Air Force, developed the basic electronic system for a machine tool which was controlled from a punched tape reader. This allowed a variety of parts to be made using the same machine simply by changing the pattern of holes (or program) on the paper tape. Tapes could be stored and then reused later to make additional parts. This technology is still used and has become widespread in manufacturing situations which require flexibility in the part design and relatively small batch production (10 - 1000 parts). Such machines are generally referred to as **numerical-control (NC)** machine tools.

Computer Numerical Control

CNC applications

While paper tape driven machine tools provide the flexibility needed in many manufacturing environments, most are limited in their ability to do other than the simplest types of computation while the program is running. Nearly all of these machines require the tape programmer to make calculations for such things as arcs and contours during the tape preparation stage. Although computers are generally used as a programming aid for such computation, the programmer must still provide the NC machine with detailed instructions about tool movement. The advent of small scale computers and the microprocessor in particular has made it possible to build the computer into the machine tool itself. Such a machine is classified as a **computer-numerical-control (CNC)** machine tool. This type of control system has been applied to a wide variety of machine tools, a partial list includes: lathes, milling machines, drilling machines, punch presses, sheetmetal forming machines, radial arm saws, and injection molding machines.

advantages of CNC

One of the most significant advantages of the CNC machine tool is the ease with which it can be programmed. Since most of the calculations required to create proper tool paths are done by the computer, the programmer only needs to supply basic geometric information about a particular part. For example, circular cuts can be created by giving the computer the location of the center of the circle and its radius. The detailed calculations needed for tool movement will be done by the computer.

CNC Programs

CNC programs are a series of commands stored in computer memory which control the operation of the machine when it is making a part. A typical program is literally a procedure list for the machine to follow in machining a particular part. The computer directs the actions of the machine by following the list one step at a time. The list of commands are created by the **CNC part programmer** using a command language and then entered directly into the machine, punched on paper tape, or stored on magnetic media.

CNC languages

There are many command languages used to program CNC machine tools. If fact, no two CNC machines will use exactly the same programming method. However, the most common programming methods can be generally divided into those which are **conversational** and those based on **word address format**.

conversational

Conversational command languages are usually designed around a set of English words and phrases. Shop words which describe conventional machining operations are generally used to create the series of commands programmed into the CNC control computer. There are many conversational programming languages used with CNC machine tools. Some of the more widely used are: APT (automatic programmed tool), COMPACT II, and MSL (machinist shop language). This type of programming will not be discussed in detail here because the Emco CNC 5 Lathe uses word address format programming. Additional information on programming languages can be found in the references section.

word address format

Unlike the conversational languages, word address format programming uses a set of numerical codes to indicate the machining operations desired. The majority of these codes have been compiled into an international standard set. Word address format programs for CNC mach`-- tools are made up of a number of basic program elements arranged into functional groups called **blocks**. The basic program elements (or **words**) in each block include: sequence number, machine function, tool path coordinate dimensions, feed rate, and any additional information. The **sequence number** is used to identify a particular block of program elements within the program. These are in simple numerical order and generally indicate the order in which actions are to be taken by the machine. **Machine functions** are used to indicate the action to be performed by the machine while it is executing a block. Each code is a number which designates a particular action. General machining operations are indicated by *preparatory functions* (G codes). Other operations are entered using the *miscellaneous functions* (M codes). **Tool path coordinate dimensions** indicate the values of the tool coordinates to be used with the preparatory function. There will be a tool path coordinate or dimension for each axis on the machine tool. In most systems, only the values for axes affected by the preceding preparatory function need to be entered in the program. The exact form of these values depend on whether the dimensioning system being used in that block is incremental or absolute. The **feed rate** for any cutting function is also required in each block. As with tool path coordinates, the exact operation called for by the preparatory function may have an understood feed rate. **Additional information** in the program block may be necessary for machine functions which require values related to tool changes, coolant on/off, spindle motors, etc..

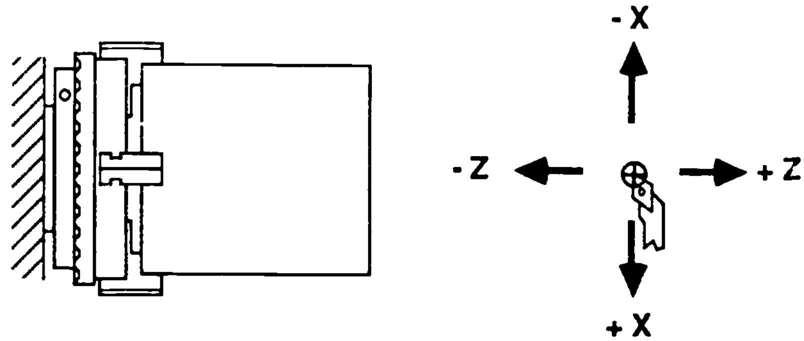
sequence number

preparatory functions

CNC Machine Axes

Z axis parallel to spindle

In order to create a CNC program, a part programmer must command the machine to move the tool or workpiece around in three dimensional space. Tool movements are defined in terms of the machine axes. The axes of a CNC machine tool are designated by a series of letters from the end of the alphabet. The Z-axis is almost always parallel to the rotating spindle of the machine. This may be vertical in the case of a vertical mill or horizontal as with the lathe or radial arm saw. The definition of the other axes depends on the machine in question. The X-axis is usually parallel with the machine's cross feed on the lathe but on a milling machine it is parallel to the longitudinal feed of the table. On most milling machines the Y-axis is parallel to the table cross feed. While any point in three dimensional space can be defined by the use of three axes or coordinates, industrial CNC machine tools may have as many as eight or more axes. These extra axes usually designate expanded functions or features of the machine, such as the control of additional tool holders, etc..



Axes of CNC Lathe

CNC Coordinate Systems

CNC machine tools may allow the use of polar coordinates (angular measurement) and/or Cartesian coordinates (linear measurement) for programming. The Emco CNC 5 Lathe uses the Cartesian coordinate system to indicate position or movement. In the Cartesian system, all locations in three dimensional space are given in terms of straight line distances. Two programming methods are used in CNC command languages to indicate these Cartesian positions or movements: **absolute** and **incremental**. Each method has its advantages and disadvantages. Because of this, many modern CNC machine tools (including the Emco CNC 5 Lathe) can be programmed in a combination of these methods.

absolute method

Using the absolute method, all tool movements and positions are given with reference to a set or "zero" reference point on the machine. Every point within the working space of the machine is defined by three unchanging numbers or coordinates. Two axis machines like the lathe will use only two numbers to indicate such absolute positions. While the starting or "**home**" position on a CNC machine tool can be defined at a number of locations, the important point to remember is that the reference point is absolute and will generally not be changed during the execution of a program. Given the known starting point, tool movements are defined in terms of the absolute coordinates of the destination. This system can be pictured much like longitude and latitude on a world globe. Every location on the earth can be defined by a pair of numbers referenced to the intersection of the prime meridian and the equator. If you know where you are starting, it is possible to move from a given location to any other by simply knowing the two values which represent the longitude and latitude of your destination. For a given destination, this pair of numbers will be the same regardless of where we start.

advantages

An important advantage of the absolute system is that when programming a move from one point to another, the distance and direction from the starting point and destination point do not have to be known. All that is required are the coordinates of the destination. The absolute system also has an advantage when several points are to be visited in succession. A change in the coordinates of an intermediate location, due to error for example, does not effect those preceding or following it. On the negative side, the absolute coordinate system is awkward to use when trying to define a geometric shape such as a circle or rectangle. Finding the absolute coordinates of significant features on the shape, like the corners of a rectangle, must be calculated from part dimensions and the current absolute position. This is a significant source of potential error.

disadvantages

incremental method

Using the incremental method, locations and paths are referenced, not to an absolute machine point, but to the current position of the tool. Movements are **incremented** or added onto previous locations. All tool movements are designated by a distance and a direction. This can be compared to the normal way people give directions for getting from one place to another. For example, start here, go two blocks north, then turn right and go four blocks east, then turn left and its the fourth house on the right. Each step in the process requires a new direction and distance beginning with the stopping point of the previous step.

advantages

A significant advantage of the incremental system is that it eliminates any problem with shape definition. Shapes can be defined directly by using their dimensions and do not require additional calculation. The system is also very convenient when a series of movements must be repeated in another location. All the values and directions of movement stay the same. The incremental system is at a disadvantage, however, when a location within a sequence of locations must be changed due to design change or error. Relocating an intermediate position requires that the previous move and all subsequent moves be changed to accommodate the new location.

disadvantages

combined methods

Most industrial CNC machine tools allow the use of both the absolute and incremental coordinate systems. Although the programming is more complicated, the systems can be combined to gain the advantages of both. For example, a shape routine such as a circle can be programmed using the incremental system then the location of the circle on the part determined by the absolute coordinates of its center. This allows not only the use of direct dimensions in the circle portion of the program but allows the movement or repetition of the circle pattern at other locations using different absolute coordinates.

Emco CNC 5 Lathe

Lathes are an excellent introduction to CNC machine tool programming because they generally have only two axes and a relatively small set of instructions. In spite of its simplicity, the lathe provides all the basic programming elements and operations common to industrial CNC machines.

Emco CNC 5 Lathe Programming

The Emco CNC 5 lathe was designed for CNC training and light production. This small lathe allows control of tool movement in the Z-axis (longitudinal feed) and X-axis (cross feed) either manually or by the microcomputer built into the lathe. A subset of the international standard G codes and tool path dimensions make up the commands to the controlling computer in the lathe. Only twelve codes are used to program tool movements and other functions. Program entry and editing are simple and can be done directly on the machine through the keyboard and display.

important programming facts

Below are some of the most significant factors to keep in mind when programming the Emco Maier CNC 5 lathe:

1. Programming can be done in the metric or conventional system of measurement.
2. The smallest programmable unit of distance is .01 mm or .00039 in. This is displayed as a unit of 1.
3. All G codes and other values are displayed <u>without</u> decimal points.
4. The dimensioning system used can be <u>absolute</u> or <u>incremental</u>. This is controlled by the appropriate G code.
5. The sign of the values entered for the X and Z axis determine the <u>direction</u> of tool movement. Negative X values cause cross feed movement toward the back of the lathe or into the workpiece. Negative Z values cause longitudinal movement toward the chuck. Positive values cause movement toward the front of the lathe or away from the chuck respectively.

program block

The Emco lathe uses five program words or elements in each **program block**. A given block may not have values for all the words, but all blocks will have at least a sequence number (N) and a preparatory function (G). The five words and their meanings are listed below:

word format

N word - Block sequence number

G word - Preparatory function code

X word - Direction and distance of movement in X axis

Z word - Direction and distance of movement in Z axis

F word - Feed rate

Emco Keyboard

The function of the keys on the Emco keyboard:

NUMBERS	Enter values of G, X, Z, and F.
[–]	Enter negative X and Z values. This key is pressed <u>after</u> the number has been entered.
[INP]	Input key. Enters value on display into computer memory.
[DEL]	Delete key. Delete the current value displayed except block sequence numbers and error codes.
[FWD]	Forward key. Advances the display from one block to the next.
[REV]	Reverse key. Steps the display back one block.
[→]	Arrow key. Advances the display to the next word.
[INP] + [FWD]	Stops program at current location similar to G20. Program can be restarted from where it left off by pressing START.
[INP] + [REV]	Stops program at current location. Program will restart from <u>beginning</u> if START is pressed.
[INP] + [REV]	Deletes an error code. The incorrect value will be displayed.
[DEL] + [INP]	Completely deletes current program from memory. Used to clear computer for a new program.
[START]	Runs the program in memory from the beginning.

Program Entry

Blocks are entered into the computer using the following sequence:

1. Press INP to enter the displayed sequence number (N).
2. Enter G code, then press INP.
3. Enter X value (if needed), then press INP.
4. Enter Z value (if needed), then press INP.
5. Enter F value (if needed), then press INP.
6. The computer will automatically display the next sequence number.

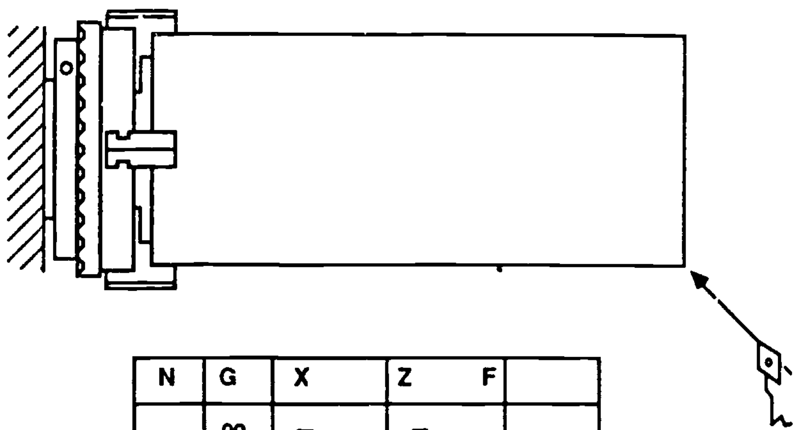
G Codes

Programs for the Emco CNC 5 lathe are written using a set of G code preparatory functions. Each code causes the machine to perform a particular type of machining operation. Each code and its related function is explained in detail below.

G 00 Rapid Traverse

The G00 code is used to move the tool rapidly from one position to another without cutting. This code allows coordinated movement of both axes at the same time or individually. After the block number and G code are input, an X value must be input. After the X value is input, the computer will request a Z value. The sign of these values will determine the direction of movement. There is no need to enter a feed rate since it is automatically set at the maximum rate of 800 mm/min.

G 00



N	G	X	Z F	
	00	–	–	

G 00 Input Format

N .. | G 00 | X ± .. | Z ± ..

Warning!!! With a feed rate of 800 mm/min., the G00 code is <u>not</u> meant for cutting. Use this code only for moves in air.

G 01 Linear Interpolation

The G01 code is used to remove material in a straight line at a controlled rate of feed. This code may be used in three ways, (1) for longitudinal turning in the Z axis, (2) for facing in the X axis, and (3) for turning tapers. This code requires an X value, Z value, and F value. The limits on these values are: X value 0-5999, Z value 0-39999, F value 1-499.

longitudinal turning

G01 <u>Longitudinal Turning</u>. To turn in only the Z axis, enter an X value of zero, then a nonzero Z value. The sign of the Z value will determine the direction of the cut and the F value will determine the feed rate.

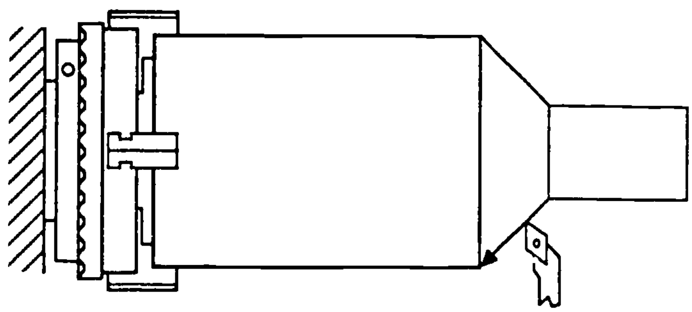
facing

G01 <u>Facing</u>. To turn in only the X axis, enter a nonzero X value, then a Z value of zero. The sign of the X value will determine the direction of the cut and the F value will determine the feed rate.

tapers

G01 <u>Tapers</u>. Tapers are turned by entering a nonzero value for both X and Z. The dimensions of the taper which will be cut are based on the <u>ratio</u> between the X value and Z value.

G 01



N	G	X	Z F	
	01	–	–	

G 01 Input Format

N .. | G 01 | X ± .. | Z ± .. | F ..

correct values

Determining the correct X and Z values: (1) if the X and Z axis dimensions of the taper are given, they can be entered directly, (2) if the taper is given in terms of an angle, a trigonometrical conversion can be used to determine the correct X and Z values.

Limits of the taper ratio: The ration of X value to Z value can range from 1:39 to 39:1. For the best surface finish, however, the recommended ratio should not exceed 1:10 or 10:1. The ratio must also be an integer number.

G 02 & G 03 Circular Interpolation

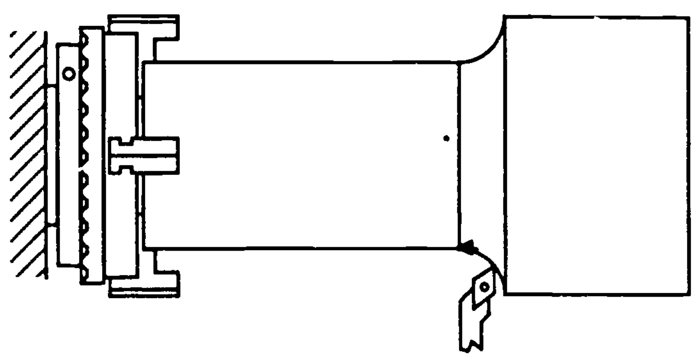
The G02 and G03 codes are used to remove material in an arc at a controlled feed rate. Both of these codes are subject to a stringent set of rules:

1. The tool will always move through a full quarter of a circle. Partial arcs are not possible.
2. The tool will always move in a minus Z direction.
3. The only radii possible in the mm mode are: .25 mm, .50 mm, then each whole mm up to 59 mm.

counterclockwise movement

G02 Counterclockwise Movement. The G02 code will cause the tool to move in a counterclockwise direction when viewed from above. The X value determines the radius of the quarter circle which will be transcribed by the tool. The sign of the X value determines which of two possible arcs will be turned. The Z axis movement will be equal to that of the X axis and always in the negative direction.

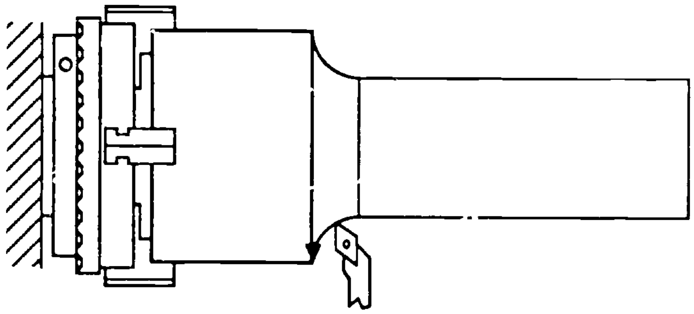
G 02



N	G	X	Z	F	
	02	–			

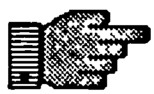
First Quadrant Circular Interpolation

G 02



N	G	X	Z	F	
	02	+			

Second Quadrant Circular Interpolation



G 02 Input Format

N .. | G 02 | X ± .. | F ..

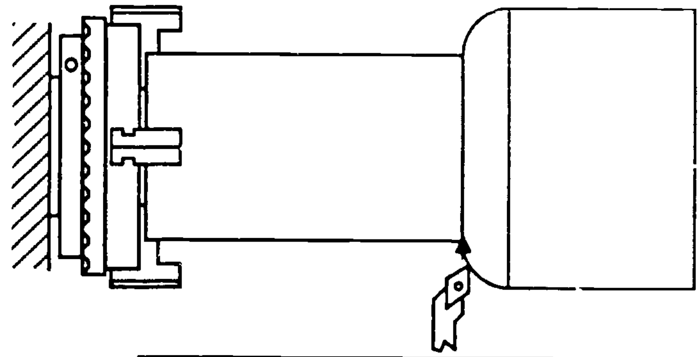
clockwise movement

G03 Clockwise Movement. The G03 code will cause the tool to move in a clockwise direction when viewed from above. As in the G02 code, the X value determines the radius of the arc. The sign of the X value determines which of two possible arcs will be turned. The Z axis movement will be equal to that of the X axis and always in the negative direction.

G 03 Input Format

N .. | G 03 | X ± .. | F ..

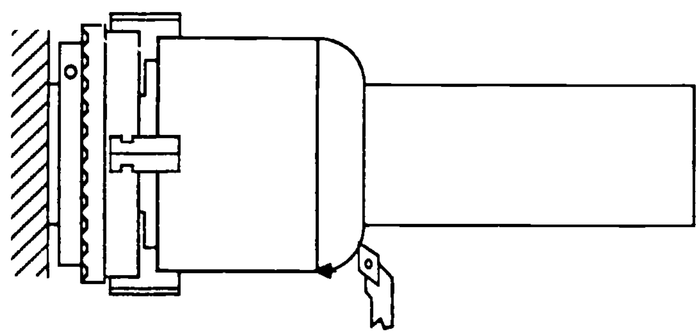
G 03



N	G	X	Z F	
	03	-		

Third Quadrant Circular Interpolation

G 03



N	G	X	Z F	
	03	+		

Fourth Quadrant Circular Interpolation

G 84 Longitudinal Turning Cycle

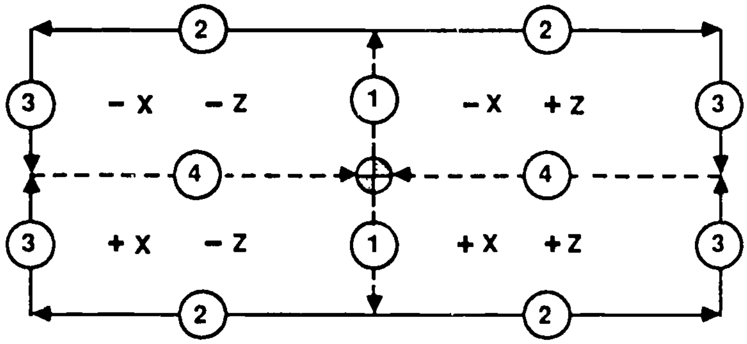
The G84 code combines a sequence of four moves commonly used on a lathe for longitudinal turning. This "canned cycle" describes a rectangle with the tool starting and stopping at the same location. The dimensions of the rectangle and the directions of tool movement are determined by the sign and quantity entered for the X and Z values.

The sequence of moves created by a single G84 code are:

G 84 Sequence of Moves

1. A rapid traverse for the distance and in the direction set by the X value.
2. A longitudinal move for the distance and in the direction set by the Z value at the feed rate set by the F value.
3. A cross feed move for the distance and opposite the direction set by the X value at the feed rate set by the F value.
4. A rapid traverse for the distance and opposite the direction set by the Z value.

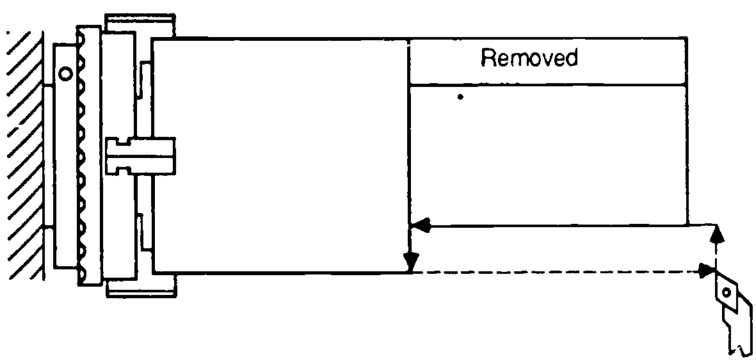
possible sequences



G 84 Input Format

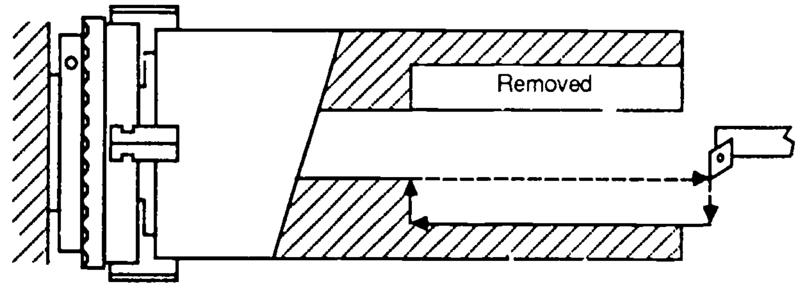
N .. | G 84 | X ± .. | Z ± .. | F ..

external turning



N	G	X	Z F	
	84	–	–	

internal boring



N	G	X	Z F	
	84	+	-	

G 20 Program Hold

A program can be stopped at any point while it is running by a block containing a G20 code. The G20 code can be used to stop a program for a tool change, measurement, or to make manual adjustments from the hand mode. After a G20 stop, the START key will restart the program from where it left off.

G 20 Input Format

N .. | G 20

G 21 Empty Block

The G21 code can be used to allow room within a program for the addition of blocks after the program is entered in the computer. When run, a program will skip over blocks containing a G21. However, the programmer can change the G code and add values to the block later.

G 21 Input Format

N .. | G 21

G 22 Program End

The G22 code indicates the end of a program. The last line of any program must be a G22 code.

G 22 Input Format

N .. | G 22

G 64 Motors Off

The G64 code is used to shut off power to the stepper motors on the lathe to prevent damage. The G64 code is not used in a program but entered directly from the keyboard.

G 64 Input Format

G 64

Tape Deck Operations

The tape deck on the Emco CNC lathe is used to save programs permanently using the G65 code. Programs are identified by a number ranging from 1 - 999. The following procedures allow programs to be loaded and saved to the tape.

Load a Program from Tape

1. Move to G input mode.
2. Delete any present G code.
3. Input a 65.
4. Press INP.
5. Press INP.
6. Input program number.
7. Press INP.
8. Tape will be searched and loaded, if found.

Save a Program on Tape

1. Move to G input mode.
2. Delete any present G code.
3. Input a 65.
4. Press INP.
5. Press FWD.
6. Input program number.
7. Press INP. Program will be saved on tape.
8. Program will then be checked.

Error Codes

Error codes or alarm signs can occur during entry of a program or when the program is run. They indicate some type of problem with the program and not with the machine itself. When an error is detected, the A light above the display will come on and a number will be displayed. The following list indicates the codes and their related errors.

During CNC Operation

A00 G code is incorrect.

A01 Incorrect radius entered for a G02 or G03 code. Correct values are: 25/50/100/200/ . . . 5900.

A02 X value incorrect. Correct value range: 0 - 5999.

A03 F value incorrect. Correct value range: 1 - 499.

A04 Z value incorrect. Correct value range: 0 - 39999.

A05 No G22 code at end of program.

A06 Spindle speed to high for threading.

A07 Taper ratio between X and Z values incorrect. Correct values are: 39:1 to 1:39.

During Tape Operation

A08 Out of space on tape during a save operation.

A09 Program not found on tape.

A10 Tape is write protected.

A11 Error during a load operation.

A12 Error during a check operation.