MANUALS

Part Programming Commands

This chapter details the part programming codes used to run your Excellon machines automatically.

The CNC-7, like all Excellon machines, has a set of part programming codes that can be used to control the machine for drilling, toolchanging, setting up machine parameters (such as feeds and speeds), and routing (if so equipped). Also, like other Excellon machines, the part program codes are backward compatible. This means that part programs from a CNC-2,4,5 or 6 can be run on your CNC-7 without modification. Since newer controls contain new features, the reverse is not necessarily true (You may not be able to run all CNC-7 programs on a CNC-2,4,5 or 6). Part programs are simply data files, coming from any one of a variety of sources or devices. This chapter will detail all available part program codes available for your use.

Part Program Headers

The M48 header is used to give your machine general information about the job. This includes the size of tools you want to drill and/or rout the PC board, the kind of measurement system you are using, the direction of the X and Y axis of the work, and other details. These instructions may be generally listed in any order in the header. The part program header is optional. Most commands that you can program into the header can also be entered at the CNC-7 console before the program runs.

Part Program Body

The set of drilling and/or routing commands is called the part program body. It is usually much longer than the header and tells the machine exactly where each hole is to be drilled, which drill bit to use, what shape you want routed, etc. The commands are laid out in the sequence you want them carried out on the PC board. For example, one line of the program will tell the machine where to drill a hole, the next line will tell where to drill the next hole, the next line will tell the machine to stop and change the drill bit. Usually the program is carried out in sequence from top to bottom. However, some commands will tell the machine to move to another location on the PC board, go back to a previous line in the program, and repeat the pattern.

Excellon Program Format vs. Other Manufacturers

Because Excellon is a pioneer in the manufacture of computerized drilling and routing equipment, it was necessary for Excellon to develop a set of commands to control the machines. The set is called Excellon Numeric Control and it uses the same commands for all Excellon machines. Some of these commands have become standard in the industry and are widely used by other manufacturers. The first machines introduced by Excellon were drilling machines. The set of commands used on drillers later became known as Format One. When Excellon introduced machines with routing capability, a set of commands called Format Two was created. Then in 1979, Excellon revised Format Two to combine drilling and routing commands into one common set. The machines introduced prior to 1979 are called generation one machines and cannot use Format Two. They do not have all the capabilities of the newer machines. However, newer generation two machines can run part programs with either Format One or Format Two commands.

What a Part Program Must Include

There is some information that the CNC-7 cannot know without being told. Some of the things that the part program must tell the machine are:

Where to drill each hole
Where to rout
What size tool to use

Additionally, if the programmer wants to change the speed of the direction of a particular tool of the worktable, without stopping the machine, the change must be made in the part program. Examples of these changes are:

Reverse the direction of routing
Change the table feed rate
Change the spindle RPM

Writing a Part Program

This section describes what you need to know to write a part program header and a part program. It identifies the mandatory requirements, as well as the options, and provides you with examples of how a part program might look.

The Header: Setting Up The Job

The header is always located at the beginning of a part program. It consists of a series of instructions (commands) that are used to give your machine general information about the job. This includes the size and speed of tools, the kind of measurement system you are using, the direction of the X and Y axis of the work, and other details. The header can have just a few commands, or dozens of them, depending on your needs. Most of these commands may be placed in any order. But one thing the header may NOT include is machine motion commands such as JOG or HOME. Do you remember that we said the header is optional? This does not mean that the commands you write into a header are optional. If you choose not to use a header, then you must either write the commands into the part program or enter them at the CNC-7 console before the program runs. Entering them manually can lead to problems. Suppose that you get an order to produce a set of the same PC boards every two or three months. Each time the program is loaded into the CNC-7, you must be given instructions on all the commands that have to be entered before the job can begin. If you put the commands in the header instead, you are assured of consistent settings for the machine.

Example of a Header

Below is a sample of a header. The PURPOSE shown to the right of the COMMAND is not part of the command, but is shown for your benefit to explain the command:

COMMAND	PURPOSE
M48	The beginning of a header
INCH,LZ	Use the inch measuring system with leading zeros
VER,1	Use Version 1 X and Y axis layout
FMAT,2	Use Format 2 commands
1/2/3	Link tools 1, 2, and 3
T1C.04F200S65	Set Tool 1 for 0.040" with infeed rate of 200 inch/min Speed of 65,000 RPM
DETECT,ON	Detect broken tools
M95	End of the header

Beginning of a Part Program Header

M48

M48 Defines the start of an M48 part program header. This command must appear on the first line of the part program header. This tells the CNC-7 that the program has a header. Please note that comment lines and blank lines are permitted in the M48 header and are ignored. Comment lines are lines of text beginning with the semicolon (;) character.

See also: Part Program Headers

End of a Part Program Header

M95

M95 Defines the end of a part program header. Either this command or the % command must follow the last header command in the part program header. This tells the CNC-7 where the header ends. When this command is used, the machine will immediately start to execute the part program body commands following the M95 command.

See also: Part Program Headers, M48

Rewind Stop

%

% Defines the end of a part program header. Either this command or the M95 command must follow the last header command in the part program header. This tells the CNC-7 where the header ends. When this command is used, the machine will stop at the end of the header and await your action. You may enter any appropriate Keyboard commands and/or press CYCLE START to continue.
Note: This command has a different meaning when used in the part program body.

See also: Part Program Headers, M48, M49

Commands Used in a Header

The following table provides you with a list of commands which (not a complete list) are the most used in a part program header. Some Operating System commands, which are discussed in the chapter on System Software, are not included here. If other commands are used, the CNC-7 will display a message when you try to run the part program. Most of the commands between the M48 and M95 or % commands may be arranged in any order, but there are some common sense exceptions. For example, the INCH/METRIC command must be specified before any commands with dimensions.

COMMAND	DESCRIPTION
AFS	Automatic Feeds and Speeds
ATC	Automatic Tool Change
BLKD	Delete all Blocks starting with a slash (/)
CCW	Clockwise or Counterclockwise Routing
CP	Cutter Compensation
DETECT	Broken Tool Detection
DN	Down Limit Set
DTMDIST	Maximum Rout Distance Before Toolchange
EXDA	Extended Drill Area
FMAT	Format 1 or 2
FSB	Turns the Feed/Speed Buttons off
HPCK	Home Pulse Check
ICI	Incremental Input of Part Program Coordinates
INCH	Measure Everything in Inches
METRIC	Measure Everything in Metric
M48	Beginning of Part Program Header
M95	End of Header
NCSL	NC Slope Enable/Disable
OM48	Override Part Program Header
OSTOP	Optional Stop Switch
OTCLMP	Override Table Clamp
PCKPARAM	Set up pecking tool,depth,infeed and retract parameters
PF	Floating Pressure Foot Switch
PPR	Programmable Plunge Rate Enable
PVS	Pre-vacuum Shut-off Switch
R,C	Reset Clocks
R,CP	Reset Program Clocks
R,CR	Reset Run Clocks
R,D	Reset All Cutter Distances
R,H	Reset All Hit Counters
R,T	Reset Tool Data
SBK	Single Block Mode Switch
SG	Spindle Group Mode
SIXM	Input From External Source
T	Tool Information
TCST	Tool Change Stop
UP	Upper Limit Set
VER	Selection of X and Y Axis Version
Z	Zero Set
ZA	Auxiliary Zero
ZC	Zero Correction
ZS	Zero Preset
Z+# or Z-#	Set Depth Offset
%	Rewind Stop
#/#/#	Link Tool for Automatic Tool Change
/	Clear Tool Linking

Duplicate Commands

If you have a command in the header and the exact same command in the part program body, there is no harm done. Nor will it matter if you enter the exact same command from the keyboard. In each case, because the commands do not contradict each other, the performance of the machine will not be affected.

Keyboard and Header Commands vs. Body Commands

Some commands allow you to specify optional information. When the options in the part program body are different from the options in the header or console, the body options are not used. Suppose you specify in the header which spindle speed you want for a particular tool. Then you repeat the tool command in the part program body and specify a different speed. The speed in the header will override the speed in the body. You could change the speed ten times in the program, but the spindle will rotate at the speed you specified in the header, each and every time.

Keyboard vs. Header Commands

Commands entered by you at the keyboard will also override duplicate commands in the part program body. Keyboard entered commands and header commands have the same authority, and they can conflict with each other. But system software uses the latest one entered as the governing authority. After a part program has been loaded, any commands entered at the keyboard will override the same command in the header. But if the command is entered at the keyboard, and then the part program is loaded, the header overrides the keyboard.

Beyond The Header: The Part Program Body

COMMAND	DESCRIPTION
A#	Arc Radius
B#	Retract Rate
C#	Tool Diameter
F#	Table Feed Rate;Z Axis Infeed Rate
G00X#Y#	Route Mode
G01	Linear (Straight Line) Mode
G02	Circular CW Mode
G03	Circular CCW Mode
G04	X# Variable Dwell
G05	Drill Mode
G07	Override current tool feed or speed
G32X#Y#A#	Routed Circle Canned Cycle
CW G33X#Y#A#	Routed Circle Canned Cycle
CCW G34,#(,#)	Select Vision Tool
G35(X#Y#)	Single Point Vision Offset (Relative to Work Zero)
G36(X#Y#)	Multipoint Vision Translation (Relative to Work Zero)
G37	Cancel Vision Translation or Offset (From G35 or G36)
G38(X#Y#)	Vision Corrected Single Hole Drilling (Relative to Work Zero)
G39(X#Y#)	Vision System Autocalibration
G40	Cutter Compensation Off
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G45(X#Y#)	Single Point Vision Offset (Relative to G35 or G36)
G46(X#Y#)	Multipoint Vision Translation (Relative to G35 or G36)
G47	Cancel Vision Translation or Offset (From G45 or G46)
G48(X#Y#)	Vision Corrected Single Hole Drilling (Relative to G35 or G36)
G82(G81)	Dual In Line Package

G83	Eight Pin L Pack
G84	Circle
G85	Slot
G87	Routed Step Slot Canned Cycle
G90	Absolute Mode
G91	Incremental Input Mode
G93X#Y#	Zero Set
H#	Maximum hit count
I#J#	Arc Center Offset
M00(X#Y#)	End of Program - No Rewind
M01	End of Pattern
M02X#Y#	Repeat Pattern Offset
M06(X#Y#)	Optional Stop
M08	End of Step and Repeat
M09(X#Y#)	Stop for Inspection
M14	Z Axis Route Position With Depth Controlled Contouring
M15	Z Axis Route Position
M16	Retract With Clamping
M17	Retract Without Clamping
M18	Command tool tip check
M25	Beginning of Pattern
M30(X#Y#)	End of Program Rewind
M45,long message\	Long Operator message on multiple\ part program lines
M47,text	Operator Message
M50,#	Vision Step and Repeat Pattern Start
M51,#	Vision Step and Repeat Rewind
M52(#)	Vision Step and Repeat Offset Counter Control
M02XYM70	Swap Axes
M60	Reference Scaling enable
M61	Reference Scaling disable
M62	Turn on peck drilling
M63	Turn off peck drilling
M71	Metric Measuring Mode
M72	Inch Measuring Mode
M02XYM80	Mirror Image X Axis
M02XYM90	Mirror Image Y Axis
M97,text	Canned Text
M98,text	Canned Text
M99,subprogram	User Defined Stored Pattern
P#X#(Y#)	Repeat Stored Pattern
R#M02X#Y#	Repeat Pattern (S&R)
R#(X#Y#)	Repeat Hole

S#	Spindle RPM
T#	Tool Selection; Cutter Index
Z+# or Z-#	Depth Offset
%	Beginning of Pattern (see M25 command)
/	Block Delete

List of Equivalent Format One Commands

FORMAT TWO COMMAND	EQUIVALENT FORMAT ONE COMMAND
G05	G81
M00	M02
M01	M24
M02	M26
M06	M01
M08	M27
M09	M00
M02X#Y#M70	M26X#Y#M23
M72	M70
M02X#Y#M80	M26X#Y#M21
M02X#Y#M90	M26X#Y#M22
R#M02	R#M26

X and Y Coordinates

The location on the PC board where a hole is to be drilled or a router begins or ends a move is called a coordinate. A coordinate is a pair of measurements used to locate that point. It is measured along an axis which runs from the front to the back of the machine, and an axis which runs from left to right. These axes are perpendicular to each other and are known as the X and Y axis. When the machine is not in the routing mode, the coordinate is also the command for a drill bit to plunge into the panel and drill a hole. The coordinate tells the CNC-7 to move the spindle to the location and drill. There are two ways to move from coordinate to coordinate and you must choose one of them when you are programming. The two ways are absolute and incremental. Absolute means that every coordinate is measured to the same location on the board. This location is called work zero. Incremental means that every coordinate is measured to the previous coordinate. Unless you specify otherwise, the CNC-7 runs in the absolute mode, and part programs must be programmed for absolute. When you program in the incremental mode, include the ICI,ON command in the part program header, or in the MACH.DAT file. The following illustrates how a set of holes are programmed in either absolute or incremental mode. Note that when either the X or Y coordinate does not change from one hole to another, it does not have to be repeated.

ABSOLUTE	INCREMENTAL
XY	XY
Y01	Y01
Y02	Y01
X012Y032	X012Y012
X024Y044	X012Y012
X034	X01

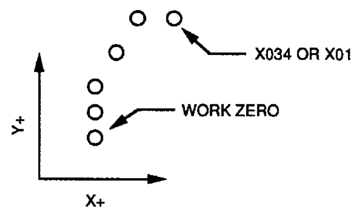


Figure. Programming Absolute or Incremental Coordinates

Inch vs. Metric

Coordinates are measured either in inch or metric (millimeters). Inch coordinates are in six digits (00.0000) with increments as small as 0.0001 (1/10,000). Metric coordinates can be measured in microns (thousandths of a millimeter) in one of the following three ways:

Five digit 10 micron resolution (000.00)
Six digit 10 micron resolution (0000.00)
Six digit micron resolution (000.000)

You specify the coordinate measurement you want by using the METRIC or INCH command in the program header. When the program is running on the machine, all X and Y coordinates will be displayed on the screen in the form you have chosen. Additionally, all other measurements will be displayed in this form, including the following:

Feed Rate
Tool Diameter
Spindle Upper and Lower Limit
Rout Depth
Spindle Retract Rate
All Zero Locations
Depth Offset
Routing Distance

Leading and Trailing Zeros

When you type coordinates into the CNC-7, it is important that you understand leading and trailing zeros. The previous section explains that the CNC-7 uses inches in six digits and metric in five or six digits. The zeros to the left of the coordinate are called leading zeros (LZ). The zeros to right of the coordinate are called trailing zeros (TZ). The CNC-7 uses leading zeros unless you specify otherwise through a part program or the console. You can do so with the INCH/METRIC command discussed later in this chapter. If you don't specify leading or trailing zeros, the CNC-7 will automatically use the last setting. With leading zeros, when you type in a coordinate, the leading zeros must always be included. If you don't, the CNC-7 will misinterpret the coordinate and move to the wrong location on PC board. Trailing zeros are unneeded and may be left off. The CNC-7 will automatically add them. This allows you to save time in typing the coordinates. If you have selected trailing zeros, the reverse of the above is true. You must show all zeros to the right of the number and can omit all zeros to the left of the number. The CNC-7 will count the number of digits you typed and automatically fill in the missing zeros.
Here are some examples of using the leading zero inch mode:

X0075	Correct
X007500	Incorrect, the two trailing zeros are unnecessary
Y014	Correct
Y014000	Incorrect, the three trailing zeros are unnecessary

Here are some examples of using the trailing zero inch mode:

X7500 = 0.75 inch	
X75 = 0.0075 inch	

The rules for typing leading and trailing zeros for other commands are discussed under each command.

Decimal Places

Decimals are not needed in either INCH or METRIC modes. But if you do use them, the decimal point will automatically override leading zero or trailing zero mode. Coordinates can be typed with or without the decimal. If you use the decimal and the coordinate distance is less than one inch or one centimeter, you can eliminate the zeros to the left of the decimal. For example, in the INCH format:

X.075	Correct
X00.075	Incorrect, the two zeros are unnecessary

The same applies to the METRIC format with three and four zeros to the left of the decimal. But in either case, if you have a whole number to the left of the decimal, it must be included. For example:

Y1.45	Correct
Y0001.45	Incorrect, the three zeros are unnecessary

If you choose to type coordinates without the decimal, all zeros to the left of the decimal must be shown. For example:

X00093 = 0.093 inch in inch format
Y00093 = 93 micron in metric format 000.00

Tool Commands

There are several commands used to select and control tools. Some are used separately and others are combined to form a single command. Whenever tool commands are used in the header, they are strictly for loading tool data into the CNC-7. When tool commands are intended for tool changing or for machine movements, they must be in the body of the program. The # in each command indicates that a number is to be used to designate quantity, distance, speed, etc. From one to six digits are used, depending on the command. The number of the tool specified with the tool command is the same as the tool number on the Tool Data Page.

Tool Commands

Tool Selection

T#

T# is used to specify which tool is to be used next in the manual or automatic tool change mode. It may be used in the part program header or body, or an M02 block step and repeat patterns. On machines with automatic tool change, the spindle will put away the tool it is using, pick up the tool number you specify in the place of #, and move to the next coordinate in the part program. On machines with manual tool change, the worktable will move to the part position and stop. The screen will display the message in the Machine Status box. After changing the tool, you press the CYCLE START button and the machine resumes operation. Tool numbers 1 through 9 may be specified with or without a leading zero. (e.g. 01 or 1)

Examples of usage:

T1	Tool number one
T01	Tool number one
T10	Tool number ten

Tool Selection with Compensation Index

T#(#) is used to select a specific tool and to set the Compensation Index for that tool. This command allows you to specify four digits. The last two are for the index number. If you omit the last two digits, or specify zeros, the index will be set equal to the tool number in the first two digits.
Compensation value is used in routing operations. Routing tools can bend and deflect away from the work, especially when moved in the

counterclockwise direction. The Compensation value offsets the path of the tool to compensate for the size and deflection of the tool. For example, a tool of 0.092" diameter might be specified for a clockwise direction. In the counterclockwise direction, however, you might need to use a diameter of 0.094". But you may not have such a diameter, or it may not be possible or practical to switch tools. Instead, you can assign an index number for a tool with a diameter of 0.094" (Refer to the CP,#,#.# command in the Keyboard Commands chapter). When you identify the index number with your 0.092" diameter routing tool, the CNC-7 will offset the path of the tool as though it were 0.094" diameter.
The Compensation Index value must be entered before the rout mode is turned on (G00 command), and may not be changed during routing moves.

Example of usage:

T0302	Tool number 3 with Compensation Index 2

See also: CP,#,#.#

Z-Axis Infeed

F#

F# is used within a routing sequence to set the worktable feed rate, or in a drilling sequence to set the spindle (Z-axis) infeed rate. Feed rate values are always entered in leading zero format, e.g.: F2 means 200 inches per minute, and F02 means 20 inches per minute. The value you assign in place of #, indicates inches per minute (IPM) or millimeters per second (mm/s). Decimals are not to be used with this command. They will produce a message when the part program runs on the machine. Drilling feed rates must be given to the CNC-7 or the machine will not run. The rate may be specified in the Tool Data Page, or through the F# command. The F# command may also be entered at the Tool Data Page to change the infeed rate for a particular tool.
The drilling feed rate can be set from 10 to 500 IPM (4 to 212 mm/s), in increments of 1 IPM (1mm/s). The routing table feed rate can be set from 10 to 150 IPM (4 to 63 mm/s), in increments of 1 IPM (1 mm/s). If you do not set a feed rate, the CNC-7 will use a maximum rate of 100 IPM for any router.

Examples of usage:

T01F2	Tool number one with a spindle infeed rate of 200 IPM or 200 mm/s
F07	Worktable feed rate of 70 IPM or 70 mm/s for routing
F03	Worktable feed rate of 30 IPM or 30 mm/s for routing

Retract Rate

B#

B# is used to set the spindle (Z-axis) retract rate, e.g., the speed at which the tool is withdrawn from the work. Retract values are always entered in leading zero format, e.g.: B02 means 200 inches per minute, and B002 means 20 inches per minute. The value you assign in place of # indicates inches per minute (in/min) of millimeters per second (mm/s). Decimals are not to be used with this command. They will produce a message when the part program runs on the machine. The B# command may also be entered at the Tool Data Page to change the retract rate for a particular tool. A default retract rate is established when the CNC-7 is started. If NO B# command is specified for a tool, the default retract rate will be used. The default rate may be changed using the RTR keyboard command. The retract rate can be set from 10 to 1000 IPM (5 to 425 mm/s), in increments of 1 IPM (1 mm/s). Unless altered by the RTR command, the default retract rate is 1000 in/min (425 mm/s).

Example of usage:

T01B02	Tool number one with a spindle retract rate of 200 IPM or 200 mm/s.

See also: RTR

Spindle RPM

S#

S# Sets the speed of spindle rotation. The value you assign in place of # indicates RPM in thousands. Trailing zeros are not shown. The S# command may also be entered at the Tool Data Page to change the rate for a particular tool. The spindle speed on most machines may be programmed from a minimum of 14,000 RPM to a maximum of 60,000 RPM for routers and 80,000 RPM for drilling tools. Some machines have spindles speeds greater than 100,000 RPM. When you specify a speed of six digits on these machines, use a decimal point, followed by a number to indicate hundreds of RPM's. This command may not be used by itself, but must be included in a tool selection block (T#S#).

Examples of usage:

T01S612	Tool number one with a speed of 61,200 RPM
T06F200S61	Tool number six with a feed rate of 200 IPM or 20 mm/s and a speed of 61,000 RPM

T03S6	Tool number three with a speed of 60,000 RPM
T04S110.5	Tool number four with a speed of 110,500 RPM

Override Current Tool Feed OR Speed

G07

When G07 is used inside the part program, the tool feed or speed can be changed after G07 command. It only affects the current part program.

Tool Diameter

C#

C# is used to select the tool diameter necessary for certain machine canned cycles. When feed and speeds are not specified with Tool Diameter, the CNC-7 will load them from the tool diameter table if a tool diameter table has been loaded. The value you specify in place of # indicates the diameter in thousandths of an inch, or microns, depending on which measurement mode the machine is set for. Trailing zeros are not shown. The C# command may also be entered at the Tool Data Page to change the diameter of a particular tool. This command should not be used by itself but must be included in a tool selection command block (T#C#).

Examples of usage:

T1C.04	Set Tool number one to .040" diameter (with feed and speed from the tool diameter page).
T1C.04F200S65	Set Tool number one to .040" diameter with an infeed rate of 200 and spindle speed of 65,000 RPM.

See also: Canned Cycle Commands

Set Maximum Hit Count

H#

H# is used to make sure that only sharp drill bits are used to drill holes. You set the maximum number of times that a drill tool may drill a hole (hit) by specifying a number in place of #. Hit counters keep track of the number of times each tool bit drills a hole. When the counter equals the maximum set by this command, the tool bit is considered to be expired, and the machine stops drilling. If other tools are linked to the expired tool, the machine will automatically change tools and continue drilling. Otherwise, the worktable will move to the park position and stop. The H# command may also be entered at the Tool Data Page to change the maximum number of hits for a particular tool. This command should not be used by itself, but must be included in a tool selection command block (T#H#). Leading and trailing zeros do not apply and decimals are not allowed. This command can also be used to turn off a hit counter so that the drill bit continues drilling. Type the H by itself without a number and the hit counter for that tool will be turned off.

Examples of usage:

T03H2000	Tool number three set at 2,000 hits maximum
T01H	Tool number one maximum hit counter is turned off

Depth Offset

Z+# or Z-#

Z+# (or Z-#) Sets the Depth Offset for tools. This command is used in conjunction with T# command. Depth Offset may be programmed for each logical tool. A mean depth, common to all tools, can be supplied through the part program header, or by you through the keyboard, or through the LOWER LIMIT or ROUT DEPTH switches on the Touch Screen. The Depth Offset is programmed as a deviation or offset from the mean depth. You supply the offset in place of #.
The offset value will be in inch or metric, LZ or TZ, depending on how the machine is set. The offset can be supplied in increments of 0.001" (0.01mm). Decimal mode may be used. Plus signs (+) may be omitted, but minus signs (-) must be used to indicate negative values. A positive value offsets the depth of the tool above the mean depth set by you or the part program header. A negative value represents a distance below the mean depth. Depth Offset permits control of drill penetration depth into the backup material. A large tool Depth Offset, requires a greater penetration depth than does an intermediate size tool, or a small tool. Accurate penetration depth is

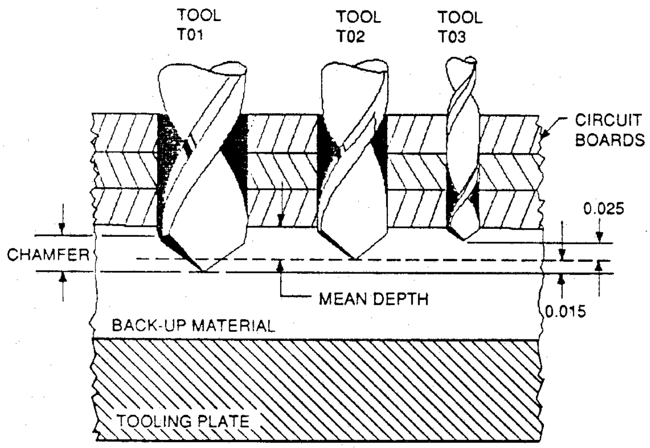
necessary to ensure that the tool chamfer clears the back of the last circuit board in the stack being drilled. The mean depth, plus the programmed Depth Offset, gives you the actual depth for that tool. The resulting actual depth must not be less than zero because this represents the lower limit of Z-axis (spindle) travel. A minimum Z-axis stroke length must be maintained. Therefore, the actual depth must be at least 0.125" (3.18mm) lower than the Upper Limit set.
The Z# command may also be entered at the Tool Data Page to change the depth offset for a particular tool. Depth Offsets may be included with preprogrammed infeed and speed information through the keyboard or a part program header. Offsets can also be stored on the Diameter Page. The Depth Offset may also be included in a part program as part of an integral feed and speed block.

Examples of usage:

T01C00125Z-00001	Sets drill penetration depth for tool 01 to 0.001" below the mean depth
T02C0009Z	Sets drill penetration depth for tool 02 to the mean depth
T03C00008Z00002	Sets drill penetration depth for tool 03 to 0.002" above the mean depth

Examples of usage:

T01C00125Z-00001	Sets drill penetration depth for tool 01 to 0.001" below the mean depth.
T02C0009Z	Sets drill penetration depth for tool 02 to the mean depth.
T03C00008Z00002	Sets drill penetration depth for tool 03 to 0.002" above the mean depth.



Link Tools for Automatic Tool Changers

#/#/#

#/#/# links tools together so that when one tool expires (too dull to drill anymore), the machine will automatically change tools and continue drilling. Naturally, all the tools linked together must be the same size. You select the tools to be linked by specifying a tool number in place of #. You may link as many of the same size tools together as you need. When the CNC-7 reads this command in your part program, it will update the Tool Data Page to show which tools are linked together.
Tools will be used in sequence from left to right, as you specify in the command.
The tool linking command may also be entered at the Tool Data Page to change the linking arrangement. Tool linking does not apply to the Tool Management System (TMS). The maximum hit counter tells the CNC-7 when it is time to replace a worn-out tool, and tool linking tells the CNC-7 which tool is to be used next. Tool linking is used in conjunction with Automatic Tool Change (ATC). When ATC is OFF, the CNC-7 will PARK the worktable and instruct you to replace the tool in the collet. If ATC is ON, but tool linking is disabled, the machine will put the tool away and request a replacement.
Example of usage:

1/5/6	Link tools number one, five, and six.

Clearing Tool Linking

A slash, all by itself in a block, will clear any previous tool linking performed by the Tool Linking command described above. When the CNC-7 reads this command in your part program, it will clear the links, which are displayed on the Tool data page if your machine is equipped with an ATC Toolchanger.

Hierarchy of Tool Commands

When several tool commands are combined into one, the order of their appearance in the combined command can be very important. The CNC-7 reads the command from left to right. The commands on the left can be overridden by the commands to the right.
For example, look at the following two sample commands:

T01F190S73C.038	T01C.038F190S73

Both commands contain the same information, but in a different order. In the first example, the CNC-7 selects tool 01, sets the feed at 190 IPM, sets the spindle speed at 73,000 RPM, and then is told that the diameter of the bit is 0.0038". The CNC-7 will now look at the Tool Diameter Page and use the feed and speed listed, if any, in the table. It may ignore the feed and speed you specified in the command. In the second sample, the opposite is true. The CNC-7 selects tool 01, looks in the Tool Diameter Page for a drill bit of 0.0038" diameter, then sets the feed at 190 IPM and the speed at 73,000 RPM. The feed and speed in the Diameter Page will be ignored.

Tool Changing

If you have only manual tool changing on your machine, then you must specify in the part program when you want to change the tool. If you have automatic tool change on your machine, you need to specify not only when to change tools, but which tool the spindle is to pick up. Changing a tool is a simple matter. When you get to the point in the program where the tool is to be changed, just type in a tool command and specify which tool is to be used for the next operation. Nothing has to be said about the tool that you are dropping. If you need to have a special RPM or infeed rate used with the tool, include it with the tool command.

Drilling and Routing Commands

When you switch from a drill bit to a router, or vice versa, the CNC-7 needs to know what mode it is in: drilling or routing. This is done with the G00 or G05 commands, which are described later in this chapter. As soon as the CNC-7 encounters one of these commands in the part program, it knows which mode it is in. Several other commands will also tell the CNC-7 whether it is in drilling or routing mode. These are the canned cycles commands which are described in the next section.

Rout Mode

G00

G00 turns the routing mode on and the drilling mode off. This command is required before any routing can be performed. An X and Y coordinate must be provided to move the worktable to a starting point for routing. When the CNC-7 encounters this command, the worktable moves to the X,Y coordinate. The spindles will not plunge into the work until a plunge command (e.g. M15) is given. Compensation is automatically turned off during the move and can be turned on again after the move. The G00 command remains in effect until another G00 command, or a G01, G02, G03, or G05 command is encountered. Do not use this command when the Z-axis is in the rout position. The tool can be damaged by a high speed move.

Format: G00X#Y#

Drill Mode

G05

G05 turns the routing mode off and returns to the default drill mode. This command is programmed in a block by itself and remains in effect until a G00 is encountered. G05 is not needed if routing has not been turned on by any rout command in the part program. Any coordinates following the G05 command will cause the worktable to move at maximum velocity to the command position and perform a drill stroke. The spindles will start to rotate above the tool holders with Automatic Tool Change (ATC) ON, and at the Drill Ready position with ATC OFF.
Special note: The G81 command, when used in Format 1, is equivalent to the G05 command. The G81 command, when used in Format 2, becomes equivalent to the G82 command.

Routing Commands

Excellon has developed a series of fourteen commands which are used strictly for routing. Each of these commands are presented here.

Linear Move

G01

G01 turns on linear interpolation mode. This means that the machine will begin routing in a straight line. If you supply an X and/or Y coordinate with the command, the machine will rout a straight line from the current position to the coordinate position. If you do not supply coordinates, the CNC-7 will look for coordinates in a succeeding block, and rout to the first coordinate found. Unless a different rate has been set, linear movement will occur at a default rate of 100 IPM (42.3 mm/s) at 100% feed rate. This can be overridden with the F# command, described in the Tool Commands section of this chapter, or with the FEED RATE buttons on the Touch Screen.

Format: G01(X#)(Y#)

Circular Clockwise Move

G02

G02 turns on circular interpolation mode and sets clockwise direction of travel. If you supply an X and/or Y coordinate with the command, the worktable will move to that coordinate position. The move will be made along an arc in a clockwise direction at a controlled velocity. If you do not supply coordinates, the CNC-7 will look for coordinates in a succeeding block, and rout to the first coordinate found. The arc must be equal to or less than 180 degrees. The arc radius or the arc center offset is specified either by the A# command or the I#J# command. These commands are indicated as optional. If they are not included in the G02 command, they must be included in a previous block of the program, either alone or with another routing command. The A# and I#J# commands are discussed in the next sections. Unless a different rate has been set, movement will occur at a default rate of 100 IPM (42.3 mm/s) at 100% feed rate. This can be overridden with the F# command, described in the Tool Commands section of this chapter, or with the FEED RATE switches on the Touch Screen.

Examples of usage (these are three separate examples):

EXAMPLE NUMBER	COMMAND	DESCRIPTION
1	G02X0245Y021A00075	Sets radius to 0.075"
2	G02X0245Y021A00075 X025567Y020567	Sets radius to 0.075" Circular clockwise move with 0.075" radius
3	G02X0245Y021A00075X025567Y020567 X0246Y0154A0015	Sets radius to 0.075"Circular clockwise move with 0.075" radius Circular clockwise move with 0.15" radius

Format: G02(X#)(Y#)(A#) G02(X#)(Y#)(I#J#)

Circular Counterclockwise Move

G03

G03 turns on circular interpolation mode and sets counterclockwise direction of travel. If you supply an X and/or Y coordinate with the command, the worktable will move to that coordinate position. The move will be made along an arc in a counterclockwise direction at a controlled velocity. If you do not supply coordinates, the CNC-7 will look for coordinates in a succeeding block, and rout to the first coordinate found. The arc must be equal to or less than 180 degrees. The arc radius or the arc center offset is specified either by the A# command or the I#J# command. If they are not included in the G03 command, they must be included in a previous block of the program, either alone or with another routing command. The A# and I#J# commands are discussed in the next sections. Unless a different rate has been set, movement will occur at a default rate of 100 IPM (42.3 mm/s) at 100% feed rate. This can be overridden with the F# command, described in the Tool Commands section of this chapter, or with the FEED RATE switches on the Touch Screen.

EXAMPLE NUMBER	COMMAND	DESCRIPTION
1	G03X0245Y021A00075	Sets radius to 0.075"
2	G03X0245Y021A00075 X025567Y020567	Sets radius to 0.075" Circular counterclockwise move with 0.075" radius
3	G03X0245Y021A00075X025567Y020567 X0246Y0154A0015	Sets radius to 0.075"Circular counterclockwise move with 0.075" radius Circular counterclockwise move with 0.15" radius

Format: G03(X#)(Y#)(A#) G03(X#)(Y#)(I#J#)

Arc Radius

A#

A# Specifies the arc radius of a circular move. You specify a radius in place of #. The digits you supply will be in inch or in metric mode, however the system is set. The arc radius command is used in conjunction with the G02, G03, G32, or G33 commands discussed in this section. If the radius you specify does not fit the X,Y coordinates supplied with these commands, the CNC-7 will adjust the arc to fit the coordinates. The following figure shows how the arc is adjusted.

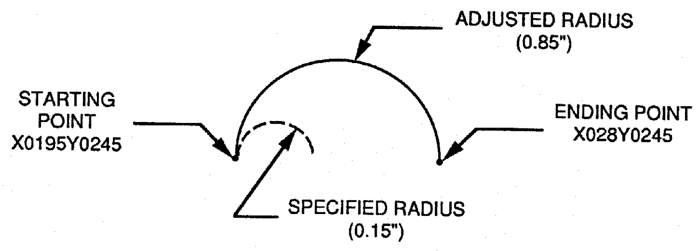


Figure. Adjusted Arc Radius

Arc Center Offset

I#J#

I#J# Specifies the distance from the arc center to the starting point of the arc to be routed. I# specifies the offset distance along the X axis and J# specifies the offset distance along the Y axis. I and J distances are measured from the arc center, not from work zero.

Routed Circle Canned Cycle CW or CCWG32

G33

G32 or G33 is used to rout out an inside circle. The G32 command routs in a clockwise (CW) direction, and the G33 command routs in a counterclockwise (CCW) direction. These commands provide automatic plunge, retract, and compensation with plunge and retract points off the circle to prevent gouging. You supply the coordinates for the center of the circle in place of X#Y#, and a radius in place of A#. A# may be omitted if the radius is the same as the previous rout move. The minimum radius size is one half of the Compensation Index value, plus 0.01" (0.254 mm). Anything less results in a message. A# can be omitted if the radius is the same as one specified several rout moves back, with no radius being specified in between. Note: Cutter compensation is always used. Commands G32 and G33 must be used for each inside circle to be cut. The pattern repeat code P cannot be used with these two commands. The G32 and G33 commands cause the machine to plunge 0.01 inch (0.254mm) off the edge of the circle, rout 540 degrees in the appropriate direction, end up 0.1 inch (2.54mm) off the edge of the opposite side of the circle, and retract. The Feed command (F#) may be entered in the block prior to the G32 or G33 commands to set the Table Feed Rate.

For example:

```
G00

F02

G32X04Y04A005
```

The G32 command will generate the following sequence internally:

```
G00X#Y#

M15

G02X1Y1Ac (where Ac = A# - one half compensation value)

Y2

Y1

XrYr

M17
```

Note: If a specific controlled table feed rate is desired, the G32 or G33 must be preceded with a G00 block containing the feed rate.

Format: G32X#Y#A# G33X#Y#A#

Cutter Compensation Off

G40

G40 turns cutter compensation off. This command is programmed in a block by itself. Cutter compensation is discussed with the G41 and G42 commands below, and with the Cutter Compensation Page.
THIS COMMAND MUST NOT BE USED WHILE PLUNGED!
Example of usage:
G40

See also: G41, G42, Cutter Compensation Page

Cutter Compensation Left

G41

G41 turns cutter compensation on for the tool being used to rout. The compensation path is left of the part relative to the direction that the tool is moving.
THIS COMMAND MUST NOT BE USED WHILE PLUNGED! A Compensation Index must be specified on the Cutter Information Page for the tool being used. Without an Index there will be no compensation applied to the tool after compensation is turned on. A value must be assigned to the index number. Compensation will continue for all routing moves until a G00, G40, or G42 command is encountered, or the part program ends. The command must be programmed in a block by itself.
Example of usage: G41

See also: G40, G42, Cutter Compensation Page, Compensation Index

---

Cutter Compensation Right

G42

G42 turns cutter compensation on for the tool being used to rout. The compensation path is right of the part relative to the direction that the tool is moving.
THIS COMMAND MUST NOT BE USED WHILE PLUNGED! A Compensation Index must be specified on the Cutter Information Page for the tool being used. Without an index there will be no compensation applied to the tool after compensation is turned on. A value must be assigned to the index number. Compensation will continue for all routing moves until a G00, G40, or G41 command is encountered, or the part program ends. This command must be programmed in a block by itself.
Example of usage: G42

See also: G00, G40, G41

Z Axis Rout Position with Depth Controlled Contouring

M14

M14 is provided for routing machines equipped with the optional router depth control scale. The M14 command performs the same function as the M15 command and also enables Depth Control Contouring. The command causes the spindle to plunge to the Rout Down position, the position from which rout moves are made. The vacuum is turned on and Depth Controlled Contouring is enabled. To perform Depth Controlled Contouring, depth control must be enabled and the tool must be declared as a depth controlled tool. A depth controlled plunge will be performed, where the machine senses the touchdown of the pressure foot to determine the proper depth. Throughout the cut, the height of the material is continuously monitored. The spindle height is adjusted automatically to maintain a constant depth into the material. Depth Controlled Contouring is turned off by G32/G33 and M15 commands, and at End of Program. With the exception of the G32 and G33 commands, a rout position command must be used before any rout moves are made. When a rout move is complete, the spindles are retracted, and the worktable moves to another rout position. A rout position command must then be used again before starting the rout move.

Z Axis Rout Position

M15

M15 causes the spindle to plunge to the Router Down position. This is the position from which rout moves are made. The vacuum is turned on and the spindle clamps are applied. For machines so equipped, if depth control is enabled and the tool is declared as a depth control tool, a depth controlled plunge will be performed. This is where the machine senses the touchdown of the pressure foot to determine the proper depth. This depth is then maintained for the duration of the cut. With the exception of the G32 and G33 commands, a rout position command must be used before any rout moves are made. When a rout move is complete, the spindles are retracted, and the worktable moves to another rout position. A rout position command must then be used again before starting the rout move.

Retract with Clamping

M16

M16 turns off vacuum, releases the spindle clamps, and causes the spindle to retract out of the Router Down position to the Upper Limit position.
The Floating Pressure Foot is activated 0.2" (5mm) before the end of the line segment preceding the M16 Retract command, unless the Pressure Foot Switch is off. The M16 command is programmed in a block by itself.

Retract without Clamping

M17

M17 turns off vacuum, releases the spindle clamps, and causes the spindle to retract out of the Router Down position to the Upper Limit position.
The Floating Pressure Foot is not activated by this command. The M17 command is programmed in a block by itself.

Canned Cycle Commands

Most PC boards have integrated circuits installed in them. These circuits use a pin pattern that is standard throughout the electronics industry. By using a simple command, you can type the coordinates of two pin holes and the CNC-7 will automatically drill the rest. This is called a "canned cycle". Excellon has supplied you with commands for drilling a large hole or a slot when you don't have a router to use. These are also canned cycles.

Excellon Supplied Stored Patterns

Excellon has built five canned cycles into system software for your use:
Dual In Line Package 8 Pin Circular L Package Drill a large hole with a drill bit Drill a slot with a drill bit
Each of these cycles is described below.

Dual In Line Package

G82

G82 Drills a dual in line integrated circuit hole pattern. The optional X and Y values in the first block of the command determine the space between the holes and the space between the rows of holes. The X value determines the spacing between the holes, and Y value determines the space between the two rows. If these parameters are omitted, the X and Y values will be 0.100 and 0.300 respectively. The next two blocks in the command contain X and Y coordinates specifying the two opposite corners of the desired pattern. The CNC-7 uses the two coordinates to determine the number of pins, and locations of the pattern, and the direction of the pattern. The G82 command drills a Dual In Line Package in both Format 1 and Format 2. The G81 and G82 commands both do the same thing in Format 2. The G81 command in Format 1 however, is equivalent to the G05 command.

Format 2	Format 1
G82/G81(X#Y#)	G82(X#Y#)
X#Y#	X#Y#
X#Y#	X#Y#

Note: Do not use the G82 command to program square packages. Since the G82 command is only given the two corners of the pattern, it will not know which side to put the pins on. You may also see different results from one machine type to another. Use the repeat hole commands to generate square patterns where necessary.

Eight Pin L Pack

G83

G83 Drills a circular eight pin package with pin spacing of 0.400 inch. You supply the coordinates for two opposite holes. They are on either the vertical or horizontal centerline of the pattern, whichever you choose.

Format: G83 X#Y# X#Y#

Canned Circle

G84

G84 Cuts a hole by drilling a set of overlapping holes around the circumference of a circle. The hole is programmed by specifying the center of the circle with an X and Y coordinate, followed by another G84 command, followed by an X dimension which specifies the diameter of the circle in thousandths of an inch or microns. This command must be in one block. It may not be broken up onto different lines (blocks) of the part program. The smallest hole diameter allowed is twice the tool diameter. If a smaller diameter is specified, the CNC-7 will display a message on the screen. The CNC-7 will use the drill size found in the tool diameter table to compensate the cutting radius. If the size is zero (not specified), a 0.125 diameter will be assumed by the CNC-7. Drilling overlapping holes around the circle creates a hole. This leaves protrusions around the edge of the hole. The holes are spaced close enough that these protrusions are less than 0.0005".

Format: X#Y#G84X#

Slot

G85

G85 Cuts a slot by drilling a series of closely spaced holes between two points. The start of the hole is programmed with an X and Y coordinate, followed by the command, followed by the ending X and Y coordinate. The tool is specified with a T command prior to the G85 command. The tool size MUST be specified prior to using this command. The size may be provided by the Operator through the console, in the part program body, or the part program (M48) header. The slot is as wide as the drill bit used. The slot is created by drilling a series of evenly spaced adjacent holes from one end of the slot to the other. This leaves protrusions around the edge of the hole. Then another set of spaced holes is drilled between the previous set. This continues until a smooth sided slot has been produced. The holes are spaced close enough that these protrusions are less than 0.0005".

Routed Step Slot Canned Cycle

The G87 code is used to rout a slot by making multiple passes. Each pass cuts deeper into the slot by a specified amount until the desired depth is reached.
The form of the G87 block is:

```
X1Y1G87X2Y2Z-#U#
```

Where:

X1Y1 - Start of slot X2Y2 - End of slot Z-# - Depth increment (must be a negative value) U# - Initial depth offset

The beginning and ending points (X1Y1, X2Y2) define the center of the slot at each end. Cutter compensation is NOT applied during step slot routing. The final depth of the slot must be specified according to the current depth mode, prior to the G87 block. This may be done either within or outside of the part program. G87 supports all depth modes, i.e. depth control and non-depth control routing. The initial depth (U code) is given as a positive offset above the final depth. The depth increment (Z code) is a negative value specifying the distance the cutter will plunge each pass through the slot. Note that the final plunge distance may be reduced in order to complete the slot at the proper depth. The G87 command internally generates the following program sequence:

```
G40
T#Z#
G00X1Y1
M15
G01
X2Y2
G00X2Y2
T#Z#
G00X2Y2
M15
G01X1Y1
```

G00X1Y1

. . . M17

Each M15 advances the cutter deeper into the slot until the desired depth is reached. Note that the spindle is not raised until the slot is completely routed.

Example of usage (Controlled penetration (Mode 3) routing):

T6Z-.05	Pick up tool 6 and set the rout depth at .05 inches into the backup
X05Y06G87X05Y07Z-.1U.2	Rout a 1 inch long slot (Y axis). The machine will rout the slot in 3 passes at the following depths: 1st pass: .15 inches above the backup 2nd pass: .05 inches above the backup 3rd pass: .05 inches into the backup

Note: The pattern repeat 'P' code cannot be used with this command.

Format: X#Y#G87X#Y#Z-#U#

See also: Setting up Depth Control

User Defined Stored Patterns

Most likely you will have many other patterns that you often use. Just imagine a board with 30 24-pin IC's. That's 720 holes to write coordinates for! It would be nice to have a simple command that would allow you to locate just one hole for each IC and let the CNC-7 figure out where the rest go. The M99 command is just such a command. It allows you to program patterns which you use frequently, store them on a device, and use them later in a part program. These are called user defined patterns. When you use the M99 command, you specify the name of the file containing the pattern, along with the XY coordinate of the first hole to be drilled. The CNC-7 copies the pattern from the file and drills the rest of the holes. User defined patterns are not just for drilling holes. They can be used to drill and/or rout, provide set-up information, and perform step-and-repeat patterns. Each of these is described in detail below.

Creating a Pattern

Patterns are created by using the Editor to type a set of X and Y coordinates. Drilling patterns locate the coordinates of each hole to be drilled. Routing patterns locate the coordinates of rout moves. Coordinates may be programmed in either absolute or incremental mode, the same as the part program. For example, if the part program is written in the incremental mode, the user defined pattern must also be incremental. It is also important to know which version of X and Y coordinates are used by the part program. Version refers to the direction of X and Y coordinates. The version of your user defined pattern must be the same as your part program. Otherwise the worktable will move in the wrong direction when it drills or routs the pattern. Once you have programmed the set of coordinates, store them as a file on the system software disk. The following figure shows how to program the coordinates for 10 pin pattern. This is a sample to illustrate the form of a pattern and how to program it. The coordinates are shown in both absolute mode with leading zeros and incremental mode with trailing zeros.

Naming Your Patterns

When you enter the Editor to program the coordinates of the pattern, you must first specify a name for your pattern. That name will be the name of the file on one of the system's devices. Along with the filename, specify the device to be used. You must specify which device you want the file stored on. Typically, you will want the M99 patterns stored with the non-replaceable data in USER (on the hard disk), or on the floppy disk containing the part program which calls the M99 file. Each file name must be different (unique), should relate to the purpose the file was created for, and should not be too long. Rules and guidelines for naming the files are covered later in this chapter.

Using a User Defined Stored Pattern

To use the User Defined Stored Pattern, type the M99 command, as described below. The CNC-7 will copy the pattern into memory from the device and drill or rout the rest of the pattern. This command requires two blocks in the part program. When the CNC-7 encounters an M99 command, it searches the device and copies the stored pattern file you identified with "name". Next the CNC-7 temporarily changes the work zero to the location specified by X#Y#. This temporary zero is dropped after M99 completion. The CNC-7 then carries out all the instructions in the pattern in the order they are presented. The X,Y coordinates in the pattern are relative to the X#Y# in the M99 command. The machine moves from one coordinate to another in succession. Upon completion of the last instruction in the pattern, the CNC-7 returns to the part program and continues with the next command. The M99 pattern may include any part program commands except the M99 command itself. This includes set-up information, such as tool feed and speed or any of the commands used in the part program body.

Format: M99,name X#Y#

Repeating Stored Patterns

Often you will need to repeat stored patterns. Earlier we presented a possible situation of a board with 30 24-pin chips. This situation lends itself to using the repeat pattern command saving time when writing programs.

Repeat Stored Pattern

P#

P tells the CNC-7 to repeat the previous Excellon supplied stored pattern. It can also be used to repeat M99 stored patterns. You specify the number of repeats (up to three digits) in place of # following the P. An X and/or Y coordinate must be used to define the spacing between the start of the patterns. These coordinates must be in the same block as the P; they may not be on a separate line.
Examples of usage:
P9X3.23 Repeat nine times, spaced by X3.23 P03X# Repeat three times P20X#Y# Repeat twenty times P200X# Repeat two hundred times The following figure illustrates how to use the repeat pattern command in a part program. This illustration uses the 10-pin pattern which we created earlier in this chapter.

Repeating a Hole

Some electrical components have so may variations of pin quantities that it would be highly impractical to create a user defined pattern for each one. As an alternative, the repeat hole command lets you locate the first pin hole and let the CNC-7 drill the rest without a stored pattern.

Repeat Hole

R#

This command drills a series of equally spaced holes from the previously specified hole. The number following the R (up to four digits) specifies the number of repeats. An X and/or Y coordinate must be used to define the spacing between hole centers. These coordinates must be in the same block as the R: they may not be on a separate line.
Examples of usage:

R9X001	Repeat nine times on X axis every 0.100"
R03Y1.5	Repeat three times on Y axis 1.5" apart
R20X00075Y00103	Repeat twenty times along a sloped line
R200X000075	Repeat two hundred times on X axis every 0.0075"
R4000X0009	Repeat four thousand times on X axis every 0.090"

This method may be easier than developing a stored pattern with 32 coordinates.

Format: R#X#(Y#)

Canned Text

M97

It is possible to drill a series of holes that spell out words or numbers. The M97 and M98 commands allow you to program the CNC-7 to write a message on the board. This feature can be used to:

Identify a company or product.
Supply a part number.
Identify the machine operator.
Date the board.

The machine will drill a series of holes to spell out the message you supply in place of text. M97 drills the text along the X+ axis and the M98 drills along the Y+ axis.
The characters you can use are:

A through Z
0 through 9 + - / *

Commas will be interpreted as spaces.
An asterisk (*) will be replaced by text which has been identified with the OPID keyboard command. The OPID command can identify up to 20 characters. If you have entered an asterisk as part of an M97/M98 command, and either OPID,OFF has been entered, or no OPID

command has been entered, the asterisk will be ignored. Any text after the asterisk in the M97/M98 command will be moved to the right to close up the gap left by the asterisk.
Both commands will start drilling at the X,Y coordinate which follows the command. If no tool diameter is specified in the Tool Page, the CNC-7 will use the default letter height of 0.25", and will drill the holes 0.0417" apart. If a diameter is specified, the holes that make up the characters will be spaced 1.2 diameters between hole centers. The characters are drilled on a 4x7 grid (4 columns in 7 rows).
Format: M97,text X#Y#
M98,text X#Y#

See also: OPID

Canned Text Offset

CAN_TEXT_OFF

It is possible to offset the original X, Y coordinate given by the M97 and M98 commands to shift the position of the text away from the edge clamps. This command will affect all M97/98 commands.

CAN_TEXT_OFF,#,#	The first parameter is X coordinate, and the second parameter is Y coordinate.
CAN_TEXT_OFF,#	Modify the X coordinate only.
CAN_TEXT_OFF	Show the current value of X, Y in machine status window.

Step and Repeat Commands

Step and Repeat means to drill or rout a pattern, move to another location and repeat the pattern. This feature is a great time saver for programmers. It can be used to repeat a large variety of patterns on the same board or to make several PC boards from one large panel. For example, let's say that you are making six boards out of one panel. You can load a tool and drill all the holes in one board for that size tool. Then step and repeat all the same size holes for the next five boards. Next, change the tool and return to the first board and repeat the procedure until all six boards are drilled with that tool. This procedure can be continued until the boards are completely finished. A Step and Repeat pattern begins with an M25 command and ends with an M01 command (M24 in Format 1). Two or more M02 commands are inserted after the pattern to identify how many repeats are to be made. The M02 commands also supply the coordinates where the repeats are to begin. If you have commands that you don't want repeated, such as screening holes and tool changes, you perform them before the M25 command. The number of things you can do between the M25 and M01 commands is almost limitless:

Repeat canned cycles
Repeat user defined patterns
Drill holes Rout Change tool
Drill and rout Repeat holes
Canned text
Turn things sideways or upside down

Mirror image Repeat entire PC boards!

Following are step and repeat commands with an explanation of how to use them.

Beginning of Pattern

M25

M25 indicates the beginning of the part program section which is to be repeated. These commands do not actually cause a repeat action by themselves, but work in conjunction with the M01 and M02 commands. The M25 and % commands are equivalent, and are programmed in a block by themselves.

End of Pattern

M01

M01 indicates the end of the part program section which is to be repeated. This command is programmed in a block by itself.

Format 2	Format 1
M01	M24

Repeat Pattern Offset

M02

The M02 command causes a repeat of all the commands between the M25 command and the M01 command. The M02 command is incremental. This means that the coordinate X#Y# is the distance from the beginning of the last pattern, not the distance from Work Zero. A separate M02 command is required for each repeat of the pattern of set of patterns. After the last M02 repeat command, an extra M02 command which requires no coordinates, must be added in a block by itself. This will clear a counter built into the system software. The M02 command must always occur after the M01 command discussed above, and before the M08 command discussed below.

Format 2	Format 1
M02X#Y#	M26X#Y#

End of Step and Repeat

M08

M08 indicates the end of all step and repeat commands. If all M02 commands have not been completed, the CNC-7 will return to the last start of pattern command and repeat. When all patterns have been completed, the program will continue on past the M08, finding either an end of program command or more program information. An M30 end of program command may be combined with this command, otherwise it is programmed in a block by itself.

Format 2	Format 1
M08	M27

Repeat Block

R#M02

R#M02 is used in place of the M02 command, discussed above, for a pattern which has the same X coordinate or the same Y coordinate as the previous pattern. It is useful when making a column of evenly spaced parts. The number following the R indicates the number of repetitions of the pattern. You specify the coordinate (X# or Y#) which changes. The X or Y coordinate which does not change can be left out of the command, at your option. The Repeat Block command may be used with the mirror image or swap axis commands which are discussed in the next section. The following figure illustrates the use of this command to produce a column of patterns with the same Y coordinate. The repeat pattern offset M02 command is also shown for comparison. They will both produce the same column of patterns.

Format 2	Format 1
R#M02X#Y#	R#M26X#Y#

See also: Mirror Image, Swap Axis

Swap AxisM70

Mirror Image and Swap Axis

You can make better use of PC board materials and reduce setup time by turning the axis of the boards by 180 degrees, or by reversing the axis to create a mirror image, or both. Excellon provides you with three commands which enable you to reverse and/or rotate the axis of a pattern, or an entire PC board. These commands are step and repeat commands and must be used in combination with the M25 and M01 commands, described earlier in this chapter. The swap rotates the pattern 90 degrees and makes a mirror image by changing the X axis to Y, and the Y axis to X. This command is used in a step and repeat offset block only, as shown.

Format 2	Format 1
M02X#Y#M70	M26X#Y#M23

Mirror Image X Axis

M80

M80 creates a mirror image of a pattern or group of patterns by reversing the sign of the X axis coordinates. All X+ coordinates will be changed to X-, and all X- coordinates will be changed to X+. The Y coordinates remain the same. This command is used in a step and repeat offset block only.

Format 2	Format 1
M02X#Y#M80	M26X#Y#M21

Mirror Image Y Axis

M90

M90 creates a mirror image of a pattern or group of patterns by reversing the sign of the Y axis coordinates. All Y+ coordinates will be changed to Y-, and all Y- coordinates will be changed to Y+. The X coordinates remain the same.
This command is used in a step and repeat offset block only.

Format 2	Format 1
M02X#Y#M90	M26X#Y#M22

Nested Step and Repeat Commands

Step and repeat programs may contain multiple step and repeat patterns, one within the other. When a pattern has been stepped and repeated, it creates two identical patterns. These two patterns can be stepped and repeated to form four patterns, etc. Or you can create three identical patterns and then step and repeat to form six. This procedure is known as nesting. An unlimited number of M01 commands can be used in any one step and repeat section. The following figure shows just how powerful a few commands can be when they are nested. Following the M25 command, a user defined stored pattern routs an L shaped hole, and an M01 commands marks the end of the pattern. Next an M02 offsets Work Zero by the distance X#Y#, and another M02 repeats any pattern between itself and the M25 command. Then an M01 ends the repeat. This produces a total of two patterns. A third M02 offsets Work Zero by the distance X#Y#, and a fourth M02 repeats any patterns between itself and the M25 command. There are now four patterns in this category, so four patterns are repeated. Then an M01 ends the repeat. This produces a total of eight patterns. An M08 ends the step and repeat.

Setup Commands

Setup commands speed set-up and reduce operator involvement when preparing your machine for a new job. As with all Excellon commands, parentheses () are used to indicate options. These commands must be used in the part program body. They cannot be used as keyboard commands.

The following table provides a list of each of the setup commands in the order they are detailed below.

COMMAND	DESCRIPTION
G90	Absolute Mode
G91	Incremental Input Mode
G93X#Y#	Zero Set M18 Command tool tip check
M45,text\	Long Operator message
M47,text	Operator Message
M60	Reference Scaling enable
M61	Reference Scaling disable
M62	Turn on peck drilling
M63	Turn off peck drilling
M71	Metric Measuring Mode
M72	Inch Measuring Mode M96 Select Spindle Group

Details of the Setup commands are described in the following sections.

Absolute Mode

G90

G90 Sets absolute measuring mode, which causes all coordinates to be referenced to work zero. G90 must be programmed in a block by itself.

Incremental Mode

G91

G91 Sets incremental mode, which causes all coordinates to be referenced to the last coordinate. This mode does not change Work Zero. The computer accumulates the coordinates into absolute dimensions, starting from Work Zero. The incremental accumulators are cleared at the end of a step-and-repeat pattern, the end of the program, or by a system reset. Clearing the accumulators sets them back to Work Zero. G91 is programmed in a block by itself.

Zero Set

G93

G93 Sets work zero relative to absolute zero. You supply a coordinate value in place of #. The CNC-7 adds the zero set coordinates to the zero correction and false zero to set up the new work zero (zero set + zero correction + false zero = work zero). The adding together of separate values allows the user to build part programs that will run on any Excellon machine, regardless of the tooling configuration.

Format: G93X#Y#

Operator Message

M47

M47 halts automatic operation of the machine and lights the red CYCLE STOP indicator light. The message you supply in place of text is displayed on the console screen, along with the M47 block. You may supply up to 20 numbers or letters for text. When the operator presses the CYCLE START button, the program will resume. This command can be used to identify a part program before the operator runs it.

Format: M47,text

See also: M45

Long Operator message

M45

M45 halts automatic operation of the machine and lights the red CYCLE STOP indicator light. The message you supply may consist of multiple lines of text, each (except the last) terminated by a backslash "\" character. A total of up to 78 characters per line may be supplied, and there is no practical limit on the length of the message. The first line which is NOT terminated by a backslash indicates the end of the M45 message. As the system encounters each M45 message in sequence, the machine will stop and bring up the display program (see TYP or HELP) to display the message. You can use the display program to page through VERY long messages. To continue operation, you must QUIT out of the display program, and press START to restart the machine. If the screen is in use when the M45 is encountered (e.g.: in the editor), the system will wait until you return to the displays before bringing up the display program. M45 messages may be displayed ONLY the first time through the program for setup purposes (e.g.: telling the operator which kinds of backup, entry material, etc to use) or they may be displayed each time the program is run (refer to the M45_REDISPLAY VSB command). If the M45 messages are displayed only the first time through, you must clear the program out of memory with an "I" or "SI" command in order to get the M45 messages to display again.
Format: M45,text\ text\ text

See also: M47, TYP

Reference Scaling enable

M60

M60 Enables Reference Scaling. Any drilled hole coordinates following this command will be adjusted per the Reference Scaling values entered by the operator and displayed on the Reference Scaling page. This command allows the part program to enable Reference

Scaling under part program control so that certain coordinates can be scaled, others not. This part program command is equivalent to entering the SCLR,ON keyboard command to enable Reference Scaling.
See also the description on setting up Reference Scaling.

Format: M60

Reference Scaling disable

M61

M61 Disables Reference Scaling. Any drilled hole coordinates following this command will NOT be adjusted per the Reference Scaling values entered by the operator and displayed on the Reference Scaling page. This command allows the part program to disable Reference Scaling under part program control so that certain coordinates can be drilled without Scaling. This part program command is equivalent to entering the SCLR,OFF keyboard command to disable Reference Scaling.
See also the description on setting up Reference Scaling.

Format: M61

Turn on peck drilling

M62

This command allows the part program to enable peck drilling under part program control. This part program command is equivalent to entering the PECK,ON keyboard command

Format: M62

Turn off peck drilling

M63

This command allows the part program to disable peck drilling under part program control. This part program command is equivalent to entering the PECK,OFF keyboard command

Format: M63

Tool tip checkProgrammed tool tip check

M18

M18 commands a tool tip check under part program control. This command allows you to request a tool tip check at any point in the program you desire. No M18 command is necessary to request the tool tip checks which are performed by the system when starting the machine or after a toolchange. These do not have to be programmed with an M18 command. The M18 is normally not needed for Depth Controlled drilling, but may be useful under certain special circumstances. For example:
1) When doing testing and you want to collect data about a number of tool tip checks without going through toolchanges.
2) When doing extremely critical depth controlled drilling where you are concerned about compensating for normally minimal factors, such as thermal spindle growth or drill wear.
M18 has no effect if Depth Control is not enabled, or if Depth Control is not on for the particular tool in question.

Format: M18

Metric Measuring Mode

M71

M71 Sets metric measuring mode. Any values following this command in the part program will be interpreted as millimeters, millimeters per second, or meters of cutting distance. This command does not translate inch values to metric; it merely assumes all values to be metric. M71 will use the digit format you last selected (000.000, 0000.00, 000.00) or, if you did not select any, the default format of 000.000. M71 must be programmed in a block by itself. It should be used only at the beginning of a part program, before the first hole is drilled or before the first rout plunge.

Inch Measuring Mode

M72

M72 Sets inch measuring mode. Any values following this command in the part program will be interpreted as inches, inches per second,

or feet of cutting distance. This command does not translate metric values to inch, it merely assumes all values to be inch. M72 must be programmed in a block by itself. It should be used only at the beginning of a part program, before the first hole is drilled or before the first rout plunge.
Format: Format 2 Format 1
M72 M70

Select Spindle Group

M96

M96 selects one of the spindle groups previously defined by the Spindle Group Assignment keyboard command (SG). M96,OFF will turn the spindle group mode off from within a part program. After an M96,OFF command is encountered, the operator can be prompted with a message (using the M47 command) to manually select the desired spindle group for further operations.

Format: M96,# or M96,OFF

Stop Commands

Stop commands are used to temporarily stop the running of the part program or to indicate the end. Many of these commands may contain X and Y coordinates which cause the worktable to position but not to drill. If the coordinates you specify in the commands exceed the worktable limits, the limits will override your coordinates.
NOTE: In each of these stop commands, the coordinate (which you provide) is relative to absolute zero, not work zero. Absolute zero is in the X and Y version (1 through 8) which is currently selected. As with all Excellon commands, parentheses () are used to indicate options. Each command is entered by pressing the RETURN key after typing the command. The following table provides a list of each of the setup commands, in the order they are detailed below.

List of Stop Commands

COMMAND	DESCRIPTION
G04X#	Variable Dwell
M09(X#Y#)	Stop for Inspection
M06(X#Y#)	Optional Stop
M00(X#Y#)	End of Program, No Rewind
M30(X#Y#)	End of Program, Rewind

Details of the Stop commands are described in the following sections.

Variable Dwell

G04

G04 Halts the machine for the time you specify in place of #. This command is used, for example, to cool a router bit after a long cut. The dwell time is interpreted as 1 millisecond per increment in the current coordinate measurement mode (inch or metric). The dwell time may be programmed from 1 to 10 seconds of 1 msec. If you program beyond these limits, or if you do not supply a value, the dwell time defaults to 10 seconds.
Format: G04X#
Example of usage:
G04X001 = 1 second

Stop for Inspection

M09

M09 Halts automatic operation of the machine and lights the red CYCLE STOP indicator light. The Machine Status box on the console screen will display the message:

If you supply a coordinate (X#Y#), the worktable will move to that position relative to absolute zero. If you do not provide a coordinate, the machine stops in its current location (it does not self park). Pressing the CYCLE START button will continue the program.
Format:

Format 2	Format 1
M09(X#Y#)	M00(X#Y#)

Optional Stop

M06

M06 this command is similar to the Stop for Inspection command, with the exception that the operator must turn the function on at the keyboard. When the operator types the OSTOP,ON command before the M06 command is encountered, the machine will stop for inspection. If OSTOP is OFF, the CNC-7 will ignore the M06 command. To continue the part program after an optional stop, the operator presses the CYCLE START switch.
Format:

Format 2	Format 1
M06(X#Y#)	M01(X#Y#)

End of Program, no rewind

M00

M00 indicates the end of the part program with no rewind of the paper tape. If you specify a coordinate (X#Y#), the worktable moves to that position relative to absolute zero. If no coordinate is supplied, the worktable will move to the park position.
Format:

Format 2	Format 1
M00(X#Y#)	M02(X#Y#)

See also: End of Program Command file

End of Program, rewind

M30

M30 indicates the end of the part program with rewind of the paper tape. The tape will rewind until it encounters an End of Rewind command (%). If none is found, the tape will rewind to the end of the tape. If you specify a coordinate (X#Y#), the worktable moves to that position relative to absolute zero. If no coordinate is supplied, the worktable will move to the park position.

See also: End of Program Command file

Binary Map Commands

M19 code inside the part program will allow the machine to drill binary map code. The characters after M19 will be treated as binary map code only when you specify the binary map input from part program. Otherwise it will be ignored. See also Binary Map Setup Commands.

Format: M19 or M19,######

Rules for Naming Your Programs

Part programs and user defined patterns are stored as files on floppy disks, hard disk directories, or on a remote file server (such as a DataWorkshop). You must give each file a name so that the CNC-7 can identify each file. When you display the list of files in the directory, the different file names will display in the file selection pop-up window. File names may be from 3 to 12 characters long on the hard disk, or up to 20 characters long on a ZOS format floppy disk. You may use the following characters, in any order:
A through Z 0 through 9 ? % - $! The characters space, tab, comma, quotes and backslash, as well as "&@(){}[]<>" are specifically prohibited. Forward slashes - "/" - usually will not work, since the operating system takes this character as a directory identifier.
Examples of file names:
IC32PIN Describes a 32-pin pattern for an IC WO6443 Work Order number

6443 PO11640 Purchase Order 11640

Sample Part Program

Following is a sample part program developed by Excellon. It was designed to demonstrate how part program commands work together to produce a PC board with a variety of drill patterns. The pattern shown below is produced from this part program.

1.	M48	Part Program Header
2.	R,C	Reset Clocks
3.	R,H	Reset Hits

4.	R,T	Reset Tool Information
5.	VER,1	Axis Version Select
6.	FMAT,2	Excellon Format Two
7.	INCH,LZ	Inch Mode, Leading Zero Format
8.	BLKD,OFF	Block Delete Switch
9.	SBK,OFF	Single Block Mode Switch
10.	SG,OFF	Spindle Group Select
11.	TCST,OFF	Tool Change Stop Switch
12.	ICI,OFF	Incremental Mode Switch
13.	OSTOP,OFF	Optional Stop Switch
14.	RSB,ON	Front Panel Reset Button Switch
15.	ATC,ON	Auto Tool Change Switch
16.	FSB,ON	Feed and Speed Button Switch
17.	T1C.0135F080S80B0500H1500	F=FeedS=SpeedB=RetractH=Max Hits
		Z=Z-Axis Depth Offset/Compensation
18.	T2C.032F070S70B0700H2000	
19.	T3C.043F160S53B1000H3000	
20.	T4C.052F132S44B1000H3000	
21.	T5C.062F109S36B1000H3000	
22.	T6C.070F099S33B1000H3000	
23.	T7C.125F040S20B1000Z-.010	
24.	T8C.250F020S20B1000Z-.020	
25.	%	End of Rewind Block
26.	M47,DRILL DEMO	Operator Message
27.	G05	Drill Mode
28.	M72	Inch Mode
29.	G93X0325Y015	Zero Set
30.	T1	Pick Up Tool One (339 Hits)
31.	M25	Beginning of Step & Repeat
32.	X042Y01	(Start Original Pattern)
33.	R9X0005	
34.	Y0105	
35.	R9X-0005	
36.	M01	End of Pattern <NESTED STEP AND REPEAT ARRAY(MAXIMUM OF 3 M01) >
37.	R2M02XY001	1st Array
38.	M01	End of Pattern
39.	R2M02X007Y	2nd Array
40.	M01	End of Pattern
41.	R2M02XY007	3rd Array
42.	M08	End of Step & Repeat
43.	G83	

44.	X003Y041	<CIRCULAR EIGHT PIN PATTERN>
45.	X007Y041	
46.	G83	
47.	X011Y041	
48.	X015Y041	
49.	T2	Pick Up Tool Two (1,322 Hits)
50.	X03Y05G85X03Y047	
51.	X034Y046G85X031Y046	
52.	X03Y042G85X03Y045	
53.	X026Y046G85X03Y045	<DRILLED SLOT PATTERN>
54.	G045X005	Variable Dwell (5 Seconds)
55.	X032Y047G85X033Y049	
56.	X031Y045G85X033Y043	
57.	X029Y049G85X027Y043	
58.	X027Y049G85X029Y047	
59.	M97,COMPLEX,PART,PROGRAM	
60.	X0115Y003	Reference Location
61.	M98,DEMONSTRATION,PANEL	<CANNED TEXT>
62.	X06Y012	Reference Location
63.	M97,EXCELLON,AUTOMATION	
64.	X0115Y052	Reference Location
65.	T3	Pick Up Tool Three (134 Hits)
66.	G82	
67.	X005Y033	16 Pin 0.1 x .3
68.	X012Y036	
69.	G82X.1Y.4	
70.	X02Y033	28 Pin 0.1 x .4
71.	X033Y036	
72.	G82X.2Y.5	
73.	X04Y033	16 Pin 0.2 x .5
74.	X054Y038	
75.	G82X.1Y.6	
76.	X04Y023	32 Pin 0.1 x .6
77.	X055Y029	
78.	G82	
79.	X037Y042	14 Pin 0.1 x .3
80.	X04Y048	
81.	P2X007	Stored Pattern Repeat
82.	T4	Pick Up Tool Four (104 Hits)
83.	M25	Beginning of Step and Repeat
84.	X01Y018	

85.	R8Y-001	Original Pattern
86.	R4X001	
87.	M01	End of Pattern
88.	M02X002Y002M70	Axis Swap
89.	M02X043M90M70	Mirror Image Y Axis Swap
90.	M02X002Y-002M80	Mirror Image X
91.	M02X-002Y038M80M90M70	Mirror Image X Y Axis Swap
92.	M02X002Y002M80M90	Mirror Image X Y
93.	M02X-047M90	Mirror Image Y
94.	M02X002Y-002M80M70	Mirror Image X Axis Swap
95.	M02	Offset Counter Control
96.	M08	End of Step and Repeat
97.	T5	Pick Up Tool Five (76 Hits)
98.	M99,LOGO	User Defined Stored Pattern
99.	X015Y014	Reference Location
100.	T6	Pick Up Tool Six (57 Hits)
101.	X0025Y01	Move to And Drill
102.	G91	
103.	Y001	
104.	Y001	
105.	Y001	< Stored Program Called LOGO Located On Disk Drive 2>
106.	Y001	
107.	Y001	
108.	X001	
109.	X001	
110.	X001 XY007	
111.	X001 R2Y-001	
112.	X001	Incremental Mode R6X001
113.	Y-001 R5Y-001	
114.	Y-001 R8X001	
115.	Y-001 R2Y001	
116.	Y-001 R6X-001	
117.	Y-001 R8Y001	
118.	X-001 R6X001	
119.	X-001 R2Y001	
120.	X-001 R8X-001	
121.	X-001 R5Y-001	
122.	X-001 R5X-001	
123.	G90	Absolute ModeX012Y007 R2X001 R2Y-001 R5X-001 R2X001Y001
124.	R5Y001 125. R5X001	REPEAT HOLE (One Axis Moves)

126.	R5Y-001	
127.	R4X-001	
128.	X016Y046	Move to and Drill
129.	R4X.1Y.1	
130.	R4X.1Y-.1	REPEAT HOLE (Two Axis Moves)
131.	R4X-.1Y-.1	
132.	R3X-.1Y.1	
133.	X03Y046	Move to and Drill
134.	T8	Pick Up Tool Eight (3 Hits)
135.	X005Y041	Move to and Drill
136.	X009Y046	Move to and Drill
137.	X013Y041	Move to and Drill
138.	T7	Pick Up Tool Seven (78 Hits)
139.	X009Y046G84X005	Drilled Circle Pattern
140.	XY	Move to and Drill Program Zero
141.	M30	End of Program, Rewind

Vision Assist Commands

Vision Assist commands are used with the Vision Assisted Drilling and/or Routing option. The additional commands described in this section allow to to do precise drilling relative to a set of visible landmarks. Several vision commands are provided, and allow you to locate reference landmarks and drill and/or rout patterns relative to them. Commands are provided for selecting vision tools, single point offsets, multipoint compensation, drilling a target, and Autocalibration. In addition, nested vision functions are provided so that you can use one set of landmarks to compensate the approximate panel locations, and nested functions to find precise targets within the panel. As with all Excellon commands, parentheses () are used to indicate options. These commands must be used in the part program body. They cannot be used as keyboard commands. The following table provides a list of each of the vision commands in the order they are detailed below.

COMMANDS	DESCRIPTION
G34,#(,#)	Select Vision Tool
G35(X#Y#)	Single Point Vision Offset (Relative to Work Zero)
G36(X#Y#)	Multipoint Vision Translation (Relative to Work Zero)
G37	Cancel Vision Translation or Offset (From G35 or G36)
G38(X#Y#)	Vision Corrected Single Hole Drilling (Relative to Work Zero)
G39(X#Y#)	Vision System Autocalibration
G45(X#Y	#) Single Point Vision Offset (Relative to G35 or G36)
G46(X#Y#)	Multipoint Vision Translation (Relative to G35 or G36)
G47	Cancel Vision Translation or Offset (From G45 or G46)
G48(X#Y#)	Vision Corrected Single Hole Drilling (Relative to G35 or G36)
M50,#	Vision Step and Repeat Pattern Start
M51,#	Vision Step and Repeat Rewind
M52(#)	Vision Step and Repeat Offset Counter Control

Details of the Vision commands are described in the following sections.

Select Vision Tool

G34,#,#

G34 Selects a vision tool for use. The number following the G34 is the number of the vision tool, which starts at one. If there are five parameters, parameters two through four are four vision tools used for inspecting a large target. In the case of more than one parameter, the last optional number is an offset used for a four point vision cycle using four consecutive vision tools. In the case of multiple tool use, the inspected center will be the average of the four results. This is useful when inspecting targets which are large relative to the field of view. G34 and its arguments are programmed in a block by themselves. This command may have one of three forms:
G34,v1 "v1" tool is used for inspection G34,v1,delta "v1" tool is used for four inspections at a spacing of "delta" around the target.
G34,v1,v2,v3,v4,delta Tools "v1" through "v4" are used to do four inspections at a spacing of "delta" around the target.

Single Point Offset

G35X#Y#

G35 is used to find a single point offset which is used to adjust a series of locations which follow. For example, you might want to drill a series of holes relative to the inspected location of a visible pad. The coordinate provided with the G35 is the location of the pad to be measured, and the amount of variance to the ACTUAL location of the pad will affect all coordinates that follow. Please note that G35 is relative to the current work zero, and is not affected by any current vision translation or offset. G35 and G36 cannot be active at the same time. G35 and its coordinate are programmed in a block by themselves.

Multipoint Vision Translation

G36X#Y#

G36 is used to adjust a series of locations relative to two or three alignment pads which can be seen by the Vision System. For example, you might want to align the drilled pattern to some alignment pads. Two or three G36 commands are used in series, depending on the setting of the ROTATE variable in the Vision Configuration file. If two G36 codes are used, the pattern is adjusted for offset and rotation only. If three G36 codes are used, the pattern is adjusted for offset, rotation, and stretch. The coordinate provided with the G36 is the location of the pad to be measured, and the amount of the variances are used together to translate all coordinates that follow. Please note that G36 is relative to the current work zero, and is not affected by any current vision translation or offset. G35 and G36 cannot be active at the same time. G36 and its coordinate are programmed in a block by themselves.

Cancel Vision Translation or Offset

G37

G37 is used to cancel a G35 or G36 adjustment which has been set up previous to this point in the part program. After this command has been used, all coordinates that follow are strictly relative to work zero, and are unaffected by any vision translation or offset. G37 is programmed in a block by itself.

Vision Corrected Single Hole Drilling

G38X#Y#

G38 is used to locate a target at the specified location and drill a hole in the middle of it. This is useful for hitting the center of a critical pad or other target. The coordinate provided with the G38 is the location of the pad to be drilled, and the amount of the variance is used to translate just this one hole. Please note that G38 is relative to the current work zero, and is not affected by any current vision translation or offset. G38 is not affected by, nor does it affect the current G35 or G36 translation or offset. G38 and its coordinate are programmed in a block by themselves.

Vision System Autocalibration

G39X#Y#

G39 is used to drill a series of holes used to calibrate the Spindle to Camera Offset. The number of holes to be drilled is specified by the COUPON setup in the Vision Configuration file. If entry material is being used, you will be instructed to install and remove the entry material at the appropriate places. This function is useful when it is desired to highly automate the use of the Vision System in a production environment, recalibrating the Vision System regularly, perhaps every board, to assure proper operation of the system.
The coordinate provided with the G39 is the location of the first calibration hole. All other variables are provided by the COUPON setup in the Vision Configuration file.
Please note that G39 is relative to the current work zero, and is not affected by any current vision translation or offset. G39 is not affected by, nor does it affect the current G35 or G36 translation or offset.
G39 and its coordinate are programmed in a block by themselves.

Nested Single Point Offset

G45X#Y#

G45 is used to find a single point offset which is used to adjust a series of locations which follow. For example, you might want to drill a series of holes relative to the inspected location of a visible pad. The coordinate provided with the G45 is the location of the pad to be measured, and the amount of variance to the ACTUAL location of the pad will affect all coordinates that follow. Please note that G45 is relative to the current G35 or G36 vision translation or offset. G45 and G46 cannot be active at the same time. G45 and its coordinate are programmed in a block by themselves.

Nested Multipoint Vision Translation

G46X#Y#

G46 is used to adjust a series of locations relative to two or three alignment pads which can be seen by the Vision System. For example, you might want to align the drilled pattern to some alignment pads. Two or three G46 commands are used in series, depending on the setting of the ROTATE variable in the Vision Configuration file. If two G46 codes are used, the pattern is adjusted for offset and rotation only. If three G46 codes are used, the pattern is adjusted for offset, rotation, and stretch. The coordinate provided with the G46 is the location of the pad to be measured, and the amount of the variances are used together to translate all coordinates that follow. Please note that G46 is relative to the current G35 or G36 vision translation or offset. G45 and G46 cannot be active at the same time. G46 and its coordinate are programmed in a block by themselves.

Cancel Nested Vision Translation or Offset

G47

G47 is used to cancel a G45 or G46 adjustment which has been set up previous to this point in the part program. After this command has been used, all coordinates that follow are relative to the original G35 or G36 vision translation or offset. It is important to understand that the G47 does not affect the G35 or G36, but simply cancels the G45 or G46 translation. G47 is programmed in a block by itself.

Nested Vision Corrected Single Hole Drilling

G48X#Y#

G48 is used to locate a target at the specified location and drill a hole in the middle of it. This is useful for hitting the center of a critical pad or other target. The coordinate provided with the G48 is the location of the pad to be drilled, and the amount of the variance is used to translate just this one hole. Please note that G48 is relative to the current G35 or G36 vision translation or offset. G48 is not affected by, nor does it affect the current G45 or G46 translation or offset. G48 and its coordinate are programmed in a block by themselves.

Vision Step and Repeat

Step and Repeat. Vision Commands

The vision step and repeat commands M50, M51 and M52 are a powerful set that help to optimize vision assisted drilling and routing.
By using these commands, all the patterns on a panel may be vision inspected first and the resulting corrections used over and over as many times as needed, usually after tool changes. For example, after inspection a drill tool may be used to drill all patterns with the regular step and repeat commands. Then the tool may be replaced and the holes of that diameter may be drilled on all patterns and so on. This procedure eliminates the need for extra tool changes which would be the case if these commands were not used.

Vision Step and Repeat Pattern Start

M50,#

M50 is used to indicate the beginning of a vision correction pattern which is expected to be repeated later on. The number that follows the command is a label or marker used to "name" that particular pattern so it may be referenced later with the command M51. This command is normally used in the part program right before an M25, so the machine will vision inspect the pads contained in the main pattern enclosed between M25 and M01, and then it will also inspect the equivalent targets contained in the stepped and repeated patterns marked by M02 creating a set of vision corrections for each of them all. As previously said, the main advantage of this command is that along with M51 and M52 allows the part-programmer to vision inspect a pattern and use the generated corrections any time later, for example after tool changes. Please note that M50 is not affected by, nor does it affect the current G36, G38, G45 or G46 translation or offset. However, it should be used in conjunction with G36 and/or G46. M50 and its label are programmed in a block by themselves.

Vision Step and Repeat Rewind

M51,#

M51 is used to rewind a vision correction "pointer" back to a previous pattern in order to use the vision corrections stored with the command M50. The number that follows the command is the same label or marker used to "name" that particular pattern with the command M51.
This command is normally used in the part program right before an M25, and after a tool change command T# so the machine will use the vision corrections for the main pattern enclosed between M25 and M01 and the subsequent step and repeat patterns marked by M02. M51 and its mandatory label are programmed in a block by themselves. Note that the label following M51 must be previously used with M50.

Vision Step and Repeat Offset Counter Control

M52[#]

M52 is used to advance the vision correction "pointer" to the next pad in a pattern that it is being corrected after an M51,#. Because G36 or G46 are used only in the main pattern with M50, any time the vision corrections for that pattern are used again, they must be replaced with M52s. There must be as many M52 commands after an M51 as G36/G46 there were in the main pattern under M50. For example, if three G36 were used after M50, three M52 would be necessary after M51. To simplify the part-program an optional number may follow the command to indicate the total number of M52 in only one block. In this case the block would look like this: M52,3. This command is normally used in the part program right after an M25, so the machine will advance the vision corrections every time it steps and repeats. M52 and its optional label are programmed in a block by themselves.

Note that M51 must precede M52.

Vision Step and Repeat Examples

There are two ways to write a part-program using these vision step and repeat commands. Both of them produce the same results, but allow the programmer to use the method with which he/she feels more comfortable. One of these methods combine vision blocks with drill blocks. In this case the part program looks very much like a standard drill program. The second method has a step and repeat section just for the vision inspection blocks and then is followed by subsequent sections where the drilling blocks are specified. An example of the first procedure follows:

COMMAND	DESCRIPTION
M48	
T1S20F09	
T2S20F09	
T3S20F09	
%	
G34,1	Vision tool number
G36X-0087Y-007	Panel pre-alignment targets
G36X-0087Y155	
G36X09Y155	
T1	
M50,1	Vision Step and Repeat Pattern Start
M25	
G46XY	Alignment targets
G46XY02	
G46X02Y02	
X005Y005	
X0051Y0151	
M01	
M02X03	
M02X03	
M02	
M08	
T2	
M51,1	Vision Step and Repeat Rewind
M25	
M52,3	Vision Step and Repeat Offset (Replaces previous G46's declared inside M50)
X01Y005	
X0101Y0151	
M01	
M02X03	
M02X03	
M02	
M08	
M50,	2 New vision Step and Repeat Pattern Start
M25	

G46X015Y	Alignment targets
G46X01Y02	
G46X02Y03	
X01Y002	
X01Y018	
M01	
M02X03	
M02X03	
M02	
M08	
T3	
M51,1	Vision Step and Repeat Rewind
M25	
M52,3	Vision Step and Repeat Offset
X015Y005	
X0151Y0151	
M01	
M02X03	
M02X03	
M02	
M08	
M30	

An example of the second procedure looks like:

COMMAND	DESCRIPTION
M48	
T1S20F09	
T2S20F09	
T3S20F09	
%	
G34,1	Vision Tool Number
G36X-0087Y-007	Panel pre-alignment targets
G36X-0087Y155	
G36X09Y155	
M50,1	Vision Step and Repeat Pattern Start
M25	
G46XY	Alignment targets
G46XY02	
G46X02Y02	
M01	
M02X03	
M02X03	
M02	
M08	
T1	
M51,1	Vision Step and Repeat Rewind
M25	

M52,3	Vision Step and Repeat Offset
X005Y005	
X0051Y0151	
M01	
M02X03	
M02X03	
M02	
M08	
T2	
M51,1	Vision Step and Repeat Rewind
M25	
M52,3	Vision Step and Repeat Offset
X01Y005	
X0101Y0151	
M01	
M02X03	
M02X03	
M02	
M08	
M50,2	New Vision Step and Repeat Start
M25	
G46X015Y	New Alignment Targets
G46X01Y02	
G46X01Y02	
X01Y002	
X01Y018	
M01	
M02X03	
M02X03	
M02	
M08	
T3	
M51,1	Vision Step and Repeat Rewind
M25	
M52,3	Vision Step and Repeat Offset
X015Y005	
X0151Y0151	
M01	
M02X03	
M02X03	
M02	
M08	
M30	

Part Program Vision Commands

In addition to the regular vision part program commands seen before, several keyboard commands may be included inside the part program to change the configuration and set up of the vision system at run time while the machine is inspecting targets. This feature adds great flexibility to the inspection process, allowing the machine to adapt itself to variations throughout the inspected panels. Most of these commands are the equivalent to screen buttons, which means that while executing these commands, the machine will behave as if there was an operator modifying the set up as the inspection phase progresses. Please note that when vision mode is enabled, the machine performs two complete passes thru the part program. In the first one, only vision commands are executed. On the second pass, the drilling/routing operations are carried on. The syntax of these commands is the same as if they were issued from keyboard. The only difference is that they must be preceded by the symbol "$". For a complete detail of the operation of these commands, see the Buttons section. A list of them and a brief description follows:

$uvis,auto,on(off)	Entry material used/not used
$uvis,box,on (off)	External video processor used/not used
$uvis,search,on (off)	Turn autosearch on
$uvis,searcharea,#	Specify the radius of the area to search
$uvis,entry,on (off)	Same as $uvis,auto,on
$uvis,insp,#	Set number of vision inspections

If your machine is equipped with Servo Controlled Variable Zoom and Focus, the following additional commands are available:

$uvis,focus,x#y#	Auto focus at the indicated location
$uvis,lamp,#	Change camera light to indicated value
$uvis,zoom,#	Set magnification to passed level
$uvis,calib	Do camera to spindle offset calibration
$uvis,alg,#,#	Specify algorithm number and fiducials

Part Program Vision Commands Examples

The following example shows a typical vision assisted part program with the addition of specific commands which permit the user to change the machine's vision configuration at run time and so further automating the inspection procedure.

COMMAND	DESCRIPTION
M48	
T1S20F09	
T2S20F09	
T3S20F09	
%	
$uvis,search,on	Turn Autosearch ON
$uvis,zoom,7	Change Magnification Level
$uvis,lamp,30	Change Camera Light Intensity
$uvis,focus,xy	Auto Focus on Target at Work Zero
$uvis,alg,0,2	Use Excellon Algorithm (0) with 2 Fiducials
$uvis,zoom,3	
$uvis,calib	Perform Auto Camera Calibration
$uvis,lamp,25	
$uvis,insp,2	Request two inspections
G34,1	
G36XY	
G36X8.6Y	
$uvis,search,off	Turn Autosearch OFF
T1	
G34,2	
$uvis,zoom,7	
$uvis,calib	
$uvis,lamp,55	

$uvis,alg,0,3	Use Excellon Algorithm (0) with 3 Fiducials
$uvis,insp,1	Make one inspection
M50,1	
M25	
G46XY	
G46X2.4Y	
G46X1.2Y2.4	
XY	
M01	
R3M02X3.1Y	
M01	
R6M02XY2.9	
M08	
T2	
M51,1	
M25	
M52,3	
X2.4Y	
M01	
R3M02X3.1Y	
M01	
R6M02XY2.9	
M08	
T3	
M51,1	
M25	
M52,3	
X1.2Y2.4	
M01	
R3M02X3.1Y	
M01	
R6M02XY2.9	
M08	
$uvis,lamp,0	Turn Camera Light Off
M30	

Excellon Automation Co.
20001 S. Rancho Way, Rancho Dominguez, CA 90220
Phone: +1 310.668.7700 Fax: +1 310.668.7800
Sales@excellon.com