Section 25
M Function Event

Topics Covered in this Section:

- Introduction
- M Function Data
- M Codes
- S Codes
- T Codes
- E Codes

AUTOCON

M FUNCTION EVENT - CONTROL WITH H and D CODES

PROGRAM MODE	N1800
0-POSITION	M FUNCTION
1-LINEAR MILL	
2-ARC MILL	M
3-FRAME MILL	
4-CIRCLE MILL	
5-BOLT CIRCLE	S
6-REPEAT	
7-SUBROUTINE	
8-DWELL	E
9-M FUNCTION	
C-CAVITY MILL	T
E-EIA	H
G-GRAPHICS	D
M-MACRO CALL	
P-PROBE	
R-ROTATE	
S-SET UP	
T-TEXT	

DEMOPART

M Function Event

The M Function event is used for:

- Activating tool offsets.
- Commanding spindle speeds.
- Activating Fixture Offsets.
- Programming miscellaneous functions.

M - Miscellaneous Codes
S - Spindle Speed
E - Fixture Offset

Tool Activation Codes:

Your control uses one of the options listed below.

T Only - Universal Activation Code
T,H,D - Individual Activation Codes

VIEW 1 OF 1

FRONT END GRAPHICS		DELETE EVENT	COPY/ STORE	PROGRAM DISPLAY	RUN CALC ASSIST	

M FUNCTION EVENT - CONTROL WITHOUT H and D CODES

PROGRAM MODE	N1800
0-POSITION	M FUNCTION
1-LINEAR MILL	
2-ARC MILL	M
3-FRAME MILL	
4-CIRCLE MILL	
5-BOLT CIRCLE	S
6-REPEAT	
7-SUBROUTINE	
8-DWELL	E
9-M FUNCTION	
C-CAVITY MILL	T
E-EIA	
G-GRAPHICS	
M-MACRO CALL	
P-PROBE	
R-ROTATE	
S-SET UP	
T-TEXT	

DEMOPART

M Function Event

The M Function event is used for:

- Activating tool offsets.
- Commanding spindle speeds.
- Activating Fixture Offsets.
- Programming miscellaneous functions.

M - Miscellaneous Codes
S - Spindle Speed
E - Fixture Offset

Tool Activation Codes:

Your control uses one of the options listed below.

T Only - Universal Activation Code
T,H,D - Individual Activation Codes

VIEW 1 OF 1

FRONT END GRAPHICS		DELETE EVENT	COPY/ STORE	PROGRAM DISPLAY	RUN CALC ASSIST	

M FUNCTION EVENT - INTRODUCTION

The M Function event is used for:

- commanding tool changes
- activating tool length and diameter offsets
- commanding spindle speeds
- activating Fixture Offsets
- programming miscellaneous functions such as: spindle (CW, CCW, or Off), program halts, end of program, coolant flow, or any other function that has been designed by your machine supplier.

One of the first events in every part program should be an M Function event (or a series of M Function events) that commands a tool change, activates the tool's length and diameter offsets, commands a spindle speed, and activates the fixture offset (E00-E32) that you wish to use. The last event in every program must be an M Function event that programs an M02 or M30 end of program command.

The degree that spindle and tool change operations are automated will vary according to your machine's capabilities and design. Your control may be equipped with additional M codes that your machine supplier has designed to operate a specific tool changer or any other mechanism that is unique to your machine.

Your machine supplier has selected the method that your control will use to program tool length and diameter offsets. Section 2 describes how to determine which method has been selected for your control. The two methods are:

- **T, H, and D codes** - Separate T, H, and D codes are used to identify, respectively, the physical tool, the tool's length, and the tool's diameter.

- **T code only** - The active T code specifies the physical tool that is in the spindle, the tool's length offset, and the tool's diameter offset.

Your machine supplier is the best source of information for instructions on programming tool changes and activating tool length and diameter offsets.

M FUNCTION DATA

Parameter	Description	Entry
N	Sequence Number	Required
M	M Code (miscellaneous)	Optional
S	S Code (spindle speed)	Optional
E	E Code (fixture offset)	Optional
T	T Code (physical tool)	Optional
H	H Code (tool length offset)[1]	Optional
D	D Code (tool diameter offset)[1]	Optional
C	C Code (provided with spindle orient feature)[2]	Optional

1. H and D codes will only be present in this event if your control uses H and D codes to activate tool length and diameter offsets. Refer to Section 2 to learn more about this issue.

ORDER OF OUTPUT TO MACHINE INTERFACE

One M, S, and T code may be programmed in the same M Function event. If all three are programmed, the output order to the machine interface is shown **below**.

1) M Code (unless M00, M01, M02, M06 or M30 is programmed)[3,4]
2) S Code
3) T Code[4]

When M00, M01, M02, M06, or M30 is programmed, the output order is S, T, M.

After one of these codes is output, the CNC waits for a "Done" signal from the machine before it will output the next code or transfer to the next part program event.

An E Code may be programmed in any M Function event. Since E Codes are not output to the machine interface and do not become active until a following motion command event, they are not included in this list. H and D codes are not included in this list for the same reason.

M CODES

Miscellaneous functions are programmed by M Codes in the range of M00-M99. Some systems permit entry of three digit M Codes in the range of M000-M999. Autocon has designated the following M Codes to perform the specific functions listed below.

M00 - PROGRAM STOP

M00 halts program execution and tool motion until you press CYCLE START. This command does not turn off the spindle.

M01 - OPTION STOP

The Optional Stop feature can be enabled by your machine supplier. If the control's Option Stop switch is ON, program execution halts when the control executes an M01 command. If the control's Option Stop switch is OFF, program execution does not halt because the M01 command is ignored. The Optional Stop feature is turned on and off with a Status screen soft key[5]. The on/off status of the feature is also listed on the Status screen. To view this screen, press the FAULT/STATUS key twice.

If the Optional Stop feature is not enabled, or is turned off, an M01 command has no affect.

---

3. Any M code that is not contained in this list may be output either first or last when M, S, and/or T codes are programmed in the same event. Check with your machine supplier about deferred M codes.
4. Macro/CPC M Codes or Macro/CPC T Codes are advanced features that may be enabled by your machine supplier. Refer to Section 2 to determine whether these features have been enabled. In general, when the corresponding feature is enabled, the control will execute an assigned Macro Catalog part program upon execution of a Macro/CPC M Code or a Macro/CPC T Code. Technical information about these features is provided later in this section.
5. Alternately, your machine supplier may provide an OPTIONAL STOP switch to turn this feature on and off. The on/off status of the feature appears on the Status Display screen. Press the FAULT/STATUS key twice to view

M02 - END OF PROGRAM

M02 has the exact same function as M30. Refer to M30 description below.

M03 - SPINDLE CLOCKWISE

If the spindle is controlled by part program commands, this code commands the spindle to turn clockwise.

M04 - SPINDLE COUNTERCLOCKWISE

If the spindle is controlled by part program commands, M04 commands the spindle to turn counterclockwise. If your spindle is not bidirectional M04 will act as M03.

M05 - SPINDLE STOP

If the spindle is controlled by part program commands, this code commands the spindle to stop.

M06 - TOOL CHANGE

Due to the diversity of spindle control and tool change applications, you should consult your machine supplier for tool change instructions. The information for the M06 - Tool Change command describes the minimum steps necessary to complete a manual tool change.

In general, your tool change sequence must include the following steps.

1) Position the spindle to provide clearance for the tool change.
2) Stop the spindle.
3) Inhibit execution of the part program until the tool change is complete.
4) Change the tool.
5) Activate the new tool's length and diameter offsets.
6) Start the spindle.
7) Allow the spindle to reach operating speed.
8) Transfer to the following part program event.

Unless the machine supplier alters the following sequence, the control's execution of an M06 event is as follows;

1) A T Code, when programmed in this event, executes first to cue a tool.[6]
2) The machine rapids the axes to the tool change point. Either the Home position (as defined in the Set Up mode) or a precise point that has been defined by your machine supplier will serve as the tool change position.
3) The control outputs M05. If the spindle is controlled, it will turn off.
4) The control outputs M06. If an auto tool changer is present, it may use this signal to begin the tool exchange.

If the tool will be manually loaded, program an M00 - Program Halt following the M06 to inhibit the program from transferring to the next event. After the tool change is complete press CYCLE START. If your cycle halts twice during the tool change, or you have a TOOL CHANGE COMPLETED pushbutton on your Operator's panel, the M00 command is redundant and need not be programmed. See the example below.

6. If your control does not use H and D codes, the T code also specifies the Tool Table length and diameter offsets for the tool. If your control uses H and D codes, you should also program an H and D code in this

(9) M06 T04 H04 D04	- rapid to the tool change position, identify tool number and tool offset entries.
(9) M00	- halt program transfer. Change tool. If the spindle is manually controlled, set its speed and turn the spindle on now. Press CYCLE START to resume operations. This event may be redundant, see above text.
(9) M03 S1000	- If the spindle is controlled by the CNC, this event will turn on the spindle clockwise at 1000 RPM.
(8) L2	- a dwell may be programmed to allow the spindle to reach operating speed before the next feed move begins.

M08 - COOLANT ON

Outputs a signal to turn coolant on. Your machine supplier determines whether this signal will be used.

M09 - COOLANT OFF

Outputs a signal to turn coolant off. Your machine supplier determines whether this signal will be used.

M30 - END OF PROGRAM

Enter this M Code as the last event in your part program. The control will execute the following series of commands.

1) The tool rapids to Home position.[7,8]
2) The control outputs M05. If the spindle is controlled, it will turn off.
3) The control outputs the M30 code.
4) The control performs a control reset. The control reset has the same effect as pressing the RESET key when the control is not in cycle. The specific actions that occur on a control reset are listed in the Auto mode section of this manual.

MACHINE SUPPLIER ASSIGNED M CODES

Your machine supplier may assign the following M Codes.

M07, M10-M29, M31-M99; and M100 to M999 if 3 digit M Codes are available.

Execution of one of these codes causes a BCD signal to appear at the machine tool interface. The machine supplier may design logic to decode the signal and actuate custom features or controls. Some applications will report the function of these M Codes on one of the ISO ASSIST pages. Refer to the machine supplier's specifications for use of these codes.

An advanced feature called "Macro/CPC M codes" allows an M code to call a specific Macro Catalog part program into execution.

7. Coordinates for the Home position are entered in the Set Up mode.
8. Machine supplier option: some machines will not return the axes to the Home position when an M30 or M02 code is output. In the this case the axes will remain in their last programmed position. Any tool clearance that

MACRO/CPC M CODES

You only need to read this topic if the Macro/CPC M Code feature has been enabled by your machine supplier. To determine if this feature is used, move to the control's System Settings screen as described in Section 2; it is enabled when the SPECIAL M CODE MACRO item is highlighted. The next line on the display provides a list of special M codes that are designated for use with this feature. As described below, the function of a Macro/CPC M Code depends on whether it is programmed in a Main Catalog or Macro Catalog part program.

Main Catalog part program - execution of a Main Catalog part program event that contains an M code which has been designated as an Macro/CPC M Code will cause the control to automatically execute the Macro Catalog part program that is assigned to the M code. The title of the Macro/CPC that is assigned to the M code is (Mxx), where xx is the number of the M code. Example: If M80 has been designated as a Macro/CPC M Code, the event

N100 (9) M80

will call the Macro/CPC program titled M80 into execution. A program of this title must reside in the Macro Catalog. You should not edit or delete the program without explicit instructions from your machine builder.

Macro/CPC M Codes always executes last when they are programmed in an event that contains any combination of M, S, T, E, H, and D codes.

Macro Catalog part program - When executed from a Macro/CPC part program, Macro/CPC M Codes are strobed to the machine interface in the exact same manner as a "normal" M code. No Macro/CPC will be called into execution.

S CODES - SPINDLE SPEED CONTROL

S Codes command the spindle speed. Most systems will have a four digit S Code (Sxxxx). Some systems permit a five digit S Code (Sxxxxx) with a maximum entry of S65535.

MANUAL SPINDLE CONTROL

When the spindle is not controlled by the CNC, programmed S Codes can prompt the operator to choose the correct RPM range for a following machining sequence. Before running a part program, the operator can preview the program to identify the required spindle speeds. The S Code that is currently active (i.e. the last executed S Code) appears on Auto mode's Event Display and Position Display screens. These displays can prompt the operator to choose the correct spindle speed when an M00 halts program execution.

PROGRAMMED SPINDLE CONTROL

The machine supplier may provide a programmable spindle. In this case the S Code may be used to program the spindle speed directly as Revolutions Per Minute.

The spindle will not respond to an S Code unless an M03 or M04 command is active. After the control sends an analog signal to the spindle the control may dwell until the spindle reaches the programmed speed. If this is not the case, you can program a following Dwell event to allow the spindle to reach operating speed.

T CODES - INTRODUCTION

The T code topic has been split into two separate descriptions depending on whether your control has been configured to use H and D codes. Refer to Section 2.

T, H, and D CODES

Read this section only if your control uses H and D codes. The following three codes are required to activate a tool. If your control has enabled the Macro/CPC T Code feature (refer to Section 2), disregard this section; your machine supplier must document the use of T codes for your control.

T code - 2 through 6 digits - The number programmed identifies the physical tool that must be put in the spindle. This number will be strobed to the machine-tool interface to command the tool changer. If your machine does not have a tool changer, this number will be displayed on the Auto and Single Event mode screens to inform the operator which tool to put in the spindle. If a Q-Lookup table is present in the Tool Tables, the programmed T code will be converted by the Q table as described in the Tool Tables Mode sections of this manual. When Tool Life Management is active, the tool number you program identifies the tool group from which the control will choose the proper tool to output to the machine-tool interface.

H code - tool length offset 2 or 3 digits - The number programmed identifies the H table tool length offset for the active tool. Activation of this offset does not cause any axes motion; instead the control stores a Z axis offset that it adds to the next event that programs a Z axis endpoint. For this reason, the first motion command following a tool code should program an absolute Z endpoint which positions the tool tip relative to the workpiece. The active H code can be canceled with an H00 command.

D code - tool diameter offset 2 or 3 digits - The number programmed identifies the D table tool length offset for the active tool. Activation of this offset does not cause any axes motion; The active D code can be canceled with an D00 command.

T CODES - CONTROLS WITHOUT H and D CODES

Read this section only if your control does not use H and D codes. If your control has enabled the Macro/CPC T Code feature (refer to Section 2), disregard this section; your machine supplier must document the use of T codes for your control.

2 or 3 DIGIT T CODE FORMAT

The T code number selects the tool's length and diameter offsets in the Tool Table, **and** is output to the machine interface. Activation of this offset does not cause any axes motion; instead the control stores a Z axis offset that it adds to the next event that programs a Z axis endpoint. For this reason, the first motion command following a tool code should program an absolute Z endpoint which positions the tool tip relative to the workpiece. Also, the control will compensate for the tool's radius during any compensated milling sequence. The control will also compensate for the tool's diameter during the execution of a Frame Mill, Circle Mill, and Cavity Mill event. Most applications allow you to cancel the active tool offsets with an T00 command.

If a Q-Lookup table is present in the Tool Tables, the programmed T code will be converted by the Q table as described in the Tool Tables Mode sections of this manual. When Tool Life Management is active, the tool number you program identifies the tool group from which the control will choose the proper tool to output to the machine-tool interface.

4 DIGIT T CODE FORMAT and a SWAPPING TOOL LOOKUP TABLE

This is a special method of programming T codes on a machine that is equipped with an automatic, random tool changer. Read this section only if your control uses a 4 digit, a swapping lookup table, but does not use H and D codes. Refer to Section 2 to learn about these features. You should also read the previous 2 or 3 Digit T Code Format topic to familiarize yourself with the manner in which the control activates tool offsets.

A four digit T Code is represented by TXXxx, where XX identifies the physical tool and xx designates the tool length and cutter diameter Tool Table offsets that will be used.

The first digit pair, XX is programmed in the range of 01-99. In some auto tool changer applications this digit pair will begin to cue the corresponding tool into position for a following M06 tool exchange.

The second digit pair, xx, designates which of the 99 Tool Table offsets will be used for the active tool.

E CODES - FIXTURE OFFSETS

E Codes assign a fixture offset to be used for a following series of machining events. E Codes range from E00 - E32. The dimensions for fixture offsets E01-E32 are stored in the Tool Tables mode. These values represent the distance that each axis is to be offset from the machine's datum[9]. Executing an E Code, causes part zero to be offset from the datum. Examples which illustrate the uses of Fixture Offsets are provided in the Fixture Offset Table section in Part 2 of this manual.

E Code execution does not cause any axes motion - instead the control stores an offset for each axis (X,Y,Z) that it will add to the part program's next motion command. The manner in which the axes move to a point relative to the fixture's part zero is chosen by your machine supplier through the selection of Immediate or Deferred offset activation.

If the control has Immediate offset activation, the first motion command that follows an E Code will cause all axes to move to a point relative to the fixture's part zero.

If the control has Deferred offset activation, the first motion command that follows an E Code will cause only the axes that are programmed in the event to move to a point relative to the fixture's part zero. Any axis that is not programmed in the event remain stationary until its endpoint is programmed in a following event.

Refer to the Immediate or Deferred Offsets topic in the Fixture Offset Table section of this manual.

Execution of E00 causes the control to cancel any active fixture offset.

Once executed, an E Code is not canceled by an M02[10], M06, M30[10] command, or by pressing RESET.

9. The datum is set with a ZERO SET operation in the Jog mode. If no ZERO SET is performed, the control assumes that machine coordinate zero is the current datum.
10. Your machine supplier may choose to configure the control so that E Codes are canceled on M02 or M30 execution. After a full part program cycle, check your Auto mode display to see if E Codes are canceled by M02 or M30.