

# APPENDICES

**DYNAPATH DELTA 40, 50, 60 MILL  
CUSTOMER INFORMATION MANUAL**



## SCREEN MESSAGES

The following messages appear as screen prompts. These errors may occur when you are entering part program data in the Program mode, or you are copying, receiving, or deleting information in the controls part program memory. You must press the CLEAR key to remove the message and correct the condition before continuing operations.

<b>CATALOG FULL:</b>	Available number of part titles has been exceeded. Delete any unwanted programs.
<b>COPY FAULT:</b>	The N number was not recognized on an attempt to copy a program event. Be certain a valid N number is used.
<b>DELETE FAULT:</b>	The Delete Program mode F or L sequence has been incorrectly entered or is illogical; program inhibit is in effect; or no N number was recognized on an attempt to delete an event from Program mode.
<b>DUPLICATE ID:</b>	The title of an imported part program already exists in the Catalog mode. This message only occurs if the title is not selected in the Catalog - otherwise a merge receive will occur.
<b>EDIT ERROR:</b>	Insufficient storage memory available for the control to relocate part program data following a Receive mode operation. Delete any unwanted programs and reload.
<b>EXECUTION FAULT:</b>	Data is missing, improperly entered or illogical.
<b>FORMAT FAULT:</b>	This fault is generated when an entry to an event contains an incorrect leading character, wrong decimal location, improper sign or incremental operand where absolute is required. Also flagged when an attempt has been made to copy a program event with an invalid N number or a specified F or L number is invalid.
<b>INVALID CONCURRENT ID:</b>	Cannot select RS232C program ID from concurrent Catalog mode.
<b>INVALID PART ID:</b>	Cannot copy or rename a program to RS232C part ID.
<b>MACRO ERROR</b>	You attempted to merge a macro program while a Macro Inhibit was set on the CNC.
<b>MACRO INHIBIT:</b>	A P11 inhibit has been entered in the Set Up mode.
<b>MEMORY FULL:</b>	Available part program storage has been exceeded. Delete any unwanted programs.
<b>NEXT N FAULT:</b>	No event number was entered prior to pressing the NEXT EVENT or -NEXT EVENT key or if NEXT EVENT key was used the next event would be outside the 0-9999.999 range.
<b>NO N FAULT:</b>	A program sequence number must be entered prior to any other data.

---

<b>MEMORY LEFT:</b>	programs.
<b>PARITY ERROR:</b>	A parity check during a Receive mode operation did not show even parity. The data is suspect.
<b>PART ID ALREADY EXISTS:</b>	The selected program title is already in use.
<b>PART ID IS CYCLING:</b>	The cycling program may not be selected from the concurrent Catalog mode.
<b>PPS FORMAT FAULT:</b>	An internal parity check shows faulty program data in a memory location. Review event data from Display Program mode. Reenter the event if necessary.
<b>PPS OVERFLOW:</b>	The attempted receive, edit or copy operation would exceed available part program storage. Delete unnecessary programs from storage.
<b>PROG INHIBIT:</b>	A P11 inhibit has been entered in the Set Up mode.
<b>RECEIVE ERROR:</b>	The program contains one or more entries in wrong format. Error was detected at while receiving data from an off-line device. The Receive operation was aborted.
<b>RESERVED PART ID:</b>	Part ID RS232C is reserved for buffered input operations.
<b>SIZE OF PART ZERO NO COPY:</b>	A program of zero content cannot be copied.
<b>STORE FAULT:</b>	See PROG INHIBIT.
<b>VALUE FAULT:</b>	An entry to an event is outside of the allowable range (e.g. improper N number or nonexistent G-cycle).

## STATUS SCREEN MESSAGES

The following messages define the current condition of selected initialization parameters or EIA/ISO modal variables. These messages report the current status of the control and machine. The Status screen may be viewed at any time by pressing the **FAULT/STATUS** key twice.

<b>X, Y, Z, 4, 5, 6 NOT REFERENCE ZEROED:</b>	Cautions that a reference zero operation has not yet been performed. Some applications may require that you Reference the machine before part programs can be executed.
<b>G CODE STATUS MESSAGES:</b>	These messages report the current setting of several modal G code groups. Modal G Codes are changed by execution of an EIA/ISO part program block. They are also changed to the machine supplier's default values when the operator presses <b>RESET</b> when the control is not in cycle, or when a part program executes an M02 or M30 end of program command.
<b>OPTIONAL STOP ON/OFF:</b>	Displays the current status of the Optional Stop feature if the machine supplier has enabled the <b>OPTION STOP</b> soft key.
<b>BLOCK DELETE ON/OFF:</b>	Displays the current status of the Block Delete feature if the machine supplier has enabled the <b>BLOCK DELETE</b> soft key.

## FAULT SCREEN MESSAGES

The following errors will halt operations and generate a fault display. You must press **RESET** to clear the message and clear the fault condition resume operations.

<b>EMERGENCY STOP:</b>	You must reset the control and correct the condition before continuing. Position information is retained unless a processor halt or memory failure has caused the fault.
<b>X, Y, Z, 4, 5, 6 SERVO ERROR:</b>	Allowable following error has been exceeded, the axis has failed to respond to a servo command, a travel limit switch has been contacted, or the servo drive has shut down.
<b>GANTRY CROSS ERROR FAULT:</b>	The allowable following error for one axis in your gantry system has been exceeded. This error should be investigated in the same manner as a servo error.
<b>X, Y, Z, 4, 5, 6 TRAVEL LIMIT:</b>	A software travel limit has been exceeded. Limits are entered in the Set Up mode. <b>RESET</b> and jog off the travel limit.
<b>4, 5, or 6 AXIS DISCONNECTED:</b>	The control has detected an attempt to execute an auxiliary axis command and the axis is not installed.
<b>PIC I/O FAULT:</b>	An electrical circuit connecting a machine switch or relay to the control has failed. Report this error to your machine supplier.
<b>MTB MESSAGES:</b>	Your machine supplier may designate halt, error, or status conditions specific to your machine. Refer to your supplier's specifications.

<b>INTERRUPT:</b>	An operating system error has occurred. Do not attempt to continue operations. Report the control's serial number and circumstances to your machine supplier.
<b>LOSS OF FEEDBACK:</b>	One or more of the axes feedback devices may have failed. Turn power Off, then On. If condition reoccurs, contact your machine supplier.
<b>X, Y, Z, 4, 5, 6 AXIS RUNAWAY:</b>	An axis has moved without a command from the CNC. Turn power Off, then On. If condition reoccurs, contact your machine supplier.
<b>CPU WATCHDOG: TIMEOUT</b>	An operating system error has occurred. Do not attempt to continue operations. Report the control's serial number and circumstances to your machine supplier.
<b>LOSS OF COMM. WITH FRONT PANEL:</b>	An operating system error has occurred. If pressing RESET does not clear the problem, cycle power to continue operations. If the problem reoccurs, report the control's serial number and circumstances to your machine supplier.
<b>PROBE CONTACT FAULT:</b>	A probe contact occurred while the Probe Protect signal was high.

## FLOPPY DISK ERROR MESSAGES:

<b>BAD DISK SECTOR:</b>	There is a bad sector on your floppy disk. Use another diskette.
<b>DISK ACCESS DENIED READ ONLY DISK?</b>	You cannot transmit a file to the floppy diskette in the drive because it is write protected.
<b>DISK FILE ALREADY EXISTS:</b>	You are attempting to overwrite a file on the disk. Either delete the disk file or transmit your file to a different filename.
<b>DISK FILE NOT FOUND :</b>	The filename that you requested is not located on the diskette. Remember that you must press <b>D ENTER</b> whenever you insert a different diskette into the drive.
<b>DISK INVALID MEDIA ID:</b>	Your floppy disk is not compatible with the disk drive. Use a disk that was formatted for a DOS computer.
<b>DISK TIME OUT: DISK IN DRIVE?</b>	After you press <b>D enter</b> in the Transmit/Receive mode the control will read the floppy disk that is in the drive. This message indicates that there is no disk in the drive.
<b>DISK WRITE PROTECTED:</b>	The control cannot transmit your part program to the floppy disk because the diskette is write protected. Slide the write protect tab to the "enable" position.
<b>INSUFFICIENT DISK SPACE:</b>	There is not enough room on your diskette to store your part program. Insert a new, formatted diskette.
<b>UNABLE TO INITIALIZE DFM MEMORY:</b>	A hardware or software error has occurred. Contact Autocon Technologies.
<b>DMA ACCESS ACROSS 64K BOUNDARY:</b>	A hardware or software error has occurred. Contact Autocon Technologies.
<b>DISK DMA OVERRUN:</b>	A hardware or software error has occurred. Contact Autocon Technologies.
<b>ILLEGAL COMMAND TO FLOPPY DISK DRIVER:</b>	A hardware or software error has occurred. Contact Autocon Technologies.
<b>CRC ERROR ON DISK READ:</b>	A hardware or software error has occurred. Contact Autocon Technologies.
<b>DISK SECTOR NOT FOUND:</b>	A hardware or software error has occurred. Contact Autocon Technologies.
<b>DISK SEEK FAILURE:</b>	A hardware or software error has occurred. Contact Autocon Technologies.

## FAULT MESSAGES

The following error codes are generated when the control's error checking routines interpret a logical error when a part program event executes. Program execution is halted and a flashing FAULT/STATUS message is displayed. The N Number of the faulty event will most likely appear in the Auto or Single Event mode's NEXT EVENT display.

### POSITION

- 001 Not used
- 002 An axis value entered for a G6 Position event is incremental.
- 003 (Aux. Axis Option) - the programmed rotary axis absolute value exceeds a maximum of 359.999°. This error code can occur in the Position, Linear Mill or Repeat event.
- 004 The value that you programmed for your rotary index axis is not a multiple of 1° or 5°.
- 005 - 009 Not used.

### LINEAR MILL

- 010 A feedrate value must be entered when the Aux. axis is programmed by itself.
- 011 - 019 Not Used.

### ARC MILL

- 020 End angle B has not been entered into the event.
- 021 The distance from center to start point does not equal the distance from center to end point of the arc. The actual start or end point may differ from either point presumed when the event was programmed.
- 022 The ratio of the feedrate to the radius is too large. The result will be a linear move with a possible servo error. Reduce the feedrate.
- 023 The generated Z feedrate relative to the arc feed would exceed the rapid rate and cause a servo error.
- 024 The Cam Mill start radius (E) must be greater than zero.
- 025 The Cam Mill start radius (R) must be greater than zero.
- 026 Illegal axis/center data
- 027 - 029 Not Used.

### FRAME MILL OR CIRCLE MILL

- 030 Letter X, Y and/or Z was not entered into the event.
- 031 The value for X or Y is less than or equal to two times the corner radius. (Frame Mill Event only).
- 032 Cutter compensation off (CO) was programmed with a finish cut (L) specified.
- 033 A Pocket Mill (G7) was programmed with cutter compensation off (CO), cutter compensation value zero, or an outside cut specified.
- 034 Frame or Circle Pocket Mill - G7 programmed with an active cutter radius less than pocket overlap value (the value J or default .010"/.254mm).
- 035 A Peck value (K) equal to zero (0) was programmed.
- 036 The radius was not entered.
- 037 An inside cut was programmed and the finish dimension L is less than the active cutter's radius.
- 038 The cutter compensation (C) was not programmed or left blank.
- 039 Invalid pocket (G7) cutter diameter.

**CIRCLE MILL**

- 040 Radius value (R) equals zero (0).
- 041 - 049 Not Used.

**BOLT CIRCLE**

- 050 No hole (H) information entered, an H value of zero, or the number of holes to be processed is greater than the number of holes declared for the bolt hole pattern.
- 051 No Bolt Circle radius was entered or a radius value of zero was programmed.
- 052 You cannot program a 2nd reference plane in an event that cancels a quill cycle.
- 053 - 059 Not Used.

**REPEAT**

- 060 The sequence number (N) was not entered in a Repeat event.
- 061 The first event (F) or number of repetitions (T) was not entered in a Repeat event.
- 062 A rotational angle (C) and an X, Y or Z step value have been entered in the same Repeat event.
- 063 The first event sequence number (F) of a repeat event is greater than or equal to the Repeat event sequence number (the N value). A Repeat event may not include events programmed after itself.
- 064 Not Used.
- 065 No event in current program storage has a sequence number equal to the F value entered in a Repeat event.
- 066 A repeat loop is beyond the nesting limit. Repeats within repeats may only be programmed a total of four levels deep.
- 067 A combination of at least one repeat and one subroutine will probably transfer control to each other infinitely if allowed to continue.
- 068 - 069 Not used.

**SUBROUTINE**

- 070 The sequence number (N) has not been entered in a Subroutine event or in an event (of any type) being tested to see if it is the last event of a Subroutine.
- 071 The first event (F) or last event (L) defining the Subroutine has not been entered in the Subroutine event.
- 072 The first event (F) programmed in a Subroutine event appears later in program storage than the last event (L) that was programmed.
- 073 Not Used.
- 074 A Subroutine is calling a range of events including itself. Specifically, a Subroutine event has a sequence number that is between the values entered for the first and last event sequence numbers. The Subroutine would infinitely call itself.
- 075 A Subroutine event is calling a section of sequence numbers where a second Subroutine event is called. The second Subroutine is then calling the first Subroutine. The result would be an endless loop.
- 076 A Repeat event within a Subroutine has a first event value which is outside the Subroutine and causes the event that called the Subroutine to be executed again. The result would be an endless loop.
- 077 The nesting limit for Subroutines has been exceeded. You cannot program more than 4 levels of Subroutines within a Subroutine.



- 078 No event is stored that has a sequence number equal to the F or L value programmed into the Subroutine.  
079 An N number within a Subroutine range of events is greater than the last N Number (L) specified by the Subroutine event.

**DWELL**

- 080 L was not programmed.

**MACRO**

- 081 You must enter the name of the macro you wish to call under the "P" parameter in the main program's Macro Call event.  
082 The macro ID specified in the calling event cannot be found in the Macro Catalog.  
083 You have nested macros greater than four deep.  
084 Your macro calls itself; it may call itself directly or a call to itself may be nested within other calls. This is known as recursive calling and is not permitted.  
085 The axis identifier in a CPC offset or probing command is out of range. See the appropriate command in Part 2 of this document for the acceptable range.  
086 A CPC probing command has been programmed on a control in which the probing option has not been installed.  
087 Cutter compensation must be turned off before a calling program calls a macro or CPC into execution.  
088 You cannot program an Arithmetic event while cutter compensation is active.  
089 Not Used.

**M FUNCTION**

- 090 An M Function event was programmed without either an M Code (M), spindle speed (S) or tool number (T) entered.  
091 The tool offset identifier digits are outside the allowed range.  
092 E Code exceeded the allowable range of E00-E14.  
093 The first two digits in the T Code exceed the maximum tool number selected in Tool Life Management.  
094 All tools within the selected group have expired. If tool life expires in the middle of a cycle, it will be allowed to finish the cycle.  
095 Special M and T macros may not be programmed in the same event. Refer to Part 1-Section 1 for further information.  
096 Not Used.  
097 Your maximum spindle speed is 65535 rpm.  
098 The tool you programmed is already in the spindle.  
099 The tool request could not be located in Random Tool Table.  
100 Tool number requested same as tool in spindle.  
101 An error has been detected in parameter PROM entries for the tool change position. Contact your machine supplier.  
102 - 109 Not Used.

**SCALE, SET UP or GRAPHICS**

- 110 A scale factor is greater than 99.9999 or a zero or negative absolute value is entered.
- 111 The programmed inch/metric option value was other than 0 or 1. Also, no other dimension whose values rely on the inch/metric declaration may be programmed into the same event with the declaration.
- 112 A mirror image option value other than 0 or 1 was programmed.
- 113 Software Travel Limit.
- 114 The plane select code (P) must be P1-P5.
- 115 The Type of Plot code must be T1 or T2.
- 116 The Graphic event scale factor must be greater than 0.0.
- 117 - 119 Not Used.

**CAVITY MILL**

- 120 The cross section radius (R) or angle (A) has not been programmed into the event.
- 121 The length (L) or linear increment (I) has not been programmed in a linear type of cavity.
- 122 The end angle (B) or rotary increment (C) has not been programmed in a rotary type of cavity.
- 123 Either R or D is less than the active cutter's radius.
- 124 A G Code has not been entered into the event.
- 125 An G Code other than G8 or G9 has been entered into the event.
- 126 A T value (ending side) is programmed, but an S value (starting side) was not programmed.
- 127 A K value (ending bottom) was programmed, but a J value (starting bottom) was not programmed.
- 128 The cavity's start point, start angle, and center of rotation are in conflict with the shape that is programmed (check L, I, C, A, B values).
- 129 Not Used.

**EVENT FORMAT**

- 130 An unrecognizable or undefined character has been entered into this event.
- 131 The first character in the stored event was not a sequence number 'N' or 'F'. (View this event in Display Program mode. To recover, delete the previous through the following event and re-enter all three events).
- 132 The event type code is an invalid character; the left or right parenthesis is missing from the entry or incorrectly entered; or an M00, M02 or M30 was not present at the end of the program.
- 133 A dimensional entry has been entered incorrectly. The fault is related to a decimal point, incremental, or minus sign, too many digits, or an invalid letter programmed within the event.
- 134 A number in the current event is too large for the allowed range.
- 135 Multiple decimal points have been entered in a given number value.
- 136 A premature end of event (\$) or end of program (END) has been detected. An M30 or M02 was not programmed at the end of the part program.
- 137 A memory parity error occurred during part program cycling. Reenter the entire event from the keyboard; reload the program; run start up diagnostics to insure memory is intact.
- 138 Both center coordinates must be programmed in a Rotate event.
- 139 Not Used.
- 140 A polar event fault: the angle of motion and/or the radial distance of the move (R) has not been entered.

- 141 The Z reference plane (event variable W) has been programmed below the final depth of cut (Z).
- 142 No more than 10 consecutive M Codes can be programmed.
- 143 The cutter compensation logic cannot compute the compensated path in a Linear Mill or Arc Mill with cutter radius compensation. Usually, the cutter size is too large to follow a tool path in a confined geometric area. Typical problems include retracing a narrow tool path with compensation, attempting to mill a notch with a cutter diameter exceeding the notch width or the cutter radius is too large to reach the start point of an arc to mill around a corner without backing into the part.
- 144 The cutter radius is greater than the circle radius in a Linear Mill or Arc Mill with cutter radius compensation.
- 145 The plane select variable (P) is not P0 (zero), P1, or P2.
- 146 An Arc Mill event Direction of cut (D) variable has been entered outside of the range of D0-D1, or a Frame Mill or Circle Mill event Direction of cut D variable has been programmed outside the range of D0-D3.
- 147 The Cutter Compensation (C) variable has been programmed outside the range of C0-C2.
- 148 An intersection cannot be calculated along a path determined by the active cutter's radius while cutter radius compensation is on. Example; a line is not tangent to a circle within the program tolerance.
- 149 Not Used.

## G CYCLES

- 150 The G value programmed is not one of the allowed values for G cycles in the faulty event.
- 151 The G value programmed is not in the range of G0-G6 for a Position event, or G0-G5 for a bolt circle event.
- 152 W may not be entered as an incremental value.
- 153 Letter W, Z, and/or K has never been entered as required for the declared G cycle in this event.
- 154 Contact Autocon Technologies with a description of events that caused error.
- 155 Invalid F value, feedrate value does not fit the ipm/mmpm format.
- 156 Report the error code, condition and serial number of the control to your machine supplier.
- 157 Report the error code, condition and serial number of the control to your machine supplier.
- 158 An internal hardware error (if unrelated to 154 handwheel operation) has occurred. Report the error code, condition and serial number of the control to your machine supplier.
- 159 Invalid Lead/Pitch (F) or RPM(S). Programmed or modal value is invalid or not programmed.

## EIA EVENT

- 160 The system does not recognize this G code.
- 161 EIA: M S T or E with illegal G codes.
- 162 EIA: M06 with motion.
- 163 Dwell format has been improperly programmed.
- 164 W cannot be programmed with another axis. W cannot be programmed when a quill cycle is active.
- 165 G93 Inverse Time is modal and time F was not programmed. It must be programmed in every block that uses Inverse Time.
- 166 You may only program G01, G21 and G00 (with G80 active) blocks when G93 Inverse Time is modal.

- 167 Cutter Compensation cannot be active when a G27, G28, G29, or G53 block executes. Cancel it with a G40 block.
- 168 The intermediate point for a return from the reference point has not been defined in a prior G28 block for a programmed axis.
- 169 - 175 Not Used.

#### **HARDWARE/OPERATIONS - SYSTEM FAILURE**

- 176 - Report the error code, condition and serial number of the control to your machine supplier.
- 179

#### **PROBE EVENT**

- 180 A required entry for the selected G cycle has not been programmed in the Probe event.
- 181 Scaling, axis rotation or mirror image cannot be active upon execution of the Probe event.
- 182 The spindle or table probe has not been calibrated.
- 183 The probe did not contact the target within the range specified in the event.
- 184 Not Used.
- 185 A fixture offset other than E00 was active upon execution of a G1 or G2 Probe cycle.
- 186 - 199 Not Used.

#### **AUTO RADIUS - LINEAR MILL**

- 200 The next event is not valid for the auto radius entry R.
- 201 The events to be joined with an auto radius form parallel lines. The auto radius cannot be performed.
- 202 - 206 Not Used.
- 207 The auto radius is too large for the programmed linear mill events.
- 208 The joining events have different P entries, are not in the same plane.
- 209 Auto radius is programmed for an out of plane axis.
- 210 The center for auto radius cannot be computed.
- 211 Incomplete event. D must always be programmed in combination with V unless a solution will be computed by Calc Assist. Without Calc Assist, the two together represent a complete move, and cannot be programmed independent of one another.
- 212 The V programmed does not agree with the direction specified, usually by axis endpoint.
- 213 - 218 See CPC Error Codes, below.

#### **HARDWARE/OPERATIONS - SYSTEM FAILURE**

- 219 Call your machine supplier with the error code and serial number of the control. Report the condition under which the error occurred.

**BUFFERED INPUT**

- 220 No baud rate selection has been entered
- 221 Incoming data failed a parity check.
- 222 Memory overflow or a character was not read. If 1600 or more characters are available (see Catalog mode) is available, select a lower baud rate.
- 223 Improper tape identifier format (for the program title). Correct format is a left parenthesis followed by up to eight characters or spaces followed by a right parenthesis.
- 224 An invalid event was detected. Subroutine and Repeat events cannot be executed through the buffer.

**CPC ERROR CODES**

- 213 The offset identifier in a GAO, GOV or SOV command is invalid. The offset letter code must equal 20 for Tool offset (T), 17 for the random tool number (Q) or 5 for a Fixture offset (E). No other values are allowed.
- 214 The Tool or Fixture (ARG2) number in a GOV or SOV command is out of range. T must be in the range 1-99; E must be in the range 1-32.
- 215 The offset letter (ARG3) in a GOV or SOV command is out of range. See the GOV or SOV topic in Part 2 of this document for a complete definition of acceptable values.
- 216 A GAG, GAO, GOV or SOV command is followed by an invalid number of arguments.
- 217 The offset value specified by a SOV command results in an offset value > 999.9999 inches or 9999.999 mm.
- 218 The argument for a GAG command must equal 0 or 1. No other values are permitted.
- 225 You have used an invalid mnemonic (such as a function) as a command (i.e. N100(A)(ABS,#34) ).
- 226 An event contains a character string that the control cannot identify as a command or expression.
- 227 Your macro contains a Symbolic Variable that is not followed by an appropriate delimiter (i.e. parenthesis, arithmetic operator, etc.).
- 228 Your macro contains a stand alone expression that is not followed by an appropriate delimiter (in most cases a missing parenthesis or comma is to blame).
- 229 Your macro contains a pound sign (#) followed by a number (or an expression that when evaluated produces a number) that is not in the range 1-99. Symbolic Variable numbers must be within this range.
- 230 Your macro contains a function (i.e. SQR, TAN, etc.) that is followed by an invalid delimiter. Generally, functions must be immediately followed by a comma (sometimes by a right parenthesis); this error appears when one is not.
- 231 Your macro contains a constant followed by an improper delimiter. Constants may be followed by arithmetic operators, commas or a right parenthesis depending on how they are used.
- 232 Expressions may be terminated by a comma only if the number of right parentheses in the expression equals the number of left parentheses. Otherwise, this error is flagged.
- 233 The only mnemonics allowed within a mathematical expression are the trig functions, the rounding functions, INT, SQR, and ABS.
- 234 A multiply (\*) or divide (/) operation has been detected without the first term (i.e. N10(A)(#5= \*#10)).
- 235 Your macro contains a character that the control does not recognize. The only characters allowed are those that appear on the controls alphanumeric

- keypad, the space, the slash (/), left paren {(), right paren ()}, pound sign (#), equal sign (=), plus sign (+) and asterisk (\*).
- 236 This error is flagged if the first character of a new expression level (the first character following a left parenthesis) is an End of Block marker.
- 237 Your macro contains expressions within expressions (ABS,(SQR,(TAN,(INT.....) beyond the limit of 4. Break the nested expression down into several separate expressions.
- 238 The last event of a Subroutine call is an Arithmetic branch event (IF, GO, GIN).
- 239 A single number contains more than one decimal point (i.e. 12.34.56).
- 240 The numeric result of an expression has overflowed the memory space allocated for it. Check that the expression is valid; it is unlikely that a valid expression would produce a number large enough to flag this error.
- 241 You can have no more than four levels of expression active at one time. This error is flagged when you exceed this limit (i.e. SQR(ABS(INT(COS(ATN(.....)))))) )
- 242 On evaluating a division operation the control has discovered that the divisor is 0. In this case, it is likely that the divisor is itself the result of an expression; check that this expression is valid. If it is, you may have to somehow modify the macro to skip the division operation when the divisor is 0.
- 243 The control's interpreter has found an arithmetic event in the master program. This is not permitted; the event must be removed.
- 244 You must have the advanced macro feature to use arithmetic events; this error indicates that an arithmetic event has been detected by a control in which advanced macros are not installed.
- 245 The result of an expression is indeterminate. Check that the expression is valid; if it is, a variable contained in the expression must be at fault. If there is no way to avoid an offending value in the variable, you may have to modify the macro to allow for it.
- 246 A branch statement in your macro specifies an invalid destination. The destination N number must be between 0 and 9999,999.
- 247 A branch statement specifies its own N number as the destination.
- 248 A branch statement in your macro is attempting to branch to a nonexistent N number. Check that all the branch destinations exist.
- 249 Your macro is attempting to take the square root of a negative number. This number is most likely the result of an expression; check that the expression is valid. If it is, one way to avoid this error is to use the ABS function on the variable which is to be evaluated for its square root.
- 250 Your macro is attempting to evaluate a trig function that is undefined, i.e. TAN,90. When evaluating variables in such cases it is important to check that they are within a defined range before applying the trig function to them.
- 251 Decimal I/O address < 0 or >1535 in a GIN statement. For SON and SOF, the decimal address must be in the range 0768-0799.
- 252 An invalid baud rate is selected in the Transmit/Receive modes.

**REFERENCE ZERO**

- 254 A required reference zero operation prior to execution of the part program has been specified for this system by your machine supplier. A reference zero is required before cycling is allowed.

**FUTURE EXPANSION**

- XXX "Refer to Manual" (error code number may vary)  
Means that an error condition has been detected but there is no attached description of the error. This will generally be related to a new feature not documented at the time your manual went to print.

Report the error code number and firmware number from your control's start up screen to Autocon's Sales Support Staff. We will make a determination of the error in question and report back to you.