
Section 10: Error Messages

Error Messages

**ABSOLUTE MODE
REQUIRED AT N =**

The CNC must be in the G90 (absolute) mode during all probe functions. Probe function L9101 is programmed in the G91 (incremental) mode.

**AMPLIFIER FAULT ON
AXIS**

This occurs at power on if the control does not detect an axis that should be operational. This is often seen when the A axis is not in use and the dummy termination plug has been installed. If the dummy termination plug is installed, this message can be disregarded.

If this message appears for any of the other axes (X, Y, or Z), then power off to check the motor overload relays. Power on and if the message appears again, call the service department, noting the error message number. The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.

This message also appears when the EMERGENCY STOP button has been pressed. The operator must release the EMERGENCY STOP button and press the JOG button to reset the amplifiers. (Depressing the EMERGENCY STOP button disconnects the axis amplifiers. This is why this message appears.)

ARM MUST BE LEFT

Before a pallet change can occur, the arm must be completely to the left with the pallet on the table. The feed back on the arm is being read at this time.

**ARRAY EXCEEDS
ALLOWABLE
NUMBER**

This message will appear when the number on the array exceeds the expected amount. This amount varies from variable to variable. For example, H(99) is acceptable, H(102) is not.

ATC FAILURE

Check to see if the Z axis alignment was properly set at cold start. The automatic tool changer has failed to respond to an M6 or the TC,1 command. Try a power off and power on. If this does not help check the fuses F5, F6, F7, and F8 to see if they are burned out. Circuit breakers 1 and 2 (if they are present) should be pressed to see if they have been tripped.



WARNING: Power off the machine before checking fuses, relays, and/or circuit breakers.

If this does not help, call the service department.

If this message appears with the turret in the extended position, it may be an indication that the sensor for turret extension is faulty; call the service department.

ATC TURRET FAILURE

Check to see if the Z axis alignment was properly set at cold start. Then check to see if anything is caught between the sheet metal guard and the turret. If not, this could be an indication that the sensor for turret rotation is faulty.

The fuses F5 and F6 should be checked to see if they are burned out. Circuit breaker 2 (if it is present) should be pressed to see if it has been tripped.



WARNING: Power off the machine before checking fuses, relays, and/or circuit breakers.

ATC WILL NOT MOVE TO POSITION

Check to see if the Z axis alignment was properly set at cold start. An attempt is made to make a tool change and something prevents the turret from going all the way out to the spindle. Check the tracks that allow the turret to slide out to the spindle. Sometimes chips stick to the track and prevent the rollers from rolling on the track.

When this message appears use the command SETCS then the command HO which will bring the tool to the Cold Start position. It is important to visually check the axis indicator markers to see if they are aligned; if not, jog the table until the markers are aligned. Use the command CS, then answer Yes (Y) to the "move to home?" question. This will bring the tool to the part's home position and enable the operator to begin running the program in AUTO again. It is also possible that the turret extended sensor is faulty.

Fuses F5, F6, F7, and F8 should be checked to see if they are burned out. Circuit breakers 1 and 2 (if present) should be pressed to see if they have been tripped.



**ATC AMBIGUOUS
TURRET POSITION,
VERIFY POCKET #
WITH SETTO,# or,
TURRET LOCATION
LOST, RESET
TURRET LOCATIONS
WITH SETTO,#**

WARNING: Power off the machine before checking fuses, relays, and/or circuit breakers.

The Tool Count sensor indicates the Turret is located between Buckets, or has been interrupted. Move Turret to next Bucket position, and apply SETTO,# where # is the Bucket number in the Bucket ready position.

1. Another system may have faulted while Turret was moving or the ATC was communicating. Correct the fault, and reset Turret locations with SETTO,#.
2. Check the Door Interlock System.
3. Check for power interruption to the Turret Motor, fuses, and circuit breakers.
4. Check the Turret Motor for liquid contamination, or inoperative brake.
5. Check the Tool Count Sensor.

**ATC AMBIGUOUS
BUCKET POSITION,
VERIFY POCKET #
WITH SETTO,# or,
BUCKET NOT UP/
DOWN, RESET
TURRET LOCATIONS
WITH SETTO,#**

The Tool Up or Tool Down sensor indicates that Bucket is out of position, and the Turret Locations may be lost. Correct Bucket Up/Down problem, and apply SETTO,# where # is the Bucket number in the Bucket Ready position.

1. Check Tool Up/Tool Down reed switches on Bucket Cylinder.
2. Check for both air valves on at the same time.
3. Check for inadequate air supply.

**ATC ARM IS NOT IN
HOME POSITION**

The Stopping Sensor or the Arm at Home Sensor indicates that the Arm is not in its Home Position, and must be before the machine can continue.

1. Check Door Interlock system.
2. Check Head Position Sensor and Head Position.
3. Check for power interruption to Arm Motor, fuses, and circuit breakers.
4. Arm Motor may need to be moved manually.

**ATC ARM SOLENOID
SENSOR/HOME
SENSOR FAILURE or,
ARM DID NOT MOVE
FROM HOME**

The Arm at Home Sensor indicates that the Arm has not moved away from ATC home position.

1. Check Door Interlock system.
2. Check Head Position Sensor and Head Position.
3. Check for power interruption to Arm Motor, fuses, and circuit breakers.

**ATC BUCKET UP
SOLENOID/SENSOR
FAILURE or, BUCKET
DID NOT MOVE UP**

The Tool Up Sensor has not reported a successful move up of the Bucket in preparation for rotating the Turret.

1. Check the lower reed switch on the Bucket Cylinder.

2. Check for inadequate air supply.
3. Check air valve.

**ATC BUCKET DOWN
SOLENOID/SENSOR
FAILURE or, BUCKET
DID NOT MOVE
DOWN**

The Tool Down Sensor has not reported a successful move down of the Bucket in preparation for exchanging the tools.

1. Check the upper reed switch on the Bucket Cylinder.
2. Check for inadequate air supply.
3. Check air valve.

**ATC
COMMUNICATION
ERROR WITH 1330
CARD**

A communication timeout has occurred between the CNC CPU and the DATC Controller (1330).

1. Route ribbon cable connecting the 1470 to 1330 away from fans.
2. If error occurs on power-up, there may not be an actual problem.

**ATC INTERLOCK IS
ON**

An inhibited ATC function has been attempted while interlocked.

**ATC TOOL CLAMPING
FAILURE or, ARM DID
NOT ARRIVE AT
SPINDLE**

The arm at Spindle Sensor or the Stopping Sensor has indicated that the Arm has not arrived at the Spindle.

1. Check for Arm Plunger latching problem.
2. Check Door Interlock system.
3. Check Head Position Sensor and Head position.
4. Check for power interruption to Arm Motor, fuses, and circuit breakers.

**ATC TOOL
UNCLAMPING
FAILURE or, ARM DID
NOT LEAVE SPINDLE**

The Arm at Spindle Sensor or the Stopping Sensor has indicated that the Arm did not leave the Spindle.

1. Check Door Interlock system.
2. Check Head Position Sensor and Head position.
3. Check for power interruption to Arm Motor, fuses, and circuit breakers.
4. Check for tool release failure.

**ATC TOOL ARM
OBSTRUCTED or,
ARM DID NOT
ARRIVE AT SPINDLE**

The Arm travel was interrupted before arriving at the Spindle.

1. Check Door Interlock system.
2. Check Head Position Sensor and Head position.
3. Check for power interruption to Arm Motor, fuses, and circuit breakers.

ATC TOOL SENSOR FAILURE or, ARM NOT AT SPINDLE

The Stopping Sensor and the Arm at Spindle Sensor both indicate that the Arm has not fully positioned at the Spindle.

1. Check Door Interlock system.
2. Check Head position Sensor and Head position.
3. Check for power interruption to Arm Motor, fuses and circuit breakers.
4. Check for proper Spindle orientation.
5. Check for Tool alignment in ATC grippers.

CAROUSEL SOLENOID FAILURE (OR SENSOR) or, TURRET HAS NOT ROTATED

The Tool Count Sensor indicates that the Turret Motor has not rotated the Turret as expected within the allotted time.

1. Check Door Interlock system.
2. Check for power interruption to Turret Motor, fuses, and circuit breakers.
3. Check Tool Count Sensor and alignment.

DRAWBAR SENSOR FAILURE or, DRAWBAR NOT CLAMPING TOOL

The Drawbar Cylinder Sensor indicates that the Drawbar is still in release position, and has not retracted from the Spindle.

1. Check Drawbar mechanism for binding or no lubrication.
2. Check Drawbar Cylinder Sensor alignment.

DRAWBAR SOLENOID FAILURE (OR NO AIR) or, DRAWBAR NOT RELEASING TOOL

The Drawbar Cylinder Sensor indicates that the Drawbar could not fully release the tool from the Spindle and the Drawbar Cylinder Piston has not moved down full stroke.

1. Check for inadequate air supply or inoperative air valve.
2. Check Drawbar Cylinder Sensor alignment.
3. Check for broken Drawbar Cylinder Piston.

ATTEMPTED DIVISION BY ZERO

In a macro statement, a division by zero was attempted.

ATTEMPTED SQR OF A NEGATIVE NUMBER

In a macro statement, a square of a negative number was attempted.

ATTEMPT TO CHANGE CRC SIDE WITHOUT G40 AT N =

This message will occur when attempting to switch from G41 to G42, or vice versa, on a linear move. However, switching from one to the other is permitted during circular interpolation.

ATTEMPT TO CHANGE TOOL WHILE IN CRC MODE, N =	This message occurs if a G41 or G42 is used in the program and a G40 code does not appear before an M6 code. The G40 cannot be on the same line with M6. An M6 will not cancel CRC.
ATTEMPT TO SWITCH PLANE DURING CRC	G40 must be in effect (CRC must not be on) when changing planes. The default plane selection is G17. If CRC is in effect and a code to change to another plane (G17, G18, G19) is used, this message will appear in the block where the plane selection was called. G40 must be in effect (CRC must not be on) when changing planes.
AXIS CONTROLLER DOES NOT RESPOND DURING POWER UP SEQUENCE	This an indication that there is a communication problem between the CPU (1400) board and the axis controller (1010). Note error number and call the service department.
AXIS CONTROLLER DOES NOT RESPOND TO NC	This an indication that there is a communication problem between the CPU (1400) board and the axis controller (1010). Note error number and call the service department.
AXIS DATA TRANSFER IS STALLED	This message is the result of a data transfer problem between the 1030 card and a 1010 card. This message does not represent a serious problem and should only be reported to the service department if it is persistent. Please note error number.
AXIS DOES NOT RESPOND	This could be a problem with the 1010 card for the stated axis. Try a power off and a power on. If this does not help, call the service department.
AXIS FAULT(S) AS FOLLOWS	<p>This appears when the EMERGENCY STOP button is pressed, a motor overload relay is tripped, or the machine places itself into the Emergency Stop mode. If any message or error number is presented on the screen, record this message and number and report them, if necessary, to the service department.</p> <p>Release the EMERGENCY STOP button if necessary, then press the JOG button which will reset the machine. If the machine does not reset, call the service department.</p> <p>If the EMERGENCY STOP button is not depressed, then power off the machine and press the reset buttons of the motor overload relays to see if they are tripped.</p> <p>The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.</p>

If the machine stalls for one reason or another, the control places itself into the Emergency Stop mode. Press the JOG button and jog the tool out of the stalled position.

An axis fault also occurs during an emergency stop because the control removes the power to each of the axis servo amplifiers as a safety feature. The control then analyzes the situation and, as a part of the report, shows a servo amplifier fault, a result of having removed power to the amplifiers.

- AXIS OVERFLOW** The axis controller detected an overflow situation. Power off the machine at the main power switch, wait ten seconds, and turn the power on again. If this problem is persistent, call the service department.
- BAD CIRCLE OR MISSING G AT N** This can indicate an incorrect starting position for the circle, an incorrect ending position, or an incorrect description of the arc center location relative to the starting position of the circle. This can also indicate that the circle is programmed in absolute or incremental and the G90 or G91 codes are incorrect.
- BAD DATA OR NO TOOL DIA.** This occurs when an H word (in format 1) or the D word (in format 2) is missing in the program. When a program uses cutter radius compensation (CRC) or a fixed subroutine, an H word or D word must be specified. This also occurs when no diameter value is placed in the tool table when using Fixed Subroutines L94NN, L95NN, L9601, L9701, L9801, L9901. A DIAMETER VALUE MUST BE USED.
- When the fixed subroutine for engraving (L9201) is used, the Z depth and the R2 variable must be included in the line. See the Users Manual for the correct format. If the R0 variable is not specified, it is assumed as R0+0. If the R1 variable is not specified, it is assumed as R1+1.
- BAD FIXTURE NO.** Fixtures are numbered E1-E48. This message is displayed when a fixture offset is programmed with a value greater than 48. Format 2: G54-G59 and E6-E48 are used.
- BAD INTERPOLATION TYPE AT N** Check to see what type of move is intended for this block. G1, G2, and G3 are the interpolation codes; G1 is for linear moves and G2 and G3 are for circular moves.
- BAD R FIELD** This occurs during input from the keyboard or through the RS-232 port (from an off line computer) when the R0 was typed incorrectly. Example: If R0 is typed (R with the letter O) or R with no zero (0) is typed (R-.05 is accepted but not R.05 or R+.05).

Correct the line in the CNC; or correct the line in the text editor and send the program again.

**BAD R1 USING L91
AT N =** The R1 parameter is not present in the block containing L9101, or it is present with an improper value. L9101 has 9 possible functions that are defined by the R1 parameter; R1+1. through R1+9.

BAD T WORD AT N= This message occurs during a SUM command. It indicates that the T word value is larger than the tool changer capacity.

**BAD Z OR R0 IN
CANNED CYCLE
CALL, N =** This is a programming error due to the position of the R plane relative to the Z axis position when initializing a fixed (canned) cycle.

Problem

- 1) The R plane is described to be above the I plane.
- 2) The R plane is described to be below the final Z depth.
- 3) The final Z depth is described to end above the I or R planes.
- 4) Canned cycle not cancelled before returning to tool change position.
- 5) Canned cycle not stated in absolute as required.

Solution

Check to see if the program has the correct positioning mode (G90 or G91), and check for correct signs and decimal point placement.

Note: The H code might not be in the program.

**CALCULATED
RADIUS ERROR OF** This is a programming error that could be due to a G90 or G91 code missing or being in the wrong place (e.g.: giving absolute positions while still in incremental). Also, check the end point description, the beginning position, and the arc center description (I, J, and K), or radius designation.

This message does not appear during the Auto mode. It appears when using the SUM command (SU).

**CANCEL CRC
BEFORE G50.1 OR
G51.1 AT N =** Before using CRC (G41, G42), mirror the desired axes (G51.1). Also, turn CRC off (G40) before canceling mirror image (G50.1).

CANCEL CRC BEFORE N =	The control requires the G40 code to cancel cutter radius compensation before the line number listed. See the cutter radius compensation section for suggestions.
CANCEL Z AXIS MIRROR BEFORE TOOL CHANGE AT N	The G50.1 code must appear on a separate line before the M6.
CANNOT RETURN FROM SUBPROGRAM OR SUBROUTINE CALLING LINE CANNOT BE FOUND	This is a background editing error message. The line that called the subprogram or subroutine was deleted or cannot be found.
CANNOT START DIRECTLY IN A SUBPROG.	<p>A mid program start must begin from the main program. If it is necessary to start the program in a subroutine, the COPY command can be used to copy the subroutine to the main program as many times as the subroutine will be repeated. The programmer or operator can now do a mid program start to where the subroutine was copied into the main program. After the part is complete, delete the copied portion and run the program as before.</p> <p>Another method to start the program in a subroutine is to use the AUTO command. Enter AU,# (see AUTO command in the Operators Manual), where # is a sequence number in the subroutine. The control will process the program from the beginning of the program up to the number specified then start execution of the program.</p>
CHAR. SIZE OR DEPTH EXCEEDS 2.5 IN.	Characters that are higher than 2.5 inches cannot be engraved using the Engraving mode (L9201).
CHECK EMERGENCY STOP SWITCH AND MOTOR OVERLOAD RELAYS	If the EMERGENCY STOP button (switch) is not pressed, it is necessary to check the motor overload relays.



WARNING: Power off the machine before checking fuses, relays, and/or circuit breakers.

The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.

Also, check the brake relay fuse (F17) to see if it is burned out. It is advised to power off the machine when checking the fuses or motor overload relays.

CHECK SPINDLE LUBE, WAY LUBE AND AIR PRESSURE

When this message is displayed, it is necessary to check the way lube level, spindle oiler level, and the air pressure gauge. It is advised to check these oil levels at least once a day.

CLEAR AN EMERGENCY STOP CONDITION

An emergency stop condition was invoked. To clear, gently rotate the red EMERGENCY STOP button (if depressed) in the direction of the arrows on the button (clockwise), then press the JOG key. Wait 2 seconds for the amplifiers to be reset, then continue.

COMMAND ERROR

This occurs because a SET (parameter) command was typed in improperly. It is suggested to look in the Users Manual or use the menu of the control to help with any command format. The following is a list of available SET commands; SETH, SETX, SETY, SETZ, SETA, SETB, SETCS, SETTO, SETP.

COMMAND PROHIBITED BY THE KEY LOCK

With the key lock in the horizontal position, editing commands (CH, CO, DE, IN) cannot be used.

CRC CALLED WITH NO TOOL ASSIGNED AT N =

This occurs when the programmer has turned CRC on without an H word in the program (or in format 2 no D word).

Table 1: CRC Called w/ No Tool Assigned at N=

Format 1	Format 2
N1 G41	N1 H1
N2 H1 This is incorrect	N2 G41 This is incorrect
N1 H1	N1 H1 D1 Diameter must be specified for Format 2
N2 G41 This is correct for Format 1 style only	N2 G41 Format 2 style programming & acceptable for Format 1 style




This is a common error in one tool programs when the Z axis is set with the tool at the gauge point. Assign an H word with no tool length offset (only a tool diameter) in the tool table.

D OR H TOO LARGE

The control maintains a table of 99 tool diameters and 99 length offsets. The control will only accept H words and D words up to 99.

DATA TRANSFER FAULT

For this error, it is important to note which axis the transfer fault is on. Note the error message number and report it to the service department. There is a data transfer problem between the 1030 card (slot 8) and the 1010 card for the stated axis.

DNC MODE	This message appears after DNC is typed at the control. The VMC is then ready for DNC operation and is waiting for CNC code from the RS-232 port.
DOOR BEGAN TO CLOSE WHILE ARM WAS MOVING	This message will occur if the door begins to close during a pallet change. The feedback of the door may need adjustment or there may have been a drop in air pressure.
DO YOU WANT TO MOVE TO THE LAST HOME POSITION?	This is not an error message, but this message comes up each time the operator uses the CS command and a home position is established (SETH) in memory.
DO YOU WANT TO ZERO TOOL TABLE?	This prompt is accessed by reinitializing the memory via the RI command. A 'Y' response zeroes the tool offset table. WARNING: All tool diameter and length offsets are deleted from memory!
	
DO YOU WANT TO ZERO FIXTURE OFFSETS?	This prompt is accessed by reinitializing the memory via the RI command. A 'Y' response zeroes the fixture offset table. WARNING: A 'Y' response deletes all fixture offsets from memory!
	
DO YOU WANT REINITIALIZE MEMORY?	This prompt is accessed by reinitializing the memory via the RI command. A 'Y' response zeroes the memory of the CNC. WARNING: A 'Y' response deletes all program data from memory! After memory is reinitialized, a cold start must be performed.
	
DRY RUN OPTION ___ IS IN EFFECT	The selected dry run option (1, 2, or 3) is displayed prior to automatic execution. See the Run Time Menu for the appropriate option definition.
DUPLICATE NAME	No two programs in the library can be identified by the same number. When copying an existing program assign an unused number to the new, or copied program. Use option 2 (display program numbers) of the PR menu to determine which numbers have been used.
DUPLICATE OR BAD PROG. NAME	This will occur if a program is being input through the RS-232 port and the program has an O word in the first line that is a duplicate of an existing number in the program library. Use option 2 (display program numbers) of the PR menu

to determine which numbers have been used. Change the O word to one not being used, or remove it. Then send the program out to the control again.

If the program currently active in memory has an O word in the first line, the program will be moved into the program library when the new program is transferred through the RS-232 port. If the program currently active does not have an O word in the first line, the program will be deleted when the new program is transferred.

**DWELL, HIT START
TO INTERRUPT**

This message is displayed when a G04 (dwell) is executed by the CNC. You can interrupt the dwell time and continue program execution by pressing the START key.

**E WORD MAY ONLY
BE USED WITH G0
OR G1 N=**

Fixture offsets cannot be on the same line with a circular move (G2, G3).

EMERGENCY STOP - TAKE APPROPRIATE ACTION:

The appropriate action to take:

- 1) Release the EMERGENCY STOP button if it has been pressed.
- 2) Press the JOG button which resets the amplifiers if the problem has been resolved.
- 3) Type HO and press ENTER, then after the waiting message appears press START (see note).
- 4) The operator may now resume the running of the part.

Note: It is not necessary to move the axes to their Cold Start position or establish the part home again unless the message, JOG AXES TO HOME POSITIONS, THEN ENTER THE CS COMMAND appears.

**EMERGENCY STOP-
TAKE APPROPRIATE
ACTION**

An emergency stop has occurred. If the emergency stop button is depressed, turn it clockwise until it returns to its normal position. Press the JOG button and the message WAIT 2 SECONDS, THE AXES ARE BEING RESET appears. If this message persists, call the service department.

**ENTER AGAIN OR HIT
MANUAL TO EXIT**

This message comes up when inserting program coding at the keyboard (using the IN command) or when in Manual Data (MD) and Change (CH) mode. It is referring to the line just typed. Something was not acceptable, for example the letter O for the number zero (0) or anything else not correct. The control is still on the line with the problem, so retype the entire line.

ENTER COMMAND SETP AND SET THE MACHINE PARAMETERS	The SETP procedure will ask questions regarding the machine and certain preferences. If any of the questions are not clear, call the service department for help.
ENTER COMMAND MU TO SEE THE MENU	<p>This happens when a command is typed incorrectly. Either it is an unrecognizable command or a comma was omitted.</p> <p>This message is only a suggestion. It is not necessary to go to the menu to use a command. The menu is there to remind the operator of the command format.</p>
ERROR	This message is a program input error. It is displayed when a program block containing an error is transmitted to the CNC, via the RS-232. It appears at the time the error is received. Therefore, list the program at the control to display the last block of code received. The following block of code in the program, which was not received, is in error. This normally is caused by back to back alpha characters, which must be separated by numeric characters. Also, a 0 could have been typed as an O(s).
ERROR IN USER PROGRAM, 'SUM' TO SEE MESSAGE	See the Operators Manual for instructions on using the SUM (SU) command. This message will occur when the control lists an error message in the Auto mode, and the operator persists in running the program by pressing the START or AUTO button. This message will appear when the line in which the error exists is at the top of the stack of lines to be executed.
ERROR N WORD EXCEEDS 99999	This error occurs when the sequence number is larger than 99999. The sequence number must be between .001 and 99999.
ERROR WHILE PROCESSING BLOCK	<p>This is an indication that there may be a problem with the CPU board.</p> <ol style="list-style-type: none">1) Send the machine back to cold start alignment markers.2) Power off the machines main power, wait 15 seconds and power on the machine.3) Cold start, then run the program again. <p>If this error repeats:</p> <ol style="list-style-type: none">1) Save all the programs and offset information currently in memory, write down the present home position2) Send the machine back to cold start alignment markers.3) Use the command RI and answer Yes (Y) to all the questions. If the above does not work, please call the Service representative in your area.

ERROR(S) READING TAPE	<p>While the control is inputting a tape or program with the TA command, error option 2, this message appears after input is complete. If the TA command and error option 1 is used (TA,0,1 or TA,1,1), the input of the program would have been stopped if an error in the program is detected.</p> <p>If errors are detected, list the program and look for missing line numbers to find which lines the control did not accept. Use the IN command to insert the missing lines where needed.</p>
FILE OVERFLOW	<p>This message could mean that the program has too many G codes and M codes on the same line. It could also mean that during CRC, there were too many blocks that the control had to look beyond for the next compensated move (comment lines, Z moves, G code and M code lines, and dwells).</p> <p>This message may also appear if a fixed cycle is in effect when attempting to initiate cutter radius compensation.</p>
FIXTURE OFFSET OUT OF RANGE	<p>This message occurs when an E word value in the program exceeds 48 (or in Format 2 exceeds G54 - G59) or exceeds E1 - E48.</p>
FIXTURE OFFSET MUST BE APPLIED WITH G0 OR G1	<p>Fixture offsets can not be applied on lines with circular interpolation (G2 or G3). Fixture offset must be applied prior to commencing arc command line (at G0 or G1) or unused fixture offset removed if not applicable to arc.</p>
G28 AND G29 USED WITH CUTTER RADIUS COMP AT SEQ	<p>Before the control can execute the codes G28 or G29, the program must turn CRC off (G40).</p>
G31 USED WITH AN INCOMPATIBLE WORD OR MODE	<p>Only G1, P, and F are allowed with a G31. No other codes are supported. CRC, mirror image, rotation, and drill cycles are not allowed during the execution of a G31 code.</p>
G45-G48 & G52 ARE NOT ALLOWED WITH ROTATION, N =	<p>Codes G45-G48 and G52 are incremental offsets. Rotation can only be used in the Absolute mode. Therefore, these codes are not allowed.</p>
G91.2 IS NOT ALLOWED IN FORMAT 1	<p>The machine is in format 1 with the G91.2 code in the program. The G91.2 code can only be used in format 2. In format 1 you must remain in the Incremental mode when G91.1 is coded.</p>
G92 CANNOT BE USED IN CRC MODE, N =	<p>The G92 code should be used in a line before CRC is turned on. The code G92 can only be used after a G40 (G40 is a default code).</p>

G92 MUST BE ONLY G CODE IN BLOCK	There can be no other G codes in the line with a G92. The program does not need to be in absolute (G90) to use the G92 code.
GNN IS AN UNSUPPORTED G CODE AT N	The G code displayed is not a supported code of the FADAL CNC 88.
HELICAL MOVE TOO SHORT, N =	See error message HELICAL RISE TOO STEEP.
HELICAL RADIUS TOO SMALL, N =	See error message HELICAL RISE TOO STEEP.
HELICAL RISE TOO STEEP, N =	The radius of the circle and the helical rise are radically different in length (usually the rise is much longer in comparison to the radius). Also, depending on the programmed feed rate, the control may or may not be able to handle the situation. Reducing the feed rate in the program can sometimes correct this problem.
I, J, OR K MUST BE SPECIFIED AT N =	This message is displayed when an X, Y, or Z is used with an R0 without a G2 or G3. Full circles must use either the I, J, or K and the direction around the circle. See the circular interpolation section for more details.
ILLEGAL O WORD	This error occurs during program input via the RS-232. The O word must have a value of 1 through 9999. No other character or symbol is allowed.
ILLEGAL G CODE DURING G91.1 MODE AT N =	This message appears when an illegal G code is programmed while in G91.1. See the Section Three for acceptable G codes.
IMPROPER USE OF CANNED SUBR.	This message may appear for the following reasons: <ol style="list-style-type: none">1) No tool diameter in the tool table or No D word was specified.2) If the programmer omits a required R word from the subroutine.3) If the programmer does not cancel a fixed cycle with a G80, G28, G29, M6, G49, or H0 before using a fixed subroutine, except for L93 (bolt hole).4) Using L96 - L99, if R1 variable is equal to the radius of the tool. The R1 variable is used for the radius on the corner of the tool (fillet radius), not the tool radius. For example, for a 1.0 diameter end mill with a .125 corner radius, the R1 variable will be R1+.125, and the tool diameter entered in the tool table will be 1.0.

5) The G41 or the G42 code has not been canceled with a G40 before using a fixed subroutine.

**INCHES MODE
REQUIRED -
OPERATOR MUST
SET**

The CNC is in the Metric mode (SETME), and the program has a G20 or G70 code to verify that the control is in the Inches mode. To run this program, the operator must set the CNC to the Inches mode by entering the SETIN command.

**INCOMPATIBLE G
CODES AT SEQ**

Some codes, even though they are modal and from different groups (families), are incompatible with one another. Break up the grouping of G codes on the line by moving some of them to the line just before or after the line they are currently on. This messages is also displayed when any G code is on the line with a G53.

EXAMPLE:

N20 G1 F20.

N21 G8 G41 X.5 The G41 and the G8 are incompatible

Move the G8 to the line with the G1

N20 G1 G8 F20. This is correct

N21 G41 X.5

**INCREMENT DIVIDED
BY TWO**

If renumbering causes a line in the program to be greater than 99999, the increment for renumbering is divided by two and renumbering is resumed.

All of the lines or program blocks must have sequence numbers. The NU command is used to renumber the lines in the program by the increment specified in the first parameter. For example, entering NU,5 will renumber the lines, incremented by five. If program is large, renumber by 1.

**INCREMENT TOO
LARGE**

This message occurs when the increment parameter of the program input (IN) command causes the sequence numbers to exceed 99999 or a typing error in the program has been given a value that is too large. Example: X-123456789.

**INCREMENT TOO
SMALL**

This message occurs when the increment parameter of the program input (IN) command is less than .001. Example: IN, 0.00001.

**INPUT XMODEM
TRANSMIT BLOCK
MISSED ERROR**

A packet has been missed or came out of order during Xmodem transmission.

**JOG AXES TO HOME
POSITIONS, THEN
ENTER THE CS
COMMAND**

This message appears at power on. If the axes are at the Cold Start position (machine home), enter the CS command. The operator needs only to jog the axes if they are not at the Cold Start position.

This message also appears when the tool changer crashes and the AUTO button is pushed or the MD command is used. However, the operator does not have to jog to Cold Start. Use the SETCS command to reference the Cold Start

position, then enter HO to return the axes to home, and when the axes are back at the Cold Start position, visually check, and then enter the CS command.

SETP will also make this message appear. If the operator used the SETP command at the part home, handle the situation in the same way as suggested for a tool changer crash (see above).

**LOOK AHEAD WAS
CANCELED BY
OPERATOR**

This indicates that the NO LOOK AHEAD buffer option of the Run Time Menu has been selected. The CNC processes only one block at a time, while displaying only two blocks of the program on the screen. This option can be selected or canceled using the Run Time Menu by typing MU in the Automatic mode.

**M,S,T LOCKOUT IS
IN EFFECT**

This message indicates the M function, Spindle, and Tool change lockout option of the Run Time Menu has been selected. In this mode, M function, Spindle or Tool Change commands will not be performed during Dry Run. This option can be selected or canceled using the Run Time Menu by typing MU in the Automatic mode.

**M FUNCTION TOO
LARGE AT N =**

M functions cannot exceed a 2 digit value. See Section Eight, Commands or the menu (MU) for a list of the accepted M functions.

**MAIN PROG. NOT
FOUND**

This is a programming error. There are subroutines written in the beginning of the program without an M30. M30 is used to separate the subroutines from the main program. Insert the M30 in the proper place. See Section Eight, Commands for the accepted format for subroutines. The M30 code on the first line of the program and a subroutine heading (L100) on the next line will also cause this error message to appear.

```

N1 O1 (START OF PROGRAM
N2 (ACCEPTABLE PROGRAM EXAMPLE
N3 L100 (SUB #1
N4 X1.
N5 *
N6 L200 SUB #2
N7 L105
N8 M46
N9 X-5.Y-1.
N10 M47
N11 *
N12 L300 (SUB #3
N13 G91
N14 L204
N15 L104

```

N16 G90

*N17 **

N18 M17 This ends the last subroutine

N19 M30 This separates the sub section from

*N20 * The main program*

N21 (MAIN PROGRAM

**MEMORY ERROR,
RELOAD PROGRAM**

This is a memory error, turn the machine's main power off. Then power on and reload the program. If this does not work, save all the programs in the memory (PU,3) and use the RI command, answering Yes (Y) to all the questions. After using the RI command, load the programs back into the control.

**MEMORY ERROR,
RESPOND WITH Y TO
DELETE BAD
BLOCKS**

There is a possible hardware problem when this message appears. If the operator answers Yes (Y) to this question, the control may delete some of the program in memory. Answer Yes (Y) only if the program in memory can be replaced (if the program is stored on paper tape or on computer disk). Answer No (N) if the program needs to be saved.

Turn the machine's main power off, then power on and reload the program. If this does not work, save all the programs in the memory (PU,3) and use the RI command, answering Yes (Y) to all the questions. After using the RI command, load the programs back into the control. If this does not help, call the service department.

**METRIC MODE
REQUIRED -
OPERATOR MUST
SET**

At power on, the control is in the Inches mode. The Metric mode must be turned on by using the SETME command.

This message will appear if the G21 or G71 code is present in the program to verify that the Metric mode has been turned on by the operator.

MOTOR OVERLOAD

This is an indication that something has stalled or overheated a motor, and is associated with an emergency stop (see EMERGENCY STOP).

**MOVE EXCEEDS AXIS
LIMIT AT N =CHECK
PROGRAM AND
TOOL OR FIXTURE
OFFSETS**

The most common reason this message appears is because of either a missing or an extra G90 or G91 code. This message will appear only when the control is in the Auto mode or Manual Data (MD) mode. This message will not appear in the Sum mode (SU).

When the control is executing the program, processing of the program is ahead of the current line that is being executed. If the control detects an over travel situation while processing the program, then execution of the program will stop immediately.

The setup person may have to consider moving the fixture to another location on the table. The program may be correct, but the fixture may be set up incorrectly. The programmer can provide information to the setup person, indicating how close to the axis limits the fixture can be located.

**MOVE TURRET TO
TOOL 1 AND ENTER
SETTO COMMAND**

When this occurs, the head will be above the tools in the turret with the turret ready for loading. Move the turret using the turret CW or CCW buttons so that the tool designated to be tool #1 is under the spindle, then remove tool #1 from the turret. Press the JOG button. The head will stay where it is (4 above the Z axis CS position), and the turret will move back to its home position. Now use the command SETTO to establish this turret position as #1. Use the HO command or jog the head down to the Z axis CS position. Next use the command TC,1 to reopen the turret. Replace tool #1 in the turret and press the MANUAL button to bring the head down over tool #1.

**NEXT ENTRY IS OUT
OF RANGE**

This message is displayed by entering pitch error compensation for the ball screw when an entry exceeds the maximum allowable difference of 9 (see Survey command in the Maintenance Manual). For example, if an entry is 15 and the next consecutive entry is 0, the difference is greater than 9.

**NO ANGLE, THE
POINTS ARE THE
SAME**

The calculated angle of either function 3 or 4 of L9101 is the same as the expected angle.

**NO MEMORY
AVAILABLE FOR NEW
PROGRAM**

When this occurs, the operator will have to remove (delete) as many programs as necessary from the program library in order to input another program from the RS-232 port. The standard memory capacity is 38K. FADAL distributors have a memory expansion board available that increases the memory capacity to 16 MEG.

**NO OTHER WORDS
ALLOWED WITH O
WORD**

If program coding is used on the first line of the program along with the O word, this message appears. A comment is allowed with an O word and is used to identify each program in the library. The first sixteen characters of this comment are shown when the program library menu displays the programs in memory.

EXAMPLE: *N1 O1 G0 G90 (P/N 1234 This line is incorrect*
N1 O1 (P/N 1234 This line is correct
N2 G0 G90

NO PARAMETERS OR PARAMETERS CORRUPTED MACHINE DEFAULTS USED	This message occurs most commonly after the memory has been zeroed from the DI diagnostics mode. This will also occur if there is a memory error. Enter the machine parameters with the SETP command. The parameters to use are usually found recorded on a check off sheet on the inside door of the pendant.
NO RESPONSE FROM AXIS	<p>This happens when powering on the machine and the CPU (1400) does not get a response from the axis controller (1010).</p> <p>If this message appears for the X, Y, Z, A, or B axes, power the machine's main power off and then on again. If this does not help, call the service department.</p>
NO TOUCH OR INCOMPLETED POINT AT N =	The move that causes the probe to touch the part was completed without a touch. Increase the length of the move so that the probe touches. Also, you must activate the selected probe by coding an M64 for an MP8 probe, or M65 for a TS20.
NOTE ERROR THEN HIT MANUAL	<p>When the programmer is inputting a program through the RS-232 port (using the command TA,1,0 or TA,1,1), the input will be terminated if the control detects a bad programming word (EXAMPLE: O for 0, or double letters (XX1.5), etc.). The programmer will have to edit the program at the computer or Teletype before inputting the program again.</p> <p>The control will ignore all errors in the program and continue input by using the error option 2, with the TA command (EXAMPLE: TA,1,2). An error count is given at the end of transmission. If there were errors detected, list the program to find missing line numbers. If the lines are numbered in sequence before transmission of the program, look for the numbers that are out of sequence. After the numbers have been located, use the IN command to insert the missing lines.</p>
NUMBER OUT OF RANGE	This message appears when a number of twelve digits or greater is typed in. This error may be detected by the machine after making a calculation. The calculated number may be out of range, for example, the macro calculation is less than .0001.
OFFSET ENTRY ERROR	This message is displayed using the Utility (UT) command, by pressing the MANUAL key when the CNC prompts you for a tool diameter. It is also displayed by entering a diameter value that is too large.
ONLY BLOCK SKIP ALLOWED WITH MACRO	The only non macro command allowed with a macro line is the block skip character.

ONLY M3, M4 & M5 ARE ALLOWED WITH M6 AT N =	This message appears when the M6 is coded with an M function other than an M3, M4, or M5. The unacceptable M function must be moved to another line.
ONLY Z, L, R & F WORDS ALLOWED	This message appears when the programmer uses the fixed subroutine L9201 for engraving, and has programmed other words in the line with the L9201 other than Z, L, R, and F.
ORIENTATION FAILURE	Visually check the air pressure, it must be between 80 and 90 PSI. If this problem persists call the service department.
OUT OF FILE SPACE, A FILE COMPRESSION IS BEING TRIED	<p>This is a message that informs the programmer that the memory of the control is almost full. It is usually displayed when editing the active program in memory, or during program input via the RS-232. This does not destroy any portion of the program. A file compression organizes the memory so that unused portions are made available. The NE command will perform a file compression before prompting you with a (Y or N) to delete the current program. Also, a file compression is performed by deleting the current program using option 5 from the PR menu. It is always best to retry the command that caused this message to appear. After the compression, the command will often work the second time. If unwanted programs are stored in the program library, remove them to create more room in memory. If there is only one program in memory, try to reduce the size of the program by removing unnecessary comments and making full use of all modal codes. Create subroutines for repetitive portions of the program wherever possible. The program may have to be broken up into separate operations.</p> <p>The standard memory capacity is 38K. FADAL distributors have a memory expansion board available that increases the memory capacity to 16 Megabytes.</p>
O WORD OUT OF RANGE	This message occurs using the program input (IN) command when typing an O word greater than 4 digits. O words must be an integer of 1 through 9999.
PALLET DISABLED IN PARAMETERS	This message occurs when a pallet changer command or code is used and the SETP parameters indicate no pallet changer exists on the machine.
PALLET IN STORAGE	This message occurs when there is an attempt to store a pallet where a pallet is already stored.
PALLET NOT FULLY STORED	This message occurs when the time to store a pallet exceeds the allotted time. The feedback on the arm may need to be adjusted. Call the service department for assistance.

PALLET MUST BE CLAMPED

This message occurs when both pallets are in storage and a command is given to make a pallet change. One of the pallets must be loaded and clamped on the table before a pallet change can occur. The feedbacks for pallet loaded and pallet stored are being read at this time.

PARAMETER ERROR

After a command is entered and a comma is typed, the control is expecting a parameter. If a parameter is not typed before pressing the ENTER button, this error message appears.

EXAMPLE: *AU, This is a parameter error*

Note: If the operator is not using a parameter, for instance, the second and third out of a possible four, then the parameters can be omitted when the fourth parameter is typed in.

EXAMPLE: *AU,50,,,1 This is acceptable*

Also if the DE command is used to delete a line that does not exist, or the CO command is used to copy lines that do not exist, this message appears.

PARITY ERROR

This message appears during input through the RS-232 port and is caused by several factors:

- 1) Unshielded cable is being used, and the cable is subject to electrical noise.
- 2) The baud rate is too fast for the length of cable used.
- 3) There is a failure in the RS-232 port (hardware problem).
- 4) There may be a problem with the computer that is sending the program.

PARITY ERROR DURING DNC

This occurs during DNC transmission of data (see the error message PARITY ERROR).

PLEASE PUT AN O WORD AT THE FIRST OF THE CURRENT PROGRAM THE FOLLOWING PROGRAMS ARE IN MEMORY

To use the PR command, the active program must have an O word and a number in the first line. The number must be different than the other numbers in the program library. This is why it lists the other programs in memory.

EXAMPLE: *N.1 O43 (P/N 34-765)
Here, a line was inserted before N1 using the command IN,.1
N1 GO G90 S10000 M3*

	<p>A comment can be added to the line with the O word. These comments help identify the program.</p>
POINTS ARE ON SAME LINE AT N =	<p>The 3 touch points of L9101 function 1 are on the same line. Check the program for positioning errors.</p>
POSSIBLE PROBE OVER TRAVEL	<p>During the UT command, this error occurs for two cases:</p> <ol style="list-style-type: none">1) when the Z axis home position is set below the Cold Start position; and the tool change is made to the Probe.2) when there is a positive offset value used, and the tool change is made to the Probe.
POSITION LIMIT	<p>This message will occur when the program has instructed the tool to move out beyond the axis limits. When the tool moves beyond the axis limit the CNC issues an emergency stop. The machine will stop at the line that caused the over travel.</p>
PRESS Y TO KEEP THIS POSITION PRESS N TO RETURN TO LAST POSITION	<p>These messages appear after jogging while in Slide Hold or Single Step modes. See Section Eight, Commands, for the proper application of the Jog Away feature.</p>
PROBE TEST = FAILURE	<p>Testing the probe by using the Utility (UT) command failed. Verify that the probe was properly interfaced. If an MP8 or MP9 is being tested, be sure the 9- volt battery has a charge. Also remove any obstructions between the probe (transmitter) and the collector unit (receiver).</p>
PROBLEM POSITIONING SLIDES TO ZERO	<p>This error only occurs during Cold Start, when the Glass Slides are not aligned close enough to the indicators during Cold Start. Place the machine in Jog and manually align the Cold Start indicators more accurately.</p>
PROGRAM DOES NOT EXIST RETRY OR HIT MANUAL TO EXIT	<p>This happens when the selected program number does not exist in the program library. Enter a program number that does exist, or press the MANUAL button, and then use the PR command again to display the programs that are stored in memory (option 2).</p> <p>The programmer may have put an O-1 or an O.1 as a program number. The number will show up when the programs are displayed in the library but the control will not be able to retrieve the program. If the operator uses the PU,3 command and presses the EMERGENCY STOP button when the lost program is being displayed on the screen, that program is now active in memory and can</p>

be edited. Delete the bad O word, from O-1 or O.1 (which appears as 1000 when it is listed in PR) to O1.

**PROGRAM NOT
FOUND**

This message is displayed when the PA command is used to display the active program, and no program is active. Use the PR command to activate the desired program stored in memory.

**PROGRAM BLOCK
NOT FOUND**

This message is displayed when the LI command is used to list selected blocks of the active program, and the selected blocks are not found or the program is not active. View the program by using the PA command.

P WORD TOO LARGE

The P word for referencing a line number using M99 is too large of a value. Line numbers can not exceed 99,999.

**RESET THE
EMERGENCY STOP
SWITCH**

The EMERGENCY STOP button has been pushed. Turn the button clockwise until it is released and then push JOG.

**RESOLVER FAULT OR
SCALE ERROR**

This message may occur in three cases as follows:

- 1) A failure of the resolver.
- 2) An axis runaway during power on.
- 3) A large Scale or Motor error, indicating a possible Scale failure.

Call the service department when this message appears.

Note: Do not run the machine when this message appears.

RAILS NOT ALIGNED

This message occurs when the rails for the pallet hanger are not aligned or the feedback switches need adjustment. Call the service department for adjustment.

**RETURN PALLET TO
THE LOAD POSITION**

This message occurs when a pallet change is attempted and the other pallet is away from the load position. The operator must slide the pallet into the load position before the other pallet on the table can be changed. The feedback from the pallet loaded and pallet stored are being read at this time.

**ROTARY AXIS MOVE
TOO LONG. N =**

The maximum incremental rotary move for one block is 1080.00 degrees. If it is necessary to go further, break up the move into several blocks.

**RS-232 ERROR
DURING DNC**

This error only occurs during DNC operations. It may occur while using 9600 baud to DNC. Slow the baud rate. It may also indicate line noise or the possibility of dropped characters. These would indicate a possible problem

with the RS-232 cable or the communications port. Check the cable and communications port and retry DNC.

Also, failure of the CPU may cause this error. If the above corrections are unsuccessful, contact the maintenance department.

SCALE ERROR	A scale error has been detected. The LEDs in the scale interface box in the back of the machine will indicate which axis gave the error. Call the service department.
SEE MENU FOR NEW CD FORMAT	The CD command is different for some versions of software. See the menu for these changes.
SEQUENCE NUMBER TOO LARGE	This error occurs using the Program Input (IN) command when the 'From' parameter is specified greater than the maximum allowable number. Sequence numbers can not exceed 99,999.
SEQUENCE NUMBER TOO SMALL	This error occurs using the Program Input (IN) command when the 'From' parameter is less than the minimum allowable number. Sequence numbers can not be less than .001.
SERVO AMPLIFIER FAULT	This message appears when the EMERGENCY STOP button is pressed. The operator must pull the button out (for '85 and older models) or turn it clockwise (for '86 to present models) and then press the JOG button to reset the amplifiers. If the amplifiers do not reset, call the service department.
SINGLE STEP	<p>This message indicates that the control is in the Single Step mode. If the green START button is pressed when in this mode, the following program block is executed. The feed rate potentiometer affects all axis motion.</p> <p>At the end of each block, the operator can press the JOG button and jog away from the current position without aborting the operation. Program execution can be continued at the new location, or the axes can be returned to the position at which the Jog function was initiated. To exit single step, press the AUTO button for continuous block execution.</p>
SLIDE HOLD	<p>Pressing the SLIDE HOLD button stops all axis motion. The distance to finish the move is presented under the blinking SLIDE HOLD message.</p> <p>It is possible to press the JOG button and jog away from the current position while in the Slide Hold mode without aborting the operation. Program execution can be continued at the new location, or the axes can be returned to the position at which the Jog function was initiated. To exit from the Slide Hold mode, the operator must press the START button if in single step, or press the AUTO button for continuous block execution.</p>

**SPINDLE
CONTROLLER DOES
NOT RESPOND**

Note the error number and see the list at the end of this chapter. This simply could be a problem that occurs when the operator is powering on the machine. Turn the machine's main power off. Press the motor overload relay button, and then power on again. If this does not help call the service department with the error message and number.

The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay. Call the service department first.

**SPINDLE
CONTROLLER OR
DRIVER FAULT**

Note the error number and see the list at the end of this chapter. This message is displayed if the spindle was commanded to turn on after an emergency stop without pressing the JOG button to reset the amplifiers.

If this message appears when not in the Emergency Stop mode, take these steps to help the service department analyze the problem:

- 1) Look at the air pressure gauge and write down the current air pressure (Do not change it at the machine!)
- 2) Check to see if the belts are on the pulleys.
- 3) Do not turn the power off.
- 4) Call the service department and report this problem (while the machine is on).

**SPINDLE
CONTROLLER
SOFTWARE UPDATE
IS REQUIRED**

The current software module (1610) detected spindle software that was not compatible. Call the service department.

**SPINDLE DRIVER
FAULT**

This is the result of the air pressure being too low while the spindle is on. Air pressure should be set between 80 and 90 PSI. Visually check the air pressure and start the program over again. If the air pressure is not the problem, power off the machine and check the motor overload relays to see if they have been tripped.

The motor overload relays can be recognized by their white reset buttons. They are located in both control enclosures. The white button must be pressed to reset a relay.

**SPINDLE FAILURE
WHILE TAPPING**

If the spindle stops while in G74, G75, or G84 modes, this message will appear. Problems such as a dull tap, undersized hole, poor tapping lubricant, or a shallow hole must be eliminated. The programmer may want to consider

thread milling as an alternative to tapping. Call the service department if this is persistent.

**SPINDLE FAILURE
DURING REVERSAL**

This will occur when an M3 and an M4 are in the same line. If the spindle stops while in G74, G75 or G84 modes, this message will appear. Problems such as a dull tap, undersized hole, poor tapping lubricant, or a shallow hole must be eliminated. The programmer may want to consider thread milling as an alternative to tapping. Call the service department if this is persistent.

**SPINDLE HAS
FAILED TO TURN ON**

Either the Hall Effect Switch on top of the spindle pulley has not recognized that the spindle is turning, or the spindle actually has not turned on and there is another problem. Call the service department.

SPINDLE FAULT LINE

The spindle inverter has detected a fault. See the fault number on the inverter and report this to the service department.

**SPINDLE MOTOR
TEMPERATURE
FAULT**

Check to see if the spindle fan is on. If it is not, turn machine's main power off and check the fuse for the spindle fan (F24). Allow time for the spindle motor to cool down and attempt to run the machine again. If this message persists, call the service department.

**SPINDLE WOULD
NOT STOP**

The spindle would not stop in the designated amount of time (12 seconds). If this message is persistent, call the service department.

STACK OVERFLOW

This message should be reported to the service department. It is caused by excessive stacking of keyboard commands.

**SUBROUTINE CALL
IS NOT ALLOWED IN
MDI**

Subroutines can be executed in AUTO mode ONLY.

**SUBR. DOES NOT
EXIST**

This message indicates that a subroutine was called that is not in the subroutine section of the program. Check to make sure that the subroutines were numbered properly.

This message also appears if the G66 code is used on a line that does not have a subroutine call on it.

**SUBR. NESTING
ERROR**

This is a message that occurs when a subroutine heading (L0100) is entered after the M30 code. This message will also occur if a Fixed Cycle is still active (use a G80 to cancel) or G68 (rotation) is still active when a tool change or end of program is coded (use a G69 to cancel).

N20 M17

N21 M30

N22 L200 This is not permitted after an M30

TAPE INPUT TERMINATED	This message reports to the operator that the tape, or program input process is complete.
TAPE IS GOOD	When verifying a punched tape, this message appears if the tape is accepted. This message also appears if no errors are detected in transmission when the TA command is used.
TEMPERATURE FAULT	For any temperature fault, the operator must check the cooling fans; if they are not operational, turn the machine's main power off and check the fan fuses (F23, F24, F27, and F28). If this message is persistent, call the service department.
TEMPORARY CONFLICT WITH AUTO	This message occurs when background editing is being used. The edit just made is in the area where AUTO is currently executing. The change will be made when AUTO is out of that area.
THE AXIS CONTROLLER IS NOT A 1010-4	See Survey in the Maintenance Manual.
THE SURVEY CONTAINS ERRORS, PLEASE REVIEW	See Survey in the Maintenance Manual.
THE SURVEY WAS NOT WRITTEN TO THE AXIS CONTROLLER	See Survey in the Maintenance Manual.
THERE IS NO SURVEY	See Survey in the Maintenance Manual.
THREAD LEAD NOT SPECIFIED AT N =	Fixed cycles G74, G75, and G84 need a Q word in the line that represents the decimal thread lead of the tap (see the Users Manual for the proper format). If the Q word is omitted then this message will appear. If the programmer uses the fixed cycles G74, G75, or G84 and does not cancel the cycle with any of the following codes: M6, G28, G29, G49, G80, H0, then this message will appear at the next Z move.

TOO MANY BLOCKS FOR GAP	This happens when the CO (copy) command is used and there are too many lines being copied between existing lines in the program. Use the NU (renumber) command and copy again, now that the program is renumbered.
TOO MANY M FUNCTIONS AT N=	This error occurs when another M function is coded with the M17 or M30. These two codes must be the only M function on the line.
TOO MANY CONSECUTIVE NON MOTION BLOCKS	<p>This error message appears if a program contains over sixteen consecutive comment blocks (with no program coding in the lines). Put coding between the comment lines or put comments on lines with program coding, so that the program can have comments on every line.</p> <p>Comment blocks in the beginning of the program can be placed in an unused subroutine. This way the control never reads the lines when executing the program, but the operator can list the program and read the comment lines.</p>
TOO MANY PARAMETERS	Commands have associated with them a certain number of parameters. If more parameters are typed in than allowed, this message appears.
EXAMPLE:	<i>AU,50,,,1, or AU,50,,,3,1</i>
	Refer to Menu (MU) on the control for the correct format of each command.
TOO MANY SUBR. CALLS	Nesting can only be seven deep. If the program goes further than seven deep, then this message appears.
TOOL BREAKAGE DETECTION = FAILURE AT N =	When using the TS-20 touch probe is used for tool breakage detection, this error message indicates the program line where the broken tool is detected. Replace the tool and continue the program.
TOOL DIAMETER TOO LARGE AT N=	<p>This error message is displayed in cutter radius compensation when the programmed move is smaller than the radius of the cutter. To correct this problem:</p> <ol style="list-style-type: none">1) Check the tool diameter entered in the tool table to see if it was entered correctly.2) Check the program to see if it contains any typographical errors.3) If the intention was to make a rough pass using a larger tool diameter than the finish pass tool diameter, the smallest programmed inside radius, on the part, should be as big as the tool radius (half the tool diameter) entered for the roughing pass.

- 4) Check the math work to see that it is correct, and was transferred correctly to the program.

When debugging a program that uses cutter radius compensation, use a zero diameter in the tool table when running the program or when using the SUM command for the first time. Then put the tool diameter in the tool table and try it again. This will make the debugging process easier.

**TOOL OFFSET NOT
ALLOWED DURING Z
MIRROR**

If mirroring the Z axis is desired, first call up the tool length offset, then code the G51.1 Z0. The H word must be on a line in the program before the G51.1 Z0.

**TOOL NN IS IN THE
SPINDLE**

Identifies the current tool that is in the spindle. This is displayed when entering the program execution (AU) and the Manual Data Input (MD) modes, and is also listed in the tool table display (DT).

**TOOL NUMBER TOO
HIGH**

The starting tool number of the utility (UT) tool setting cycle must not exceed that of the tool changer capacity.

**TOOL TURRET
LOCATION IS NOT
SET**

Rotate the tool turret so that tool #1 is adjacent to the spindle and then type SETTO.

**TRANSFER ERROR,
PLEASE RETRY**

See Survey command in Maintenance Manual.

**TROUBLE READING
THE EXTERNAL
SLIDE HOLD SWITCH**

The feedback from the external slide hold switches is erratic or inconsistent. This is an indication that the external slide hold switch system is faulty. Call the service department.

**TYPE A Y TO
IGNORE, OR TURN
POWER OFF AND
THEN ON AGAIN**

This error message appears only after the message AXIS CONTROLLER DOES NOT RESPOND DURING POWER UP SEQUENCE. This is an indication that there is a communication problem between the CPU and the axis controller. Call the service department.

**UNDEFINED MACRO
ERROR**

This message will appear when situations during a macro execution cause the system to fail, but there is no specific error or message associated to the situation.

**WAIT 2 SEC., THE
AXIS DRIVERS ARE
BEING RESET**

This message occurs when the JOG key is pressed after an axis fault. The control returns to the Command mode after 2 seconds. The axis drivers are reset and the machine can continue normal operation.

WAITING	In the WAITING mode, the operator has the choice of pressing either the START or AUTO button to execute automatic motion. Or, the operator can press the MANUAL button to exit to the Command mode.
WAITING ON AIR VALVE	When this occurs, the control is waiting on a signal from one of the air valves indicating that the valve has completed its job; usually air pressure to the machine has dropped below 80 PSI. If this message persists, call the service department.
WARNING POSSIBLE GOUGE AT N =	Generally caused by math errors in the program. The CNC has processed a move that possibly could cause the cutting tool to damage the part. The error usually is not large enough to make any noticeable difference on the machined part. Therefore, only a warning is displayed. If there is a noticeable difference, review your program math.
WAY LUBE PRESSURE SWITCH FAILURE	Check the way lube oil level if this message appears.
WRITING ERROR, WAIT 2 MIN. AND RETRY	See Survey in the Maintenance Manual.
X,Y OR Z MOVE MUST BE SPECIFIED AT N =	Probe function #1 (L9101) has been programmed without an axis specified for the given plane. For example, if probing is to be used in the XY plane (G17), X and/or Y must be programmed in the same block as L9101 R1+1. Z can be used if in the ZX (G18) or YZ (G19) plane.
YOU CANNOT DELETE THE PROGRAM THAT IS CURRENTLY ACTIVE	To delete a program in memory, it must be in the library. Use the PR command and switch the program currently in memory with a program in the library. The program can then be deleted. If there are not any programs in the library, use option 3 to start a new program. This will put the currently active program in the library and then the program can be deleted. Or just simply use the DE command and delete starting from the first block in the program to the last.
YOU HAVE A VERSION UPGRADE OR MEMORY HAS BEEN CORRUPTED... MEMORY NEEDS TO BE ZEROED	This occurs when the control detects that a new module has been inserted. See the Maintenance Manual for proper zeroing procedures. This may also occur because of a memory error and the memory needs zeroing.
YOU MUST ENTER THE BACKLASH TABLE	The CNC does not have a valid backlash table. Enter the backlash for each axis by using the BL command. The backlash set at the factory is documented on the inside surface of the pendant door.

The backlash information is retained by the battery backed memory card. If this message appears again after the BL command has been used, call the service department.

**YOUR VERSION OF
CNC MAIN NEEDS TO
BE UPDATED**

The control has detected an incompatibility with the CNC memory manager software on the 32MP pendant. Call the service department for an update.

**Z, Q, OR F MISSING
IN CANNED CYCLE
CALL, N =**

See Section Four, Fixed Cycles for the proper format for each fixed cycle.

**Z & M6 LOCKOUT IS
IN EFFECT**

This message indicates the Z axis and Tool Change Lockout option of the Run Time Menu has been selected. In this mode, Z axis movement and tool change commands will not be performed during Dry Run. This option can be selected or canceled by using the Run Time Menu.

**Z AXIS MUST BE AT
COLD START FOR
TC,1**

Check to see if the Z axis alignment was properly set at cold start. The operator cannot use the TC,1 command unless the Z axis is at the Z axis CS position. Use the HO command or jog the axis to the Cold Start (Z0).

If the SETZ command has been used, first use the HO command to bring all axes home, then enter the SETCS command. Jog the Z axis to Z0 and then use the command SETH so the X and Y home position is not lost. The TC,1 command can now be used to open the tool changer. If necessary return the Z axis to the previous setting and use the SETZ command. (See the error message JOG AXES TO HOME POSITIONS if the tool changer crashes.)

**Axis/Spindle
Controller Error
Messages**

Descriptions

Error numbers help discern the source of the problem the axis or spindle is reporting. The error number and text will follow the message CALL THE SERVICE DEPARTMENT AND REPORT THE MESSAGE AND NUMBER ON THE SCREEN TO CORRECT THESE PROBLEMS.

Error numbers from the AXIS or SPINDLE CONTROLLER

(see MU pages 40-41)

**1. NO MOTOR
FEEDBACK**

This message appears on power up if no motor feedback (resolver or encoder) or no motor motion is detected by the axis controller.

- 2. ENCODER NOT RESPONDING PROPERLY**

This message appears during the rigid tap operation if the encoder feedback is interrupted during the spindle operation for any period longer than 3 seconds. The spindle operation is halted and this error message is displayed.
- 3. LOGIC JUMPER INCORRECT or COMMAND SIGNAL MISSING**

This message appears on power up. An illogical motor-tach jumper configuration or failed component may cause the axis to run away in the opposite direction of the command signal. To prevent runaway, axis operation is halted and this error message is displayed.
- 4. ENCODER IS NOT 1024 LINE**

This message appears on power up, only for axes with a 1024-line encoder as motor feedback, if the count of the encoder lines per revolution of the screw is not 1024 lines per turn.
- 5. NO INDEX MARK DETECTED**

This message appears during the cold start process, if the index mark of the axis' primary feedback device, encoder, or scale is not detected.
- 6. SWITCH 1 DISABLED**

This message is not used.
- 7. RIGID TAP PRECYCLE ENCODER COUNT PROBLEM (NOT ENOUGH COUNT)**

This message indicates that the spindle feedback is not functioning properly. Prior to the rigid tap cycle, the spindle feedback is checked for operational accuracy. Malfunction of the feedback results in termination of the cycle and display of this error message.
- 8. SPINDLE MAGNET NOT DETECTED or SPINDLE NOT RUNNING**

This message appears during spindle operation if the spindle magnet is not detected for any period longer than 3 seconds. This problem could be related to either command signal, spindle not turning, or spindle magnet malfunction.
- 9. SPINDLE FAULT LINE DOWN**

The fault line signal down to the inverter should stay high during spindle operation. If this signal is held low when the spindle is running then this error message is displayed.
- 10. ENCODER AND MAGNET NOT RESPONDING or SPINDLE NOT RUNNING**

This is a spindle operation error message. During spindle operation, the encoder feedback and spindle magnet are monitored. If no proper response from either of them is detected for any period longer than 3 seconds, then this error message is displayed. This error could also be caused by the command signal, i.e., if the spindle is not turning.
- 11. MOTOR OVERLOAD. EXCESSIVE FOLLOWING ERROR**

This message is not used.

- 12. MOTOR OVERLOAD. PULSE COMMAND STEP COMMAND CHECKSUM ERROR** This message appears at the completion of a move if the number of pulses required to complete the move does not match the number of pulses received from the CPU by the axis controller, usually indicating a hardware-related problem.
- 13. MOTOR OVERLOAD. FOLLOWING ERROR GREATER THAN THE OVERLOAD FACTOR** This message is not used.
- 14. MOTOR OVERLOAD. MISSING 0 OR 1 COMMAND** This message appears if an axis move command is not initialized properly (missing 0 or 1 command), usually indicating a hardware-related problem.
- 15. TIME OUT ON RIGID TAP PRE-CYCLE TEST** This message appears during rigid tap precycle test if the spindle rotation is interrupted or the spindle magnet is not detected for a certain length of time.
- 16. TIME OUT ON RESOLVER RESPONSE** This message appears any time the resolver zero-crossing pulse is not detected within the allowable time (4-10 msec). The zero-crossing pulse is hardware-generated every 1 msec. This can also be caused by motor over temperature or by severe motor vibrations.
- 17. BAD READING ON RESOLVER PORT** This message appears if the value of resolver “counts”, read from the resolver port when the “zero-crossing” pulse is detected is too large, indicating possible failure of resolver cables, 1010-4(-5) card, or the resolver itself. This can also be caused by motor over temperature or by severe motor vibrations.
- 18. SERVO AMPLIFIER FAULT LINE DOWN** This message appears if the axis card cannot detect the fault line signal from the amplifier, or the fault line signal strength drops below 10V, indicating a possible amplifier fault.
- 19. STACK OVERFLOW. OVERLOADED WITH STEP COMMAND** This message appears if the stack on the axis card (used to store CPU commands until they have been serviced) overflows.
- 20. MOVE TRANSFER FAULT. INCOMPLETE DATA FOR MOVE COMMAND** This message appears when an axis move command is improperly transferred to the axis card, as indicated by a checksum comparison between what the CPU sent and what the axis card received.

- 21. SPURIOUS INTERRUPTS. NOT SERVICEABLE** This message appears if the interrupt currently being processed is an undefined interrupt, and thus cannot be serviced, indicating possible failure of the 1010-4(-5), 1030, or main CPU. This could also indicate a software problem.
- 22. BAD SCALE READING** This message appears if the feedback from the scale port is outside allowable limits, indicating possible failure of the EXE box, scales, cables, or 1010-4(-5) card. This can also be caused by severe motor vibrations.
- 23. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN STANDBY MODE** This message appears (in stand-by mode) when the following error exceeds the user-defined overload factor, which sets, in motor turns, the maximum following error for the axis operation. MESSAGES 23, 24, 25, AND 26 ARE NO LONGER RELATED TO ERROR 13.
- 24. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN POINT TO POINT MODE** See ERROR 23.
- 25. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN CONTOURING MODE** See ERROR 23.
- 26. MOTOR OVERLOAD. ERROR > OVERLOAD FACTOR IN JOB MODE** See ERROR 23.
- 27. RETURN TO MAGNET TIME-OUT** This message appears at the completion of the rigid tap cycle if the spindle controller could not detect the magnet within 6 seconds while returning to home position.
- 28. ORIENTATION TIME-OUT OR TAPPING CALIBRATION FAILURE** This message appears if the spindle controller could not detect the magnet within 10 seconds, either during spindle orientation (M19, M6) or on the last rotation during rigid tap precycle.

**29. RIGID TAP
PRECYCLE ENCODER
FAILURE (TOO MANY
COUNTS)**

See ERROR 7.

**30. SURVEY
CLEARED DUE TO
BLANK OR CORRUPT
SURVEY**

This message appears during cold start if the checksum, calculated from reading the survey table, is not equal to the checksum stored in the survey table. The survey will be zeroed.

**31. TIME-OUT ON
ORIENTATION**

If the spindle magnet does not align within 10 seconds of an orient spindle then this error message is displayed.

**32. SPINDLE WILL
NOT STOP. CHECK
INVERTER ZERO
SPEED**

If the spindle does not stop in 3 seconds from entering a spindle stop command then this error message is displayed.

**33. ENCODER
CHANNELS ARE
REVERSED (RIGID
TAP)**

This message will display if the encoder connection for spindle motor is reversed at spindle control card.