

APPENDIX 1

EIA Codes

PREPARATORY FUNCTIONS

- G00—Denotes rapid traverse for point-to-point positioning.
- G01—Linear interpolation.
- G02—Circular interpolation clockwise.
- G03—Circular interpolation counterclockwise.
- G04—Dwell.
- G05–07—Unassigned.
- G08—Acceleration at a smooth rate.
- G09—Deceleration at a smooth rate.
- G10–16—Unassigned.
- G13–16—Axis selection codes. - Don't use
- G17—XY plane selection.
- G18—ZX plane selection.
- G19—YZ plane selection.
- G20–32—Unassigned.
- G33—Thread cutting, constant lead.
- G34—Thread cutting, increasing lead.
- G35—Thread cutting, decreasing lead.
- G36–39—Unassigned.
- G40—Cutter diameter compensation cancel.
- G41—Cutter diameter compensation left.
- G42—Cutter diameter compensation right.
- G43—Cutter compensation inside corner (used to adjust for differences in programmed and actual cutter size). Don't use
- G44—Cutter compensation outside corner (used to adjust for differences in programmed and actual cutter size). - Don't use
- G45–49—Unassigned.
- G50–59—Used with adaptive controls.
- G60–69—Unassigned.
- G70—Inch programming. - automatic when coded in U.S.
- G71—Metric programming.
- G72—Three-dimensional circular interpolation clockwise.
- G73—Three-dimensional circular interpolation counterclockwise.
- G74—Multiquadrant circular interpolation cancel. automatic

Family

- G75—Multiquadrant circular interpolation.
- G76–79—Unassigned.
- G80—Cycle cancel.
- G81—Drill cycle.
- G82—Drill cycle with dwell.
- G83—Intermittent or deep hole drilling cycle.
- G84—Tapping cycle.
- G85–89—Boring cycles.
- G90—Absolute positioning.
- G91—Incremental positioning.
- G92—Register preload code.
- G93—Inverse time feedrate.
- G94—Inches (millimeters) per minute feedrate. *- Milling machine*
- G95—Inches (millimeters) per revolution feedrate. *- Lathe default*
- G96—Unassigned. *- Direct cutting*
- G97—Revolutions per minute spindle speed. *constant*
- G98–99—Unassigned.

MISCELLANEOUS FUNCTIONS

- M00—Program stop.
- M01—Optional (planned) stop.
- M02—End of program.
- M03—Spindle on clockwise.
- M04—Spindle on counterclockwise.
- M05—Spindle off.
- M06—Tool change.
- M07—Coolant on (flood).
- M08—Coolant on (mist).
- M09—Coolant off.
- M10—Automatic clamp.
- M11—Automatic unclamp.
- M12—Synchronize multiple axes.
- M13—Spindle clockwise and coolant on.
- M14—Spindle counterclockwise and coolant on.
- M15—Rapid motion positive direction.
- M16—Rapid motion negative direction.
- M17–18—Unassigned.
- M19—Spindle orient and stop.
- M20–29—Unassigned.
- M30—End of tape, will rewind tape automatically.

M31—Interlock bypass.
M32–39—Unassigned.
M40–46—Gear changes if used, otherwise unassigned.
M47—Continues program execution from the start of program.
M48—Cancel M47.
M49—Deactivate manual speed or feed override.
M50–57—Unassigned.
M58—Cancel M59.
M59—RPM hold.
M60–99—Unassigned.

OTHER ADDRESSES

A—Rotary motion about the X axis.
B—Rotary motion about the Y axis.
C—Rotary motion about the Z axis.
D—Angular dimension around a special axis. Also used for a third feed function.
E—Angular dimension around a special axis, or special feed function.
H—Unassigned.
I—X axis arc centerpoint.
J—Y axis arc centerpoint.
K—Z axis arc centerpoint.
L—Unassigned.
O—Used on some controllers in place of N address for sequence numbers.
P—Special rapid traverse code, or a third axis parallel to the X axis.
Q—Special rapid traverse code, or a third axis parallel to the Y axis.
R—Special rapid traverse code, or a third axis parallel to the Z axis. Also used for radius designation.
U—Secondary axis parallel to X.
V—Secondary axis parallel to Y.
W—Secondary axis parallel to Z.