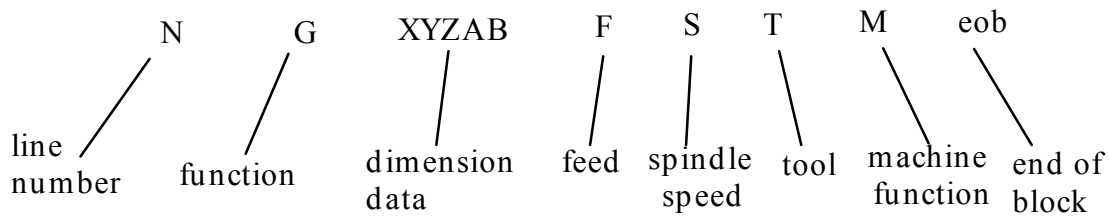


G-Codes to Know

G-Code is the almost universal language of CNC. Industry has standardized on this basic set of codes for the operation of computer numerical controlled machines, and its basic elements are used very widely. The Haas machines run exclusively on G-code, so knowing these codes will help you understand what your CNC program is doing and will also let you try programming at the most primitive level. A complete list of G-codes specific for each machine is available, but this list is a good starting point.

Basic G-Code Format



Good G-Codes to Know

- G00 Rapid position
- G01 Linear cut
- G02 Arc cw
- G03 Arc ccw
- G04 Dwell
- G40 Cutter comp cancel
- G41 Cutter comp left
- G42 Cutter comp right
- G28 Return to machine reference point
- G92 Absolute zero reset

Good M-Codes to know

- M02 End of program
- M03 Spindle cw
- M04 Spindle ccw
- M05 Spindle off
- M06 Tool change
- M30 End of tape

History

ver 1	12/93	John Wadsworth	original text
ver 1.1	8/01	Katherine Kuchenbecker	minor revisions