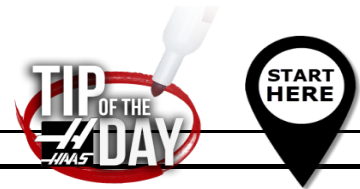


Start Simply.

The majority of parts can be machined using only the codes shown on this page.



Mill. Simple 9-line Program

```

%
O1234 (MILL EXAMPLE PROGRAM) Program 'O' number, and comment
M06 T1 Tool Change, to Tool #1
M03 S7500 Start Spindle, Clockwise, at 7,500 RPM
G54 G00 G90 G17 Use Work Offset G54, Safe Startup Line
X0. Y0. XYZ Positioning
G43 H01 Z.1 M08 Use Tool Offset #1, move to Z position, turn on coolant
G01 F5.0 Z-.01 Feed the tool, in a straight line, at 5 inches per minute

{ DOT-TO-DOT XYZ LOCATIONS } Can add more dot-to-dot XY location points here

G00 Z2.0 Rapid to location.
M30 End program
%
    
```

Lathe. Simple 9-line Program

```

%
O1235 (LATHE EXAMPLE PROGRAM) Program 'O' number, and comment
T101 Change to Tool #1, use Offset #1
G50 S2000 Set maximum spindle RPM to 2000
G97 M03 S1000 Start spindle, clockwise, at 1000 RPM
G54 G00 G99 G18 Use Work Offset G54, Safe Startup Line
X3. Z1. M08 Position in XZ, turn on coolant. Lathe X values are in diameter
G01 F.006 Z.1 Feed the tool, in a straight line, at .006 inches per revolution

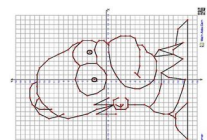
{ DOT-TO-DOT XZ LOCATIONS } Can add more dot-to-dot XZ location points here

G00 Z2.0 Rapid to location.
M30 End program
%
    
```

Study the codes shown in the simple programs above.
Once these codes are mastered, move on and study the following:

Mill	Lathe	
M00 / M01	M00 / M01	Program / Optional Stop
G53	G53	Non-Modal Machine Position
G02 / G03	G02 / G03	Arcs, right and left
G83	G83	Peck Drilling Cycle
G84	G84	Tapping Cycle, right-hand taps
G41 / G42 / D	G41 / G42	Cutter Compensation
G90 / G91	XZ / UW	Absolute vs Incremental
G94 / G95	G98 / G99	Feed per Minute / Revolution
	G96 / G97	Constant Surface Speed On / Off
	G70	Finish Turning Cycle
	G71	Rough Turning Cycle
	G76	Threading Cycle

Tip: Google "Graphing Coordinate Pairs"



Do a Google search for: "Graphing Coordinate Pairs"