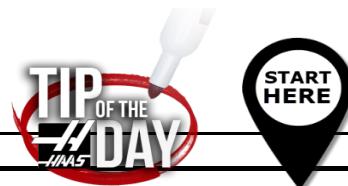


## Start Simply.

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



### Mill. Simple 9-line Program

|                                     |             |   |
|-------------------------------------|-------------|---|
| %                                   |             |   |
| <b>O1234 (MILL EXAMPLE PROGRAM)</b> |             | 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

|                                      |             |  |
|--------------------------------------|-------------|--|
| %                                    |             |  |
| <b>O1235 (LATHE EXAMPLE PROGRAM)</b> |             | 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  |
| %                                    |             |  |

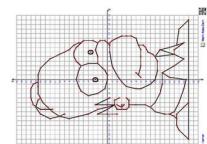
Study the codes shown in the simple programs above.

Once these codes are mastered, move on and study the following:

| Mill          | Lathe     |
|---------------|-----------|
| M00 / M01     | M00 / M01 |
| G53           | G53       |
| G02 / G03     | G02 / G03 |
| G83           | G83       |
| G84           | G84       |
| G41 / G42 / D | G41 / G42 |
| G90 / G91     | XZ / UW   |
| G94 / G95     | G98 / G99 |
|               | G96 / G97 |
|               | G70       |
|               | G71       |
|               | G76       |

Program / Optional Stop  
 Non-Modal Machine Position  
 Arcs, right and left  
 Peck Drilling Cycle  
 Tapping Cycle, right-hand taps  
 Cutter Compensation  
 Absolute vs Incremental  
 Feed per Minute / Revolution  
 Constant Surface Speed On / Off  
 Finish Turning Cycle  
 Rough Turning Cycle  
 Threading Cycle

Tip: Google "Graphing Coordinate Pairs"



Do a Google search for: "Graphing Coordinate Pairs"