CNC Programming for Mill

- Example CNC program - bores a center hole and drills bolt circle
- Haas manual at: k:\class\engr\480\haas\usermanual.pdf or at http://engr.wallawalla.edu/engr480/docs/Haas

```
%
O00100
(Maxon motor bolt circle program);
(center hole 0.256R);
T3 M06;                          (Get tool T3 from toolchanger);
G00 G90 G54 X0.74 Y0.63;         (Move to 0.74,0.63 in G54 coord sys);
S1100 M03;                       (Set spindle speed at 1100 RPM and start CW);
G43 H03 Z0.3 M08;                (Compensate for tool length, Set return height);
G01 Z0 F10;                      (drop to surface);
G12 G91 Z0.05 I0.1 K0.256 Q0.01 L3 D03 F1.0; (Bore center hole);
G00 G90 Z1. M09;                 (return to 1" above surface, turn off coolant);

(bolt circle - 0.374R, peck 0.05);
G70 I0.374 J60.0 L6;             (0.374R, 60deg, 6 holes);
T1 M06;                          (Get tool T1);
G00 G90 G54 X0.74 Y0.63;           (Move to 0.5,0.5 in G54 coord sys);
S1050 M03;                       (Set spindle to 1050 RPM);
G43 H01 Z1. M08;                 (Compensate for tool len, set ret height, coolant on);
G82 G98 Z-0.05 P1. R0.1 F5.; (Execute drill cycle for each hole);
G00 G80 Z1. M09;                 (Cancel cycle, turn off coolant);
G28 G91 Z0 M05;                  (Return home, turn off spindle);
M30;                             (End of program);
```

G-Codes for Milling

G00	Rapid positioning
G01	Linear interpolation (feeding)
G02	CW Circular interpolation
G03	CCW Circular interpolation
G04	Dwell
G12	Circular Pocket (CW)
G20	Inch system
G21	Metric system
G28	Return to reference point
G43	Tool Length Compensation

G-Codes for Milling

G54-G59	Select work coord system
G70	Bolt circle cycle
G81	Drill canned cycle
G82	Spot drill canned cycle
G83	Peck drill canned cycle
G84	Tapping canned cycle
G90	Absolute coordinates
G91	Incremental coordinates
G98	Initial point return

M-Codes for Milling

M00	Program Stop
M01	Opt. Program Stop (panel controlled)
M03	Start spindle (normal rotation)
M04	Start spindle (reverse rotation)
M05	Stop spindle
M06	Tool change
M08	Start coolant
M09	Stop coolant
M30	Program end

Loading CNC Program

- Copy to Z: as haascode.txt
- Press List Prog
- Type program number (O01234)
- Press RECV RS232