- Quick review
 - rapid traverse - G0 X Z
 - G1 X Z F feed w/ linear interpolation
 - G2/G3 X Z I K F circular interpolation

limit max speed

- G28 U0 W0 return home
- G50 S
- G99/G98
- G21
- M3/M5
- M8/M9
- M0/M1/M30

- G96/G97 S const. surf. speed/fixed speed
 - feed per rev/feed per min
 - metric units
 - spindle on/off
 - coolant on/off
 - stop/optional stop/program end

- Quick review
 - G blocks are modal G0 is rapid traverse mode, G1 is linear feed mode.

G0 X10.0 Z20.0 Z10.0 X15.0 G1 F0.1 X4.5 Z12.0

M codes are on/off, only one per line M3 M8

• Variables:

- #1-33 (local vars for macros)
- #100-199 (zeroed on power cycle)
- #500 to 999 (survive power cycle)
- Don't change vars above #1000: these are system variables.
- math:
- •[#500 + 1.0]
- •[#500 + #510]
- \bullet [#500 + [#510/2]]

Example Program

```
%
01 (THAT'S AN OH NOT A ZERO)
  (EXAMPLE PROG - TURN PLUG 20.0MM OD X 17MM LONG)
  (STOCK: ALUM 1.5" X 17MM + 10MM)
 (T1- CNMG 55DEG DIAMOND)
 (T2- VNMG 15DEG DIAMOND)
  (T10- 3.175MM CUTOFF)
 (VARIABLES)
#500=38.1
             (STOCK DIAMETER)
#501=17.0 (STOCK LENGTH)
             (SURFACE M/MIN FOR CUTTING ALUM)
#502=150.0
#503=0.2
             (ROUGHING FEED: MM/REV)
N1
             (WORK OFFSET)
G54
G21
             (METRIC)
G28 U0 W0
             (GO HOME)
             (MAX SPINDLE SPEED)
G50 S2000
```

```
(ROUGH OD 38.0 TO 20.5MM)
  (USE 2.0MM DEPTH OF CUT -> 8 PASSES)
N2
                  (55DEG DIAMOND TOOL, TOOL 1)
G00 T0101
                    (CLAMP SPEED AT MAX 2000)
G50 S2000
G96 S#502
                    (CONST SURF SPEED)
                    (FEED PER REV)
G99
G00 \times [#500 + 0.5] \times Z#501 (INITIAL POINT FOR ROUGHING)
                    (SPINDLE ON)
M03
M08
                    (TURN ON COOLANT)
G00 X36.0 Z#501
G01 X36.0 Z5.5 F#503 (FIRST PASS)
G01 X[#500+0.5] F#503 (RETRACT X)
G00 Z#501
                       (RETRACT Z)
G00 X34.0
```

```
G01 X34.0 Z5.5 F#503 (SECOND PASS)
G01 X[#500+0.5] F#503G00 Z#501
G00 X32.0
G01 X32.0 Z5.5 F#503 (THIRD PASS)
G01 X[#500+0.5] F#503
 (FINISH OD 20.0MM)
Ν3
G00 T0202
                     (VNMG 15DEG DIAMOND)
                     (POSITION FOR START OF CHAMFER)
G00 X19.0
G00 Z[#501+0.5]
G01 X20.0 Z[#501-0.5] F#504 (CHAMFER 0.5MM)
G01 Z0.0 F#504
                            (FINISH TURN)
                             (BACK OFF)
G01 X[#500+0.5] F0.1
M09
M05
                             (GO HOME)
G28 U0 W0
M01
```

- NEVER DO A TOOL CHANGE AWAY FROM HOME! (G28 U0 W0)
- ALWAYS PUT A DECIMAL POINT AFTER DIMENSIONAL NUMBERS (no decimal --> microns)
- USE ALL CAPS IN YOUR PROGRAM (lowercase gets dropped)
- DON'T GET "OHS" AND "ZEROS" MIXED UP. PROGRAM NUMBER STARTS WITH "OH", NOT ZERO

Planning a Program

- Critical dimensions
 - if your most critical dimension is an OD, do
 OD turning first
 - if your most critical dimension is an ID, do boring first
- Machining ID
 - Drill, rough bore (with end mill), finish bore (with boring bar)
 - be very careful retracting from ID operations.
 It is too easy to crash a tool inside the part and break the tool. ID tools are much more expensive as well as much more fragile than OD tools.

- Finish passes
 - if your tolerance is tight or your part is slender (so it may deflect noticeably), run multiple finish passes at the same diameter.

G-Code Looping & Branching

- Loops:
 - WHILE [#500 GT #510] DO 1
 - G01 ...
 - #500 = #500 0.2
 - END 1
- Conditionals:
 - IF [#502 LE #515] GOTO N100
 - ...
 - N100

ROUGHING CYCLE - G71



Roughing Cycle - G72



(Diameter designation, metric input) N010 G50 X220.0 Z190.0 ; N011 G00 X176.0 Z132.0 ; N012 G72 W7.0 R1.0 ; N013 G72 P014 Q019 U4.0 W2.0 F0.3 S550 ; N014 G00 Z58.0 S700 ; N015 G01 X120.0 W12.0 F0.15 ; N016 W10.0 ; N017 X80.0 W10.0 ; N018 W20.0 ; N019 X36.0 W22.0 ; N020 G70 P014 Q019 ;

Finishing Cycle - G70

G70P (ns) Q (nf) ;

- (ns) : Sequence number of the first block for the program of finishing shape.
- (nl) : Sequence number of the last block for the program of finishing shape.

NOTE

- 1 F, S, and T functions specified in the block G71, G72, G73 are not effective but those specified between sequence numbers "ns" and "nf" are effective in G70.
- 2 When the cycle machining by G70 is terminated, the tool is returned to the start point and the next block is read.
- 3 In blocks between "ns" and "nf" referred in G70 through G73, the subprogram cannot be called.

G71/G70 Cycle (OD example)

G0 T0101	M09
G50 S2000	M05
G96 S#502	G28 U0 W0
G99	
G0 X[#500 + 10.0] Z[#501 + 10.0] (CLOSE)	G0 T0202
M3	G50 S4000
G0 X#500 Z[#501 + 2.0] (INITIAL POINT)	G96 S#502
M8	G99
	G0 X[#500 + 10.0] Z[#501 + 10.0] (CLOSE)
G71 U2.0 R0.75 (2MM DOC, 0.75MM RET)	M3
G71 P100 Q110 U0.5 W0.25 F#503 (LEAVE	G0 X#500 Z[#501 + 2.5] (INITIAL POINT)
0.5MM ON OD 0.25MM ON FACES)	M8
N100 G0 X0.0 Z[#501 + 2.0] (START POINT)	G70 P100 Q110 (EXACT PROFILE FOR
G1 Z#501 F#504	FINISH)
X10.0	
Z7.0	M9
X14.0 Z5.0	M5
Z-3.2	G28 U0 W0
N110 G0 X[#500 + 2.0] (EXIT)	

Arcs

- G2 = CW arc
- G3 = CCW arc

Parameters:

- X,Z = end point of arc
- I,K = distance to center point of arc from start

(I is in radius units, not diameter units)

F = feedrate

G0 X0.0 Z19.0 (START POINT) G3 X38.0 Z0.0 I0.0 K-19.0 F0.1 (CCW)



Convex Arc Example

Convex Arc Example

G71 U2.0 R0.75 (2MM DOC, 0.75MM RETRACT) G71 P100 Q110 U0.5 W0.25 F#503 N100 G00 X0.0 Z19.0 (ENTRANCE POINT) G03 X38.0 Z0.0 I0.0 K-19.0 F#504 G01 X38.0 Z-3.2 N110 G00 X[#500+2.0] (EXIT)

G70 P100 Q110 F#504 (FINISH PASS)

Concave Arc Example

- Drill center hole
- Use boring tool to rough and finish



G0 X0.0 Z15.0 G2 X30.0 Z10.0 I0.0 K25.0 F0.2

Threading Cycle

- Threading requires multiple passes in perfect synchronization to reach final thread depth.
- Must make sure you are in constant spindle speed mode, not constant surface speed!

Computing Thread Parameters

Overall thread height

- H = Pitch*(0.5 / tan 30deg) = 0.866025*Pitch

- Ext thread depth = (17/24)*H = 0.613435*Pitch
- Int thread depth = (5/8)*H = 0.541266*Pitch







 $G76X(u) = Z(W) = R(i) P(k) Q(\Delta d) F(L);$

- Repetitive count in finishing (1 to 99)
 This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5142, and the parameter is changed by the program command.
- r : Chamtering amount

When the thread lead is expressed by L, the value of L can be set from 0.0L to 9.9L in 0.1L increment (2-digit number from 00 to 90). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5130, and the parameter is changed by the program command.

a : Angle of tool tip

One of six kinds of angle, 80-, 60-, 55-, 30-, 29-, and 0-, can be selected, and specified by 2-digit number.

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5143, and the parameter is changed by the program command. m, r, and a are specified by address P at the same time.

(Example)

When m=2, r=1.2L, a=60°, specify as shown below (L is lead of thread).

- $P = \frac{02}{m} = \frac{12}{r} = \frac{60}{a}$
- ∆dmin : Minimum outling depth (specified by the radius value) When the outling depth of one cycle operation (∆d – ∆d –1) becomes smaller than this limit, the outling depth is clamped at this value. This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter No. 5140, and the parameter is changed by the program command.
- Finishing allowance This designation is modal and is not changed until the other value

is designated. Also this value can be specified by parameter No. 5141, and the parameter is changed by the program command.

- Difference of thread radius It i = 0, ordinary straight thread cutting can be made.
- k : Height of thread
 - This value is specified by the radius value.
- ∆d : Depth of cut in 1st cut (radius value)
- L : Lead of thread (same as G32).

- G76 Pffppaa Qddd Rfff (first line of G76)
 - ff = # of finish passes (01-99)
 - pp = # revolutions for gradual pull-out (0.0-9.9=00-99)
 - -aa = angle (60 degrees for us)
 - ddd = last depth of cut (min.cutting depth) (positive radial value, no decimal point – microns)
 - fff = finish allowance (mm, dec.point allowed)

- G76 Xxx Zzz Ppp Qqq Fff (2nd line)
 - -xx = last thread pass diameter
 - -zz = Z endpoint of thread
 - pp = height of thread (pos.radial value, microns)
 - qq = 1st threading pass depth (max.cutting depth, pos.radial value, microns)
 - ff = feedrate (same as thread lead)

- Example: M24x2 external thread
 - G00 X30.032 Z[#501 + 6.0]
 - G76 P010060 Q005 R0.0
 - G76 X24.43 Z[#501-10.0] P1360 Q0544 F2.0

Multistart Threading

- Example: M24x2 three start external thread
 - #1=2.0 (PITCH MM)
 - #2=0.5/TAN[30.0] * #1 (H)
 - #3=#2 * 17.0/24.0 (THREAD DEPTH)
 - #4=3 (NUMBER OF STARTS)
 - #5=0 (START NUMBER)
 - G00 X32.0 Z[#501 + 3 * #1 * #4] (START 3X LEAD AWAY)
 - WHILE [#5 LT #4] DO 1
 - G00 X30.032 Z[#501 + #1 * #5 + 3 * #1 * #4]
 - G76 P100060 Q005 R0.0
 - G76 X23.39 Z[#501-10.0] P[FIX[#3*1000]] Q0544 F[#1*#4]
 - #5 = #5 + 1
 - END 1