G-Code Primer - Review

- Quick review
  - G00 X__ Z__ rapid traverse
  - G01 X__ Z__ F__ feed w/ linear interpolation
  - G28 U0 W0 return home
  - G50 S____ limit max speed
  - G96/G97 S__ const.surf. speed/fixed speed
  - G99/G98 feed per rev/feed per min
  - G21 metric units
  - M03/M05 spindle on/off
  - M08/M09 coolant on/off
  - M01/M30 optional stop/program end

G-Code Primer - Review

- 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
```

G-Code Primer - Review

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

G-Code Primer - Review

- 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 (lc gets dropped)
- DON'T GET "OHS" AND "ZEROS" MIXED UP. PROGRAM NAME STARTS WITH "OH", NOT ZERO

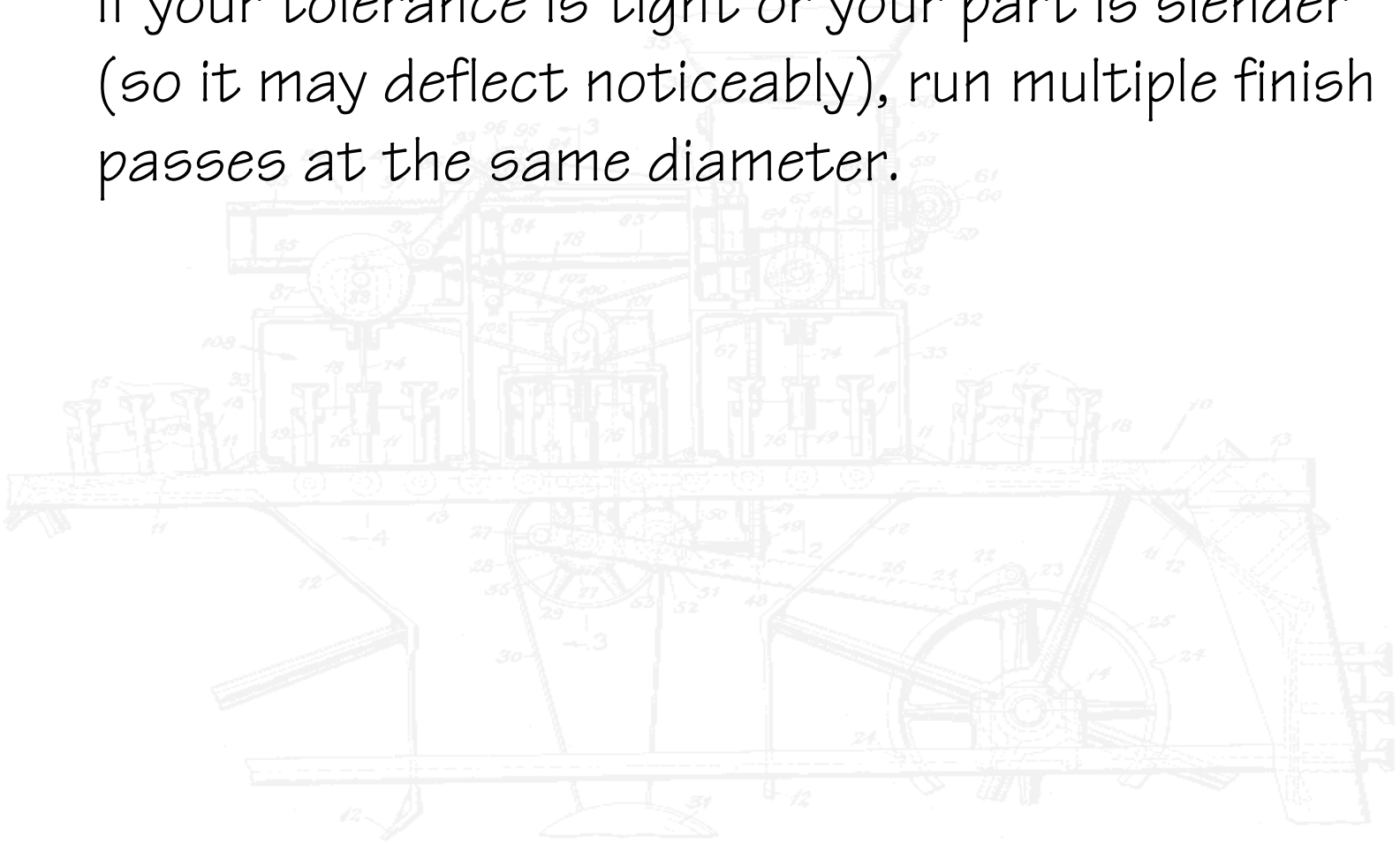
Planning a Program

- 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.

Planning a Program

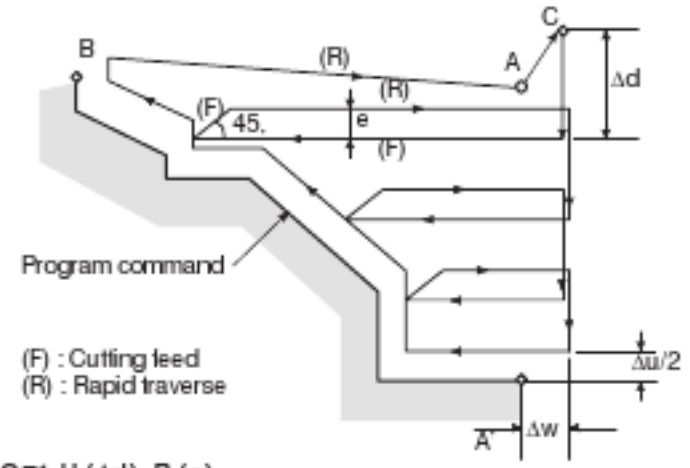
- Finish passes
  - if your tolerance is tight or your part is slender (so it may deflect noticeably), run multiple finish passes at the same diameter.



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



G71 U (Δd) R (e) ;

G71 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t)

N (ns)·····

·····

F____
S____
T____

The move command between A and B is specified in the blocks from sequence number ns to nf.

N (nf)·····;

Δd : Depth of cut (radius designation)
Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

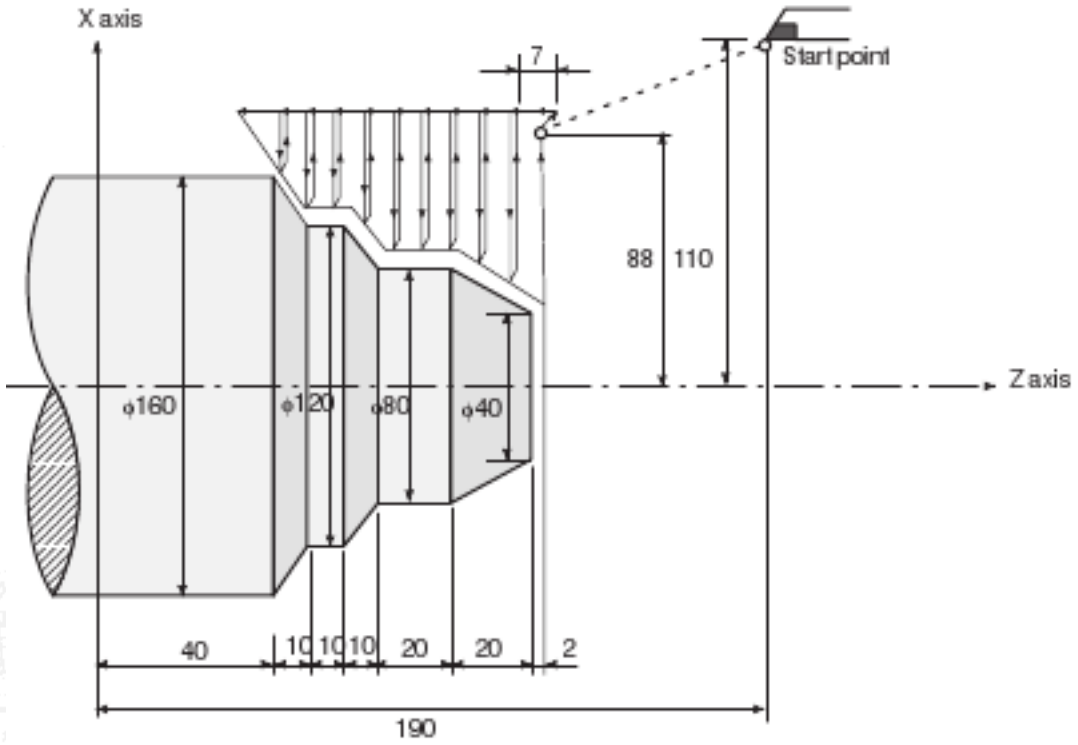
nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).

Δw : Distance and direction of finishing allowance in Z direction.

f,s,t : Any F, S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

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.

(nf) : 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.