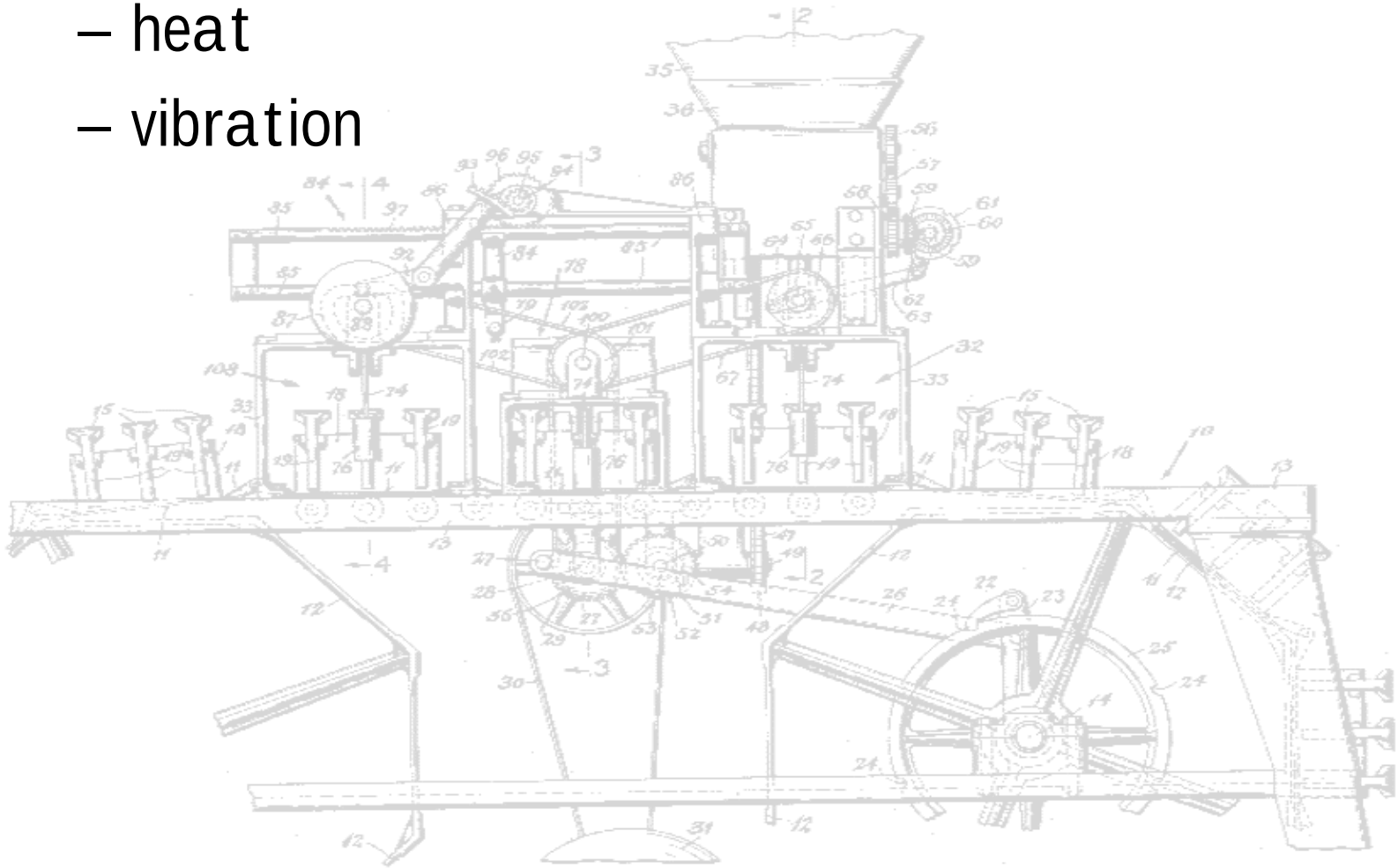
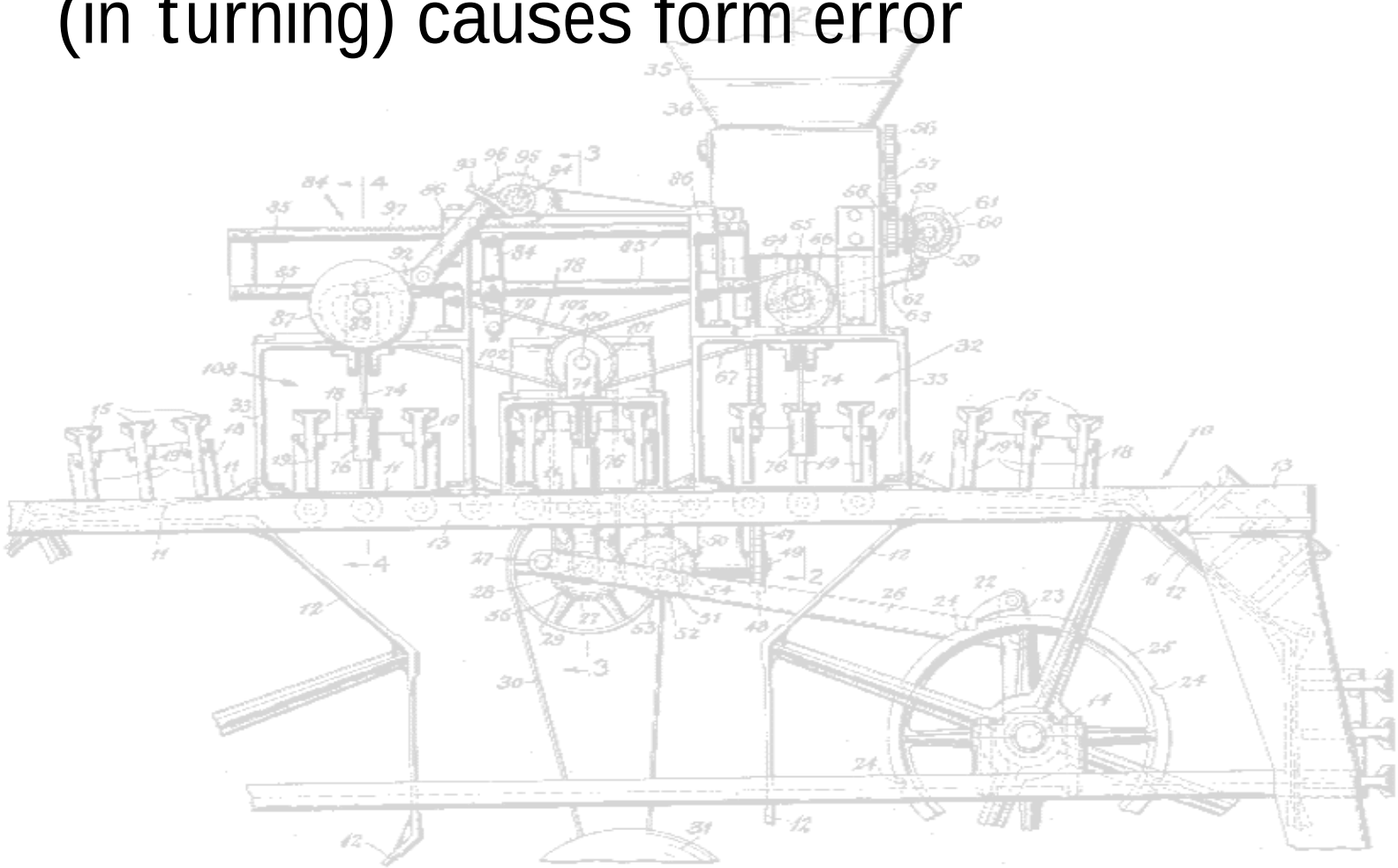
Machinability

- The enemies:
  - heat
  - vibration

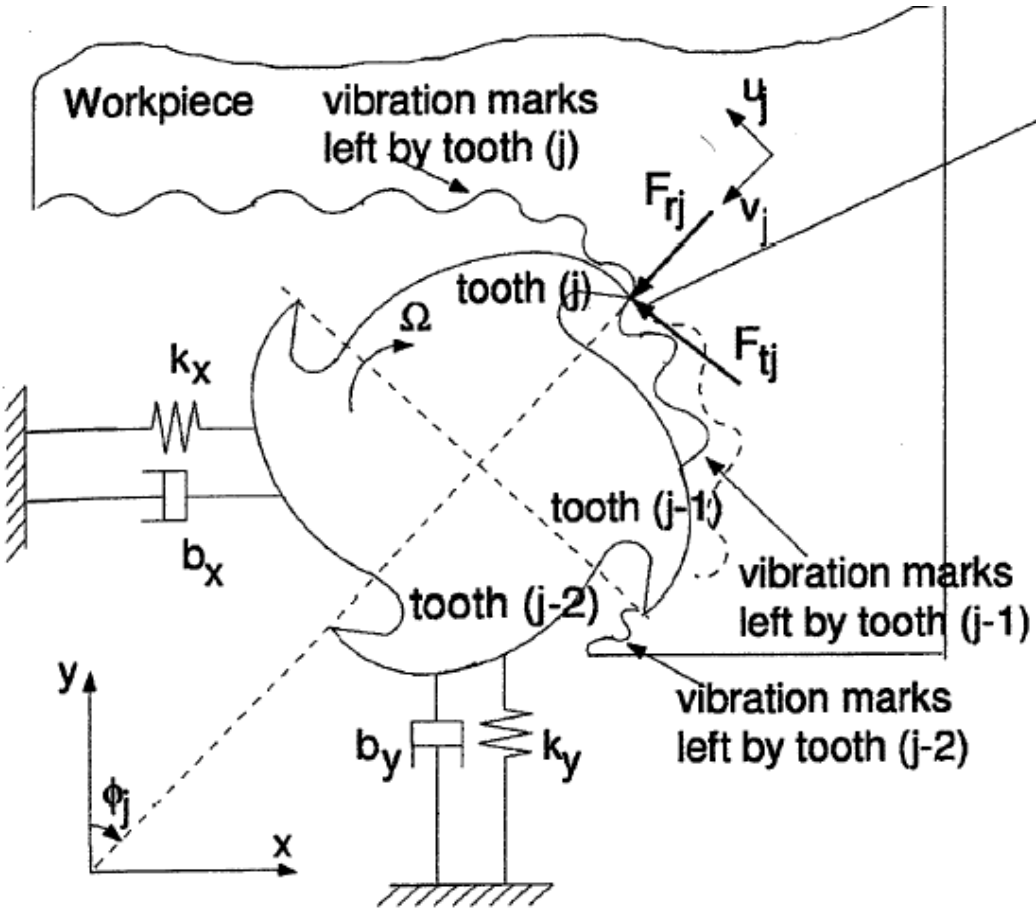


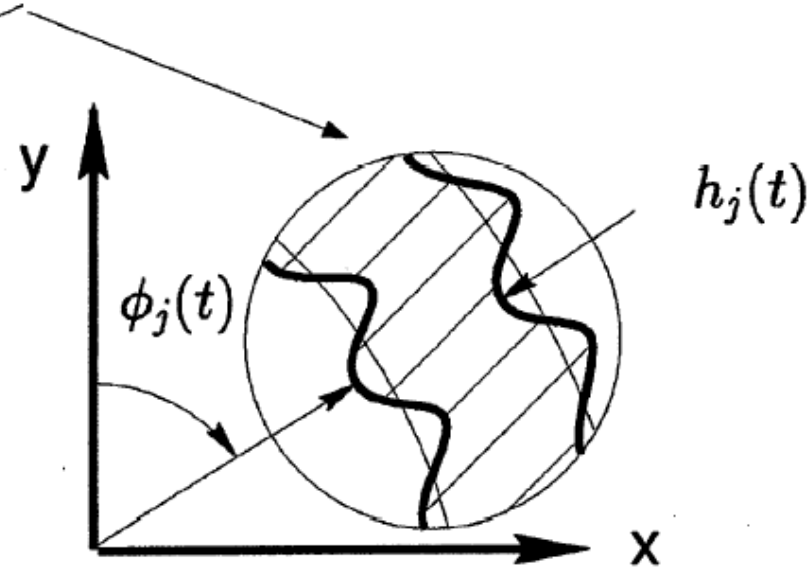
Form Errors

- Deflection of cutter (in end milling) or work (in turning) causes form error



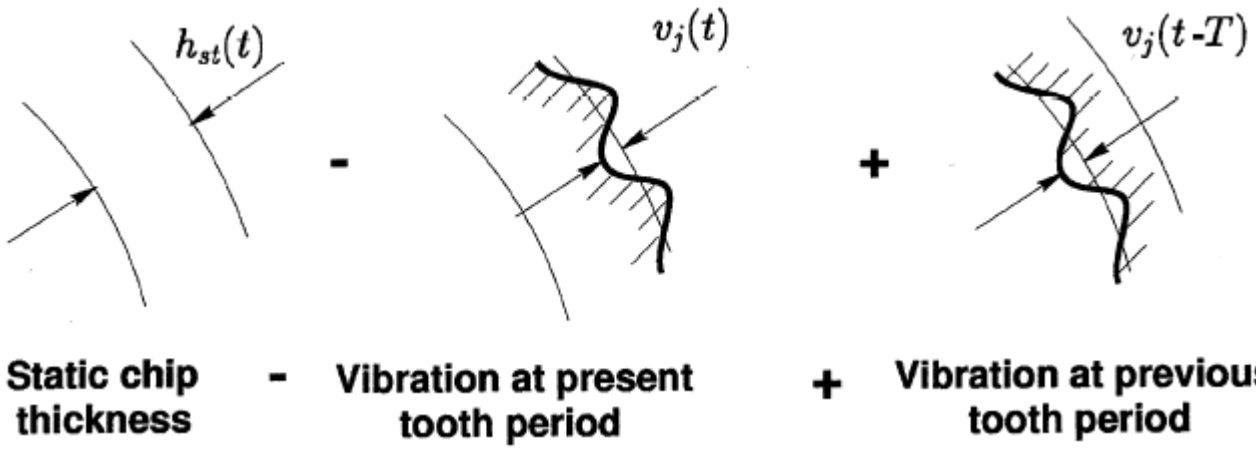
Vibration (chatter)

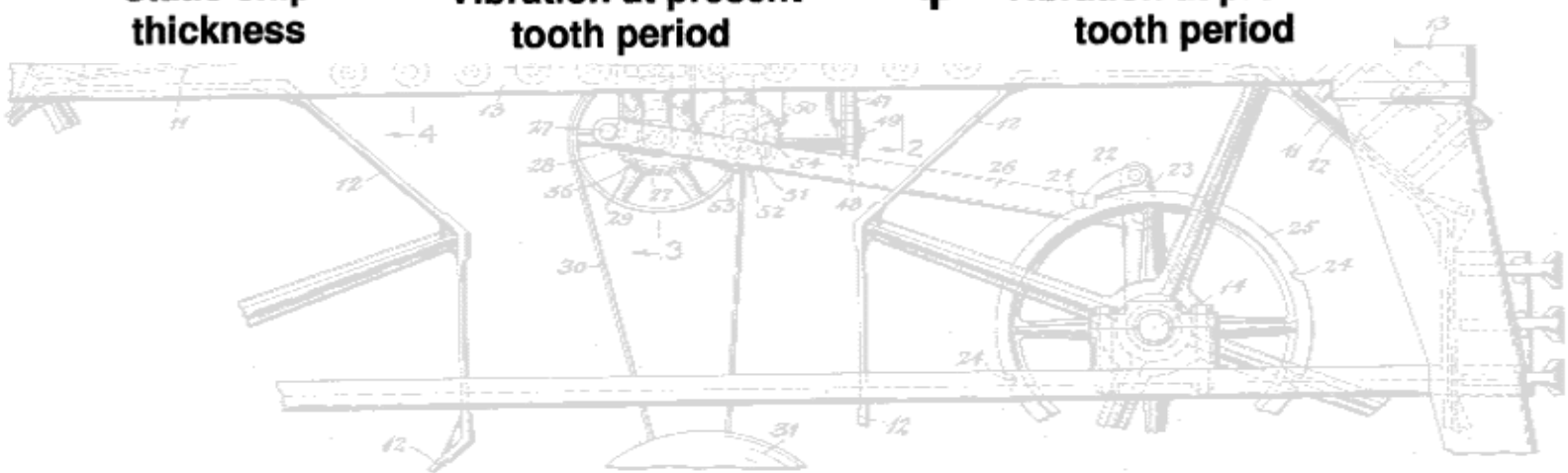




Dynamic chip thickness

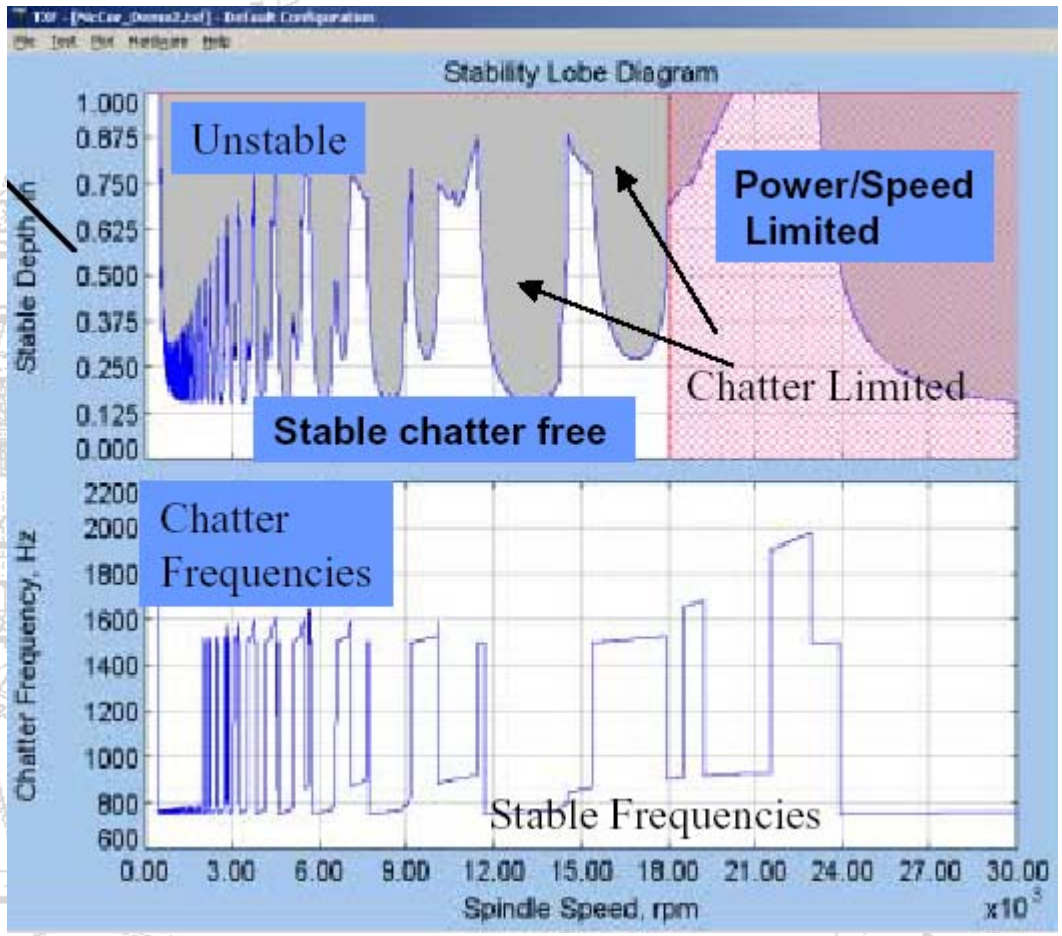
Vibration


$h_{st}(t)$
-
$v_j(t)$
+
$v_j(t-T)$
Static chip thickness
-
Vibration at present tooth period
+
Vibration at previous tooth period

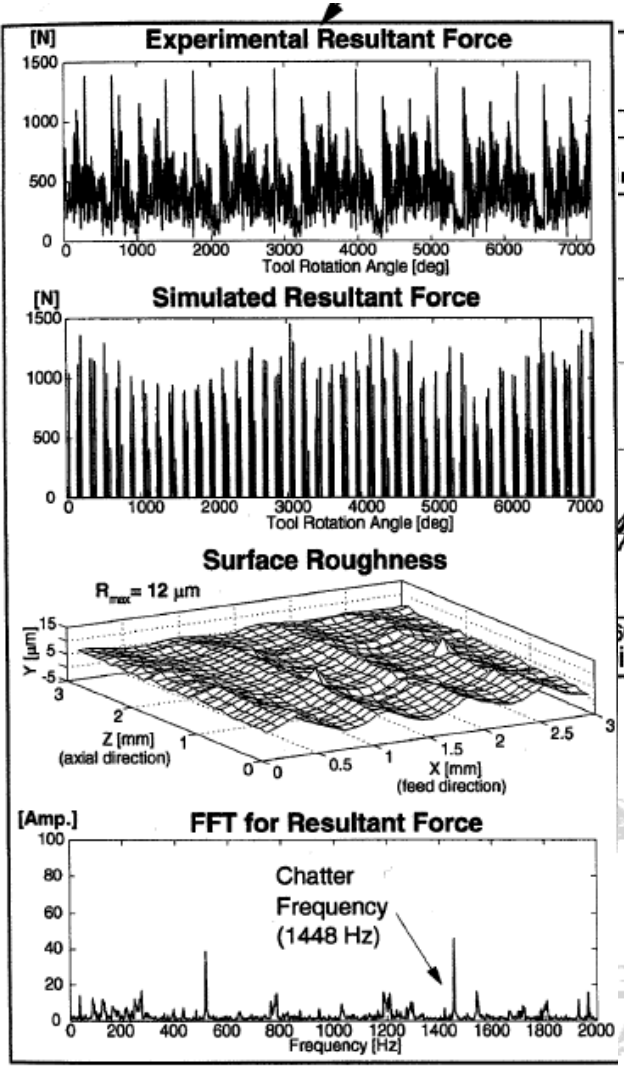


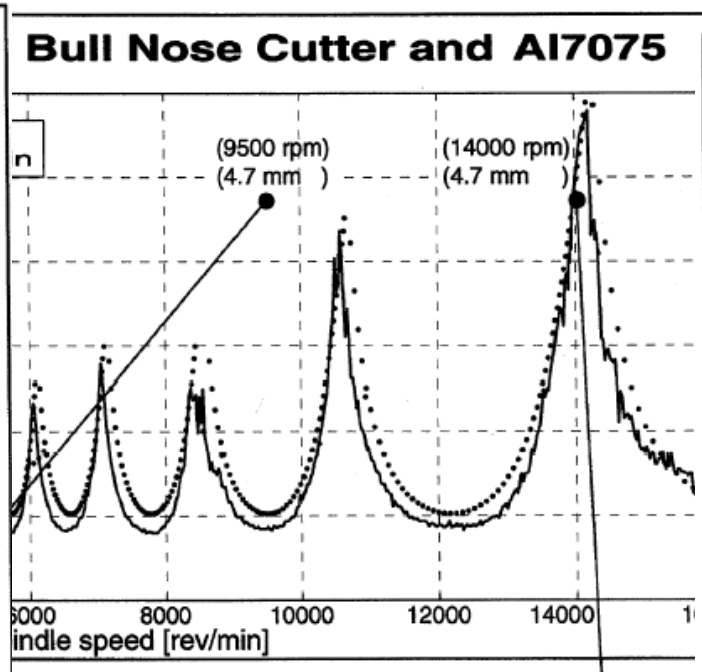
Vibration

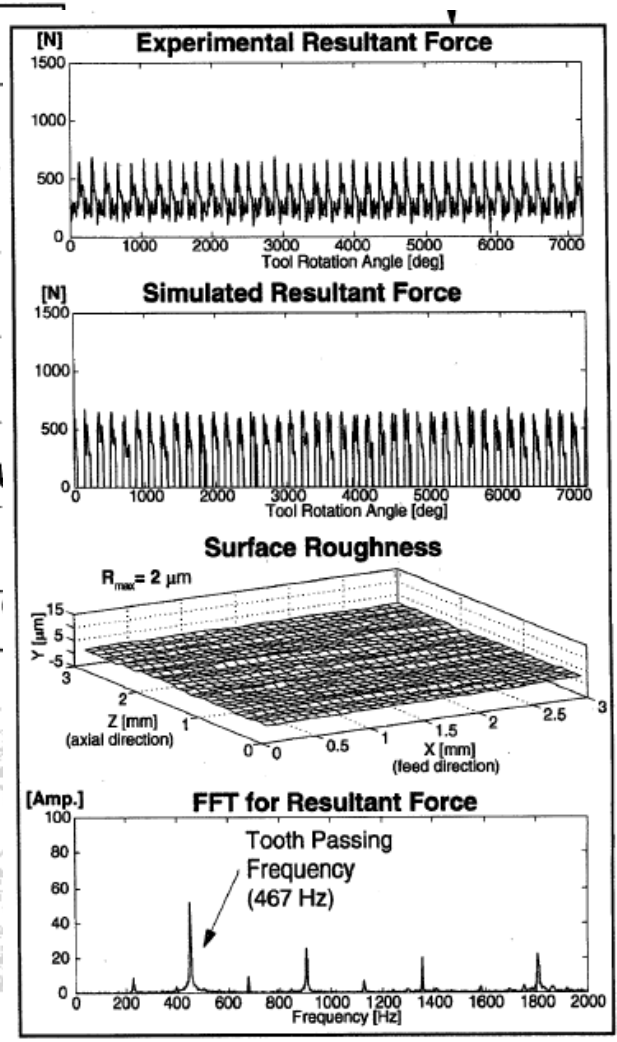
- Choose highest spindle RPM
- Tune tool length to stay in a stable lobe at top spindle RPM



Vibration


[N]
Experimental Resultant Force
Tool Rotation Angle [deg]
[N]
Simulated Resultant Force
Tool Rotation Angle [deg]
Surface Roughness
R_{max}= 12 μm
Y [μm]
Z [mm]
(axial direction)
X [mm]
(feed direction)
[Amp.]
FFT for Resultant Force
Chatter
Frequency
(1448 Hz)
Frequency [Hz]


Bull Nose Cutter and Al7075
(9500 rpm)
(4.7 mm)
(14000 rpm)
(4.7 mm)
indle speed [rev/min]

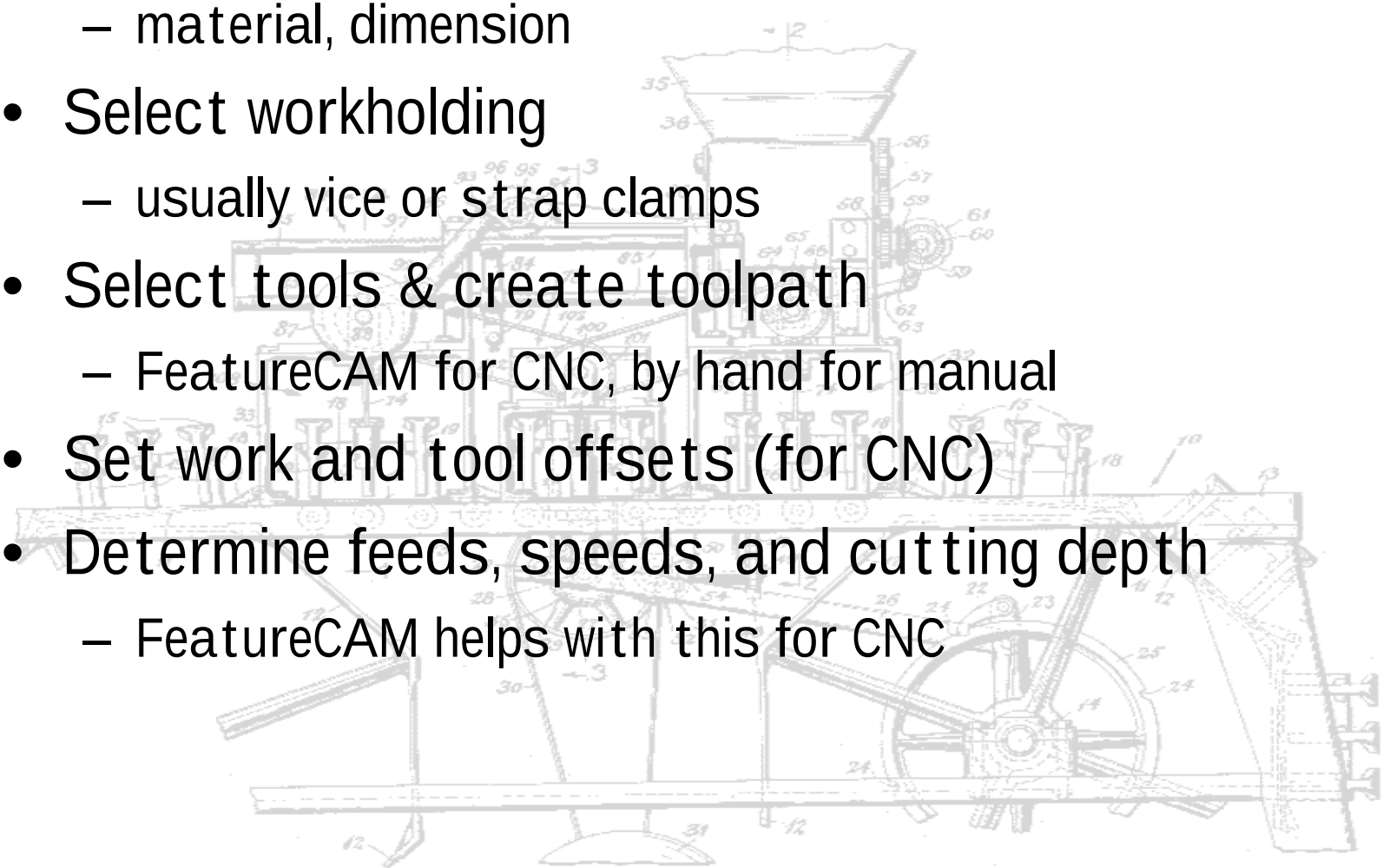

[N]
Experimental Resultant Force
Tool Rotation Angle [deg]
[N]
Simulated Resultant Force
Tool Rotation Angle [deg]
Surface Roughness
R_{max}= 2 μm
Y [μm]
Z [mm]
(axial direction)
X [mm]
(feed direction)
[Amp.]
FFT for Resultant Force
Tooth Passing
Frequency
(467 Hz)
Frequency [Hz]

Process

- Rigidity:
  - use shortest tool and tool holder
    - deflection of tool or work causes *form* error
  - keep workpiece firmly clamped and supported
  - avoid speed/feed/depth combos that chatter
- Heat:
  - use coated tools when heat is a problem
  - keep chips cleared (liquid or air coolant)
    - hard chips get harder
    - soft chips stick to tool
  - don't go too fast OR too slow
- Chip load:
  - keep volume removed constant!
  - especially watch tool entry, exit, corners

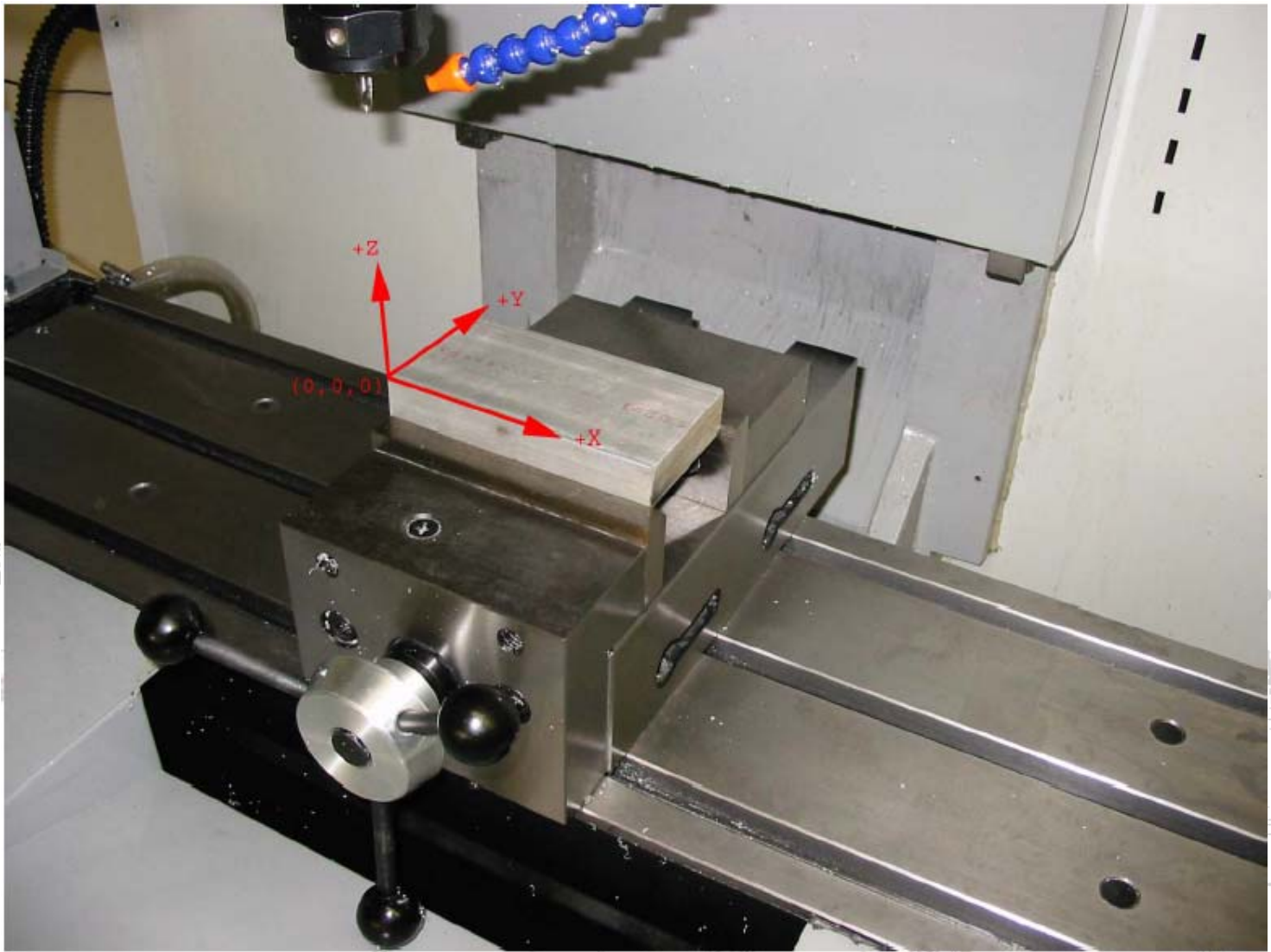
Doing Vertical Milling

- Select stock
  - material, dimension
- Select workholding
  - usually vice or strap clamps
- Select tools & create toolpath
  - FeatureCAM for CNC, by hand for manual
- Set work and tool offsets (for CNC)
- Determine feeds, speeds, and cutting depth
  - FeatureCAM helps with this for CNC



Vertical Milling

Coordinate System



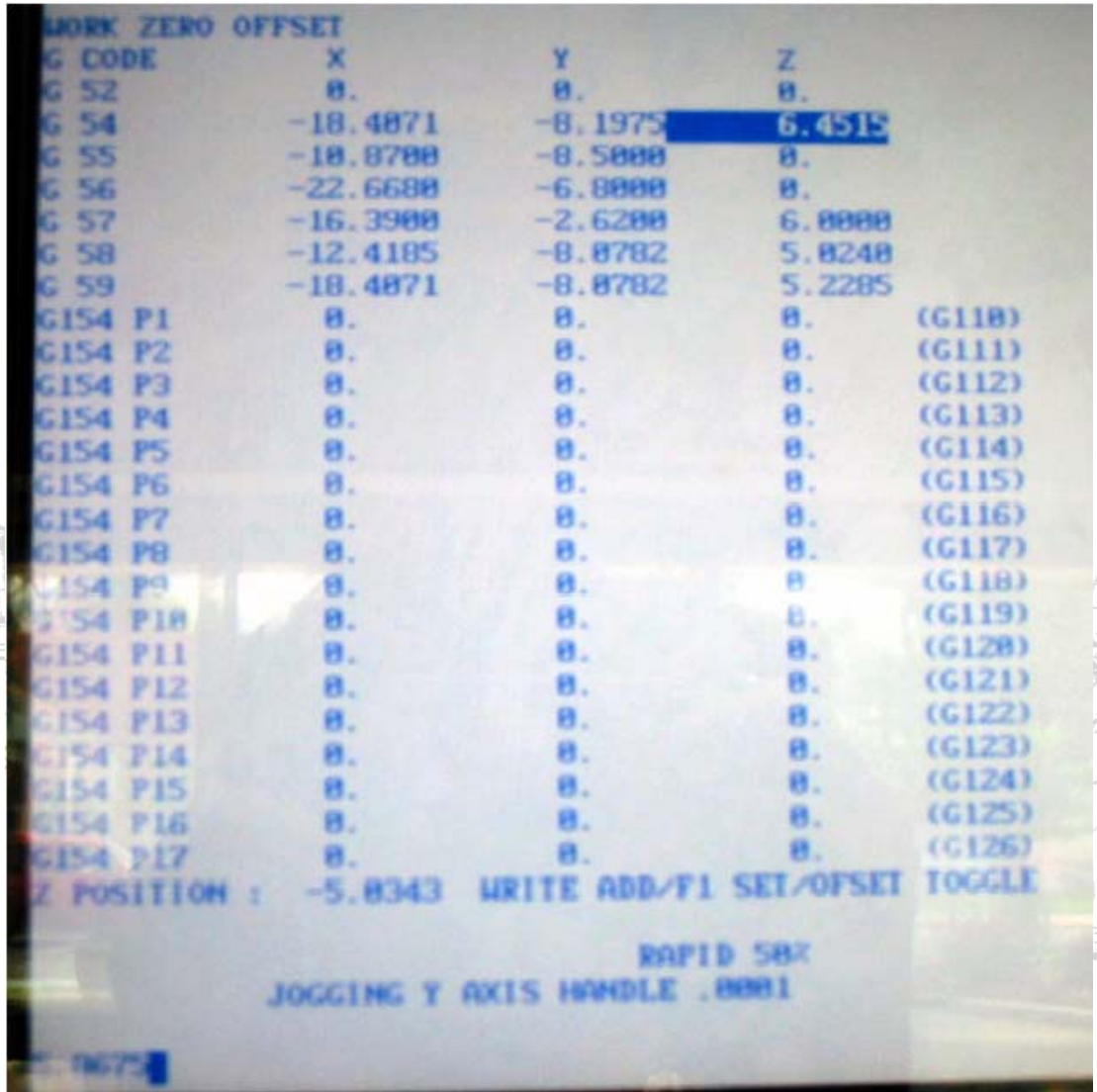
Work Offsets

- Work Offsets
  - G54-G59
  - G54 X & Y aligned with vice step jaw left front
  - Set G54 Z to height of top of work (type number, press F1)

Offsets


RESET
POWER UP
TOOL CHANGER
DISPLAY
PRGRM
POSIT
OFFSET
CURNT COMDS
ALARM
PARAM
SETNG GRAPH
HELP CALC
F1
F2
F3
F4
HOME
PAGE UP
END
PAGE DOWN
JOG LOCK
EDIT
INSERT
ALTER
DELETE
UNDO
MEM
SINGLE BLOCK
DRY
OPTION STOP
BLOCK DELETE
MDI DNC
COOLNT
ORIENT SPINDLE
ATC FWD
ATC REV
HAND JOG
.0001
.001
.01
.1
ZERO RET
ALL AXES
ORIGIN
SINGL AXIS
HOME G28
LIST PROG
SELECT PROG
SEND RS232
RECV RS232
ERASE PROG
OVERRIDES
CNTRL FEED
CNTRL SPIN
SPINDLE
CW
STOP
CCW
RAPID
SHIFT
A
B
C
D
E
F
G
H
I
J
K
L
M
N
O
P
Q
R
S
T
U
V
W
X
Y
Z
EOB
CANCEL
SPACE
WRITE ENTER

Setting Work Offset

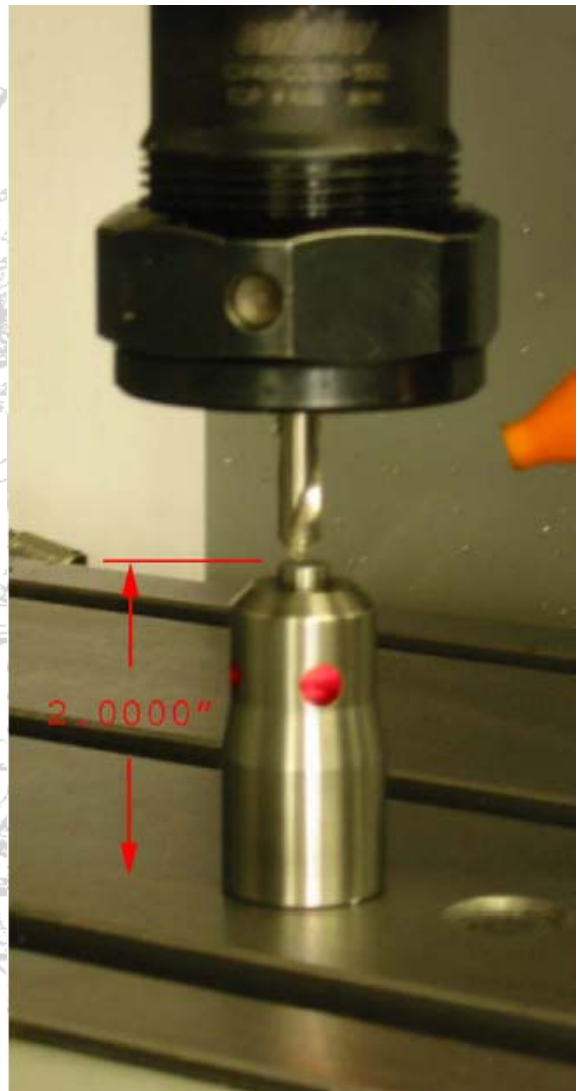


Tool Offsets

- Select tool #
- Jog until touch
- Press “Tool Offset Measure”
- Subtract 2.000” (-2.0 Enter)

TOOL	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR	FLUTES
1		-16.1442	0.	0.1250	0.	2
2		-16.7966	0.	0.1250	0.	2
3		-16.7883	0.	0.1250	0.	2
4		-12.2747	0.	0.1000	0.	2
5		-13.9795	0.	0.3750	0.	4
6		-16.2011	0.	0.2500	0.	2
7		-14.4871	0.	0.3125	0.	2
8		-14.4289	0.	0.1250	0.	2
9		-16.9389	0.	0.0625	0.	2
10		-17.0816	0.	0.0625	0.	2
11		0.	0.	0.	0.	2
12		0.	0.	0.	0.	2
13		0.	0.	0.	0.	2
14		0.	0.	0.	0.	2
15		0.	0.	0.	0.	2
16		0.	0.	0.	0.	2
17		0.	0.	0.	0.	2
18		0.	0.	0.	0.	2
19		0.	0.	0.	0.	2
20		0.	0.	0.	0.	2

Z POSITION : -14.1442 WRITE ADD/F1 SET/OFSET TOGGLE



CNC Programming

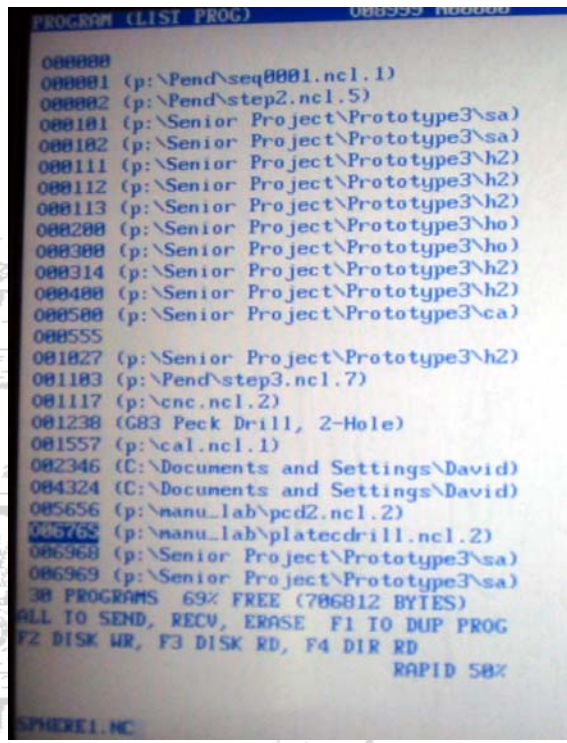
- Example CNC program - bores a center hole and drills bolt circle
- Haas manual at: k:\class\engr\480\haas\usermanual.pdf

```
%
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);
```

Loading CNC Program from Floppy

- Name program with 8 or fewer letters
- Copy to a: drive
- Put in Haas
  - press List Prog, type in name, press F3



Mill/Turn Video