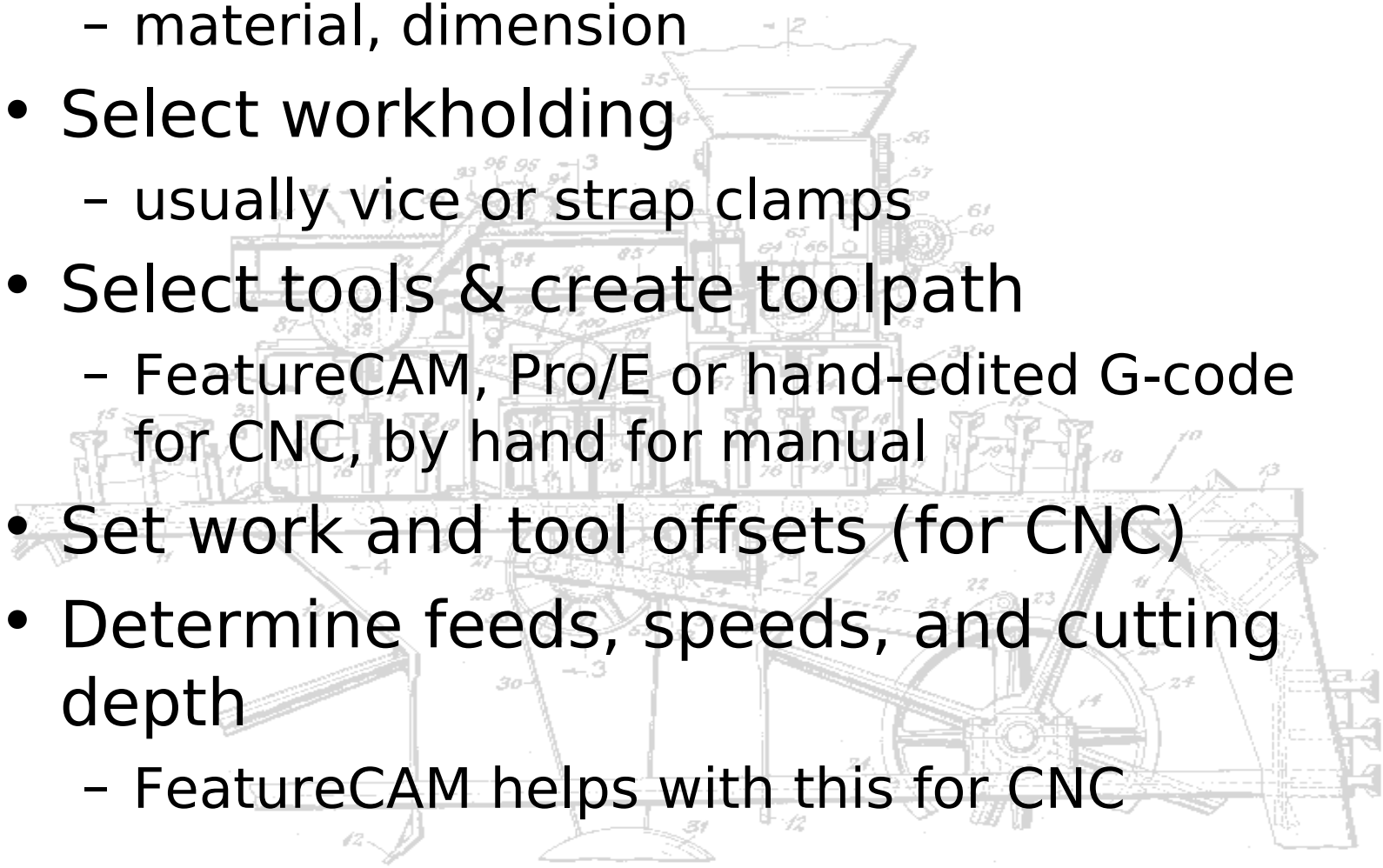
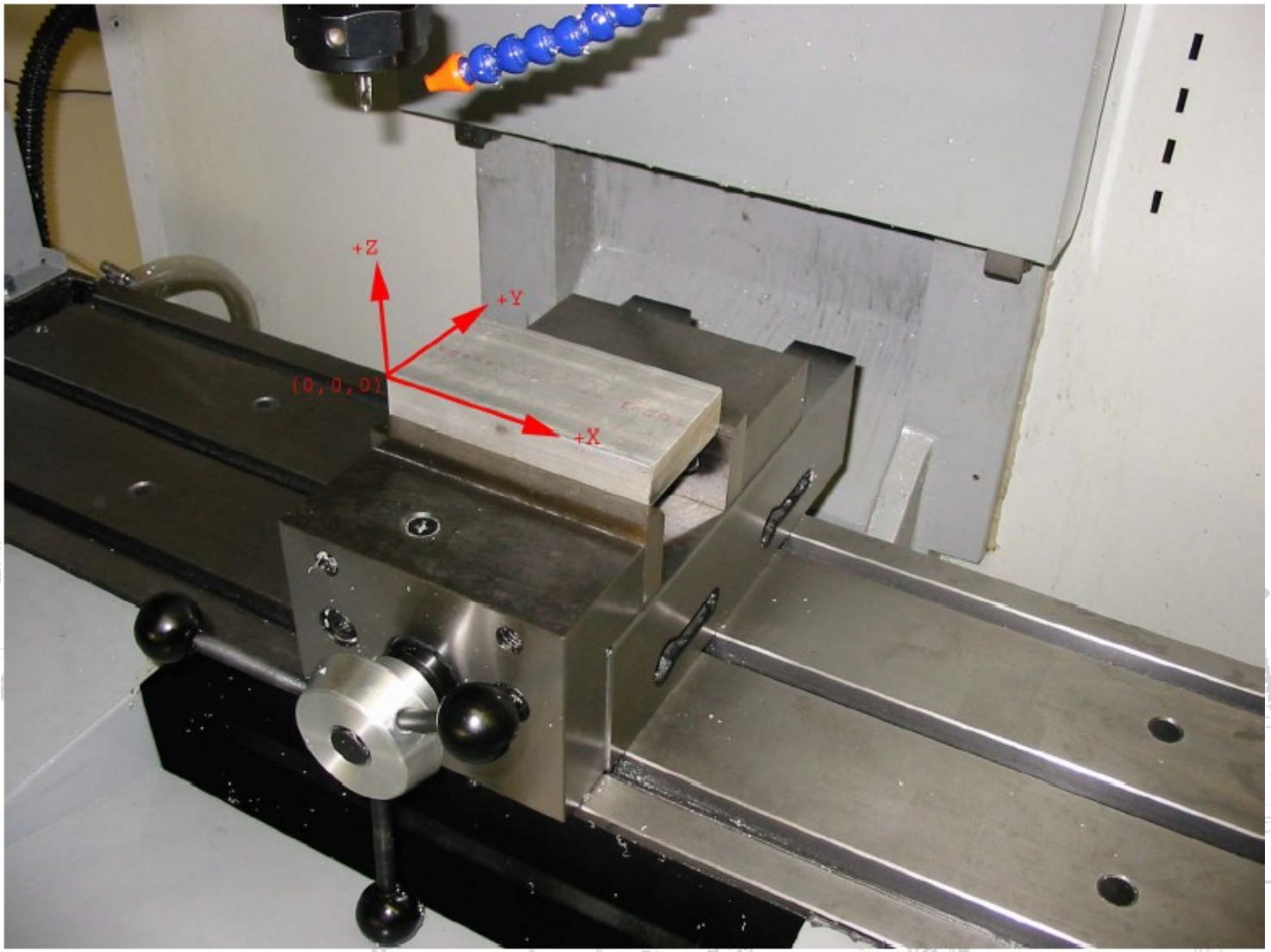
Doing Vertical Milling

- Select stock
  - material, dimension
- Select workholding
  - usually vice or strap clamps
- Select tools & create toolpath
  - FeatureCAM, Pro/E or hand-edited G-code 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

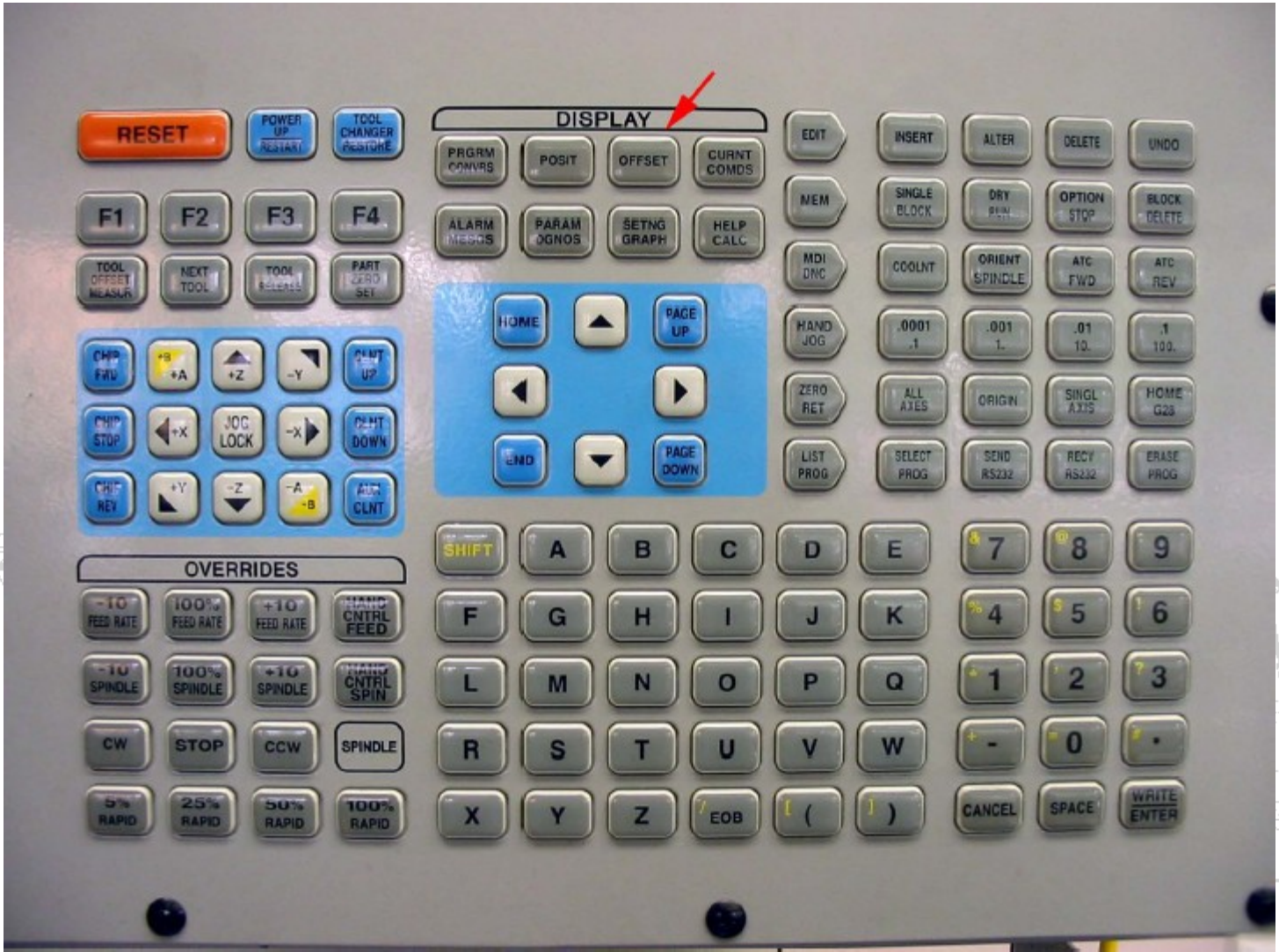
Coordinate System



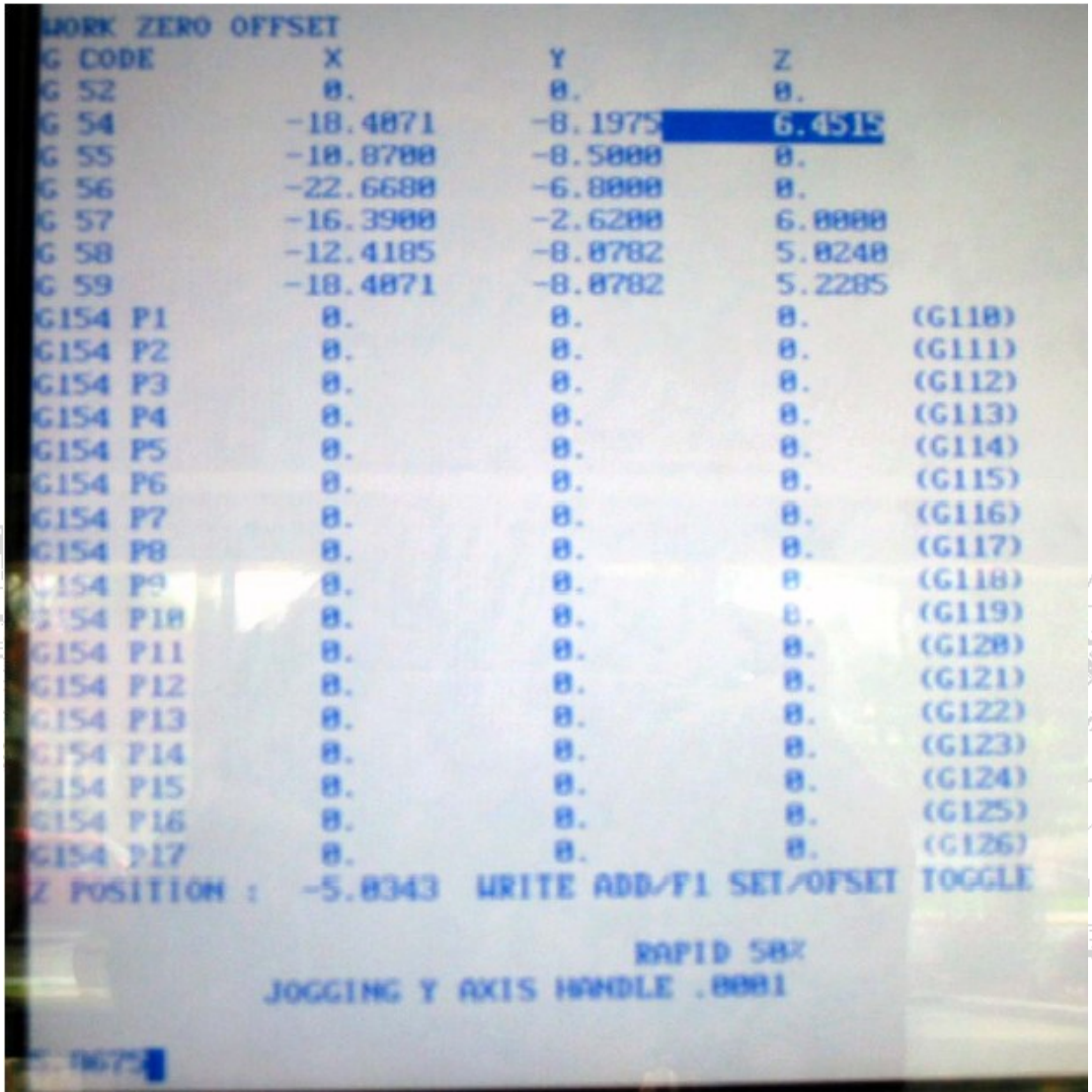
Work Offsets

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

Offsets


RESET
POWER UP RESTART
TOOL CHANGER RESTORE
DISPLAY
PRGRM CONVRS
POSIT
OFFSET
CURNT COMDS
ALARM MESGS
PARAM DGNOS
SETNG GRAPH
HELP CALC
F1
F2
F3
F4
TOOL OFFSET MEASUR
NEXT TOOL
TOOL RELEASE
PART ZERO SET
HOME
PAGE UP
END
PAGE DOWN
JOG LOCK
CHIP FWD
CHIP STOP
CHIP REV
CLNT UP
CLNT DOWN
AUX CLNT
EDIT
MEM
MDI DNC
HAND JOG
ZERO RET
LIST PROG
INSERT
ALTER
DELETE
UNDO
SINGLE BLOCK
DRY RUN
OPTION STOP
BLOCK DELETE
COOLNT
ORIENT SPINDLE
ATC FWD
ATC REV
.0001 .1
.001 1.
.01 10.
.1 100.
ALL AXES
ORIGIN
SINGL AXIS
HOME G28
SELECT PROG
SEND RS232
RECV RS232
ERASE PROG
OVERRIDES
FEED RATE
CNTRL FEED
SPINDLE
CNTRL SPIN
CW
STOP
CCW
SPINDLE
RAPID
25% RAPID
50% RAPID
100% 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

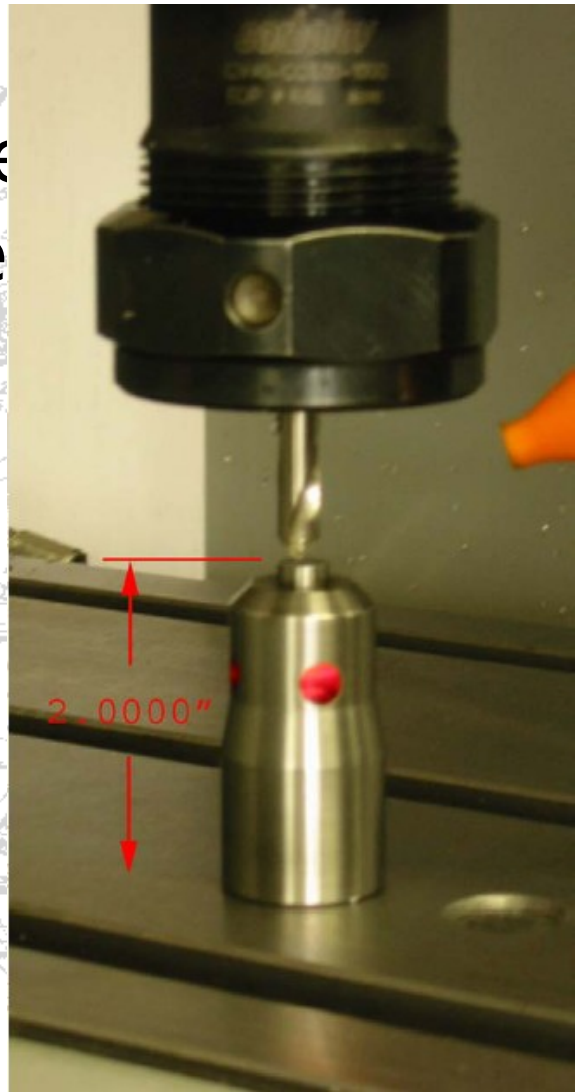


Tool Offsets

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

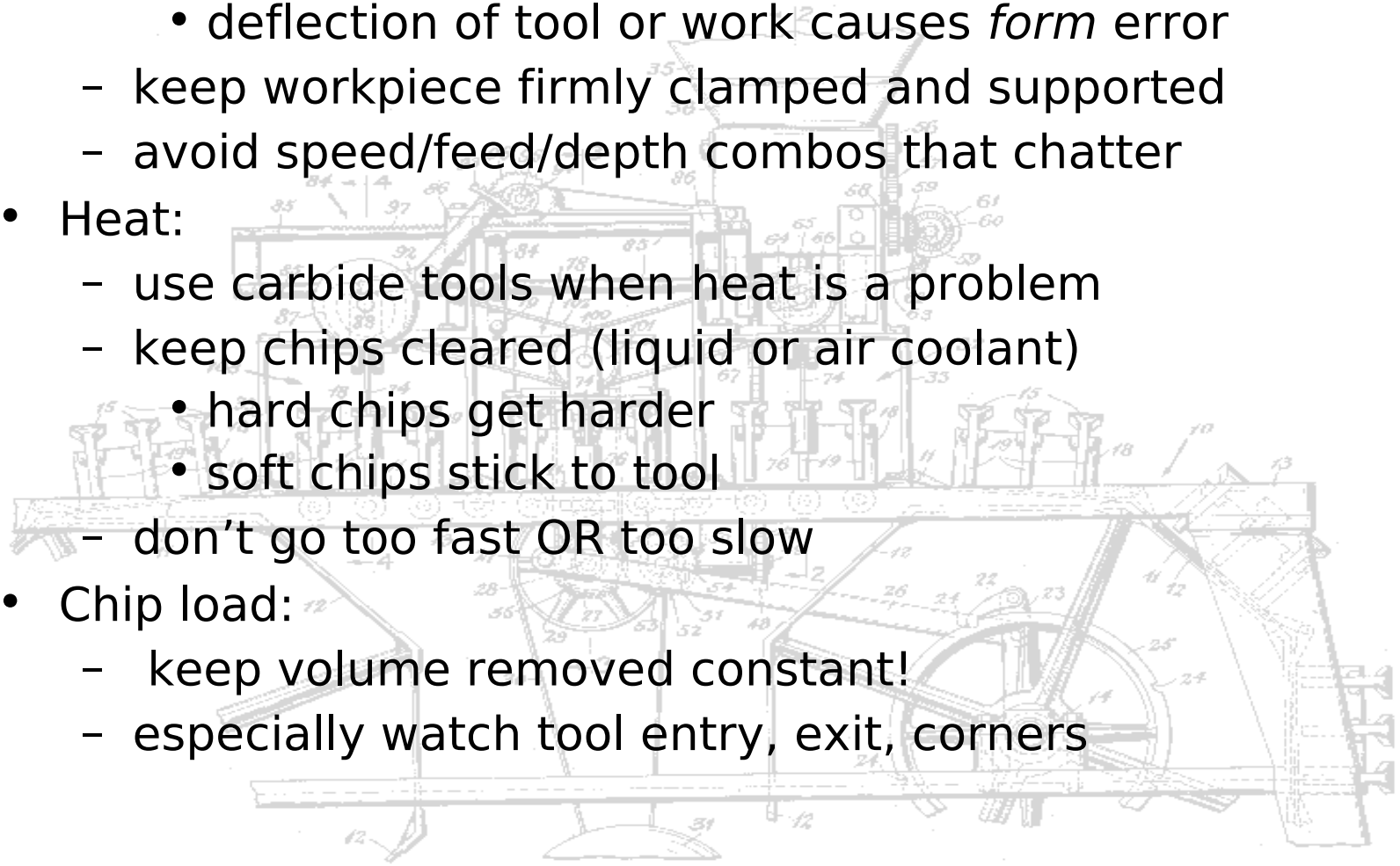
TOOL	POSITION	GEOMETRY	WEAR	GEOMETRY	WEAR	FLUTES
1		-16.1442	0.	0.1250	0.	2
2		-16.7966	0.	0.1250	0.	2
3		-16.7883	0.	0.1250	0.	2
4		-12.2747	0.	0.1000	0.	2
5		-13.9795	0.	0.3750	0.	4
6		-16.2011	0.	0.2500	0.	2
7		-14.4871	0.	0.3125	0.	2
8		-14.4289	0.	0.1250	0.	2
9		-16.9389	0.	0.0625	0.	2
10		-17.0816	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



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 carbide 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

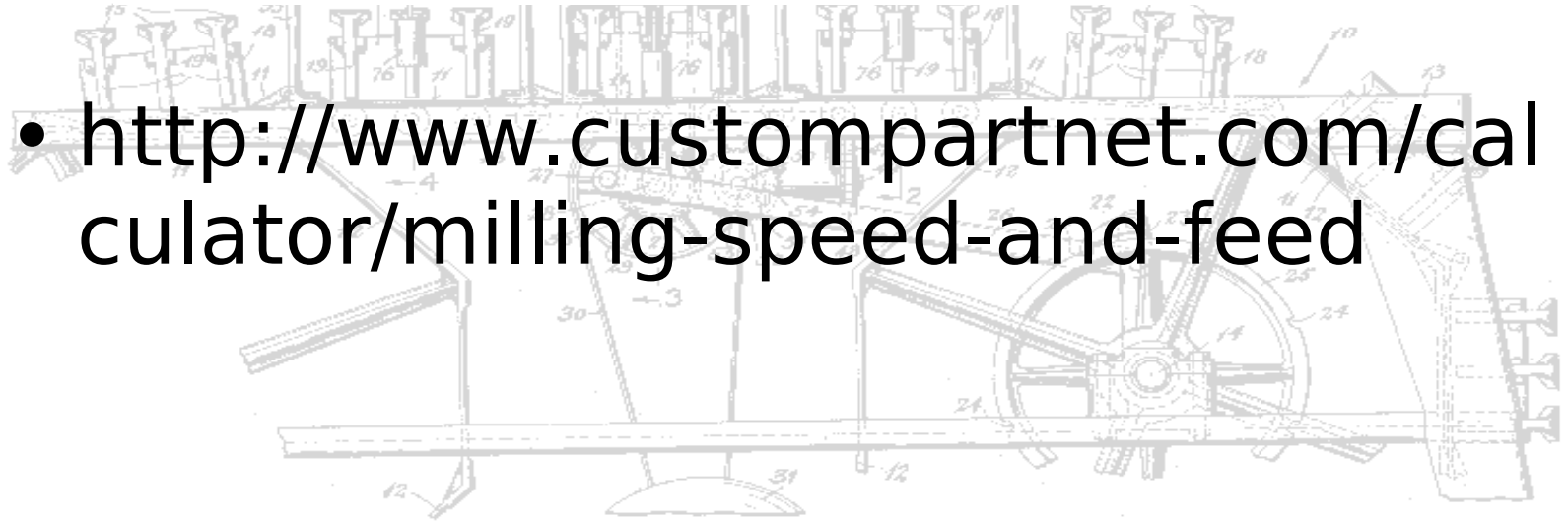


Setting Feeds & Speeds

Aluminum (6061, 2024, 7075)				
SFM	Chipload Per Tooth			
2, 3, & 4 Flute	up to .125 dia.	.125-.250 dia.	.250-.500 dia.	.500-1.0 dia.
300-500	.0008-.0020	0015-.0040	0020-.0060	0030-.0090

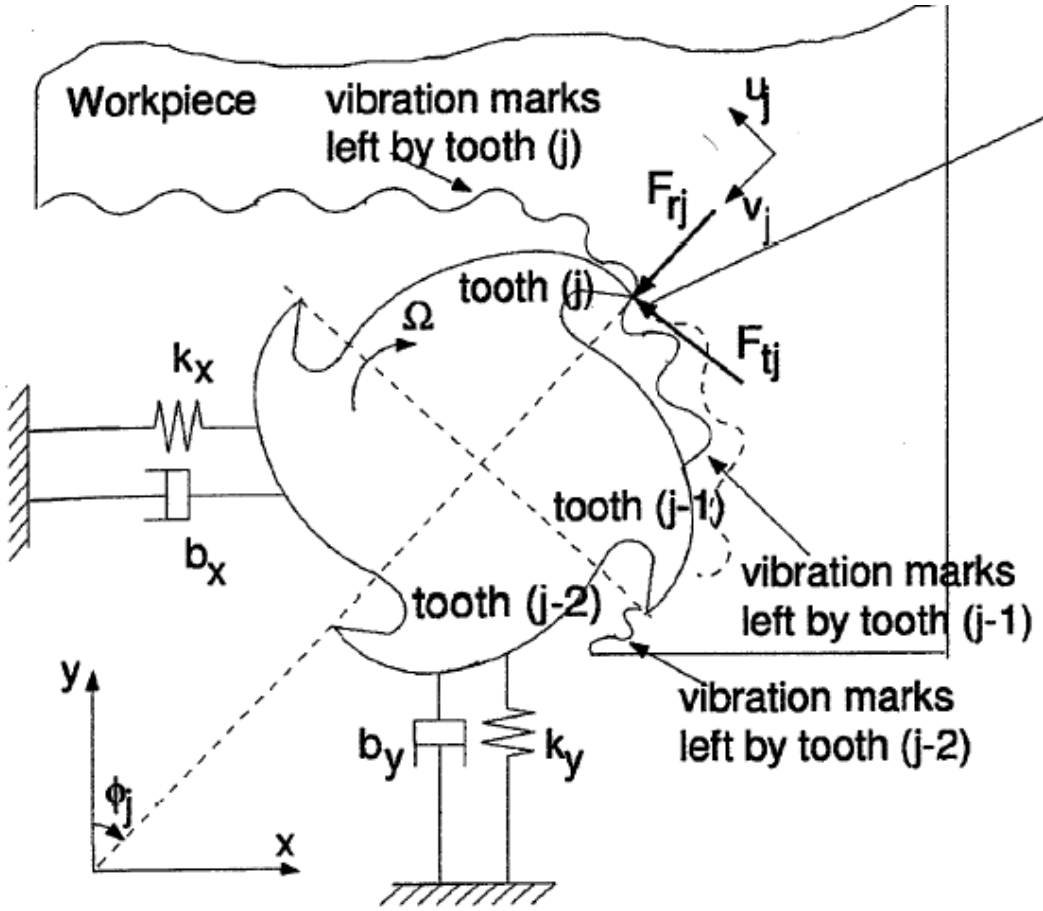
Tool Steels <30 RC (4140, 4340, A2, D2, O1, S7, P2,H13)				
SFM	Chipload Per Tooth			
2, 3, & 4 Flute	up to .125 dia.	.125-.250 dia.	.250-.500 dia.	.500-1.0 dia.
150-225	.0005-.0010	0008-.0020	0010-.0030	0020-.0040

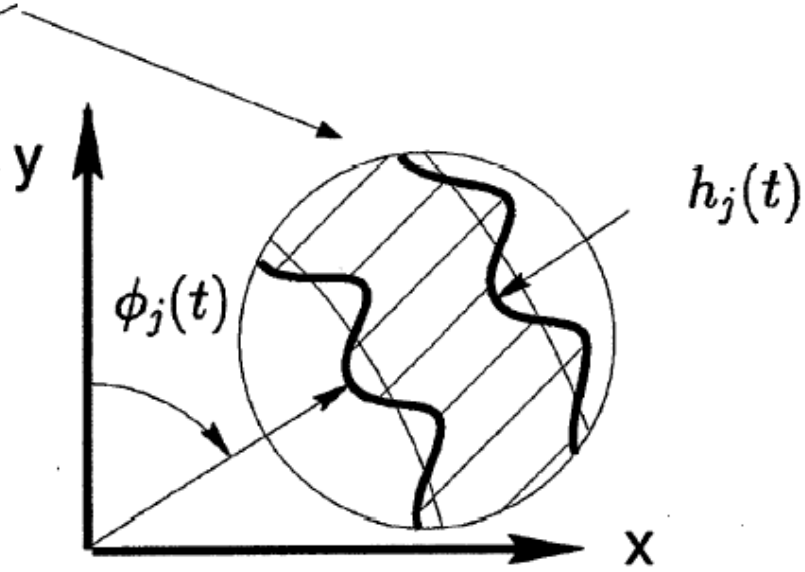
Carbon Steels <35 RC (A36, 1000's, 1100's, 1300's)				
SFM	Chipload Per Tooth			
2, 3, & 4 Flute	up to .125 dia.	.125-.250 dia.	.250-.500 dia.	.500-1.0 dia.
175-250	.0006-.0015	0010-.0025	0015-.0040	0020-.0050



- http://www.custompartnet.com/calculator/milling-speed-and-feed

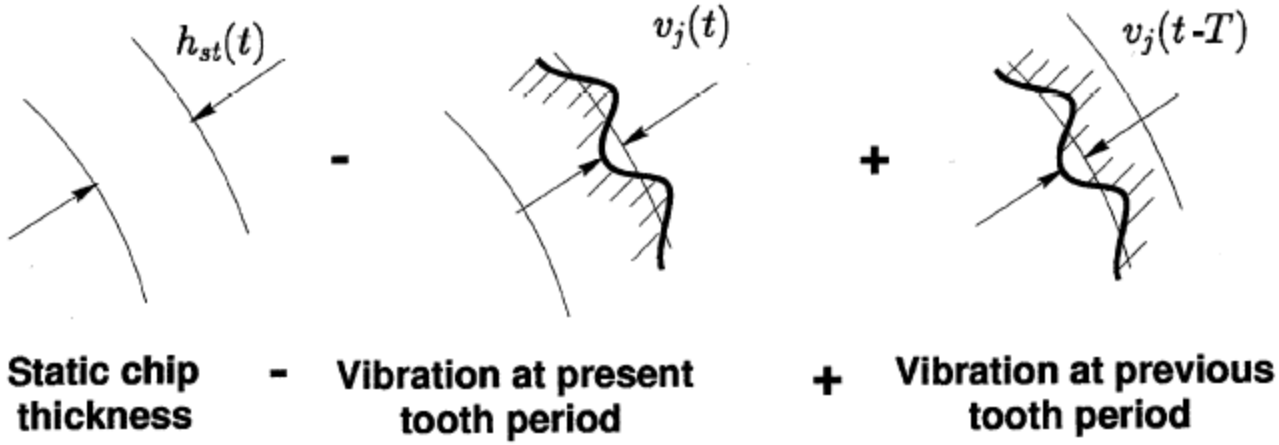
Vibration (chatter)





Dynamic chip thickness

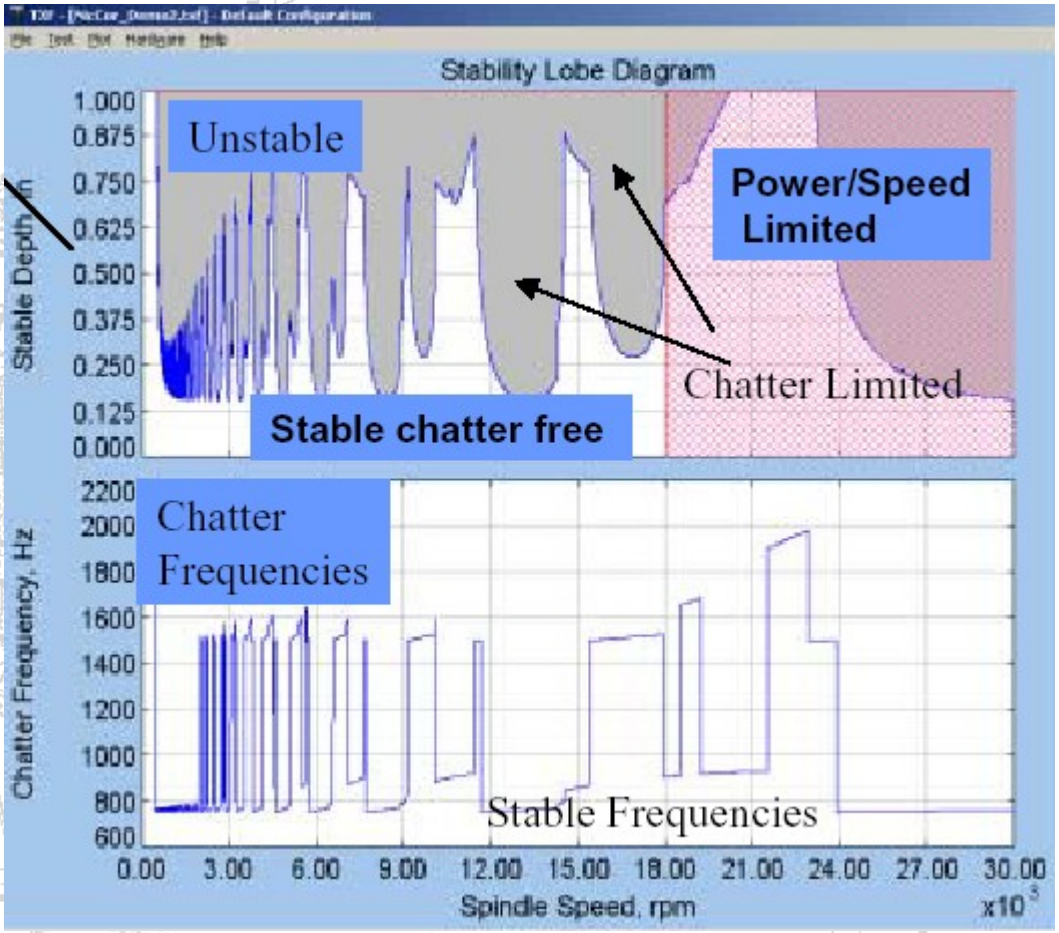
Vibration



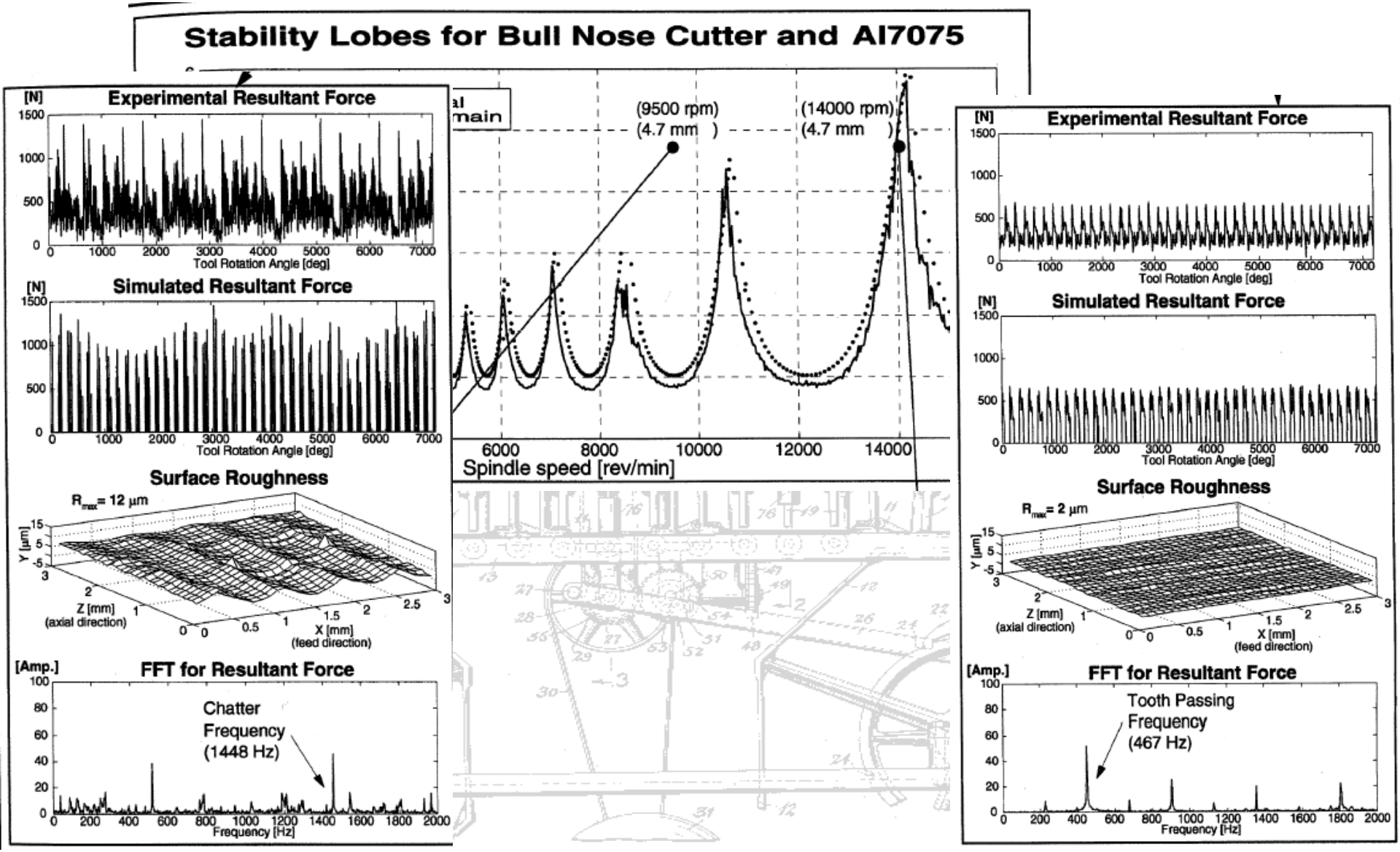
Static chip thickness - **Vibration at present tooth period** + **Vibration at previous tooth period**

Vibration

- For Max Material Removal Rate:
  - Choose highest spindle RPM
  - Tune tool length to stay in a stable lobe at top spindle RPM

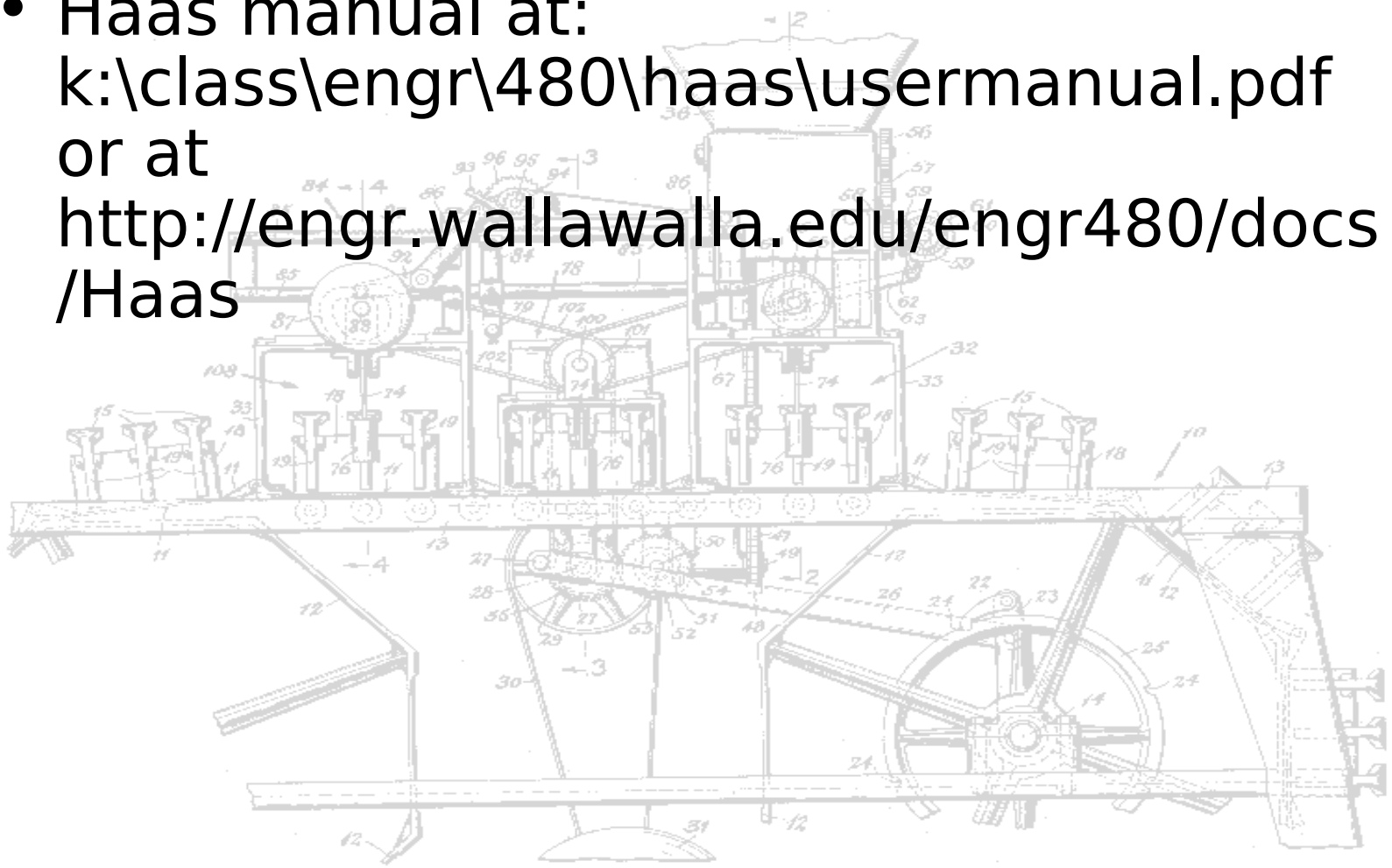


Vibration


Stability Lobes for Bull Nose Cutter and Al7075
(9500 rpm)
(4.7 mm)
(14000 rpm)
(4.7 mm)
6000
8000
10000
12000
14000
Spindle speed [rev/min]
[N]
Experimental Resultant Force
Tool Rotation Angle [deg]
Simulated Resultant Force
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]
R_max= 2 µm
Tooth Passing
Frequency
(467 Hz)

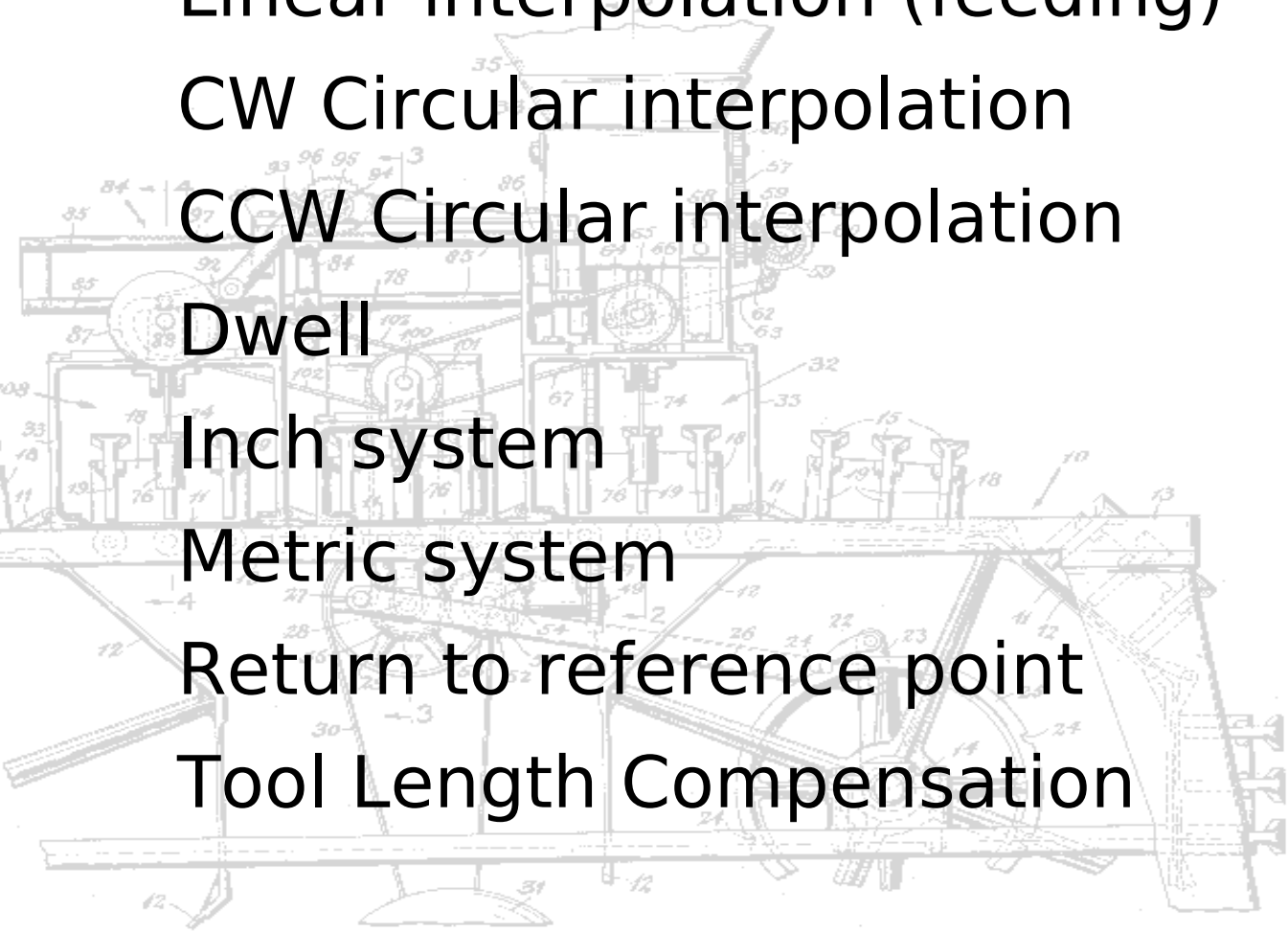
CNC Programming for Mill

- Example CNC program – drills four holes
- Haas manual at:
  k:\class\engr\480\haas\usermanual.pdf
  or at
  http://engr.wallawalla.edu/engr480/docs/Haas



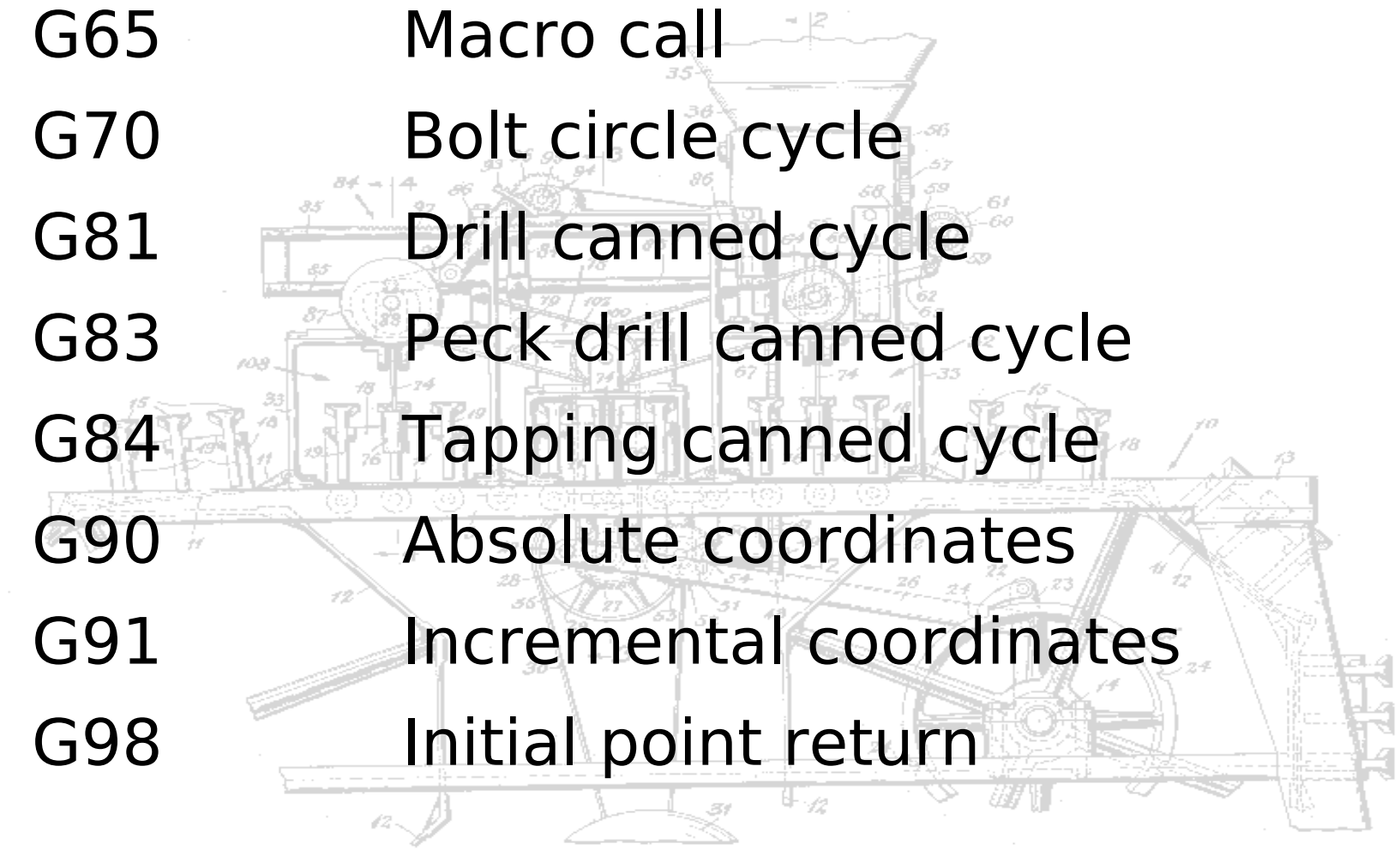
G-Codes for Milling

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



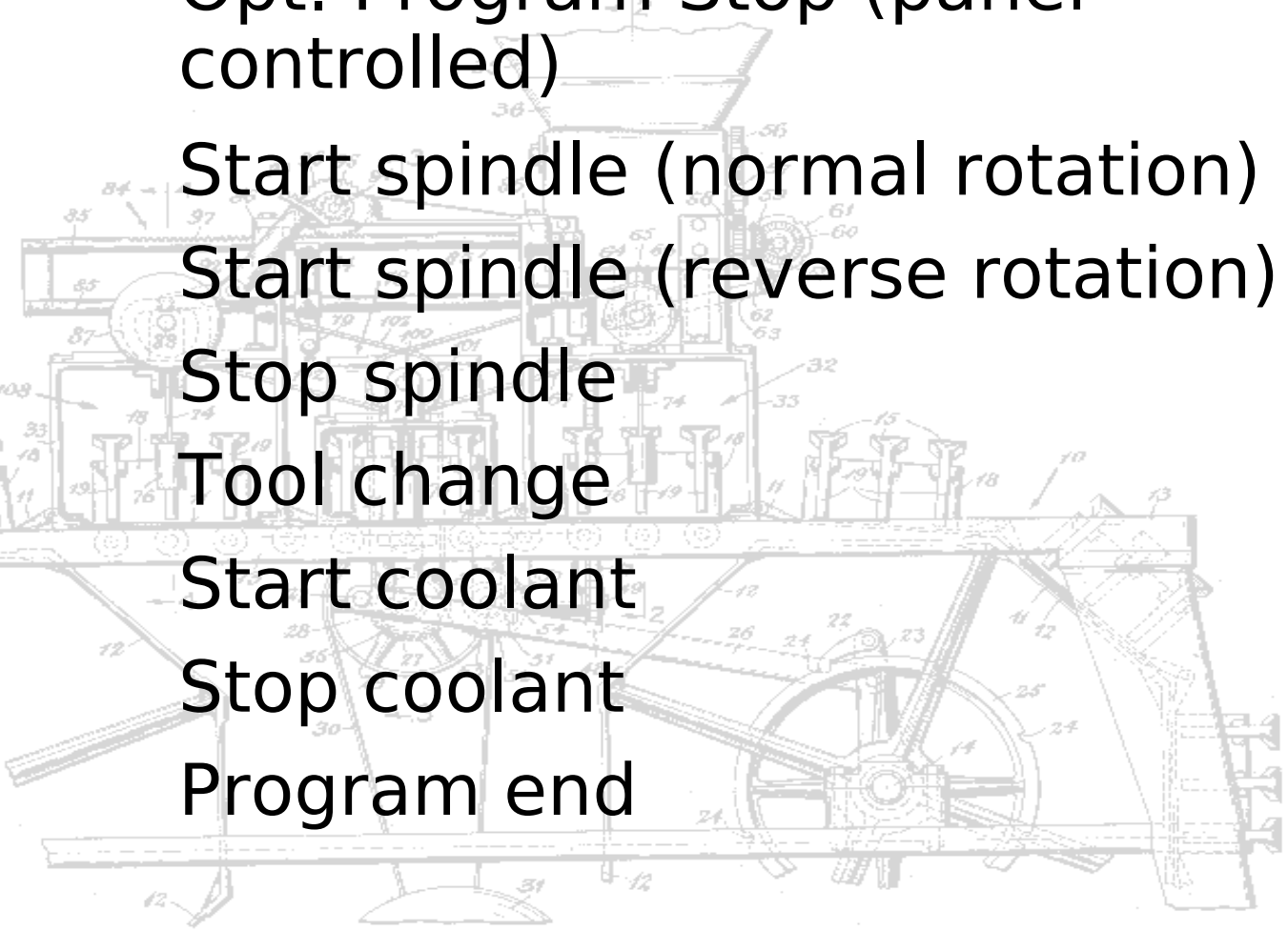
G-Codes for Milling

G54-G59	Select work coord system
G65	Macro call
G70	Bolt circle cycle
G81	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



Example Program

```
%
O01001
(DRILL EXAMPLE - 2013-04-05)
(OP1- TOP SIDE)
(STOCK- 2IN X 2IN X 1/4IN)
(G54 ZERO- LEFT FRONT TOP)

G54
G00 G40 G90 G17 G20

(SPOT DRILL HOLES)
T7 M6
G43 H7
S2000 M3
M8
G0 X0.50 Y0.50 Z0.1
G81 Z-0.1 R0.1 F10.; (CANNED
```

```
(DRILL 1/4IN HOLES)

T9 M6
G43 H9
S2000 M3
M8
G0 X0.50 Y0.50 Z0.1
G83 Z-0.35 Q0.1 R0.1
   F10.0
X0.50 Y1.00
X1.00 Y1.00
X1.00 Y0.50
G80 G00 Z0.1

G00 Z6.0

M9
```

Loading CNC Program

- Copy to K:\class\engr\cnc as haascode.txt
- Press List Prog
- Type program number (O01234)
- Press RECV RS232