Vertical Milling

Doing Vertical Milling

- Select stock
  - material, dimension
- Select workholding
  - usually vise or strap clamps
- Select tools & create toolpath
  - Esprit, Fusion360, Inventor HSM or hand-edited G-code for CNC, by hand for manual
- Set work and tool offsets (for CNC)
- Determine feeds, speeds, and cutting depth
  - CAM tools help with this for CNC (Esprit, HSM Works, Fusion360)
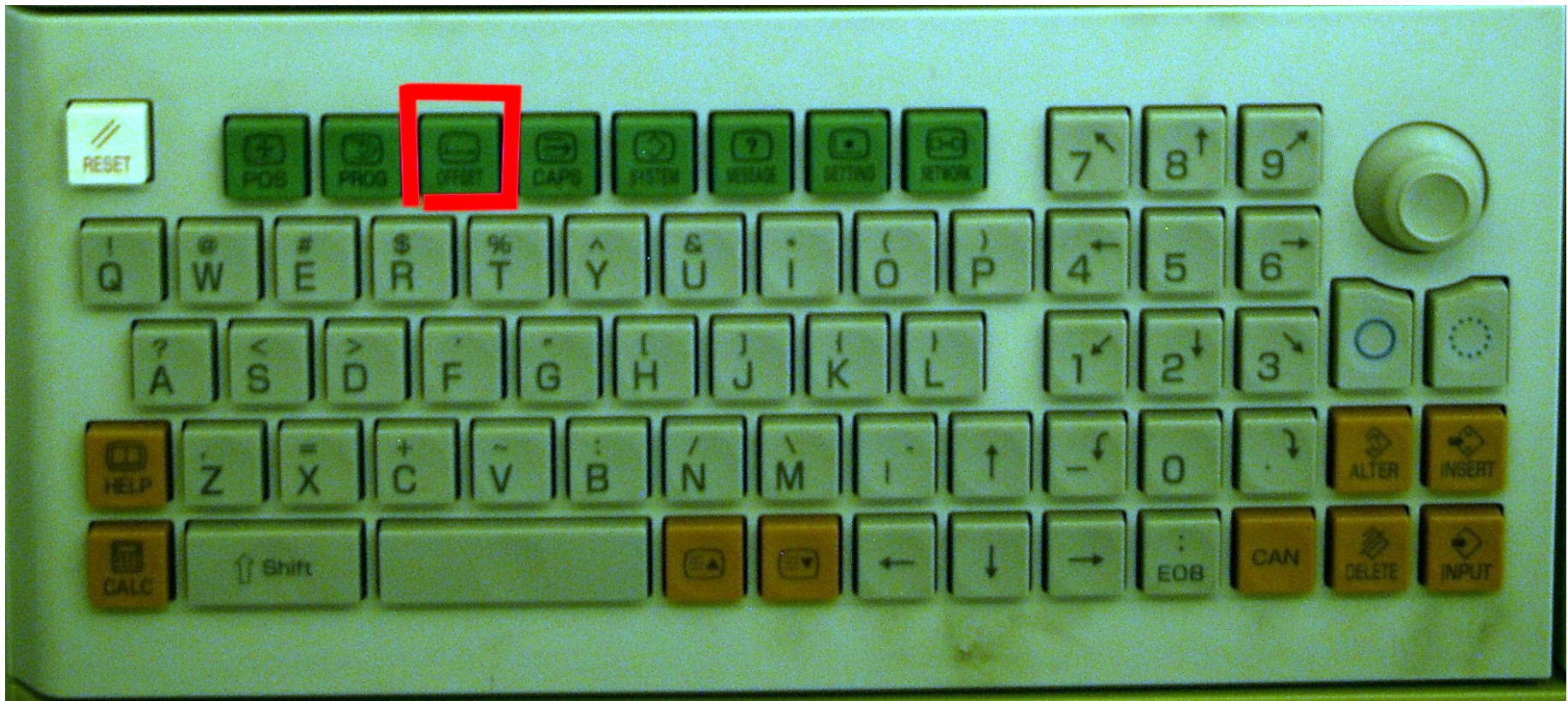
Coordinate System
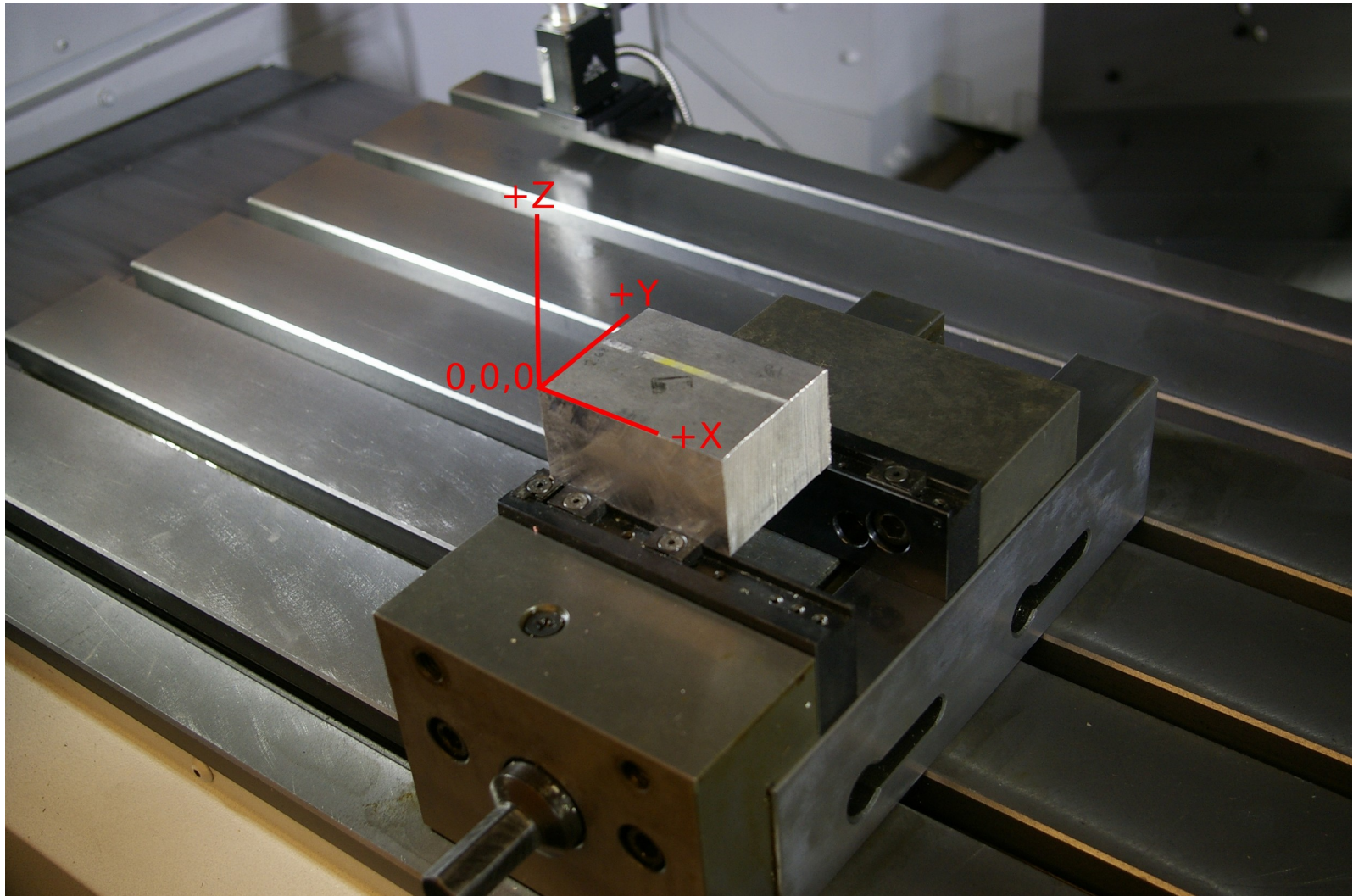
Work Offsets

- G54-G59
- Different offsets will be identified for different vise jaw corners.
- If Z0 is top of stock, measure from table and subtract 600.0mm.
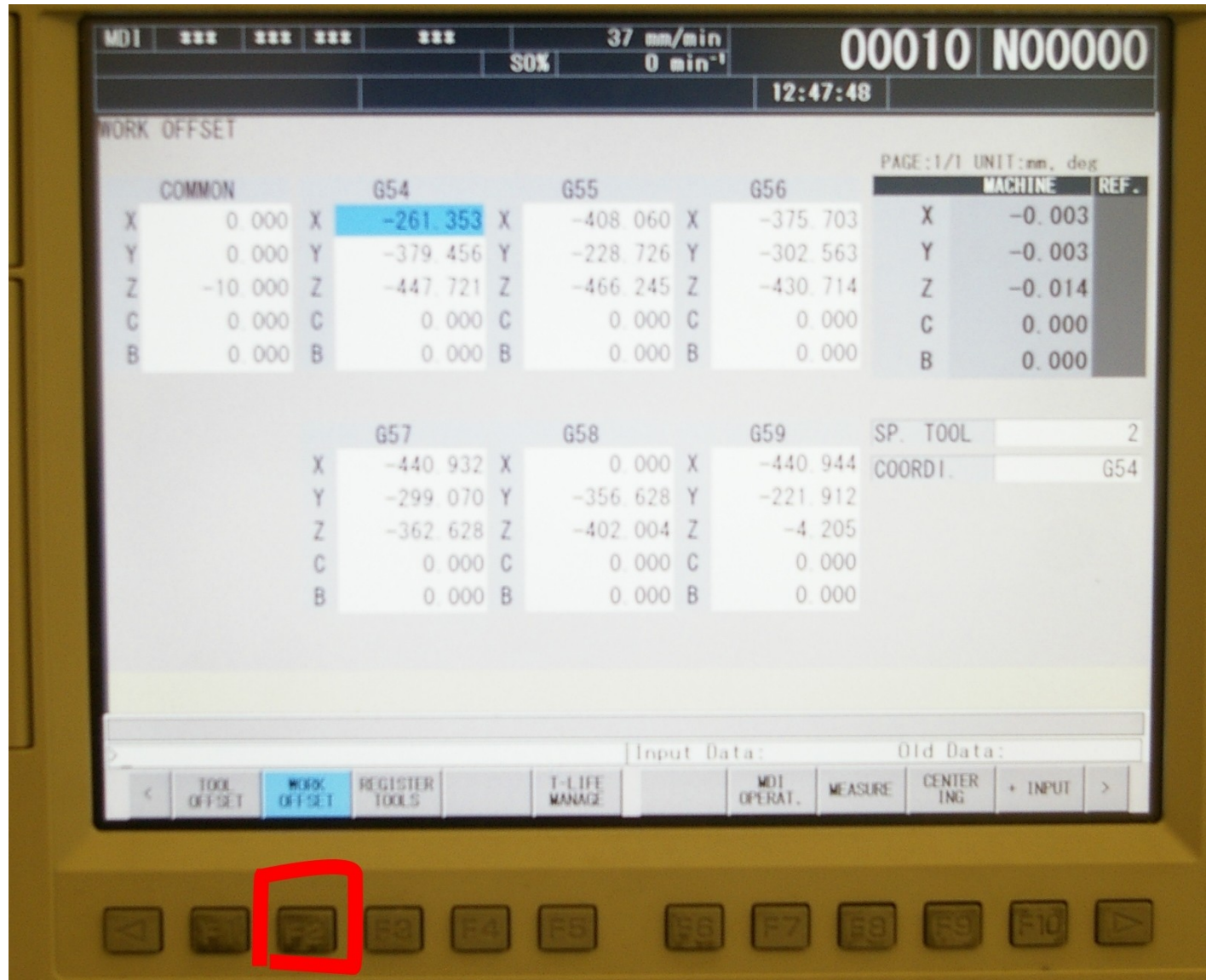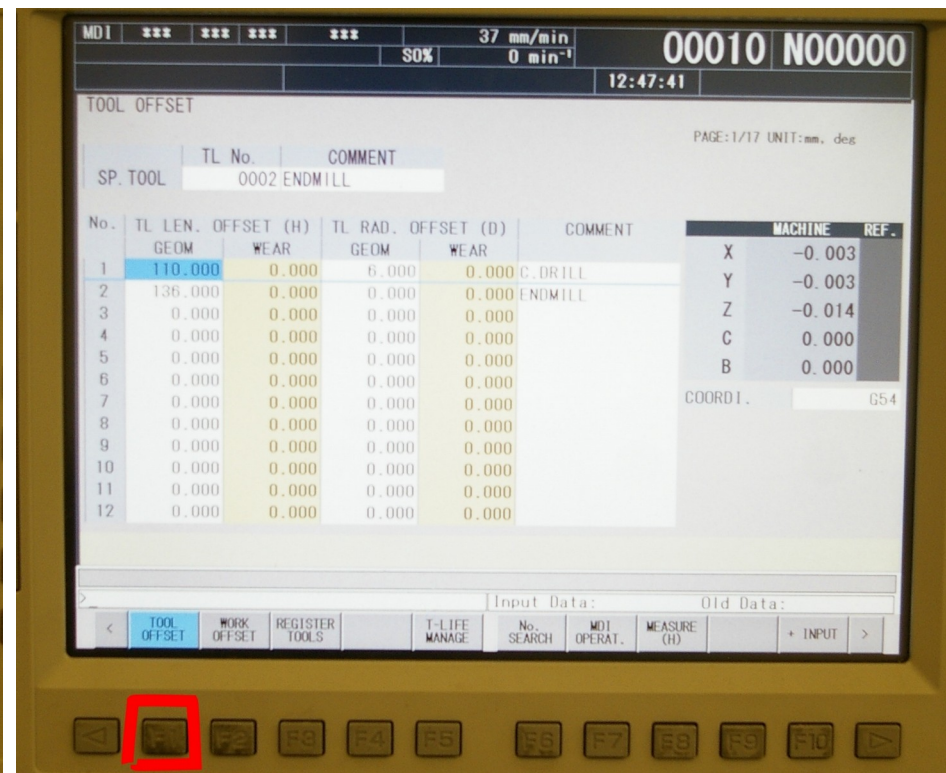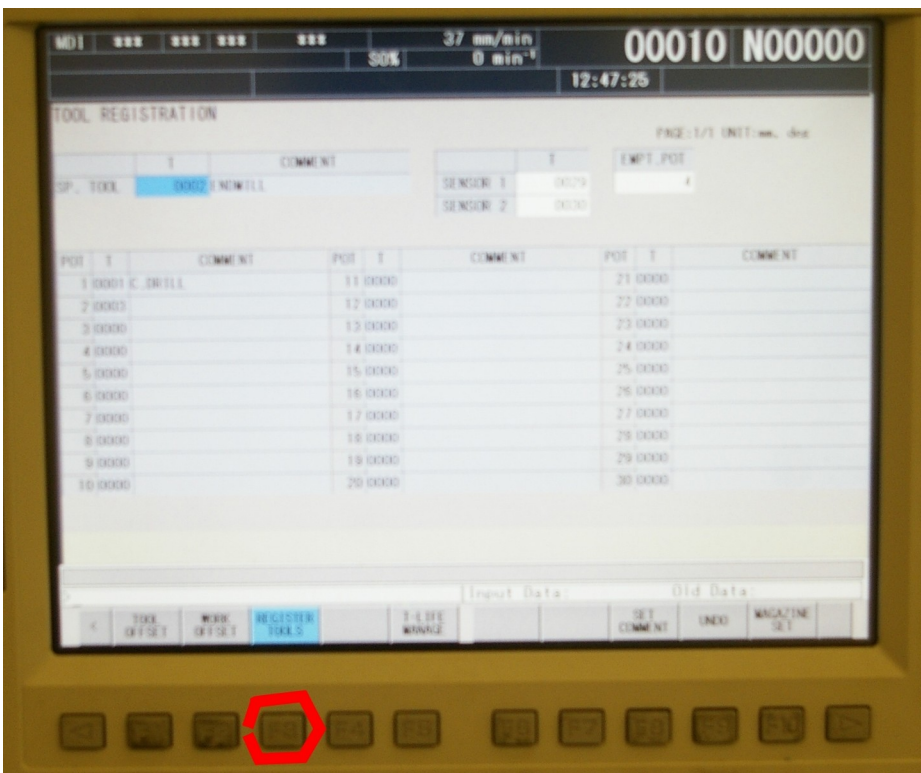- Work probe should be available soon.
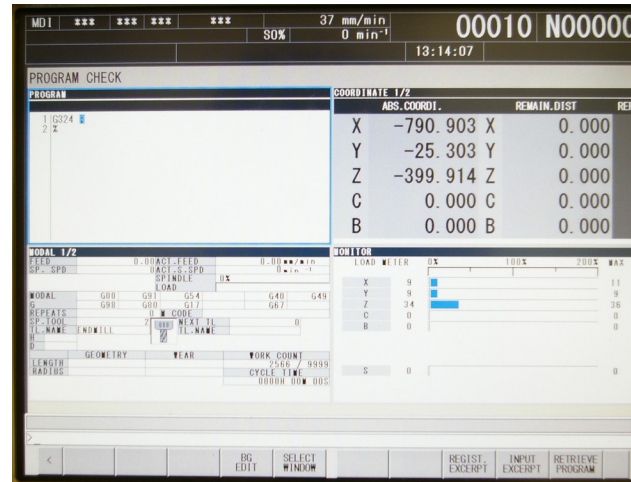
Offsets

Setting Work Offset

Tool Offsets

- Enter tool data in offset table
- Put tool in spindle
- Indicate tool number in TOOL REGISTRATION screen

Tool Offsets

- Run program O0002, move tool close to sensor, press START again.
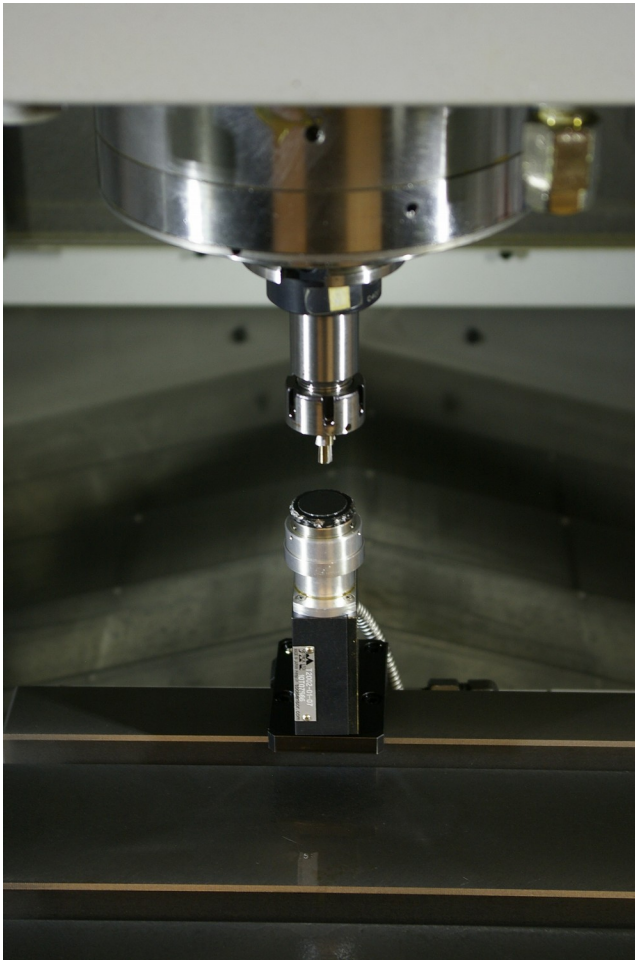
Process

- 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

$h_{st}(t)$ $v_j(t)$ $v_j(t\text{-}T)$

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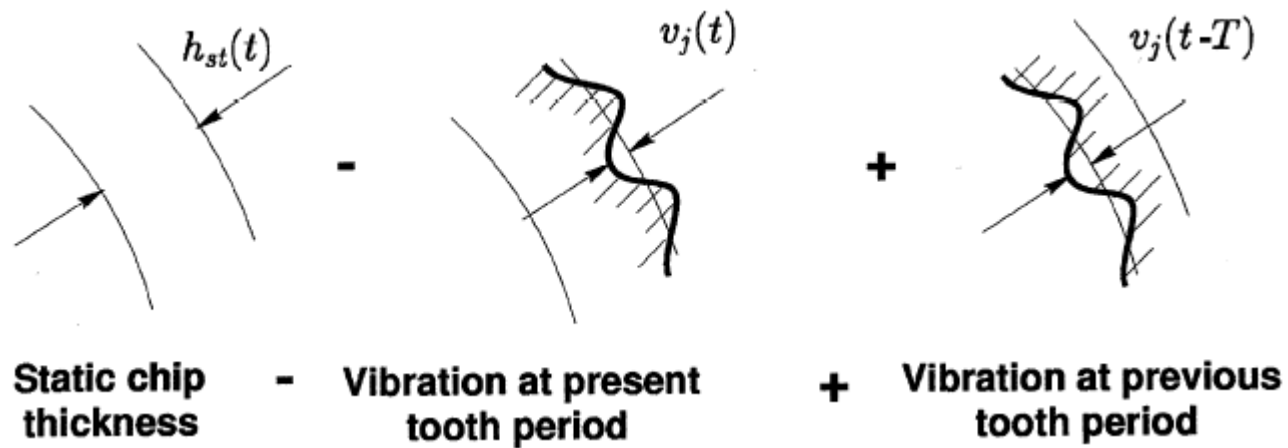
Vibration

- 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)

Spindle speed [rev/min]

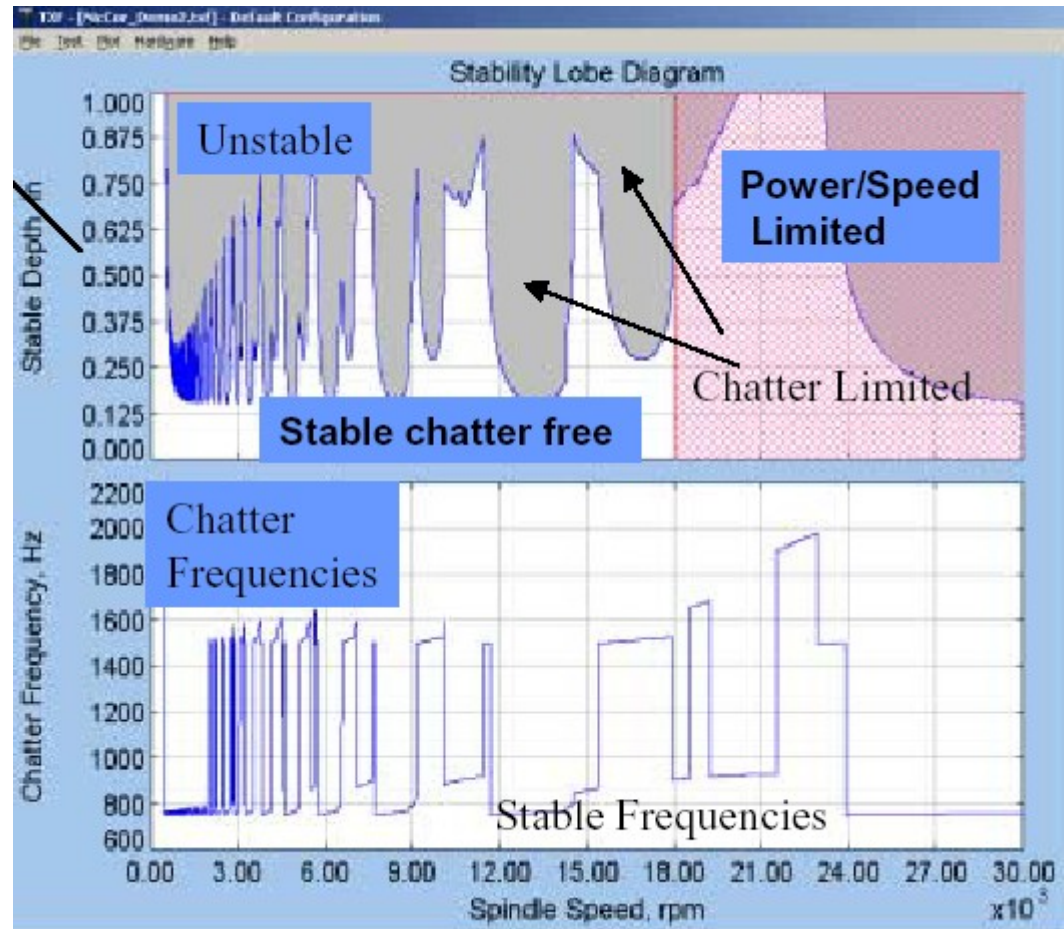**Experimental Resultant Force**

Tool Rotation Angle [deg]

Simulated Resultant Force

Tool Rotation Angle [deg]

Surface Roughness

R_{max} = 12 μm

Z [mm] (axial direction) X [mm] (feed direction)

FFT for Resultant Force

Chatter Frequency (1448 Hz)

Frequency [Hz]

Experimental Resultant Force

Tool Rotation Angle [deg]

Simulated Resultant Force

Tool Rotation Angle [deg]

Surface Roughness

R_{max} = 2 μm

Z [mm] (axial direction) X [mm] (feed direction)

FFT for Resultant Force

Tooth Passing Frequency (467 Hz)

Frequency [Hz]

Example Program

```
%

O0010 (SIMPLE EXAMPLE FOR NVX)

(OP1- TOP SIDE)

(STOCK- 50MM X 40MM X 50MM)

(G54 ZERO- LEFT FRONT TOP)

G54

G00 G40 G90 G17 G21

(SPOT DRILL HOLES)

T7 M6

G43 H7

S2000 M3

M8

G0 X12.5 Y12.5 Z2.5

G81 Z-2.5 R2.5 F250.0;   (CANNED DRILL CYCLE)

    X12.5 Y25.0

    X25.0 Y25.0

    X25.0 Y12.5

G80 G0 Z2.5;               (CANCEL CANNED CYCLE)

M9

M5

(DRILL 6.35MM HOLES)

G28 Z0

T9 M6

G43 H9

S2000 M3

M8

G0 X12.5 Y12.5 Z2.5

G83 Z-2.5 Q2.5 R2.5 F250.0;   (CANNED DRILL CYCLE)

    X12.5 Y25.0

    X25.0 Y25.0

    X25.0 Y12.5

G80 G0 Z2.5;     (CANCEL CANNED CYCLE)

M9

M5

G28 Z0

M30

%
```

G-Codes for Milling

G00	Rapid positioning
G01	Linear interpolation (feeding)
G02	CW Circular interpolation
G03	CCW Circular interpolation
G04	Dwell
G21	Metric system
G30	Return to reference point
G40	Cancel tool radius compensation

G-Codes for Milling

G54-G59   Select work coord system

G65   Macro call

G81   Drill canned cycle

G83   Peck drill canned cycle

G84   Tapping canned cycle

G90   Absolute coordinates

G91   Incremental coordinates

M-Codes for Milling

M0   Program Stop

M1   Opt. Program Stop

M3   Start spindle (normal rotation)

M4   Start spindle (reverse rotation)

M5   Stop spindle

M6   Tool change

M8   Start coolant