

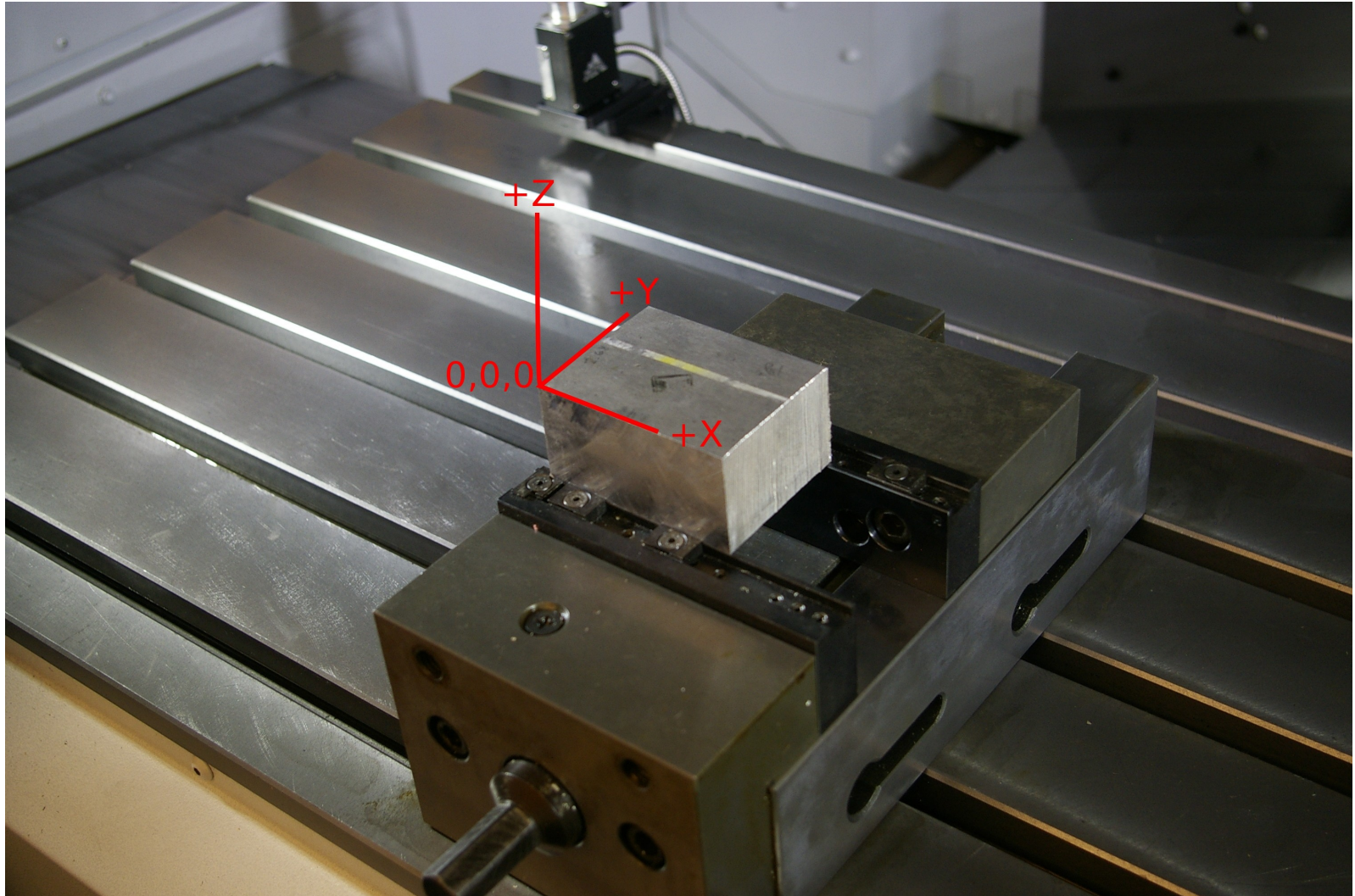
Vertical Milling



Doing Vertical Milling

- Select stock
 - material, dimension
- Select workholding
 - usually vise or strap clamps
- Select tools & create toolpath
 - Esprit, FeatureCAM, Creo or hand-edited G-code for CNC, by hand for manual
 - MfgSuite does accurate simulation of G-code.
- Set work and tool offsets (for CNC)
- Determine feeds, speeds, and cutting depth
 - Esprit and FeatureCAM help with this for CNC

Coordinate System



Example Program

%	(DRILL 6.35MM HOLES)
O0010 (SIMPLE EXAMPLE FOR NVX)	
(OP1- TOP SIDE)	G28 Z0
(STOCK- 50MM X 40MM X 50MM)	T9 M6
(G54 ZERO- LEFT FRONT TOP)	G43 H9
G54	S2000 M3
G00 G40 G90 G17 G21	M8
(SPOT DRILL HOLES)	G0 X12.5 Y12.5 Z2.5
T7 M6	G83 Z-2.5 Q2.5 R2.5 F250.0; (CANNED DRILL CYCLE)
G43 H7	X12.5 Y25.0
S2000 M3	X25.0 Y25.0
M8	X25.0 Y12.5
G0 X12.5 Y12.5 Z2.5	
G81 Z-2.5 R2.5 F250.0; (CANNED DRILL CYCLE)	G80 G0 Z2.5; (CANCEL CANNED CYCLE)
X12.5 Y25.0	
X25.0 Y25.0	M9
X25.0 Y12.5	M5
G80 G0 Z2.5; (CANCEL CANNED CYCLE)	
M9	G28 Z0
M5	M30
	%

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

G81 Drill canned cycle

G83 Peck drill canned cycle

G84 Tapping canned cycle

G90 Absolute coordinates

G91 Incremental coordinates

M-Codes for Milling

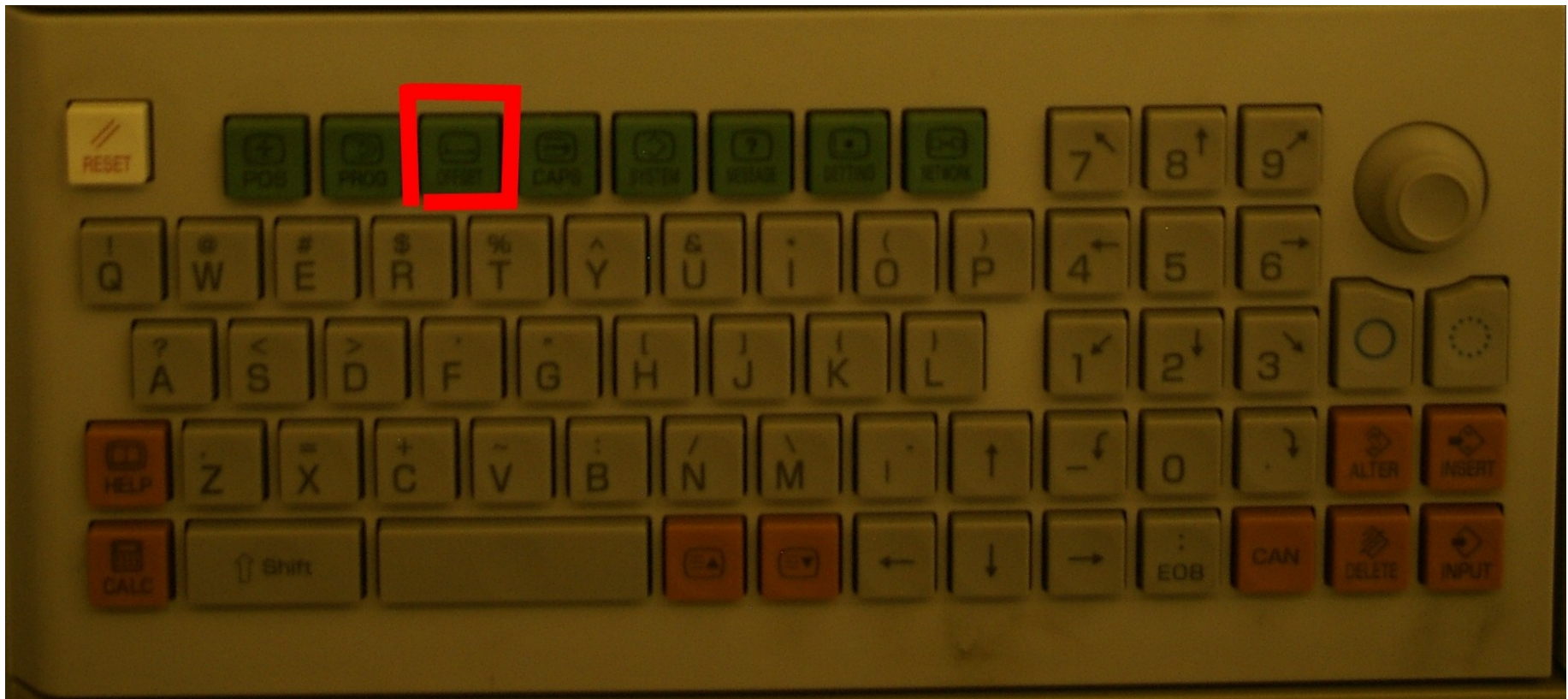
M0	Program Stop
M1	Opt. Program Stop (panel controlled)
M3	Start spindle (normal rotation)
M4	Start spindle (reverse rotation)
M5	Stop spindle
M6	Tool change
M8	Start coolant
M9	Stop coolant
M30	Program end

Work Offsets

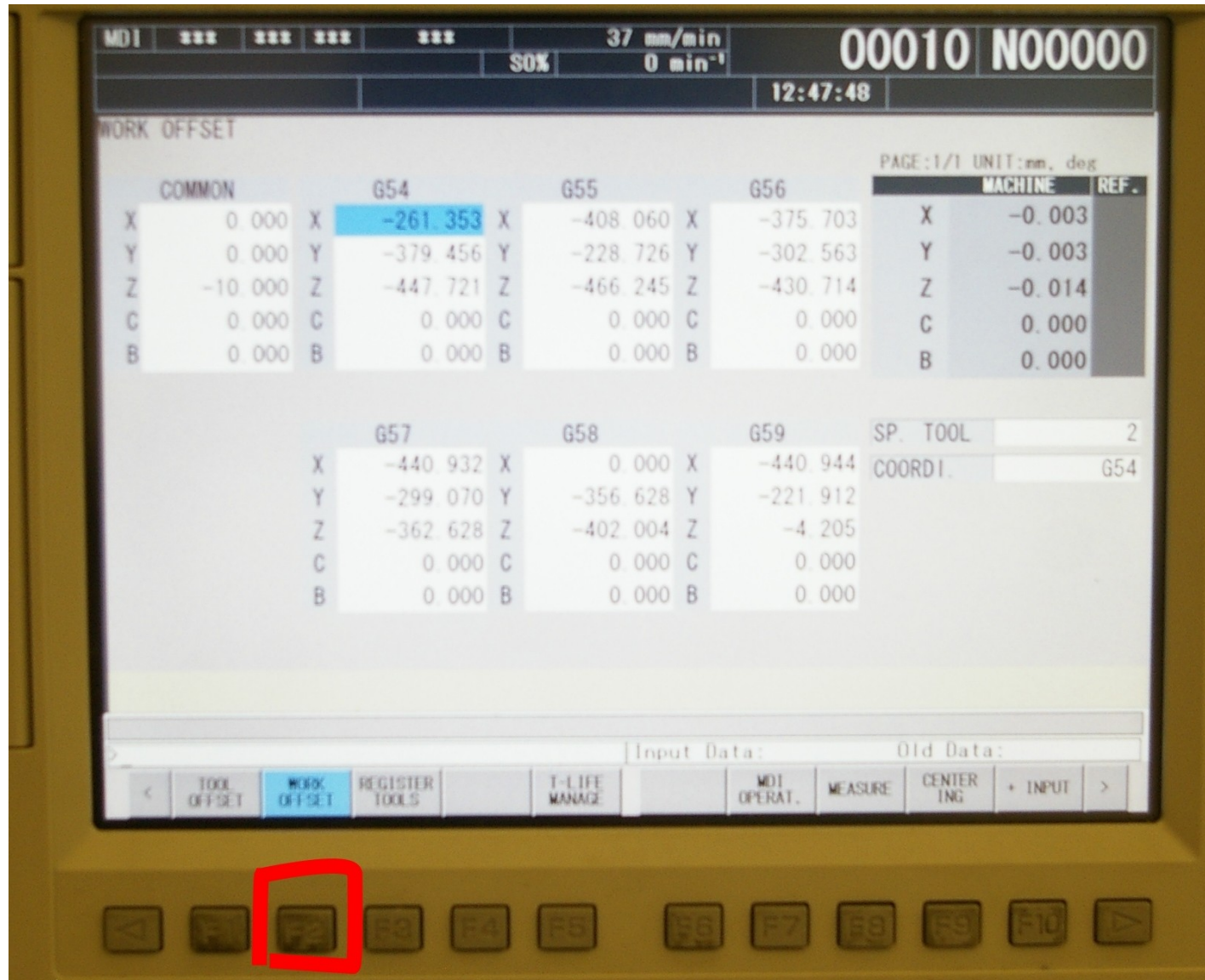
- Work Offsets
 - G54-G59
 - G54 X & Y aligned with vice jaw left front
 - Set G54 Z =
-660.0 + height to desired Z0



Offsets

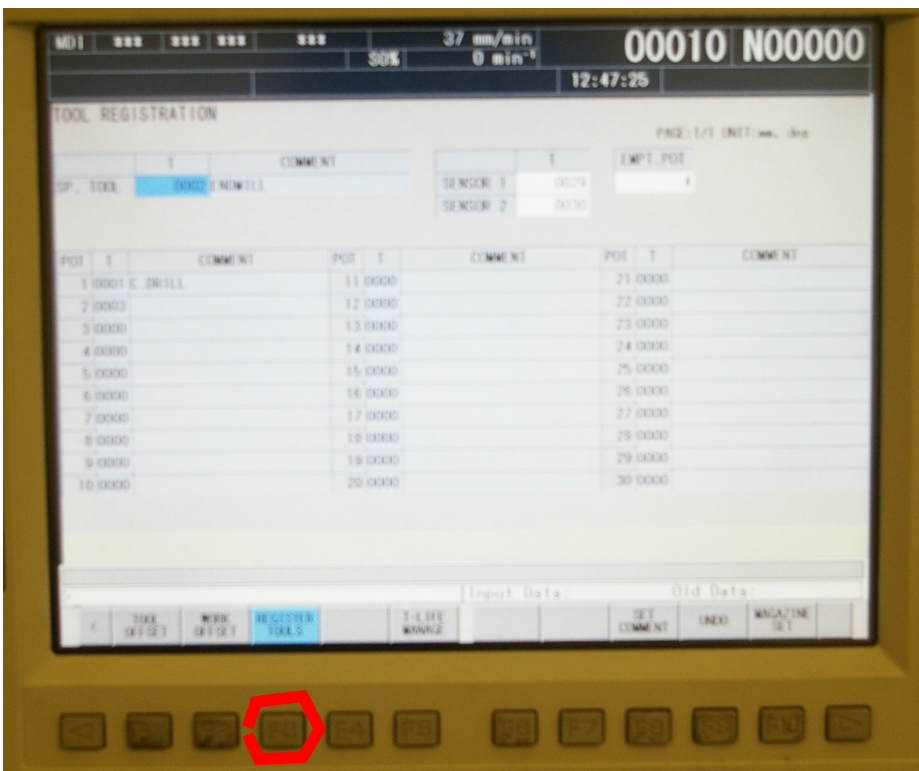


Setting Work Offset



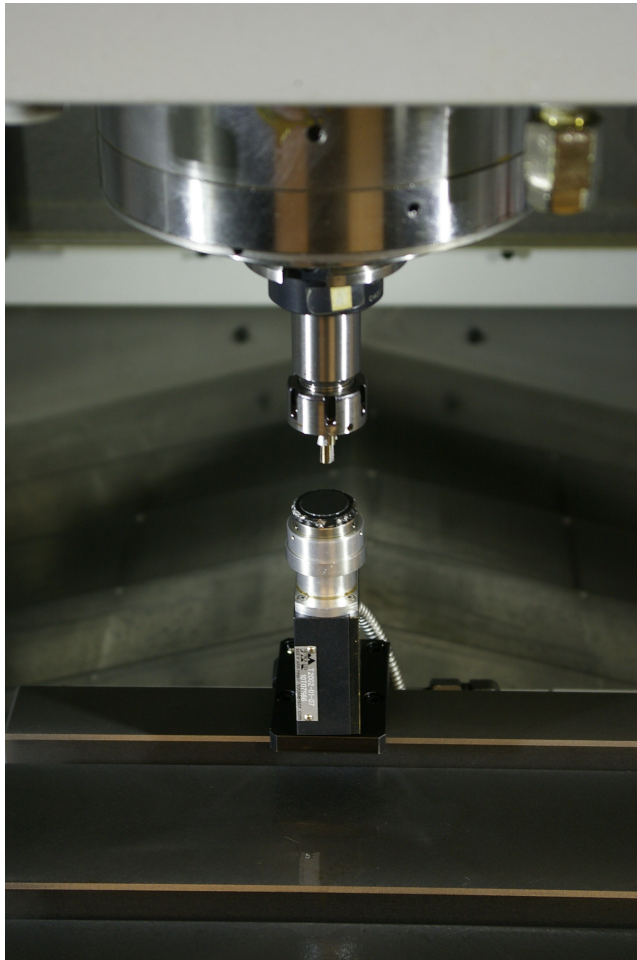
Tool Offsets

- 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

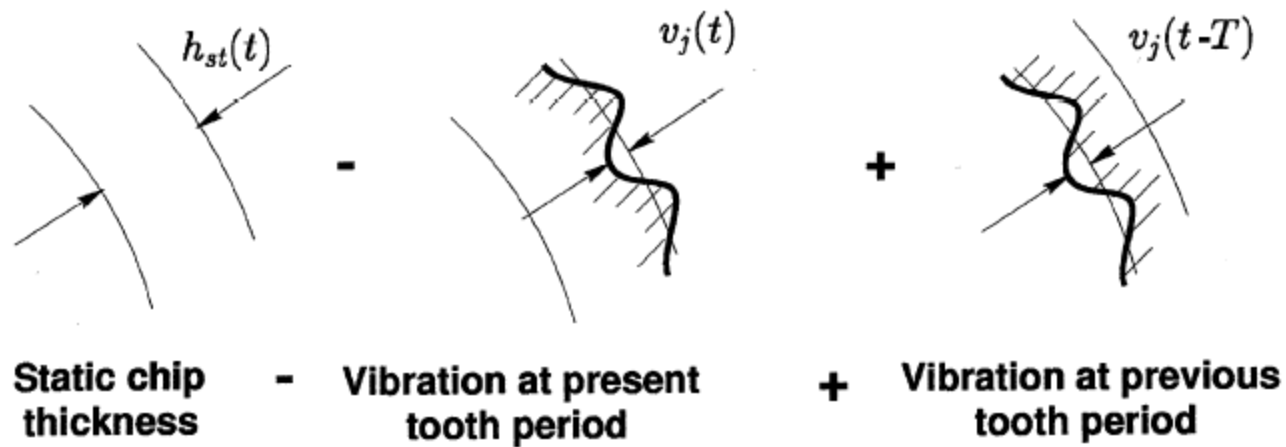
<u>Aluminum (6061, 2024, 7075)</u>				
SFM	Chipload Per Tooth			
<u>2, 3, & 4 Flute</u>	<u>up to .125 dia.</u>	<u>.125-.250 dia.</u>	<u>.250-.500 dia.</u>	<u>.500-1.0 dia.</u>
300-500	.0008-.0020	0015-.0040	0020-.0060	0030-.0090

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

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

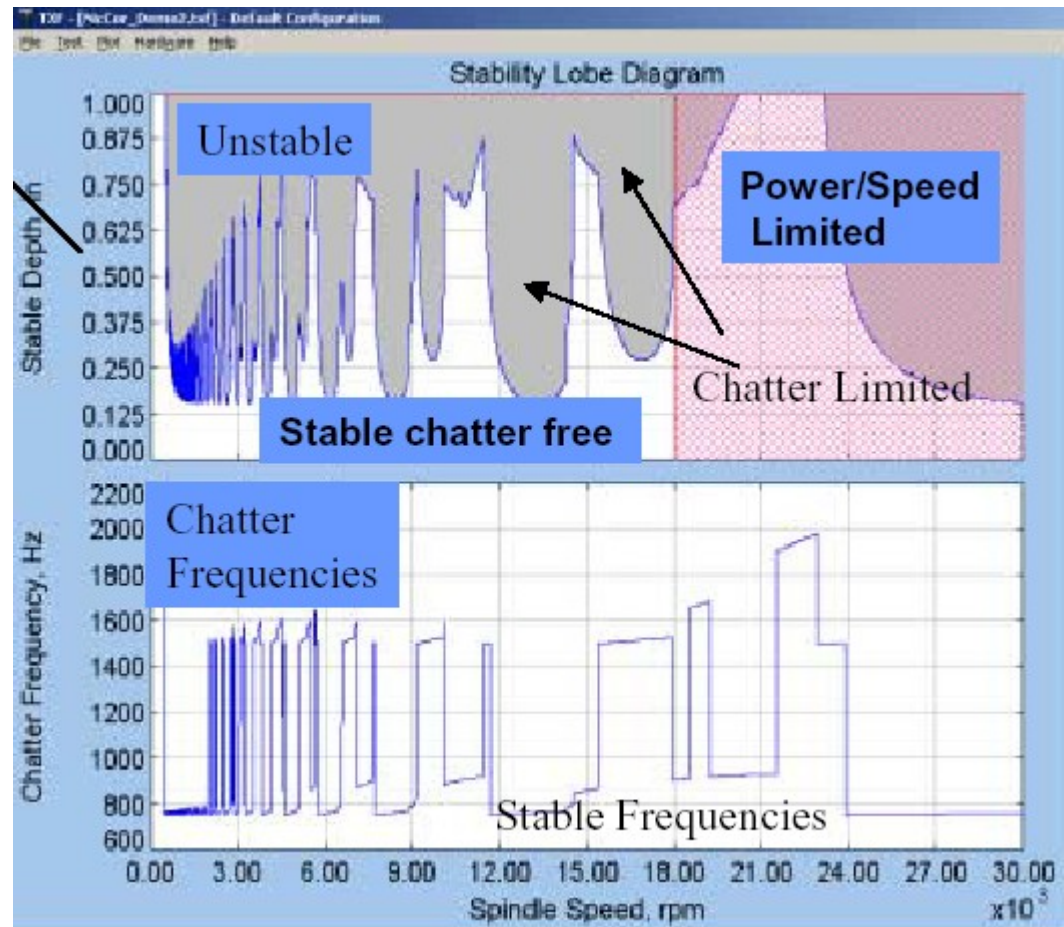
- <http://www.custompartnet.com/calculator/milling-speed-and-feed>

Vibration



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

