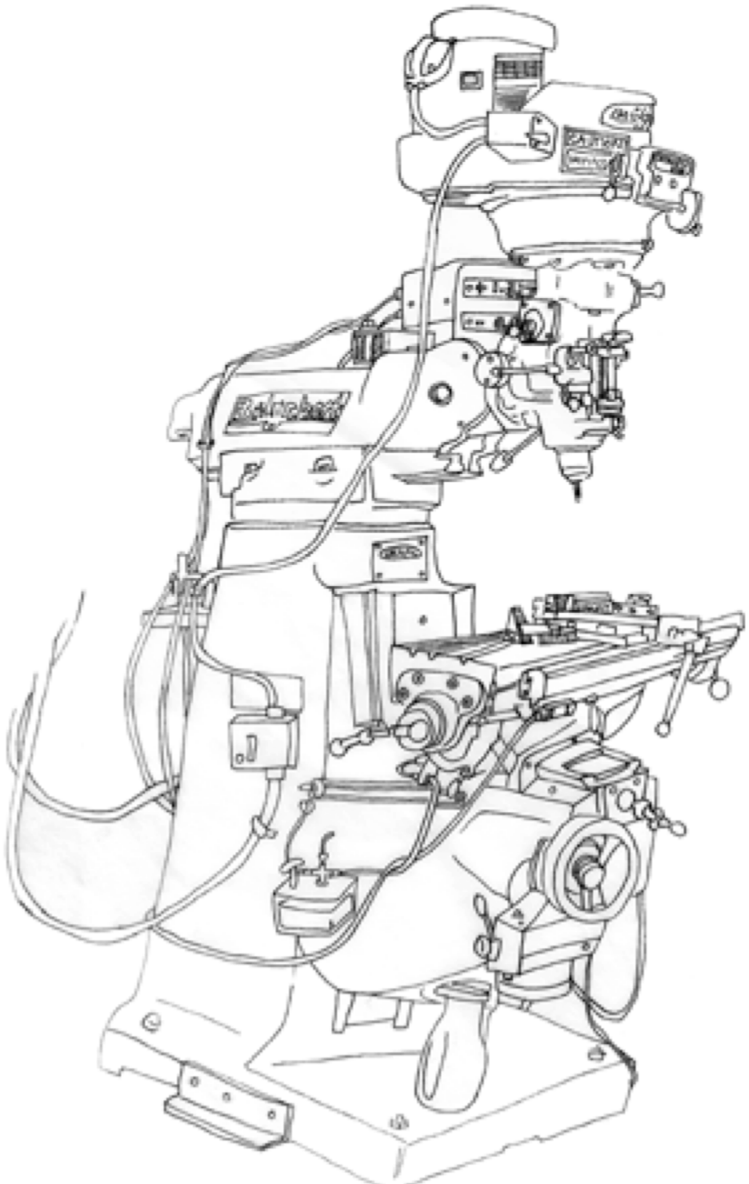
Machining Handout



Learning to use the Mill and Lathe

Machine Shop Terminology

Boring	Boring is the enlarging of a hole by means of a single-point cutting tool. Boring may be done to obtain a continuous inside diameter, or a stepped, tapered, contoured or recessed diameter. Boring may be performed on either a lathe or a milling machine.
Counter Boring	Counter boring is the enlarging of a portion of a hole. Counter boring may be performed on either a lathe or a drilling machine.
Countersinking	Countersinking is the process of making a cone shaped enlargement at the entrance of a hole. Countersinking may be performed on either a lathe or a drilling machine.
Drilling	Drilling is the making of a hole in a work piece where none previously existed. Drilling may be performed on either a lathe or a drilling machine.
Facing	Facing is the process of machining a flat surface across the face of a work piece. The faced surface (usually an end of the work piece) is at right angles to the lathe axis and the part itself. Facing is performed on a lathe.
Reaming	Reaming is the enlarging of a hole to accurate size. Reaming must be proceeded by a drilling or boring operation. Reaming may be performed on either a lathe or a drilling machine.
Tapping	Tapping is the process of cutting internal threads. Tapping may be performed on either a lathe or a drilling machine.
Threading	Threading is the making of a thread on a work piece (such as a screw). Thread cutting can produce either inside or outside threads that are either straight or tapered. Threading is performed on a lathe.
Turning	Turning is the removal of material from the outside diameter of a work piece to form a cylindrical surface. The surface may be straight (one continuous diameter), tapered or contoured (as a concentric but irregularly shaped surface). Turning is performed on a lathe.

Meaurement:

Decimal Equivalents: To convert a fraction to a decimal you divide the numerator by the denominator. So the decimal equivalent of ¼ is .25.

- A 64th of an inch is about .015 (spoken as “15 thousandths”, actual .0156)
- A 32nd of an inch is about .030 (spoken as “30 thousandths”, actual .0312)
- A 16th of an inch is about .060 (spoke as “ 60 thousandths”, actual .063)
- And an 8th of an inch is .125 (spoken as “a hundred twenty-five thousandths”)

Dial Calipers - The bottom scale on both the direct reading and digital (automatic) calipers runs up to 6” and is divided into .1 (1/10)” increments. The dial on the direct reading calipers is divided into .001” units (one thousandth of one inch).

Mill – the dials on the Mill can be used to measure movement in the X, Y or Z axis. The dials are in increments of thousandths of an inch. X and Y axis.001” , Z axis .01”.

DRO - A digital Read-Out continuously displays the position of the milling table in X and Y coordinates. Some DRO’s are also capable of showing the "Z" position of the knee or quill. The display on the read-out typically uses a red or green LED numbers. The resolution of the display is 0.0005" (one half of one-thousandth)

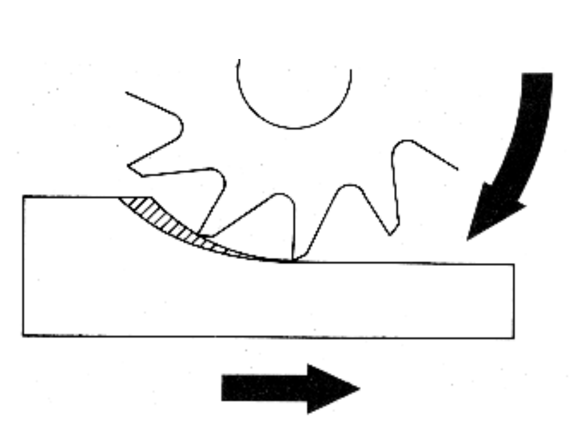
Conventional vs. Climb Milling

Conventional Milling is also called "Up Milling". The direction of the motion of the milling cutter tooth as it engages the work is opposite from the direction of the movement of the work caused by the table feed. Because of this the table and the work piece will never have a tendency to pull towards the work piece because of loss motion between the nut and the table screw.

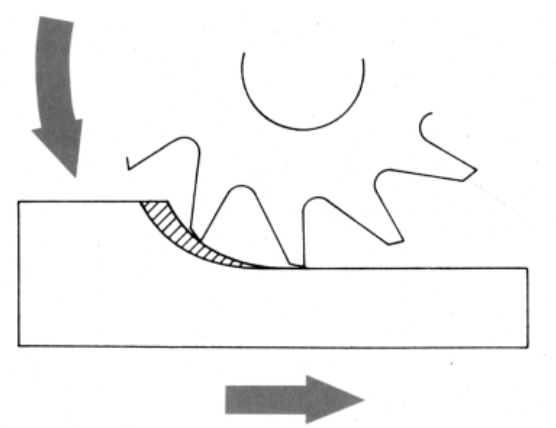
Climb Milling is also called "Down Milling". The Milling Cutter and the work piece move in the same direction. The velocity of the cutters teeth is greater than the velocity of the table feed, which moves the work into the cutter, producing the chip. This pulls the work piece into the cutter by the action of the cutting forces. This can damage the work piece, cutter, and the Machine.

When milling, one should be aware of the difference between conventional, and climb milling. In conventional milling, since the work piece is fed into the rotation of the cutter, this type of cut requires lower forces and is preferred for roughing cuts. In climb milling, since the work moves with the rotation of the cutter it produces a better finish. HOWEVER, it is NOT recommended if the work piece cannot be held securely or cannot support high forces.

Conventional Milling



Climb Milling



Machining Rules of Thumb

Calculating Speeds and Feeds

Cutting speed refers to the speed at which the tool point of the cutter moves with respect to the work measured in feet per minute. Feed is the rate at which the work moves into the cutter measured in feed per tooth revolution. Feeds and speeds affect the time to finish a cut, tool life, finish of the machined surface and power required of the machine. The cutting speed is mostly determined by the material to be cut in addition to the material of the tool. To find the right speed for any task, refer to the Machinery's Handbook or other reference. To calculate the proper spindle speed, divide the desired cutting speed by the circumference of the tool expressed in feet. The feed rate depends on the width and depth of cut, finish desired and many other variables. To calculate the desired feed setting from the feed rate, multiply feed per tooth per revolution by number of teeth and rpm of the spindle.

Setting Spindle Speed

Spindle speed is varied by changing the geometry of the drive train. On many modern machines, it can be adjusted continuously with a hand crank. The spindle must be turning to make the adjustment. A dial indicator reads the speed in rpm. The spindle speed dial indicator shown above has two scales, one for low range, and one for high range. The machine is switched between ranges with a lever. Sometimes, the spindle must be rotated slightly to allow the gears to mate properly.

How fast should the mill spindle spin?

Calculate mill spindle speed using the following formula:

$$Speed = \frac{(CS)*4}{D}$$

1) **Speed** is the calculated spindle speed in revolutions per minute (rpm).

2) **CS** is the cutting speed of the material, a property that you can find in reference books. It is measured in surface feet per minute (sfpm). Our shop uses the following values:

- 500 plastic
- 300 aluminum
- 200 brass
- 100 mild steel
- 50 stainless steel

3) **D** is the diameter of the tool in inches.

Example: If you were using a 3/8"-diameter endmill on mild steel, you would perform the following calculation

$$Speed = \frac{(100 sfpm)*4}{0.375"} = 1067 RPM$$

Note that when using plastic or very small endmills, you may calculate speeds far higher than our machines can reach. Use the highest speed available and reduce your feed rate accordingly.

REMEMBER, this formula is only a guideline for selecting a speed. Always use common sense, pay close attention to your work, and adjust your settings to the situation. If you have a question, always ask a TA.

What is a good mill feed rate?

Calculate mill feed rate using the following formula:

$$Feed = (ChipLoad)*(n)*(Speed)$$

1) **Feed** is the calculated feed rate in inches per minute (ipm).

2) **ChipLoad** is the amount of material that each flute will remove on each revolution. Each material/operation has a suggested value, which you can find in reference books. It is measured in inches, sometimes written inches per tooth (ipt). Our shop uses the following typical values:

- 0.005” for roughing
- 0.001” to 0.002” for finishing

  n is the number of flutes on the tool.

3) **Speed** is the spindle speed in revolutions per minute (rpm).

Example: If you were doing a roughing pass with a four-fluted endmill turning at 1000 rpm, you would perform the following calculation

$$Feed = (0.005") * (4) * (1000rpm) = 20ipm$$

REMEMBER, this formula is only a guideline for selecting a feed rate. Always use common sense, pay close attention to your work, and adjust your settings to the situation. If you have a question, always ask a TA.

Rules of Thumb:

Speed, Feed, and Depth of Cut

1. Machining Speed in rpm = (CS x 4) / dia, where CS = cutting speed in surface feet per minute =

- 500 for plastic
- 300-800 for aluminum
- 200-400 for brass
- 40-70 for high carbon steel
- 30-80 for stainless steel

2. Drilling Speed in rpm = (CS x 4) / dia where CS = cutting speed in sfm =

- 300 for aluminum
- 120 for brass and bronze
- 90 for low carbon steel
- 70 for cast iron
- 50 for alloy steel

3. Counterboring / Countersinking Speed = 1/3 drilling speed

4. Machine Reaming Speed = 1/2 - 1/3 drilling speed

5. Facing Feed on a Lathe for Roughing = 0.005-0.015” / rev, for Finishing = 0.003-0.005” / rev

6. Machining Feed on a Mill in inches per minute = (chip load) x (number of flutes) x (speed in rpm) where chip load is 0.005” for roughing and 0.002” for finishing

7. Machine Reaming hand feed = two times drilling hand feed

8. When Hand Tapping, turn in 1/4 - 1/2 turn, then out 3/4 turn

9. When Cutting Threads with a Die, turn on 1 full turn, then back 1/2 turn

10. When Boring, drill 1/32 - 1/16 undersize, then bore

11. When Flycutting, take off 0.025 -0.100” per pass with a spindle speed of 1500-2000 rpm. Faster will leave a better surface finish, but use 1200 rpm or so with delrin and other plastics because otherwise the material will melt. Also, having a nice radius on the tip of your tool will improve surface finish.

12. Reaming stock allowance according to reamer diameter:

- 1/32 - 1/8” = 0.003-0.006”
- 1/8 - 1/4” = 0.005-0.009”
- 1/4 - 3/8” = 0.007-0.012”
- 3/8 - 1/2” = 0.010-0.015”
- 1/2 - 3/4” = 1/64-1/32”
- 3/4 - 1” = 1/32”

13. Depth of cut on soft steel = up to 1/8 inch

Fasteners

1. Minimum thread engagement = diameter of screw (1.5 diameters is better)

2. #0 machine screw diameter = 0.060”, add 0.013” for each # above

e.g. diameter of a #6 machine screw = 0.060” + (6 x 0.013”) = 0.138”

3. The diameter of a bolt head is 1.5 times the diameter of the screw

4. The diameter of a counterbore is .0313” (1/32) greater than diameter of the bolt head

5. The length of the bolt head is equal to the diameter of the screw

6. The practical strength limit for threads is 75%

Tapping & Clearance Hole Information

Machine Screw Size		# of Threads Per Inch	Minor Dia.	Tap Drills				Clearance Hole Drills			
				Aluminum, Brass & Plastics 75% Thread		Stainless Steel, Steels & Iron 50% Thread		All Materials			
								Close Fit		Free Fit	
No. or Dia.	Major Dia.			Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.
0	.0600	80	.0447	3/64	.0469	55	.0520	52	.0635	50	.0700
1	.0730	64	.0538	53	.0595	1/16	.0625	48	.0760	46	.0810
		72	.0560	53	.0595	52	.0635				
2	.0860	56	.0641	50	.0700	49	.0730	43	.0890	41	.0960
		64	.0668	50	.0700	48	.0760				
3	.0990	48	.0734	47	.0785	44	.0860	37	.1040	35	.1100
		56	.0771	45	.0820	43	.0890				
4	.1120	40	.0813	43	.0890	41	.0960	32	.1160	30	.1285
		48	.0864	42	.0935	40	.0980				
5	.1250	40	.0943	38	.1015	7/64	.1094	30	.1285	29	.1360
		44	.0971	37	.1040	35	.1100				
6	.1380	32	.0997	36	.1065	32	.1160	27	.1440	25	.1495
		40	.1073	33	.1130	31	.1200				
8	.1640	32	.1257	29	.1360	27	.1440	18	.1695	16	.1770
		36	.1299	29	.1360	26	.1470				
10	.1900	24	.1389	25	.1495	20	.1610	9	.1960	7	.2010
		32	.1517	21	.1590	18	.1695				
12	.2160	24	.1649	16	.1770	12	.1890	2	.2210	1	.2280
		28	.1722	14	.1820	10	.1935				
		32	.1777	13	.1850	9	.1960				
1/4	.2500	20	.1887	7	.2010	7/32	.2188	F	.2570	H	.2660
		28	.2062	3	.2130	1	.2280				
		32	.2117	7/32	.2188	1	.2280				
5/16	.3125	18	.2443	F	.2570	J	.2770	P	.3230	Q	.3320
		24	.2614	I	.2720	9/32	.2812				
		32	.2742	9/32	.2812	L	.2900				
3/8	.3750	16	.2983	5/16	.3125	Q	.3320	W	.3860	X	.3970
		24	.3239	Q	.3320	S	.3480				
		32	.3367	11/32	.3438	T	.3580				
7/16	.4375	14	.3499	U	.3680	25/64	.3906	29/64	.4531	15/32	.4687
		20	.3762	25/64	.3906	13/32	.4062				
		28	.3937	Y	.4040	Z	.4130				
1/2	.5000	13	.4056	27/64	.4219	29/64	.4531	33/64	.5156	17/32	.5312
		20	.4387	29/64	.4531	15/32	.4688				
		28	.4562	15/32	.4688	15/32	.4688				
9/16	.5625	12	.4603	31/64	.4844	33/64	.5156	37/64	.5781	19/32	.5938
		18	.4943	33/64	.5156	17/32	.5312				
		24	.5114	33/64	.5156	17/32	.5312				
5/8	.6250	11	.5135	17/32	.5312	9/16	.5625	41/64	.6406	21/32	.6562
		18	.5568	37/64	.5781	19/32	.5938				
		24	.5739	37/64	.5781	19/32	.5938				
11/16	.6875	24	.6364	41/64	.6406	21/32	.6562	45/64	.7031	23/32	.6562

Cutting Speeds:

Milling

Material	AISI/SAE/ASTM Designation	Feet/Minute
Free machining plain carbon steels (resulfurized)	1212, 1213, 1215	130 - 140
Free machining plain carbon steels	1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	115 - 130
Free machining plain carbon steels	1132, 1137, 1139, 1140, 1144, 1146, 1151	35 - 70
Free machining plain carbon steels (Leaded)	11L17, 11L18, 12L13, 12L14	110 - 140
Plain carbon steels	1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	65 - 110
Plain carbon steels	1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	25 - 100
Free machining alloy steels (resulfurized)	4140, 4150	35 - 100
Water hardening tool steel	W1, W2, W5	85
Cold work, oil hardening tool steel	O1, O2, O6, O7	50
Cold work, air hardening tool steel	A2, A3, A4, A6, A7, A8, A9, A10	40 - 50
Stainless steels (Austenitic)	201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	70 - 75
Cast Iron	ASTM Class 20, 25, 30, 35, 40	70 - 100
Brass	C33200, C34000, C34200, C35300, C35600, C36000, C37700, C48500	100 - 200
Bronze	C22600, C65100, C65500, C67500	30 - 80
Wrought aluminum	6061-T6, 5000, 6000, and 7000 series.	165
Cast aluminum	Sand and permanent mold casting alloys	165

Drilling & Reaming

Material	AISI/SAE/ASTM Designation	Drilling Feet/Minute	Reaming Feet/Minute
Free machining plain carbon steels (resulfurized)	1212, 1213, 1215	120 - 125	80
Free machining plain carbon steels	1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	100 - 120	75 - 80
Free machining plain carbon steels	1132, 1137, 1139, 1140, 1144, 1146, 1151	35 - 70	20 - 45
Free machining plain carbon steels (Leaded)	11L17, 11L18, 12L13, 12L14	90 - 130	60 - 85
Plain carbon steels	1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	60 - 100	40 - 65
Plain carbon steels	1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	25 - 90	15 - 60
Free machining alloy steels (resulfurized)	4140, 4150	30 - 90	15 - 60
Water hardening tool steel	W1, W2, W5	85	55
Cold work, oil hardening tool steel	O1, O2, O6, O7	45	30
Cold work, air hardening tool steel	A2, A3, A4, A6, A7, A8, A9, A10	30 - 50	20 - 35
Stainless steels (Austenitic)	201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	50 - 55	30 - 35
Cast Iron	ASTM Class 20, 25, 30, 35, 40	80 - 100	50 - 65
Brass	C35600, C37700, C36000, C33200, C34200, C35300, C48500, C34000	160 - 175	160 - 175
Bronze	C65500, C22600, C65100, C67500	120 - 140	110 - 120
Wrought aluminum	6061-T6, 5000, 6000, and 7000 series.	350 - 400	350 - 400
Cast aluminum	Sand and permanent mold casting alloys	350 - 400	350 - 400

Turning: Turning speeds are adjusted to the feed rate of the mini lathe (0.004/rev.), a depth of cut of 0.040, and a tool life of 180 minutes.

Material	**AISI/SAE/ASTM Designation**	**HSS Feet/Minute**	**Carbide Feet/Minute**
Free machining plain carbon steels (resulfurized)	1212, 1213, 1215	270 - 290	820 - 1045
Free machining plain carbon steels	1108, 1109, 1115, 1117, 1118, 1120, 1126, 1211	215 - 235	950
Free machining plain carbon steels	1132, 1137, 1139, 1140, 1144, 1146, 1151	70 - 215	670 - 800
Free machining plain carbon steels (Leaded)	11L17, 11L18, 12L13, 12L14	200 - 260	800 - 820
Plain carbon steels	1006, 1008, 1009, 1010, 1012, 1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1513, 1514	125 - 215	800 - 885
Plain carbon steels	1027, 1030, 1033, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043, 1045, 1046, 1048, 1049, 1050, 1052, 1524, 1526, 1527, 1541	55 - 180	670 - 970
Free machining alloy steels (resulfurized):	4140, 4150	70 - 200	430 - 685
Water hardening tool steel	W1, W2, W5	180	590
Cold work, oil hardening tool steel	O1, O2, O6, O7	125	590
Cold work, air hardening tool steel	A2, A3, A4, A6, A7, A8, A9, A10	80 - 125	355 - 365
Stainless steels (Austenitic)	201, 202, 301, 302, 304, 304L, 305, 308, 321, 347, 348	115 - 135	570
Cast Iron	ASTM Class 20, 25, 30, 35, 40	145 - 215	410
Brass	C35600, C37700, C36000, C33200, C34200, C35300, C48500, C34000	300 - 350	1170
Bronze	C65500, C22600, C65100, C67500	200 - 250	715
Wrought aluminum	6061-T6, 5000, 6000, and 7000 series.	500 - 600	2820
Cast aluminum	Sand and permanent mold casting alloys	600 - 750	2820