LIST NUMBER INDEX

Drill Feeds and Speeds

Material	Hardness	Drill Type	Point Style	Point Angle	Speed (SFM)	Feed	Coolant
Aluminum / Aluminum Alloys	–	High Helix	Standard	118°	200-300	Medium to Heavy	Water Soluble
Aluminum Bronze	–	High Helix	Standard	118°	50-100	Medium to Heavy	Water Soluble
Brass, Free Machining	–	Low Helix	Standard	118°	100-250	Medium to Heavy	Water Soluble
Bronze, Soft Medium	Below 200	General Purpose	Standard	118°	70-150	Medium to Heavy	Water Soluble
Bronze, High Tensile	–	Low Helix or Half RD	Standard	118°	50-100	Medium to Heavy	Water Soluble
Cooper / Cooper Alloys	–	General Purpose	Standard	118°	100-200	Medium	Water Soluble
High Temp. Alloys, Cobalt Base	180-300	Cobalt	Split Point	135°	5-20	Medium	Activated Oils
High Temp. Alloys, Iron Base	180-300	Cobalt	Split Point	135°	5-20	Medium	Activated Oils
High Temp. Alloys, Neckel Base	180-300	Nitro® or Cobalt	Split Point	135°	5-15	Medium	Activated Oils
Iron, Soft Case	Up to 150	Heavy Duty or Nitro®	Split Point	135°	75-150	Medium	Dry / Air
Iron, Medium Case	150-250	Heavy Duty or Nitro®	Split Point	135°	50-100	Medium	Dry / Air
Iron, Hard Case	Over 250	Carbide	Standard	118°	40-75	Light to Medium	Dry / Air
Plastic & Related Materials	–	Low Helix	Low Angle	90°	100-200	Medium to Heavy	Dry / Air
Steel, Alloyed Low Carbon	125-275	General Purpose	Standard	118°	50-70	Medium	Water Soluble
Steel, Alloyed Medium Carbon	275-325	Heavy Duty or Nitro®	Split Point	135°	40-55	Medium	Water Soluble
Steel, Alloyed High Carbon	Over 325	Cobalt	Split Point	135°	30-50	Medium	Water Soluble
Steel Low & Medium carbon	Below 175	General Purpose	Standard	118°	45-95	Medium to Heavy	Water Soluble
Steel High Carbon	175-225	General Purpose	Standard	118°	45-65	Medium	Water Soluble
Stainless Steel, Austenitic	Below 300	Heavy Duty or Nitro®	Split Point	135°	40-60	Medium to Heavy	Water Soluble
Stainless Steel Precipiation	Over 300	Cobalt	Split Point	135°	20-40	Medium	Cutting Oil

Drill Dia.	Light	Medium	Heavy
Feed Per Drill Revolution			
1/16" to 1/8"	.0005 - .0010	.0010 - .0020	.0020 - .0040
1/8" to 1/4"	.0010 - .0030	.0030 - .0050	.0040 - .0060
1/4" to 3/8"	.0030 - .0050	.0050 - .0070	.0060 - .0100
3/8" to 1/2"	.0040 - .0060	.0050 - .0080	.0080 - .0120
1/2" to 3/4"	.0050 - .0070	.0070 - .0100	.0090 - .0140
3/4" to 1"	.0070 - .0100	.0090 - .0140	.0140 - .0200

Feeds – Feed rates for drilling are governed by the drill diameter, machinability of materials and depth of hole.

Speeds – The speed of a drill is determined by the rate which the outer periphery of the tool rotates in relation to material being cut. In general, the SFM at which a drill will operate is within a range based upon the workpiece material, its condition, hardness and depth of hole. It is usually advisable to start the drilling process at a slower SFM and then increase to the maximum.

RPM = $\frac{SFM}{TD}$ x 3.82 = Revolutions Per Minute

SFM = RPM x TD x .26 = Surface Feet Per Minute

TD – Tool diameter in inches

710 Drill Feeds and Speeds

Workpiece Material Groups	Hard-ness	Speed (SFM)	1/32"	1/16"	1/8"	1/4"	3/8"	1/2"	5/8"	3/4"
Free Machining & Low Carbon Steels 1006, 1008, 1015, 1018, 1020, 1022, 1025, 1117, 1140,1141, 11L08, 11L14, 1213, 12L13, 12L14, 1215, 1330	up to 28 Rc	330	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Medium Carbon & High Carbon Steels, Alloy Steels & Easy to Machine Tool Steels 1030, 1035, 1040, 1045, 1050, 1052, 1055, 1060, 1085, 1095, 1541, 1551, 9255, 2515, 3135, 3415, 4130, 4137, 4140, 4150, 4320, 4340, 4520, 5015, 5115, 5120, 5132, 5140, 5155, 6150, 8620, 9262, 9840, 52100, 01, 02, 06, S2,W1 to W310	28 to 38 Rc	265	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Tool Steels & Die Steels 07, M1, M2, M3, M4, M7, T1, T2, T4, T5, T8, T15, A2, A3, A6, A7, H10, H11, H12, H13, H19, H21, L3, L6, L7, P2, P20, S1, S5, S7, 52100, A 128, D2, D3, D4, D5, D7	28 to 44 Rc	230	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Hardened Steel	45 to 65 Rc	50	0.0003	0.0010	0.0010	0.0010	0.0020	0.0020	0.0020	0.0030
Stainless Steel - Moderately Difficult 301, 302, 303 High Tensile, 304, 304L, 305, 420, 15-5PH, 17-4PH, 17-7PH	up to 28 Rc	150	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Stainless Steel - Difficult to Machine 302B, 304B, 309, 310, 316, 316B, 316L, 316Ti, 317, 317L,321, PH13-8Mo, Nitronics	up to 28 Rc	100	0.0003	0.0005	0.0020	0.0040	0.0050	0.0060	0.0080	0.0100
High Temp Alloys Nimonics, Inconel, Monel, Hastelloy	up to 42 Rc	70	0.0003	0.0005	0.0020	0.0040	0.0050	0.0060	0.0080	0.0100
Titanium Alloys 6Al-4V, 5Al-2.5 Sn, 6Al-2 Sn-4Zr-6Mo, 3Al-8V-6Cr4Mo-4Zr,10V-2Fe-3Al, 13V-11Cr-3Al	up to 42 Rc	180	0.0003	0.0005	0.0020	0.0040	0.0050	0.0060	0.0080	0.0100
Cast-Iron - Gray CG, ASTM A48, CLASS 20, 25, 30, 35, SAE J431C, GRADES G1800, G3000, G3500, GG 10, 15, 20, 25, 30, 35, 40	up to 240 HB	365	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Cast Iron - Ductile & Malleable CGI 60-40-18, 65-45-12, D4018, D4512, D5506, 32510, 35108,M3210, M4504, M5503, 250, 300, 350, 400, 450	over 240 HB	365	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Plastics		300	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120
Kevlar		300	0.0010	0.0020	0.0030	0.0060	0.0080	0.0100	0.0110	0.0120

Annular Cutters

Feeds and Speeds for Annular Cutters

Material To Be Drilled		HSS Cutter	Nitro® Cutter	CT Cutter
Cutting Speed In Feet/Min. (SFPM)				
Steel	Freecutting-Leaded	100	150	175
	Up To 180 Brinell	80 to 110	130	165
	A-36	80 to 110	130	165
	180 To 285 Brinell	45 to 70	80	90
	285 To 375 Brinell	30 to 45	55	60
	375 & Up Brinell	15 to 25	35	40
	Manganese (low)	15 to 20	25	30
	Stainless-Freecutting	30 to 60	70	100
	Stainless 304,316,	–	–	–
	320 & Tougher	20 to 40	50	60
Inconel, Hastelloy		25 to 40	55	70
Inconel - X		20 to 25	35	45
Titanium		30 to 40	50	60
A-286		25 to 35	50	60

Material To Be Drilled		HSS Cutter	Nitro® Cutter	CT Cutter
Speed In Feet/Min. (SFPM)				
Aluminum		150	250	450-600
Brass	Ordinary	150	250	250-550
	Leaded	200	300	300-600
Bronze	Ordinary	100	200	200-500
	Hi-tensile	70	100	100-200
Cast Iron	Soft	100	150	175
	Medium	80	90	100
	Hard	50	70	80
	Chilled	25	35	40
Copper		100	200	250
Magnesium		Up to 300	400	400-800
Malleable Iron		70	80	100-130
Monel		40	50	50

Cutter Dia.	Feed Ins./Rev.
Operating Feed*	
7/16" to 9/16"	.002 to .006
5/8' to 3/4"	.003 to .003
1-3/16"- 1-1/16"	.004 to .008
1-1/8" - 1-7/16"	.006 to .010
1-1/2" to 2"	.008 to .012
2" to 2-1/2"	.012 to .025

*Always commence operations at the lower end of RPM.

CUTTING SPEEDS (SFPM)	7/16"	1/2"	9/16"	5/8"	11/16"	3/4"	13/16"	7/8"	15/16"	1"	1-1/8"	1-1/4"	1-3/8"	1-1/2"	1-5/8"	1-3/4"	2"	2-1/4"	2-1/2"
10	87	76	68	61	56	51	47	44	41	38	34	31	28	25	24	22	19	17	15
20	175	153	136	122	111	102	94	87	82	76	68	61	56	51	47	44	38	34	31
30	262	229	204	183	167	153	141	131	122	115	102	92	83	76	71	65	57	51	46
40	349	306	272	245	222	204	188	175	163	153	136	122	111	102	94	87	76	68	61
50	437	382	339	306	278	255	235	218	204	191	170	153	140	127	118	109	95	85	76
60	524	458	407	367	333	306	282	353	245	229	204	183	167	153	141	131	115	102	92
70	611	535	475	428	389	357	329	306	285	267	238	214	194	178	165	153	134	119	107
80	700	611	543	489	444	408	376	350	326	306	272	244	222	204	188	175	153	136	122
90	786	688	611	550	500	458	426	393	367	344	306	275	250	229	212	196	172	153	138
100	873	764	679	611	556	509	470	436	408	382	340	306	278	255	235	218	191	170	153
120	1048	917	815	733	667	611	564	524	489	458	407	367	333	306	282	262	229	204	183
140	1222	1070	950	856	778	713	658	611	571	535	475	428	390	356	329	306	267	238	214
150	1310	1146	1018	917	833	764	705	655	611	573	509	458	417	382	353	327	286	255	229
160	1397	1222	1086	978	889	815	752	698	652	611	543	490	444	407	376	350	306	272	244
180	1572	1375	1222	1100	1000	917	846	786	734	688	611	550	500	458	423	393	344	306	275
200	1747	1528	1358	1222	1111	1020	940	874	815	764	680	611	556	510	470	437	382	340	306
220	1920	1681	1494	1345	1222	1121	1034	960	897	840	747	672	611	560	517	480	420	374	336
240	2096	1833	1630	1467	1333	1222	1128	1048	978	917	815	733	667	611	564	524	458	407	367
250	2183	1910	1697	1528	1389	1274	1175	1091	1020	955	850	764	694	637	588	546	477	424	382
260	2270	1986	1765	1590	1444	1325	1221	1135	1060	993	883	795	722	662	611	568	497	441	397
280	2445	2140	1900	1712	1556	1426	1316	1222	1140	1070	950	856	778	713	658	611	535	475	428
300	2620	2292	2037	1834	1667	1528	1410	1310	1222	1146	1020	917	833	7	705	655	573	510	458

SPEED AND FEED FORMULAS

SPEED OF CUTTER IN SURFACE FEET PER MINUTE (SFM)	REVOLUTIONS PER MINUTE (RPM)	FEED IN INCHES PER MINUTE (IPM) OR (F)
SFM = D x .26 x RPM	RPM = SFM x 3.82 divided by Dia.	F = f x T x RPM

End Mill Speeds & Feeds

Workpiece Material	Slotting			Peripheral			Roughing		
	Tool Geometry	Speed (SFM)	Feed Range	Tool Geometry	Speed (SFM)	Feed Range	Tool Geometry	Speed (SFM)	Feed Range
Aluminum, Wood	A	200 - 600	1	A	200 - 600	1	D	200 - 600	1
Brass, Bronze, Alloyed Aluminum	A	100 - 200	2	A	100 - 200	2	D	150 - 400	2
Mild Steel, Cast Iron	B	80 - 100	3	B	80 - 100	3	E	90 - 110	3
Mild Steel Forgings (100 - 195 BNH)	B	60 - 80	3	B	60 - 80	3	E	60 - 80	3
Stainless Steel, Alloy Steel (195 - 260)	B	40 - 60	4	B	40 - 60	4	E	40 - 60	4
High Strength Stainless, Alloy Steels (260 - 310 BNH)	C	20 - 40	4	C	20 - 40	4	E	25 - 40	4
Nickel Base Alloys, High Strength Titanium Alloys	C	15 - 20	5	C	15 - 20	5	E	15 - 20	5
Alloy Steels (320 - 345 BHN)	C	10 - 15	6	C	10 - 15	6	E	10 - 20	6
Alloy Steels (345 - 375 BHN)	C	5 - 10	6	C	5 - 10	6	E	10 - 20	6
	2 or 3 Flute Axial = 1/2 Diameter Radial = 1/2 Diameter			4 + Flutes Axial = 1-1/2 Diameter Radial = .1 Diameter			4 + Flutes Axial = 1 Diameter Radial = 1/2 Diameter Slot = 1/2 Diameter Axial		

A = Aluminum
B = GP - HSS
C = GP - Cobalt
D = 3-flute High Helix Cobalt Rougher
E = Cobalt Rougher - Square Cut/Knuckle

Feed Range (inches/tooth)

Dia. Size	1/16" - 3/32"	1/8" - 3/16"	1/4" - 1/2"	9/16" - 11/16"	3/4" - 1"	1-1/8" - 1-3/4"	2" - 3"
1	.0002 - .0005	.0002 - .001	.0005 - .002	.0005 - .003	.001 - .004	.002 - .004	.003 - .006
2	.0002 - .0005	.0002 - .001	.0005 - .003	.0005 - .004	.001 - .006	.002 - .004	.003 - .006
3	.0002 - .0005	.0002 - .001	.0005 - .003	.001 - .004	.002 - .006	.003 - .004	.003 - .006
4	.0002 - .0005	.0002 - .0005	.0005 - .003	.001 - .004	.002 - .006	.002 - .004	.003 - .006
5	–	.0002 - .0005	.0003 - .002	.0005 - .002	.001 - .004	.002- .004	.003 - .005
6	–	–	.0002 - .002	.0005 - .002	.001 - .003	.002 - .004	.002 - .004

Drill Surface Treatments

Chemical Compositions of High Speed Steels Drills Taps Reamers End Mills						
	Chemical Composition					
Type	Carbon %	Tungsten %	Molybdenum	Chromium %	Vanadium %	Term
M.50	.83	0.10	4.25	4.00	1.00	HSS
M1	.80	1.50	8.00	4.00	1.00	HSS
M2	.85	6.00	5.00	4.00	2.00	HSS
M7	1.00	1.75	8.75	4.00	2.00	HSS
M35	.80	6.00	5.00	4.00	2.00	5% Cobalt
M42	1.0	1.50	9.50	3.75	1.05	8% Cobalt

Surface Treatments

Surface treatments for high speed steel tools function to condition them, so in certain applications they will outperform tools that have not been treated. Surface treatments do not, however, alter the functional structure of the tool itself.

Black Oxide - This treatment is applied to finished tools and produces a thin black iron oxide surface coating. It also provides additional tempering and stress relieving. This coating reduces galling and chip welding and also increases the ability of the tool to retain lubricants. Recommended in iron and steel drilling applications. It should not be used in non-ferrous metals such as aluminum because it increases the loading tendencies of the tool.

Nitride - This treatment produces a hard case which is highly resistant to abrasion. It also retards the tendency of softer materials to cling or load on tools. Recommenced for tools that are used for ferrous, non-ferrous, and non-metallic materials which are abrasive and have loading characteristics.

Nitrade and **Oxide** - Combines the advantages of the lubricity of oxide with the abrasion resistance of nitriding. Recommended for abrasive ferrous applications. Not recommended for soft materials such as aluminum, magnesium or similar non-ferrous applications.

Chrome Plating - This treatment deposits an extremely thin layer of chromium on the surface of tools. It reduces the coefficient of friction, and resists chip weld and abrasion. Recommended for non-ferrous and non-metallic materials.

Titanium Nitrade - This surface treatment improves tool life by acting as a wear resistant and thermal barrier. It also gives the tool a low coefficient of friction and very high surface hardness. It reduces friction and chip welding and acts as a thermal insulator between the chip and the tool. Recommended for use in ferrous materials below Rc 40 and in non-ferrous materials.

Drill Points

Conventional Point - Conventional points with 118° included point angles are the most commonly used because they provide satisfactory results in a wide variety of materials. A possible limitation is that the straight chisel edge contributes to walking at the drill point, often making it necessary to spot the hole for improved accuracy.

Split Point - Split points were originally developed for use on drills designed for deep oil holes in automotive crankshafts. Since its inception, the split point has gained widespread use and is applied to both 118° and 135° included point angles. Its main advantages are the ability to reduce thrust and eliminate walking at the drill point. This is a distinct advantage when the drill is used in a portable drill or in drilling applications where busings cannot be used. The split point also has tow positive rake cutting edges extending the center of the drill, which can assist as a chipbreaker to produce small chips which can readily be ejected.

Notched Point - Notched points were developed for drilling tough alloys. Commonly incorporated on heavy web drills, which allow the point to withstand the higher thrust loads required in drilling these materials. As with the split point, the notched point contains two additional positive rake cutting edges extending toward the center of the drill. These secondary cutting lips, which extend no further than half the original cutting lip, can assist in chip control and reduce the torque required in drilling tough materials. Notched points can be incorporated on both 118° and 135° included point angles, making them suitable for drilling a broad variety of materials.

Helical Point - Helical points change the flat blunt chisel to a "S" contour with a radiused crown effect which has its highest point at the center of the drill axis. This crown contour creates a continuous cutting edge from margin to margin across the web. The advantage is its self-centering ability which allows the chisel to cut and enables the drill to cut closer to actual drill diameter. Helical Points are not available under 1/16" diameter.

Reduced Rake Point - Reduced rake points are generated by flattering or dubbing both cutting lips from the outer periphery to the chisel. This reduces the effective axial rake to 0–5, positive which translates to a plowing rather than shearing action. This reduction in shearing is an effective method of preventing the drill from grabbing in low tensile strength materials such as brass. Reducing the rake also strengthens the cutting lip and can assist in breaking chips.

Decimal Equivalent Chart

Drill Size	MM	Decimal Inches	Drill Size	MM	Decimal Inches	Drill Size	MM	Decimal Inches	Drill Size	MM	Decimal Inches
–	0.10	.0039	45	2.08	.0820	5	5.22	.2055	7/16	11.11	.4375
–	0.20	.0079	44	2.18	.0860	4	5.31	.2090	29/64	11.15	.4531
–	0.25	.0098	43	2.26	.0890	3	5.41	.2130	15/32	11.91	.4688
–	0.30	.0118	42	2.37	.0935	7/32	5.56	.2188	–	12.00	.4724
80	0.34	.0135	3/32	2.38	.0938	2	5.61	.2210	31/64	12.30	.4844
79	0.37	.0145	41	2.44	.0960	1	5.79	.2280	1/2	12.70	.5000
1/64	0.40	.0156	40	2.50	.0980	A	5.94	.2340	–	13.00	.5118
78	0.41	.0160	39	2.53	.0995	15/64	5.95	.2344	33/64	13.10	.5156
77	0.46	.0180	38	2.58	.1015	–	6.00	.2360	17/32	13.49	.5312
–	0.50	.0197	37	2.64	.1040	B	6.05	.2380	35/64	13.89	.5469
76	0.51	.0200	36	2.71	.1065	C	6.15	.2420	–	14.00	.5512
75	0.53	.0210	7/64	2.78	.1094	D	6.25	.2460	9/16	14.29	.5625
74	0.57	.0225	35	2.79	.1100	1/4	6.35	.2500	37/64	14.68	.5781
–	0.60	.0236	34	2.82	.1110	E	6.35	.2500	–	15.00	.5906
73	0.61	.0240	33	2.87	.1130	F	6.53	.2570	19/32	15.08	.5938
72	0.64	.0250	32	2.95	.1160	G	6.63	.2610	39/64	15.48	.6094
71	0.66	.0260	–	3.00	.1181	17/64	6.75	.2656	5/8	15.88	.6250
–	0.70	.0276	31	3.05	.1200	H	6.76	.2660	–	16.00	.6299
70	0.71	.0280	1/8	3.18	.1250	I	6.91	.2720	41/64	16.27	.6406
69	0.74	.0282	30	3.26	.1285	–	7.00	.2756	21/32	16.67	.6562
–	0.75	.0295	29	3.45	.1360	J	7.04	.2770	–	17.00	.6693
68	0.79	.0310	28	3.57	.1405	K	7.14	.2810	43/64	17.07	.6719
1/32	0.79	.0313	9/64	3.57	.1406	9/32	7.14	.2812	11/16	17.46	.6875
–	0.80	.0315	27	3.66	.1440	L	7.37	.2900	45/64	17.86	.7031
67	0.81	.0320	26	3.73	.1470	M	7.49	.2950	–	18.00	.7087
66	0.84	.0330	25	3.80	.1495	19/64	7.54	.2969	23/32	18.26	.7188
65	0.89	.0350	24	3.86	.1520	N	7.67	.3020	47/64	18.65	.7344
–	0.90	.0354	23	3.91	.1540	5/16	7.94	.3125	–	19.00	.7480
64	0.91	.0360	5/32	3.97	.1562	–	8.00	.3150	3/4	19.05	.7500
63	0.94	.0370	22	3.99	.1570	O	8.03	.3160	49/64	19.45	.7656
62	0.97	.0380	–	4.00	.1575	P	8.20	.3230	25/32	19.84	.7812
61	0.99	.0390	21	4.04	.1590	21/64	8.33	.3281	–	20.00	.7874
–	1.00	.0394	20	4.09	.1610	Q	8.43	.3320	51/64	20.24	.7969
60	1.02	.0400	19	4.22	.1660	R	8.61	.3390	13/16	20.64	.8125
59	1.04	.0410	18	4.31	.1695	11/32	8.73	.3438	–	21.00	.8268
58	1.07	.0420	11/64	4.37	.1719	S	8.84	.3480	53/64	21.03	.8281
57	1.09	.0430	17	4.39	.1730	–	9.00	.3543	27/32	21.43	.8438
56	1.18	.0465	16	4.50	.1770	T	9.09	.3580	55/64	21.84	.8594
3/64	1.19	.0469	15	4.57	.1800	23/64	9.13	.3594	–	22.00	.8661
55	1.32	.0520	14	4.62	.1820	U	9.35	.3680	7/8	22.23	.8750
54	1.40	.0550	13	4.70	.1850	3/8	9.53	.3750	57/64	22.62	.8906
53	1.51	.0595	3/16	4.76	.1875	V	9.56	.3770	–	23.00	.9055
1/16	1.59	.0625	12	4.80	.1890	W	9.80	.3860	29/32	23.02	.9062
52	1.61	.0635	11	4.85	.1910	25/64	9.92	.3906	59/64	23.42	.9219
51	1.70	.0670	10	4.91	.1935	-	10.00	.3937	15/16	23.81	.9375
50	1.78	.0700	9	4.98	.1960	X	10.08	.3970	–	24.00	.9449
49	1.85	.0730	–	5.00	.1968	Y	10.26	.4040	61/64	24.21	.9531
48	1.93	.0760	8	5.05	.1990	13/32	10.32	.4062	31/32	24.61	.9688
5/64	1.98	.0781	7	5.11	.2010	Z	10.49	.4130	–	25.00	.9843
47	1.99	.0785	13/64	5.16	.2031	27/64	10.72	.4219	63/64	25.00	.9844
–	2.00	.0787	6	5.18	.2040	–	11.00	.4331	1"	25.40	1.0000
46	2.06	.0810	–	–	–	–	–	–	–	–	–

Common Drilling Terms

Axis - The imaginary straight line which forms the lengthwise center of a drill

Body - The section of a drill from the shank to the outer edges of the cutting lips

Body Clearance Diameter - The portion of the land that has been cut away so it will not bind against the walls of the hole

Chip Removal - The ability of a drill to pull material that has been cut away from the point, up the flutes and out of the hole

Chisel Edge - The edge at the end of the web that connects the cutting lips

Chisel Edge Angle - The angle between the chisel edge and cutting lip, as viewed from the end of a drill

Cobalt Steel - A heat-resistant steel that produces increased drill life

Drill Diameter - The diameter over the margins of a drill, measured at the point (Dia.)

Feeds - Feed rates for drilling are determined by the drill diameter, machinability of materials and depth of the desired hole. Small drills, harder materials and deeper holes require additional considerations in selecting proper feed rates

Flute - Groove cut in the body of drills to provide cutting surfaces, permit removal of chips and allow cutting fluid to reach cutting surfaces

Flute Length - The distance from the outer edges of the cutting lips to the extreme back of the flutes

Helix Angle - The angle formed between the leading edge of the land and the axis of a drill

High Speed Steel - The high quality steel used in drills for most maintenance and industrial applications

IPM - Feed rate in Inches Per Minute = IPR X RPM

IPR - Inches Per Revolution (Feed)

Land - The outer portion of the body between two adjacent flutes

Land Width - The distance between the leading edge and heel of the land, measured at a right angle to the leading edge

Lip Relief Angle - The relief angle at the outer corner of the lip

Lip - The cutting edge of a two-flute drill which extends from the chisel edge to the outer edge

Margin - The narrow raised surface on a drill body along the flute that stabilizes the drill in the hole

Neck - The section of reduced diameter between the body and the shank of a drill

Overall Length - The length from the end of the shank to the outer corners of the cutting lip

Point - The cone-shaped cutting end of a drill, made from the ends of the lands and the web

Point Angle - The angle of the cutting surfaces on a drill point, commonly 118° or 135°

RPM - Revolutions Per Minute = $\frac{\text{SFM}}{\text{Dia.}}$ x 3.83

SFM - Surface Feet Per Minute = RPM X Dia. X .26

Shank - The part of a drill by which it is held and driven

Size - Measurement reference for diameter size of a drill, expressed as either fractional, wire, letter or metric

Speed - The speed of a drill is determined by the rate which the outer edge of the tool rotates in relation to the material being cut. In general, the SFM is within a range based upon the workpiece material, its condition, hardness and depth of the hole. The deeper the hole, the greater tendency for more heat to be generated. Speed reduction is often recommended to minimize the amount of heat. It is usually best to start drilling at a slower speed and then increase.

Split Point - A special point configuration that eliminates “walking” so holes stay on center

Web - The central portion of the body that joins the lands. The extreme end of the web forms the chisel edge of a two-flute drill

Web Thickness - The thickness of the web at the point, unless otherwise specified

Drill Troubleshooting Guide

Use this chart to help you find the best drill for your job

Simply choose your material and then the corresponding tool

Works Best in This Material	*Nitro®* Black and Gold	TiN	Heavy-Duty Black Oxide	Heavy-Duty Cobalt Bronze Oxide	General Purpose Bright	General Purpose Black Oxide
Aluminum/Aluminum Alloys; Bronze, Soft and Medium		•			•	
Iron, Cast	•	•	•	•		•
Steel, Low and Medium Carbon	•	•	•	•		•
Steel, High Alloy	•	•	•	•		
Steel, Stainless and PH	•	•		•		•
Tough, Medium and High-Tensile Strength Alloys	•	•	•	•		
Wood and Plastic		•		•	•	
						
Point:	135° Split	135° Split	135° Split	135° Split	118°	118°

Drill Troubleshooting Guide

Trouble	Possible Causes	Corrections
Drill Breaks	Spring or back lash in press or work	Test press and work for rigidity and alignment
	Too little lip relief	Regrind properly
	Speed to low in proportion to the feed	Increase or decrease speed
	Dull drill	Sharpen drill
	Possible chip congestion	Peck drill
Outer Corner Breakdown	Uneven hardness in work material	Reduce speed
	Too much Speed	Reduce Speed
	Improper cutting compound	Use proper cutting compound
Drill Breaks in Brass or Wood	Chips clog up flutes	Increase speed. Use drills designed for these materials
Cutting Lips Chipped	Too much feed	Reduce feed
	Too much lip relief	Regrind properly
Cracking or Checking in Cutting Edges	Heated and cooled too quickly	Warm slowly before using. Do not throw cold water on hot drill while grinding or drilling
	Too much feed	Reduce feed
Hole too Large	Unequal angle or length of the cutting edges or both	Regrind properly
	Loose spindle	Test spindle for rigidity
Only One Lip Cutting	Unequal length or angle of cutting lips or both	Regrind drill properly
Drill Splits up Center	Too little lip relief	Regrind with proper relief
	Too much feed	Reduce feed
Rough Hole	Dull or improperly ground drill	Regrind properly
	Lack of lubricant or wrong lubricant	Lubricate or change lubricant
	Improper set-up	Check set-up
	Too much feed	Reduce feed

Common Tapping Terms

Angle of Thread - The angle included between the flanks of the thread measured in an axial plane

Back Taper - A slight axial relief on the thread of the tap which makes the pitch diameter of the thread near the shank somewhat smaller than that of the
chamfered end

Bottoming Chamfer - The shortest chamfer (1-2 threads long) for threading close to the bottom of blind holes

Chamfer - The tapering of the threads at the front end of each land of a tap, by cutting away and relieving the crest of the first few teeth, to distribute the cutting action over several teeth

Chamfer Relief - The gradual decrease in land height from cutting edge to heel on the chamfered portion, to provide clearance for the cutting action as the tap advances

Chordal Hook Angle - The angle between the chord passing through the root and crest of a thread form at the cutting face, and a radial line through the crest at the cutting edge

Crest - The top surface joining the two flanks of a thread. The crest of an external thread is at its major diameter, while the crest of an internal thread is at its minor diameter

Cutting Face - The leading side of the land in the direction of rotation for cutting on which the chip impinges

Flutes - The longitudinal channels formed in a tap to create cutting edges on the thread profile, and to provide chip spaces and cutting fluid passages

Height of Thread - The distance between the crest and the base of a thread measured normal to the axis

Lead - The distance a screw thread advances axially in one complete turn. On a single lead screw or tap, the lead and pitch are identical. On a double-lead screw or tap, the lead is twice the pitch, etc.

Pitch - The distance from any point on a screw or tap thread to a corresponding point on the next thread, measured parallel to the axis. The pitch equals one divided by the number of threads per inch

Pitch Diameter - On a straight thread, the diameter of any imaginary coaxial cylinder, the surface of which would pass through the thread profile at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder
- On a taper thread, the diameter at a given distance from a reference plane perpendicular to the axis of an imaginary co-axial cone, the surface of which would pass through the thread profile at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cone

Plug Chamfer - The most common chamfer (3-5 threads long), for hand or machine tapping in through or blind holes

Rake - Any deviation of a straight cutting face of the tooth from a radial line. *Positive Rake* means that the crest of the cutting face is angularly advanced ahead of the balance of the face of the tooth. *Negative Rake* means that the same point is angularly behind the balance of the cutting face of the tooth.
Zero Rake means that the cutting face is directly on the center line

Spiral Point (Chip Driver) - A supplementary angular fluting cut in the cutting face of the land at the chamfer end. It is slightly longer than the chamfer on the tap, and of the opposite hand to that of rotation

Taper Chamfer - A long chamfer (8-10 threads long) for easier starting and engagement in the work piece. More working teeth means that less tapping torque is required, and the chip load is reduced on each tooth

Threads Per Inch - The number of threads in one inch of length

Tap Troubleshooting Guide

Trouble	Possible Causes	Corrections
Oversize Pitch Diameter	Incorrect tap	Use correct “H” limit Use longer chamfered tap
	Chip packing	Use spiral pointed tap Use larger drill size Shorten length of thread in blind holes
	Galling	Reduce tapping speed Use lubricants
Torn or Rough Threads	Dull tap	Resharpen or replace
	Chamfer too short	Increase chamfer length
	Galling	Use lubricant Reduce speed Use larger drill
Breakage	Incorrect tap selection	Increase chamfer length Use spiral pointed taps in through holes Tapping too deep
	Excessive tapping torque	Increase hole size Shorten thread length Increase chamfer length
	Operating conditions	Reduce tapping speed Avoid bottoming out the tap
Chipping	Incorrect tap selection	Increase chamfer length Avoid chip packing in flutes
	Operating conditions	Reduce tapping speed Use larger drill Ensure adequate lubricant
Excessive Wear	Incorrect tap selection	Increase chamfer length
	Operating conditions	Reduce tapping speed Ensure adequate lubricant Use larger drill

Tap Drill Sizes – Cut Taps

Tap Size	Drill Size	Probable % Thread	Tap Size	Drill Size	Probable % Thread	Tap Size	Drill Size	Probable % Thread
0-80	3/64"	71-81	**10-32**	#21	68-76	**5/8-18**	37/64"	58-65
M1.6-.35	1.25 MM	69-77	**M5 x .8**	4.2 MM	69-77	**M16 x 2**	35/64"	76-81
1-64	#53	59-67	**12-24**	#16	66-72	**3/4-10**	21/32"	68-72
M2 x .4	1/16"	72-79	**12-28**	#15	70-78	**3/4-16**	11/16"	71-77
1-72	#53	67-75	**M6 x 1**	#10	76-84	**M20 x 2.5**	11/16"	74-78
2-56	#50	62-69	**1/4-20**	#7	70-75	**7/8-9**	49/64"	72-76
2-64	#50	70-79	**1/4-28**	#3	72-80	**7/8-14**	13/16"	62-67
M2 x .45	2.05 MM	69-77	**5/16-18**	F	72-77	**M24 x 3**	53/64"	72-76
3-48	5/64"	70-77	**5/16-24**	I	67-75	**1" -8**	7/8"	73-77
3-56	#46	69-78	**M8 x 1.25**	6.7 MM	74-80	**1"- 12**	59/64"	67-72
4-40	#43	65-71	**3/8-16**	5/16"	72-77	**1"- 14**	15/16"	61-67
4-48	#42	61-68	**3/8-24**	Q	71-79	**1-1/8"- 7**	63/64"	72-76
M3 x .5	#40	70-79	**M10 x 1.5**	8.4 MM	76-82	**1-1/8-12**	1-3/64"	66-72
5-40	#38	65-72	**7/16-14**	U	70-75	**M30 x 3.5**	1-3/64"	75
5-44	#37	63-71	**7/16-20**	25/64"	65-72	**1-1/4 - 7**	1-7/64"	76
M3.5 x .6	#33	72-81	**M12 x 1.7**	13/32"	69-74	**1-1/4-12**	1-11/64"	72
6-32	#36	71-78	**1/2-13**	27/64"	73-78	**1-3/8 - 6**	1-7/32"	72
6-40	#33	69-77	**1/2-20**	29/64"	65-72	**1-3/8-12**	1-19/64"	72
M4 x .7	3.25 MM	74-82	**M14 x 2**	15/32"	76-81	**M36 x 4**	1-1/4"	82
8-32	#29	62-69	**9/16-12**	31/64"	68-72	**1-1/2 - 6**	1-11/32"	72
8-36	#29	70-78	**9/16-18**	33/64"	58-65	**1-1/2-12**	1-27/64"	72
10-24	#25	69-75	**5/8-11**	17/32"	75-79	**–**	–	–

Tap Size	Taperpipe NPF & NPTF	Decimal Equivalent	Straight Pipe NPS	Decimal Equivalent
1/8-27	Q	.3320	11/32	.3438
1/4-18	7/16	.4375	7/16	.7375
3/8-18	9/16	.5625	37/64	.5781
1/2-14	45/64	.7031	23/32	.7188
3/4-14	29/32	.9062	59/64	.9219
1"- 11-1/2	1-9/64	1.1406	1-5/32	1.1562
1-1/4"- 11-1/2	1-31/64	1.4844	1-1/2	1.5000
1-1/2"- 11-1/2	1-23/32	1.7188	1-3/4	1.7500
2"- 11-1/2	2-3/16	2.1875	2-7/32	2.2188
2-1/2"- 8	2-39/64	2.6094	2-21/32	2.6562
3"- 8	3-15/64	3.2344	3.2344	33/64

Tap Drill Sizes – Thread Formers

Note: Recommended thread percentage for various (Inch standard) tap sizes is shown in the "60% Thread " columns below. This is also the average percentage that is desirable for metric sizes. Use the "Probable Percent of Thread" column in the "Metric Sizes" tables below.

Machine Screw Sizes

Tap Size	75% Thread		70% Thread		65% Thread		60%Thread		55% Thread		50% Thread	
	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size
0-80	0.0536	1.35 mm	0.0540	1.35 mm	0.0545	–	0.0549	54	0.0554	54	0.0558	1.0 mm
1-64	0.0650	1.65 mm	0.0655	1.65 mm	0.0661	–	0.0666	–	0.0672	51	0.0677	51
1-72	0.0659	1.65 mm	0.0663	–	0.0669	1.7 mm	0.0673	51	0.0679	51	0.0683	–
2-56	0.0769	1.65 mm	0.0774	1.95 mm	0.0669	23498	0.0787	47	0.0794	2.0 mm	0.0799	–
2-64	0.0780	5/64	0.0785	47	0.0791	2.0 mm	0.0796	2.0 mm	0.0802	–	0.0807	2.05 mm
3-48	0.0884	2.25 mm	0.0890	43	0.0898	43	0.0905	2.3 mm	0.0913	2.3 mm	0.0919	–
3-56	0.0899	43	0.0904	–	0.0911	2.3 mm	0.0917	2.8 mm	0.0924	2.35 mm	0.0929	2.35 mm
4-40	0.0993	2.5 mm	0.1000	39	0.1010	39	0.1018	38	0.1028	2.6 mm	0.1035	2.6 mm
4-48	0.1014	38	0.1020	38	0.1028	2.6 mm	0.1035	2.6 mm	0.1043	37	0.1049	37
5-40	0.1123	34	0.1130	33	0.1140	33	0.1148	2.9 mm	0.1158	32	0.1165	32
5-44	0.1134	33	0.1130	2.9 mm	0.1150	2.9 mm	0.1157	–	0.1166	32	0.1173	32
6-32	0.1221	3.1mm	0.1230	3.1 mm	0.1243	–	0.1252	40916	0.1264	3.2 mm	0.1274	–
6-40	0.1253	1/8	0.1260	3.2 mm	0.1270	2.9 mm	0.1278	3.25 mm	0.1288	30	0.1295	30
8-32	0.1481	3.75 mm	0.1490	–	0.1503	25	0.1512	3.8 mm	0.1524	24	0.1534	3.9 mm
8-36	0.1481	25	0.1507	3.8 mm	0.1518	25	0.1526	24	0.1537	3.9 mm	0.1546	23
10-24	0.1481	–	0.1507	18	0.1717	23682	0.1729	23682	0.1746	–	0.1758	–
10-32	0.1481	17	0.1750	–	0.1763	–	0.1772	16	0.1784	4.5 mm	0.1794	–
12-24	0.1481	10	0.1960	9	0.1977	5.0 mm	0.1989	8	0.2006	5.1 mm	0.2018	7
12-28	0.1481	5.0 mm	0.1989	8	0.2003	8	0.1989	7	0.2028	–	0.2039	13/16

Fractional Sizes

Tap Size	75% Thread		70% Thread		65% Thread		60% Thread		55% Thread		50% Thread	
	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size	Theor. Hole Size	Nearest Drill Size
1/4-20	.2245	5.7 mm	.2260	–	.2280	1	.2295	1	.2315	–	.2330	5.9 mm
1/4-28	.2318	–	.2329	5.9 mm	.2343	A	.2354	15/64	.2368	6.0 mm	.2379	B
5/16-18	0.2842	7.2 mm	.2861	7.25 mm	.2879	7.3 mm	.2898	L	.2917	7.4 mm	.2936	–
5/16-24	0.2912	7.4 mm	.2927	–	.2941	M	.2955	7.5 mm	.2969	19/64	.2983	7.6 mm
3/8-16	.3431	11/32	.3452	8.75 mm	.3474	S	.3495	8.9 mm	.3516	–	.3537	9.0 mm
3/8-24	.3537	9.0 mm	.3552	9.0 mm	.3566	–	.3580	T	.3594	23/64	.3608	–
7/16-14	.4011	–	.4035	Y	.4059	13/32	.4084	–	.4108	–	.4132	Z
7/16-20	0.4120	Z	.4137	10.5 mm	.4154	–	.4171	–	.4188	–	.4205	–
1/2-13	.4608	–	.4634	–	.4660	–	.4686	15/32	.4712	12 mm	.4738	12 mm
1/2-20	.4745	–	.4762	–	.4779	–	.4796	–	.4712	–	.4830	31/64

Metric Sizes

Metric Tap Size	Tap Drill Size	Decimal Equiv. of Tap Drill (inches)	Theoretical Percentage of thread %	Probable Mean Oversize (inches)	Probable Hole Size (inches)	Probable Percent of Thread %
M3 x 0.5	36	0.1065	86	.0026	.1091	67
–	2.7 mm	0.1062	88	.0026	.1088	70
M4 x 0.7	27	0.1440	72	.0032	.1472	54
–	3.6 mm	.1417	84	.0032	.1449	67
–	9/64	.1406	90	.0032	.1438	73
M5 x 0.8	14	0.1820	69	.0035	.1855	53
–	4.6 mm	.1811	74	.0035	.1846	57
–	15	.1800	79	.0035	.1835	62
–	16	0.1770	92	.0035	.1805	76
M6 x 1	7/32	.2188	65	.0038	.2226	51
–	5.4 mm	.2126	88	.0038	.2164	74

Metric Tap Size	Tap Drill Size	Decimal Equiv. of Tap Drill (inches)	Theoretical Percentage of thread %	Probable Mean Oversize (inches)	Probable Hole Size (inches)	Probable Percent of Thread %
M8 x 1.25	7.4 mm	0.2910	71	.0042	.2952	59
–	L	0.2900	75	.0042	.2942	62
–	7.3 mm	.2874	82	.0042	.2916	70
M10 x 1.5	U	0.3680	64	.0046	.3726	53
–	9.3 mm	0.3660	69	.0046	.3706	58
–	9.2 mm	0.3620	78	.0046	.3666	67
–	23/64	.3594	85	.0046	.3640	74
M12 x 1.5	11.3 mm	.4449	70	.0047	.4496	57
–	7/16	.4375	86	.0047	.4422	75
M12 x 1.75	7/16	.4375	75	.0047	.4422	65
–	11 mm	.4331	84	.0047	.4378	73

Material & Speed Recommendations

Nitro®-Max can be used in a wide range of materials. CNC type equipment is recommended to achieve maximum performance of the taps.

Conditions that effect SFM:

- Rigidity of fixture
- Coolant flow
- Spindle runout
- Feed mechanism

Workpiece Material	2150N (Sp.Pt)	2160N (Sp.Fl)	2250N (Sp.Pt)	2260N (Sp.Fl)	2270N (T.F)
	Speed (SFM)	Speed (SFM)	Speed (SFM)	Speed (SFM)	Speed (SFM)
A. Stainless Steel (Free Machining)	25-50	25-50	50-80	40-70	60-100
B. Carbon Steels					
Low Carbon	50-80	40-70	50-80	30-60	80-120
Medium Carbon	30-60	20-50	30-60	15-40	60-90
High Carbon	10-30	10-20	10-30	10-20	20-50
C. Non-Ferrous Materials					
Zinc Die Cast	75-100	60-90	80-120	80-120	150-200
Copper	70-120	60-110	80-120	80-120	100-150
Brass	75-100	60-90	80-120	80-120	100-150
Plastic	40-80	40-80	60-100	60-100	80-120
D. Aluminum Alloys					
Wrought	75-100	60-90	80-120	80-120	150-200
Cast	60-100	60-100	80-120	80-120	120-180
Unalloyed	75-100	75-100	80-120	80-120	150-200

Carbide Bur Selection & Use

Series	Head Style	Description	Use	Picture
7000A	SA	Cylindrical	Deburring	
7000B	SB	Cylindrical with end cut	Interior Contour Work	
7000C	SC	Cylindrical Ball Nose	Surface Milling & Contour Work	
7000D	SD	Ball Shape	Surface Milling, Deburring, & Contour Work	
7000E	SE	Oval Shape	Contour Work	
7000F	SF	Round Tree Shape	Narrow Contour Work	
7000G	SG	Pointed Tree Shape	Surface Milling & Narrow Contour Work	
7000H	SH	Flame Shape	Contour Work	
7000J	SJ	60° Included Angle	Machining of Acute Angle Areas & Countersinking	
7000K	SK	90° Included Angle	Machining of Acute Angle Areas & Countersinking	
7000L	SL	14° Included Angle	Surface Machining & Narrow Contour Work	
7000M	SM	Cone Shape	Surface Machining & Narrow Contour Work	
7000N	SN	Inverted Cone	Machining from the Rear in hard to reach areas	
7000NE	SNE	Inverted Cone with end cut	Machining from the Rear in hard to reach areas	

Technical

Carbide Bur Selection & Use

MAGNUM SOLID CARBIDE

Single Cut

Drillco's Single Cut produces a smooth finish for general purpose use on steel, cast iron and other ferrous and non-ferrous materials. Most of Drillco's Carbide Burs are available in the Single Cut right hand spiral design.

Double Cut

Drillco's Double Cut, with it's chisel type cutting edge, is a machine ground tool built to exacting tolerances of concentricity, size and shape. This accuracy, when combined with precision grinders, results in smooth running, fast metal removal and fine finishes.

The right and left hand helical flutes combined to produce an chisel type cutting tooth. This results in faster penetration and stock removal with minimal bounce or chatter. The Double Cut design also produces an easy to handle granular type chip, in most metals, as opposed to the conventional sliver type chips. Throughout it's life, the Double Cut gives faster stock removal, less operator fatigue, and maintains a good finish on the widest possible variety of materials.

Chip Breaker

The addition of a chip breaker will reduce sliver size and improve operator control at a slightly reduced surface finish. Available as a special.

Diamond Cut

The Diamond Cut is very effective in heat treated and tough alloy steels, producing extremely small chips. It offers very good operator control with good stock removal at the sacrifice of finish. The diamond grind will reduce tool life, however. Available as a special.

Aluminum Cut

Drillco's Aluminum Cut burs are outstanding on soft or nonferrous type materials. Use the Aluminum Cut design on aluminum, magnesium, brass, lead and most plastics.

RPM Chart

Bur Diameter	RPM	Bur Diameter	RPM	Bur Diameter	RPM
1/16	60,000 - 90,000	**3/8**	20,000 - 40,000	**1**	7,500 - 20,000
1/8	40,000 - 70,000	**7/16**	15,000 - 40,000	**1-1/8**	7,000 - 13,000
3/16	35,000 - 60,000	**1/2**	15,000 - 40,000	**1-1/2**	5,000 - 10,000
1/4	30,000 - 50,000	**5/8**	12,000 - 25,000	**1-3/4**	4,500 - 9,000
5/16	20,000 - 40,000	**3/4**	10,000 - 20.000	**2**	4,000 - 8,000

Speed

High speeds are essential for efficient and economical use of a carbide bur. At high speeds there will be less tendency for chips to build up in the flute. Also, the bur will cut more freely in corners or pockets, and reduce the chances of jamming or wedging.

The selection of operating speeds for a carbide bur is usually less exacting than for most other cutting tools. Rather than specify a different grinder speed for every bur diameter used, one grinder will normally be used for many different bur sizes. However, for most efficient operations, the grinder should be specified with consideration given to diameter of the burs that will be used.

Carbide burs should operate between 1,500 and 3,000 surface feet per minute. Using these guidelines, a grinder can be selected that will efficiently work with a fairly broad range of bur sizes. For example, a 30,000 RPM grinder can be used with a 3/16" to a 3/8" diameter bur. A 22,000 RPM grinder will be satisfactory for burs ranging from 1/4" to 1/2".

Speeds are an important factor in achieving desired stock removal rates and workpiece finish. Increasing the speed will improve finishes, while lower speeds will remove material more rapidly.

EDP# Index

EDP# Index

Notes:	PAGE #

Notes

Notes:	PAGE #

Terms and Conditions

Freight Policy
Freight prepaid via economical route of our choice on individual orders valued at $300 net ($500 to Canada) billing or more, within the continental US (except Alaska). It is the responsibility of the consignee to check shipment for damage and notify the shipper within 5 days.

Payment Terms
Net 30 days. Special terms printed on customer's orders will not be recognized where they differ from our terms. Discounts taken on remittance not meeting Drillco terms will be considered unearned and will remain on statement as a past due balance until subsequent payment is received. Accounts that are not maintained in accord with accounts receivable policy are subject to being placed on C.O.D.

Minimum Billing
Minimum billing is $25.00 net.
$5.00 under minimum fee.

Pricing Policy
All orders accepted are subject to prices in effect on the day the order is received. Our representatives have no authority to quote special prices. Prices and discounts are subject to change without notice. Any item not listed in our Price List, once ordered, is non-cancellable and non-returnable.

Sales Policy
All Drillco products are sold exclusively through authorized distributors. Possession of our catalogs and or price list does not infer intent to sell.

Returned Merchandise Policy
Prior authorization covering specific items and quantities must be obtained before any goods are returned to the factory for credit. Authorized returns are subject to a 15% restocking charge. No returns for specials will be accepted unless merchandise is found to be defective in quality or workmanship.

Broken Package Charges
Orders will be shipped in standard packaging units. Orders for less than standard packs will be increased to standard packs. Please refer to standard packaging on each page of catalog. Tools are packaged in standard packs unless your purchase order specifically stated "broken package required"; an additional 15% charge will apply to these orders. Tools are packaged in envelopes and tubes unless otherwise indicated.

Claims
All claims must be made within 5 days after receipt of goods. Any exterior damage is responsibility of carrier.

Warranty
Drillco, Inc (THE COMPANY) warrants to distributors of its products that each new product manufactured by the Company shall be free from material defects and workmanship for one year after delivery by the Company. The Company's exclusive obligation under this warranty is limited to furnishing a replacement for, repairing, or issuing credit for, any product which is determined by the Company to be materially defective in material or workmanship.

The warranty does not apply to tools deemed to have been misused, altered without authorization, or operated with improper conditions. The company may request information regarding their operation during the inspection process. All defective products returned to the Company under this warranty shall be the Company's property.

No sales representative of the Company, Distributor of the Company's products, or other entity may make claims on product warranty that supersede the terms found in this written warranty.

The Company's sole liability of any claim of any kind, whether in contract, tort, or otherwise, for any loss or damage arising out of, connected with, or resulting from the manufacture, sale, delivery or use of the products sold hereunder shall in no case exceed the amount actually paid to the Company for the applicable products. Our liability is limited to replacement of the tools sold by us.

California PROP 65
Drillco Cobalt and Solid Carbide tools both contain Cobalt, which is listed as a cancer causing ingredient on California's Proposition 65 list. This warning does not apply to High Speed Steel (HSS) tools, as they do not contain cobalt or any other ingredients found on California's Proposition 65 list.

All Drillco Solid Carbide and Cobalt tools are labelled with the following:

WARNING: This product can expose you to chemicals including Cobalt, which is [are] known to the State of California to cause cancer. For more information go to www.P65Warnings.ca.gov