

Speeds & Feeds

ProDrill

Speeds & Feeds available at deboertool.com/app/speeds/drills, or by scanning the QR code



in Essential

	Cutting Speed (SFM)		Feed per tooth (IPT). For slotting reduce by 20%									
	Min	Max	1/8 0.125	3/16 0.1875	1/4 0.250	5/16 0.3125	3/8 0.375	7/16 0.4375	1/2 0.500	5/8 0.625	3/4 0.750	1 1.000
Aluminum Alloys	600	800	0.0011	0.0017	0.0022	0.0027	0.0032	0.0038	0.0042	0.0060	0.0085	0.0092
Soft Steels (>35Rc)	500	600	0.0006	0.0008	0.0010	0.0012	0.0016	0.0024	0.0030	0.0034	0.0036	0.004
Alloy Steels < 35Rc (4140-4340)	350	400	0.0006	0.0010	0.0016	0.0020	0.0025	0.0031	0.0036	0.0037	0.0039	0.0045
Alloy Steels > 35Rc (4140-4340)	250	350	0.0004	0.0009	0.0014	0.0019	0.0020	0.0028	0.0031	0.0032	0.0036	0.0042
Tool Steels < 36Rc (A2, D2, S7)	100	200	0.0004	0.0007	0.0014	0.0020	0.0020	0.0025	0.0033	0.0031	0.0035	0.0042
Tool Steels > 36Rc (A2, D2, S7)	250	350	0.0003	0.0009	0.0017	0.0020	0.0025	0.0029	0.0031	0.0037	0.004	0.005
Die Steels (P20, H13)	200	300	0.0005	0.0010	0.0017	0.0020	0.0023	0.0032	0.0033	0.0037	0.0037	0.004
Easy to cut Stainless Steels (303)	250	350	0.0006	0.0006	0.0008	0.0008	0.0014	0.0021	0.0026	0.0031	0.0038	0.0042
Mod. Difficult to cut Stainless Steels	225	275	0.0005	0.0005	0.0005	0.0007	0.0012	0.0020	0.0024	0.0028	0.0034	0.0041
Difficult to cut Stainless Steels (316L)	200	250	0.0003	0.0004	0.0006	0.0007	0.0013	0.0019	0.0023	0.0028	0.0031	0.0038
Gray Cast Iron	450	550	0.0006	0.0008	0.0010	0.0012	0.0016	0.0025	0.0030	0.0035	0.0045	0.0055
High Temperature Alloys	50	100	0.0003	0.0004	0.0005	0.0007	0.0008	0.0014	0.0016	0.0020	0.0023	0.0028

in X-Mill

	Cutting Speed (SFM)		Feed per tooth (IPT)									
	Min	Max	1/8 0.125	3/16 0.1875	1/4 0.250	5/16 0.3125	3/8 0.375	1/2 0.500	5/8 0.625	3/4 0.750	1 1.000	
Soft Steels (>35Rc)	550	550	0.0006	0.0008	0.0010	0.0012	0.0016	0.0024	0.0024	0.0034	0.0029	
Alloy Steels < 35Rc (4140-4340)	375	375	0.0006	0.0010	0.0016	0.0020	0.0025	0.0031	0.0036	0.0037	0.0039	
Alloy Steels > 35Rc (4140-4340)	300	300	0.0004	0.0009	0.0014	0.0019	0.0020	0.0028	0.0031	0.0032	0.0036	
Tool Steels < 36Rc (A2, D2, S7)	300	300	0.0004	0.0007	0.0014	0.0020	0.0020	0.0025	0.0033	0.0031	0.0035	
Tool Steels > 36Rc (A2, D2, S7)	150	150	0.0003	0.0009	0.0017	0.0020	0.0025	0.0029	0.0031	0.0037	0.0040	
Die Steels (P20, H13)	250	250	0.0005	0.0010	0.0017	0.0020	0.0023	0.0032	0.0033	0.0037	0.0037	
Easy to cut stainless steels (303)	300	300	0.0006	0.0006	0.0008	0.0008	0.0014	0.0021	0.0026	0.0031	0.0038	
Mod. Difficult to cut stainless Steels	255	255	0.0005	0.0005	0.0005	0.0007	0.0012	0.0020	0.0024	0.0028	0.0034	
Difficult to cut Stainless Steels (316L)	220	220	0.0003	0.0004	0.0006	0.0007	0.0013	0.0019	0.0023	0.0028	0.0031	
Gray Cast Iron	500	500	0.0006	0.0008	0.0010	0.0012	0.0016	0.0025	0.0029	0.0034	0.0045	
High Temperature alloys	80	80	0.0003	0.0004	0.0005	0.0007	0.0008	0.0014	0.0016	0.0020	0.0023	
Titanium	140	140	0.0004	0.0005	0.0007	0.0008	0.0013	0.0019	0.0024	0.0026	0.0037	

NOTE:

Side milling applications - for longest reach (L3 LON) tools, reduce max step over by 30%

Slot milling applications - for longest reach (L3 LON) tools, reduce max slotting depth by 30%

Lower value of cutting speed (SFM) is used for high-stock removal applications or for higher hardness (machinability) within group

Higher value of cutting speed (SFM) is used for finishing applications or for lower hardness (machinability) within group

Above parameters are based on ideal conditions. For smaller taper machining centers, please adjust parameters accordingly

RPM	$(3.82 \times \text{SFM}) / \text{Dia.}$
IPR	IPM / RPM
SFM	$(\text{RPM} \times \text{Dia.}) / 3.82$
IPM	$\text{IPT (Chip Load)} \times \text{No. of Teeth} \times \text{RPM}$
IPT (Chip Load)	$(\text{Chip Load}) = \text{IPM} / (\text{No. of Teeth} \times \text{RPM})$

Speeds & Feeds

** Factor in chip thinning for any cut under 40% of the tool's diameter for optimal results*

in XM											
1xD Side Milling		SFM	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	1
			0.125	0.1875	0.250	0.3125	0.375	0.500	0.625	0.750	1.000
P0	Low Carbon Steels, Long Chipping	575	0.00067	0.00139	0.00147	0.00181	0.00210	0.00253	0.00290	0.00320	0.00355
P1	Low Carbon Steels, Short Chipping	575	0.00067	0.00139	0.00147	0.00181	0.00210	0.00253	0.00290	0.00320	0.00355
P2	Medium & High Carbon Steels	550	0.00067	0.00139	0.00147	0.00181	0.00210	0.00253	0.00290	0.00320	0.00355
P3	Alloy Steels & Tool Steels	475	0.00058	0.00118	0.00125	0.00154	0.00178	0.00217	0.00254	0.00285	0.00328
P4	Alloy Steels & Tool Steels	400	0.00057	0.00110	0.00116	0.00142	0.00160	0.00195	0.00227	0.00251	0.00286
P5	Ferritic, Martensitic & PH Stainless Steels	275	0.00052	0.00099	0.00105	0.00128	0.00146	0.00179	0.00209	0.00233	0.00266
P6	High Strength Ferritic, Martensitic & PH Stainless Steel	215	0.00047	0.00088	0.00093	0.00112	0.00125	0.00152	0.00174	0.00199	0.00213
K1	Cast Iron	450	0.00067	0.00139	0.00147	0.00181	0.00210	0.00253	0.00290	0.00320	0.00355

** It is strongly recommended to run flood coolant when slotting*

in XM											
1xD Slotting		SFM	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	1
			0.125	0.1875	0.250	0.3125	0.375	0.500	0.625	0.750	1.000
P0	Low Carbon Steels, Long Chipping	575	0.00057	0.00115	0.00121	0.00149	0.00170	0.00206	0.00237	0.00260	0.00287
P1	Low Carbon Steels, Short Chipping	575	0.00057	0.00115	0.00121	0.00149	0.00170	0.00206	0.00237	0.00260	0.00287
P2	Medium & High Carbon Steels	550	0.00057	0.00115	0.00121	0.00149	0.00170	0.00206	0.00237	0.00260	0.00287
P3	Alloy Steels & Tool Steels	475	0.00051	0.00099	0.00104	0.00128	0.00147	0.00178	0.00206	0.00232	0.00267
P4	Alloy Steels & Tool Steels	400	0.00049	0.00092	0.00096	0.00117	0.00133	0.00160	0.00185	0.00200	0.00233
P5	Ferritic, Martensitic & PH Stainless Steels	275	0.00045	0.00084	0.00088	0.00107	0.00122	0.00147	0.00171	0.00190	0.00217
P6	High Strength Ferritic, Martensitic & PH Stainless Steel	215	0.00040	0.00074	0.00078	0.00094	0.00105	0.00127	0.00143	0.00156	0.00174
K1	Cast Iron	450	0.00057	0.00115	0.00121	0.00149	0.00170	0.00206	0.00237	0.00260	0.00287

** All below speed & feeds are calculated with 10% radial step-over*

in XM											
Dynamic Milling		SFM	1/8	3/16	1/4	5/16	3/8	1/2	5/8	3/4	1
			0.125	0.1875	0.250	0.3125	0.375	0.500	0.625	0.750	1.000
P0	Low Carbon Steels, Long Chipping	700 - 1200	0.0012	0.0023	0.0025	0.0030	0.0035	0.0042	0.0048	0.0053	0.0060
P1	Low Carbon Steels, Short Chipping	700 - 1200	0.0012	0.0023	0.0025	0.0030	0.0035	0.0042	0.0048	0.0053	0.0060
P2	Medium & High Carbon Steels	650 - 1100	0.0012	0.0023	0.0025	0.0030	0.0035	0.0042	0.0048	0.0053	0.0060
P3	Alloy Steels & Tool Steels	550 - 925	0.0010	0.0020	0.0022	0.0025	0.0030	0.0037	0.0042	0.0048	0.0055
P4	Alloy Steels & Tool Steels	475 - 800	0.0010	0.0018	0.0020	0.0023	0.0027	0.0033	0.0038	0.0042	0.0048
P5	Ferritic, Martensitic & PH Stainless Steels	325 - 550	0.0008	0.0017	0.0018	0.0022	0.0025	0.0030	0.0035	0.0038	0.0045
P6	High Strength Ferritic, Martensitic & PH Stainless Steel	275 - 450	0.0008	0.0015	0.0015	0.0018	0.0022	0.0025	0.0028	0.0033	0.0035
K1	Cast Iron	550 - 925	0.0012	0.0023	0.0025	0.0030	0.0035	0.0042	0.0048	0.0053	0.0060

SFM	Surface Feet Per Minute
IPM	Inch Per Minute
IPT	Inch Per Tooth
Z	Number of Flutes
RPM	Revolutions Per Minute
DOC	Depth of Cut
WOC	Width of Cut
MRR	Metal Removal Rate

Calculations:	
RPM	$3.82 * SFM / D = RPM$
IPM	$IPT * Z * RPM = IPM$
MRR	$WOC * DOC * IPM = MRR$

Speeds & Feeds

in T-Mill	Cutting Speed (SFM)		Feed per tooth (IPT)						
	Min	Max	1/8	3/16	1/4	3/8	1/2	5/8	3/4
			0.125	0.1875	0.250	0.375	0.500	0.625	0.750
Ferric, martensitic, and 15-5 PH stainless steels <35 Rc	235	270	0.0007	0.0010	0.0014	0.0021	0.0026	0.0030	0.0035
Ferric, martensitic, and 15-5 PH stainless steels >35 Rc	195	230	0.0007	0.0010	0.0014	0.0021	0.0026	0.0030	0.0035
Austenitic stainless steel (302, 303, 304)	300	375	0.0008	0.0011	0.0018	0.0023	0.0029	0.0031	0.0034
Austenitic stainless steel (316)	195	270	0.0006	0.0009	0.0013	0.0018	0.0024	0.0025	0.0029
Nickel based heat-resistant alloys	90	140	0.0003	0.0004	0.0007	0.0011	0.0016	0.0020	0.0023
Titanium Alloys (Ti6Al4V)	170	220	0.0005	0.0008	0.0008	0.0015	0.0021	0.0028	0.0036

NOTE:

Side milling applications - for longest reach (L3 LON) tools, reduce max step over by 30%
 Slot milling applications - for longest reach (L3 LON) tools, reduce max slotting depth by 30%
 Lower value of cutting speed (SFM) is used for high-stock removal applications or for higher hardness (machinability) within group
 Higher value of cutting speed (SFM) is used for finishing applications or for lower hardness (machinability) within group
 Above parameters are based on ideal conditions. For smaller taper machining centers, please adjust parameters accordingly

in Thread Mill	Cutting Speed (SFM)		Feed per tooth (IPT) for nominal thread size							
	Min	Max	< 1/8	1/8 - 3/16	3/16 - 1/4	1/4 - 5/16	5/16 - 3/8	3/8 - 1/2	1/2 - 5/8	5/8 - 3/4
			0.125	0.1875	0.250	0.3125	0.375	0.500	0.625	0.750
Copper / Brass / Bronze	490	660	0.0004	0.0007	0.0010	0.0015	0.0018	0.0022	0.0030	0.0037
Aluminum Alloys	460	620	0.0004	0.0008	0.0012	0.0017	0.0021	0.0025	0.0035	0.0040
Soft Steels (<35Rc)	390	520	0.0003	0.0006	0.0010	0.0013	0.0017	0.0020	0.0027	0.0035
Alloy Steels < 35Rc (4140-4340)	460	620	0.0003	0.0006	0.0010	0.0012	0.0015	0.0018	0.0025	0.0032
Alloy Steels > 35Rc (4140-4340)	390	520	0.0002	0.0004	0.0007	0.0009	0.0012	0.0015	0.0020	0.0028
Tool Steels < 36Rc (A2, D2, S7)	300	490	0.0001	0.0003	0.0005	0.0007	0.0010	0.0012	0.0015	0.0018
Tool Steels > 36Rc (A2, D2, S7)	200	330	0.0001	0.0002	0.0004	0.0005	0.0007	0.0009	0.0012	0.0015
Die Steels (P20, H13)	160	250	0.0001	0.0003	0.0005	0.0007	0.0010	0.0012	0.0015	0.0018
Difficult to cut Stainless Steels (316L)	200	260	0.0003	0.0004	0.0006	0.0008	0.0011	0.0015	0.0020	0.0025
Gray Cast Iron	390	520	0.0005	0.0007	0.0010	0.0013	0.0017	0.0025	0.0032	0.0040
High Temperature Alloys	160	300	0.0003	0.0005	0.0007	0.0009	0.0012	0.0015	0.0021	0.0028
Titanium	260	460	0.0003	0.0005	0.0007	0.0009	0.0012	0.0015	0.0021	0.0028

**Due to the nature of thread milling, the thread mill will range from 140° tooth engagement for internal threads, to 60° engagement for external threads. Because of this, once you have calculated feed rate you must adjust it by the factors below

NOTE:

For internal thread - programmed Feed Rate = Feed Rate X (hole diameter - cutter diameter) / hole diameter
 For external thread - programmed Feed Rate = Feed Rate X (hole diameter + cutter diameter) / hole diameter
 Lower value of cutting speed (SFM) is used for high-stock removal applications or for higher hardness (machinability) within group
 Higher value of cutting speed (SFM) is used for finishing applications or for lower hardness (machinability) within group
 Above parameters are based on ideal conditions. For smaller taper machining centers, please adjust parameters accordingly

RPM	$(3.82 \times \text{SFM}) / \text{Dia.}$
IPR	IPM / RPM
SFM	$(\text{RPM} \times \text{Dia.}) / 3.82$
IPM	$\text{IPT (Chip Load)} \times \text{No. of Teeth} \times \text{RPM}$
IPT (Chip Load)	$(\text{Chip Load}) = \text{IPM} / (\text{No. of Teeth} \times \text{RPM})$

Speeds & Feeds

in HV	Cutting Speed (SFM)		Feed per tooth (IPT). For slotting reduce by 20%									
	Min	Max	1/32	1/16	3/32	1/8	3/16	1/4	5/16	3/8	7/16	1/2
			0.03125	0.0625	0.9375	0.125	0.1875	0.250	0.3125	0.375	0.4375	0.500
Soft Steels (<35Rc)	490	660	0.0004	0.0005	0.0006	0.0008	0.0010	0.0012	0.0016	0.0024	0.0027	0.0033
Alloy Steels < 35Rc (4140-4340)	460	620	0.00035	0.0005	0.0006	0.0010	0.0016	0.0020	0.0025	0.0031	0.0032	0.0035
Alloy Steels > 35Rc (4140-4340)	390	520	0.0003	0.0004	0.0005	0.0009	0.0014	0.0019	0.0020	0.0028	0.0030	0.0033
Tool Steels < 36Rc (A2, D2, S7)	300	490	0.00025	0.0003	0.0004	0.0006	0.0009	0.0013	0.0016	0.0025	0.0027	0.0030
Tool Steels > 36Rc (A2, D2, S7)	200	330	0.0002	0.00025	0.0003	0.0009	0.0017	0.0020	0.0025	0.0029	0.0030	0.0033
Die Steels (P20, H13)	160	250	0.00025	0.0003	0.0005	0.0010	0.0017	0.0020	0.0023	0.0032	0.0035	0.0037
Easy to cut Stainless Steels (303)	260	330	0.0003	0.0004	0.0006	0.0006	0.0008	0.0008	0.0014	0.0021	0.0028	0.0033
Mod. Difficult to cut Stainless Steels	200	260	0.00025	0.00035	0.0005	0.0005	0.0005	0.0007	0.0012	0.0020	0.0025	0.0030
Difficult to cut Stainless Steels (316L)	200	260	0.00015	0.0002	0.0003	0.0004	0.0006	0.0007	0.0013	0.0019	0.0023	0.0025
Gray Cast Iron	390	520	0.0003	0.0004	0.0006	0.0008	0.0010	0.0012	0.0016	0.0025	0.0027	0.0035
High Temperature Alloys	160	300	0.00015	0.00025	0.0003	0.0004	0.0005	0.0007	0.0008	0.0014	0.0023	0.0029
Titanium	260	460	0.0002	0.0003	0.0004	0.0005	0.0007	0.0008	0.0013	0.0019	0.0020	0.0026

in HV Feed Mill	Cutting Speed (SFM)		Feed per tooth (IPT)						
	Min	Max	1/8	3/16	1/4	5/16	3/8	1/2	5/8
			0.125	0.1875	0.250	0.3125	0.375	0.500	0.625
Alloy Steels < 35Rc (4140-4340)	800	1000	0.0082	0.0113	0.0125	0.0156	0.0187	0.0197	0.0235
Alloy Steels > 35Rc (4140-4340)	800	990	0.0082	0.0113	0.0124	0.0153	0.0184	0.0193	0.0231
Tool Steels < 36Rc (A2, D2, S7)	720	790	0.0060	0.0076	0.0082	0.0097	0.0119	0.0126	0.0156
Tool Steels > 36Rc (A2, D2, S7)	590	720	0.0052	0.0062	0.0066	0.0078	0.0096	0.0102	0.0129
Die Steels (P20, H13)	490	590	0.0039	0.0039	0.0039	0.0038	0.0054	0.0059	0.0078
Easy to cut Stainless Steels (303)	700	800	0.0061	0.0077	0.0082	0.0097	0.0122	0.0130	0.0156
Gray Cast Iron	750	820	0.0064	0.0080	0.0087	0.0105	0.0130	0.0138	0.0164

in A-Mill	Cutting Speed (SFM)		Feed per tooth (IPT)					
	Min	Max	1/4	3/8	1/2	5/8	3/4	1
			0.250	0.375	0.500	0.625	0.750	1.000
Rougher Series (510)								
Aluminum Alloys	1200	1600	0.0036	0.0057	0.008	0.0098	0.0119	0.016
Aluminum Alloys 440, 356, 380, C61300	600	1000	0.0027	0.0043	0.006	0.0074	0.0089	0.012
Semi-Finisher Series (520)								
Aluminum Alloys	1200	1600	0.0054	0.00855	0.012	0.0147	0.01785	0.024
Aluminum Alloys 440, 356, 380, C61300	600	1000	0.00405	0.00645	0.009	0.0111	0.01335	0.018
Finisher Series (550)								
Aluminum Alloys	1200	1600	0.00342	0.005415	0.0076	0.00931	0.011305	0.0152
Aluminum Alloys 440, 356, 380, C61300	600	1000	0.002565	0.004085	0.0057	0.00703	0.008455	0.0114

NOTE:

Side milling applications - for longest reach (L3 LON) tools, reduce max step over by 30%
 Slot milling applications - for longest reach (L3 LON) tools, reduce max slotting depth by 30%
 Lower value of cutting speed (SFM) is used for high-stock removal applications or for higher hardness (machinability) within group
 Higher value of cutting speed (SFM) is used for finishing applications or for lower hardness (machinability) within group
 Above parameters are based on ideal conditions. For smaller taper machining centers, please adjust parameters accordingly