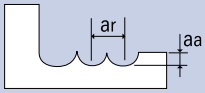
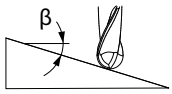
List 8430, 8530: 4 Flute, Regular Length, Ball End

Roughing - Contouring

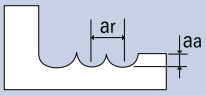
Hardness	Up to 45 HRC		45-55 HRC		55-62 HRC		62-66 HRC		66-70 HRC	
Work Material	Tool Steels Hardened Steels Alloy Steels		Hardened Steels							
Cutting Speed	425 - 500 SFM		375 - 450 SFM		325 - 400 SFM		300 - 360 SFM		200 - 260 SFM	
Depth of Cut			Dia / aa / ar D<6: 0.15D / 0.2D 6≤D: 0.1D / 0.2D		Dia / aa / ar D<6: 0.12D / 0.15D 6≤D: 0.07D / 0.15D		aa=0.05D ar=0.15D			
Mill Dia. Inch	Mill Dia. mm	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min
–	2	20,850	131	18,450	73	16,000	63	14,550	57	9,700	31
–	3	13,900	110	12,300	68	10,650	58	9,700	53	6,450	31
1/8	–	13,150	108	11,600	67	10,100	58	9,150	53	6,100	31
–	4	10,450	99	9,200	65	8,000	57	7,300	52	4,850	31
3/16	–	8,750	103	7,750	73	6,700	63	6,100	57	4,050	35
–	5	8,350	105	7,350	75	6,400	65	5,800	59	3,900	37
–	6	6,950	110	6,150	77	5,350	68	4,850	61	3,250	36
1/4	–	6,550	103	5,800	73	5,050	64	4,600	58	3,050	34
5/16	–	5,250	98	4,650	72	4,050	63	3,650	57	2,450	34
–	8	5,200	98	4,600	72	4,000	63	3,650	57	2,450	35
3/8	–	4,700	95	4,200	72	3,650	62	3,350	57	2,350	35
–	10	4,450	91	4,000	69	3,500	61	3,200	55	2,250	34
–	12	4,050	89	3,650	69	3,250	62	2,900	55	2,100	33
1/2	–	3,800	84	3,450	65	3,050	58	2,750	52	2,000	31

1. Use a rigid and precise machine and holder.
2. We suggest using air blow or MQL (mist).
3. The above parameters are applicable to an overhang of 4xD maximum. When the overhang is longer, please reduce feed, speed, and cutting depth.
4. The above parameters are standard starting values for contouring and side milling operations. If vibration or chatter occurs due to machine or part setup, please adjust the speed, feed, and depth of cut accordingly.
5. If contouring includes corners of radius less than 1.5 times the tool diameter, reduce speed and feed to 50-80% of above and reduce Ar to 20-60% of above.
6. When the part incline angle (β) is more than 15°, reduce the speed to 40-60% of above parameters, the feed to 30-50% of above parameters, and Aa to 30-60% of above parameters.
7. If the cutting depth is small it is possible to increase the speed and feed above the recommended parameters.

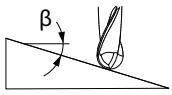


List 8430, 8530: 4 Flute, Regular Length, Ball End

Finishing - Contouring

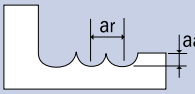
Hardness		Up to 45 HRC		45-55 HRC		55-62 HRC		62-66 HRC		66-70 HRC	
Work Material		Tool Steels Hardened Steels Alloy Steels		Hardened Steels							
Cutting Speed		550 - 625 SFM		500 - 575 SFM		450 - 525 SFM		375 - 425 SFM		275 - 350 SFM	
Depth of Cut		 a_a=0.02D a_r=0.05D									
Mill Dia. Inch	Mill Dia. mm	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min
–	2	25,000	157	24,750	97	22,300	70	18,450	58	13,600	43
–	3	18,100	143	16,500	91	14,900	70	12,300	58	9,050	43
1/8	–	17,100	140	15,600	90	14,050	70	11,600	58	8,550	43
–	4	13,600	129	12,350	87	11,150	70	9,200	58	6,800	43
3/16	–	11,400	135	10,400	98	9,350	81	7,750	67	5,700	49
–	5	10,850	137	9,900	101	8,950	85	7,350	69	5,450	52
–	6	9,050	143	8,250	104	7,450	82	6,150	68	4,550	50
1/4	–	8,550	135	7,800	98	7,050	78	5,800	64	4,300	47
5/16	–	6,850	127	6,250	97	5,600	69	4,650	58	3,400	42
–	8	6,800	128	6,200	98	5,600	71	4,600	58	3,400	43
3/8	–	6,000	121	5,600	95	5,000	69	4,200	58	3,150	44
–	10	5,700	116	5,350	93	4,750	67	4,000	57	3,000	43
–	12	5,000	110	4,600	87	4,200	66	3,500	55	2,850	45
1/2	–	4,750	105	4,350	82	3,950	62	3,300	52	2,650	42

1. Use a rigid and precise machine and holder.
2. We suggest using air blow or MQL (mist).
3. The above parameters are applicable to an overhang of 4xD maximum. When the overhang is longer, please reduce feed, speed, and cutting depth.
4. The above parameters are standard starting values for contouring and side milling operations. If vibration or chatter occurs due to machine or part setup, please adjust the speed, feed, and depth of cut accordingly.
5. If contouring includes corners of radius less than 1.5 times the tool diameter, reduce speed and feed to 50-80% of above and reduce Ar to 20-60% of above.
6. When the part incline angle (β) is more than 15°, reduce the speed to 40-60% of above parameters, the feed to 30-50% of above parameters, and Aa to 30-60% of above parameters.
7. If the cutting depth is small it is possible to increase the speed and feed above the recommended parameters.

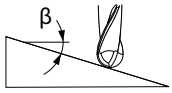


List 8430, 8530: 4 Flute, Regular Length, Ball End

Roughing - High Speed Contouring

Hardness		Up to 45 HRC		45-55 HRC		55-62 HRC		62-66 HRC		66-70 HRC	
Work Material		Tool Steels Hardened Steels Alloy Steels		Hardened Steels							
Cutting Speed		750 - 1000 SFM		675 - 900 SFM		600 - 800 SFM		525 - 725 SFM		350 - 525 SFM	
Depth of Cut		aa=0.1D ar=0.2D		aa=0.08D ar=0.2D		 aa=0.05D ar=0.1D					
Mill Dia.		Speed	Feed	Speed	Feed	Speed	Feed	Speed	Feed	Speed	Feed
Inch	mm	RPM	in/min	RPM	in/min	RPM	in/min	RPM	in/min	RPM	in/min
–	2	25,000	157	25,000	98	25,000	98	25,000	79	17,000	54
–	3	24,900	196	22,000	121	19,100	105	17,150	81	11,300	53
1/8	–	23,550	186	20,800	115	18,050	99	16,200	77	10,700	51
–	4	20,600	195	18,200	129	16,000	113	14,300	90	9,600	60
3/16	–	17,300	204	15,300	136	13,450	119	12,000	94	8,050	63
–	5	16,500	208	14,550	149	12,800	131	11,450	108	7,650	72
–	6	13,750	217	12,150	153	10,650	134	9,550	105	6,400	71
1/4	–	13,000	205	11,450	144	10,100	127	9,000	99	6,050	67
5/16	–	10,400	203	9,300	145	8,050	126	7,200	99	4,850	66
–	8	10,300	195	9,200	145	8,000	126	7,150	101	4,800	68
3/8	–	9,150	196	8,400	150	7,350	131	6,700	108	4,700	76
–	10	8,750	179	8,000	139	7,000	121	6,400	101	4,450	70
–	12	8,100	179	7,300	138	6,400	121	5,800	91	4,200	66
1/2	–	7,650	169	6,900	130	6,050	114	5,500	87	3,950	62

1. Use a rigid and precise machine and holder.
2. We suggest using air blow or MQL (mist).
3. The above parameters are applicable to an overhang of 4xD maximum. When the overhang is longer, please reduce feed, speed, and cutting depth.
4. The above parameters are standard starting values for contouring and side milling operations. If vibration or chatter occurs due to machine or part setup, please adjust the speed, feed, and depth of cut accordingly.
5. If contouring includes corners of radius less than 1.5 times the tool diameter, reduce speed and feed to 50-80% of above and reduce Ar to 20-60% of above.
6. When the part incline angle (β) is more than 15°, reduce the speed to 40-60% of above parameters, the feed to 30-50% of above parameters, and Aa to 30-60% of above parameters.
7. If the cutting depth is small it is possible to increase the speed and feed above the recommended parameters.

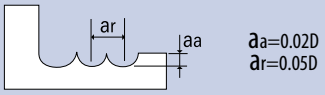


ABOUT OSG
DRILLING
THREADING
MILLING
HOLDERS
INDEX

CONTINUED

List 8430, 8530: 4 Flute, Regular Length, Ball End

Finishing - High Speed Contouring

Hardness		Up to 45 HRC		45-55 HRC		55-62 HRC		62-66 HRC		66-70 HRC	
Work Material		Tool Steels Hardened Steels Alloy Steels		Hardened Steels							
Cutting Speed		825 - 1125 SFM		750 - 1050 SFM		700 - 950 SFM		575 - 775 SFM		425 - 625 SFM	
Depth of Cut		 ar=0.02D ar=0.05D									
Mill Dia.		Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min	Speed RPM	Feed in/min
Inch	mm										
–	2	25,000	157	25,000	98	25,000	79	25,000	79	20,400	64
–	3	25,000	197	24,600	136	22,300	105	18,100	86	13,600	64
1/8	–	25,000	197	23,250	128	21,100	106	17,100	86	12,850	64
–	4	24,250	229	22,300	158	20,150	127	16,500	104	12,150	77
3/16	–	20,350	240	18,750	166	16,900	146	13,850	120	10,200	88
–	5	19,400	244	17,850	183	16,100	152	13,200	125	9,700	92
–	6	16,150	254	14,900	188	13,400	148	11,000	121	8,100	89
1/4	–	15,300	241	14,050	177	12,700	140	10,400	115	7,650	84
5/16	–	12,200	238	11,250	176	10,150	139	8,300	114	6,100	83
–	8	12,150	230	11,150	176	10,050	127	8,250	104	6,050	76
3/8	–	10,800	231	9,950	178	9,050	125	7,550	105	5,700	79
–	10	10,300	211	9,450	164	8,650	123	7,200	102	5,450	77
–	12	9,050	199	8,350	158	7,600	120	6,250	99	5,000	79
1/2	–	8,550	188	7,850	148	7,200	114	5,900	93	4,750	75

1. Use a rigid and precise machine and holder.
2. We suggest using air blow or MQL (mist).
3. The above parameters are applicable to an overhang of 4xD maximum. When the overhang is longer, please reduce feed, speed, and cutting depth.
4. The above parameters are standard starting values for contouring and side milling operations. If vibration or chatter occurs due to machine or part setup, please adjust the speed, feed, and depth of cut accordingly.
5. If contouring includes corners of radius less than 1.5 times the tool diameter, reduce speed and feed to 50-80% of above and reduce Ar to 20-60% of above.
6. When the part incline angle (β) is more than 15°, reduce the speed to 40-60% of above parameters, the feed to 30-50% of above parameters, and Aa to 30-60% of above parameters.
7. If the cutting depth is small it is possible to increase the speed and feed above the recommended parameters.