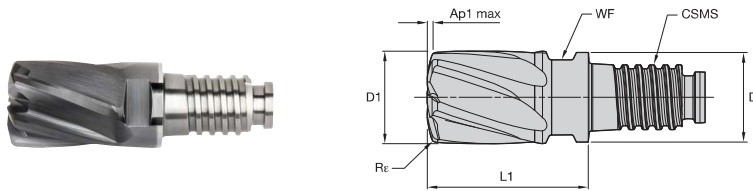


DUO-LOCK™ • KENFEED™ • 6 FLUTES

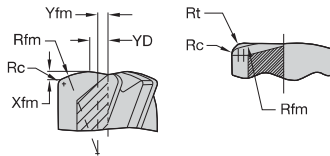


- first choice
- alternate choice

P	●
M	○
K	●
N	○
S	●
H	○

order number	catalogue number	D1	D	Ap1 max	L1	CSMS	WF	Rc	KC643M
6625741	KSDB1000X6BQX	10,00	9,60	0,53	17,50	DL10	8,00	0,63	●
6625742	KSDB1200X6BQX	12,00	11,50	0,63	21,00	DL12	9,50	0,75	●
6625743	KSDB1600X6BQX	16,00	15,50	0,84	28,00	DL16	13,00	1,00	●
6625744	KSDB2000X6BQX	20,00	19,30	1,05	35,00	DL20	16,00	1,25	●

DUO-LOCK™ • KENFEED • 6 FLUTES • PROGRAMMING DATA



ramping guide for circular and linear interpolation																
geometrical parameters									circular interpolation		linear interpolation					
									allowed range of hole diameter		calculated length per ramp angle					
catalogue number	D1	Ap1 max	Rfm	Rt	Rc	Xfm	Yfm	YD	Number of flutes	smallest	largest	1°	2°	3°	4°	5°
KSDB1000X6BQX	10,00	0,53	10,00	1,04	0,625	0,53	1,25	2,20	6	14,40	20,00	30,20	15,09	10,06	7,54	6,02
KSDB1200X6BQX	12,00	0,63	12,00	1,24	0,750	0,63	1,50	2,64	6	17,28	24,00	36,24	18,11	12,07	9,05	7,23
KSDB1600X6BQX	16,00	0,84	16,00	1,66	1,000	0,84	2,00	3,52	6	23,04	32,00	48,31	24,15	16,09	12,06	9,64
KSDB2000X6BQX	20,00	1,05	20,00	2,07	1,250	1,05	2,50	4,40	6	28,80	40,00	60,39	30,19	20,11	15,08	12,05
recommended degree of programmed feed rate to use while ramping												100%	70%	50%	30%	10%

NOTE: YRC = distance from centreline to the crown of the R radius.
 RCN = distance from centreline to the start of the cutting edge. This dimension can also help determine the minimum circle size when helical ramping.
 R = the head radius size.
 Rc = the shoulder radius or radius at the corner of the cutter.

DUO-LOCK™ • KENFEED • 6 FLUTES • APPLICATION DATA

Material Group	Side Milling (A)		short			medium			long			Recommended feed per tooth (fz = mm/th) for side milling (A).					
	A		adaptor reach									D1 – Diameter					
			KC643M			KC643M			KC643M								
	ap	ae	Cutting Speed – vc m/min		Cutting Speed – vc m/min		Cutting Speed – vc m/min		Cutting Speed – vc m/min		mm	10,0	12,0	16,0	20,0		
P	5	0,05 x D	0,55 x D	60	–	100	51	–	85	48	–	80	fz	0,290	0,337	0,419	0,485
	6	0,05 x D	0,55 x D	50	–	75	43	–	64	40	–	60	fz	0,242	0,279	0,342	0,389
M	1	0,05 x D	0,55 x D	90	–	115	72	–	92	63	–	81	fz	0,363	0,421	0,523	0,606
	2	0,05 x D	0,55 x D	60	–	80	48	–	64	42	–	56	fz	0,290	0,337	0,419	0,485
S	3	0,05 x D	0,55 x D	60	–	70	48	–	56	42	–	49	fz	0,242	0,279	0,342	0,389
	1	0,05 x D	0,55 x D	50	–	90	40	–	72	30	–	54	fz	0,363	0,421	0,523	0,606
	2	0,05 x D	0,55 x D	25	–	40	20	–	32	15	–	24	fz	0,192	0,223	0,278	0,324
	3	0,05 x D	0,55 x D	25	–	40	20	–	32	15	–	24	fz	0,192	0,223	0,278	0,324
	4	0,05 x D	0,55 x D	50	–	60	40	–	48	30	–	36	fz	0,267	0,310	0,385	0,445

NOTE: These guidelines may require variations to achieve optimum results.
 Lower value of cutting speed is used for high stock removal applications or for higher hardness (machinability) within group.
 Higher value of cutting speed is used for finishing applications or for lower hardness (machinability) within group.
 Above parameters are based on ideal conditions. For smaller taper machining centres, please adjust parameters accordingly on diameters greater than 12mm.
 For cutting aluminium with high silicon TiCN coating is recommended.
 For better surface finish reduce feed per tooth.
 For tools with reach >3 x D, reduce fz by 20%.
 For tools with reach >5 x D, reduce fz by 30%.
 For tools with reach >10 x D, reduce Vc and fz by 30%.

