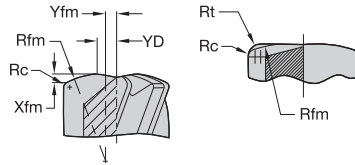


## DUO-LOCK™ • KENFEED • 6 FLUTES • PROGRAMMING DATA



### 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												Recommended feed per tooth (fz = mm/th) for side milling (A).					
	Side Milling (A)		short			medium			long								
	A		adaptor reach									D1 – Diameter					
			KC643M			KC643M			KC643M								
		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%.

