kennametal.com

KENNAMETAL®

MASTER CATALOG 2018

VOLUME TWO | **ROTATING** TOOLS

HOLEMAKING | TAPPING | SOLID END MILLING | INDEXABLE MILLING

High-Performance Solid Carbide Thread Mills

Our solid carbide thread mills are designed to deliver high-quality internal threading on 3-axis CNC machines. Because these mills are made of carbide, they are capable of easily cutting most difficult materials up to 63 HRC. Thread mills make interrupted cuts and short chips.

The combination of these design elements offers a range of benefits to improve overall thread quality and tool production. Short, easily evacuated chips generate less heat and friction, so there is a lower risk of damage to threading. Also, the superior carbide grades make threading easier and machining times shorter.

Features and Benefits

System Requirements

- 3-axis CNC machine.
- Good clamping for tool and workpiece.
- Internal coolant supply.

Advantages

- Versatile.
- Better surface quality.
- No chip problems.
- No need to reverse the spindle.
- More production safety.

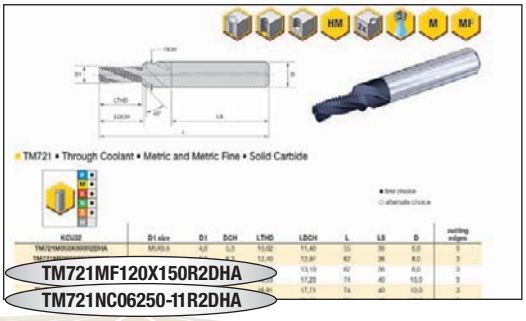
Features

- Interrupted cut.
- Short chips.
- Optimized carbide grades.
- Drill, thread, chamfer.

Through and Blind Holes (2 x D)

					
		Thread Mill	Thread Mill and Chamfer	Drill, Thread Mill, and Chamfer	Mill, Thread Mill, and Chamfer
P	<35 HRC	TM711	TM721	—	TM741_RHSF
P	35–43 HRC	—	TM721	—	TM741_RHSF
M		TM711	TM721	—	TM741_RHSF
K		TM711	TM721	TM731	TM741_RHSF
N	Wrought	TM711	TM721	—	TM731
N	Cast	TM711	TM721	TM731	TM741_RHSF
S		TM711	TM721	—	TM741_LHSF
H	44–63 HRC	—	—	—	TM741_RHSF TM741_LHSF

Solid Thread Mills Identification System



Metric

TM721	MF	120	X	150	R	2D	HA

Inch

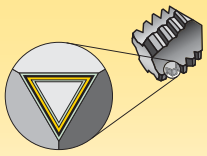
TM721	NC	06250	-	11	R	2D	HA

Thread Mill Design	Type of Thread	Nominal Diameter of Thread		Pitch	Cutting Direction	Depth of Cut	Shank
		mm or inch (depending on type)		mm or TPI (depending on type)			

M = Metric coarse-pitch thread (ISO form)
MF = Metric fine-pitch thread (ISO form)
NC = Unified coarse series thread
NF = Unified fine series thread

DIN 6535
HA = Plain Shank
HB = Weldon® Shank
HE = Whistle Notch Shank

Style
TM711 = Solid Thread Mill; Through Coolant
TM721 = Solid Thread Mill and Chamfer; Through Coolant
TM731 = Solid Thread Mill, Chamfer, and Drill; Through Coolant
TM741 = Solid Thread Mill, Chamfer, and Mill; Through Coolant

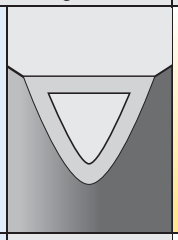
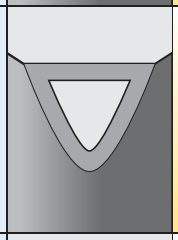
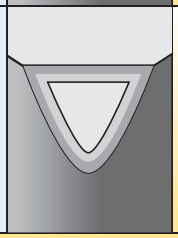


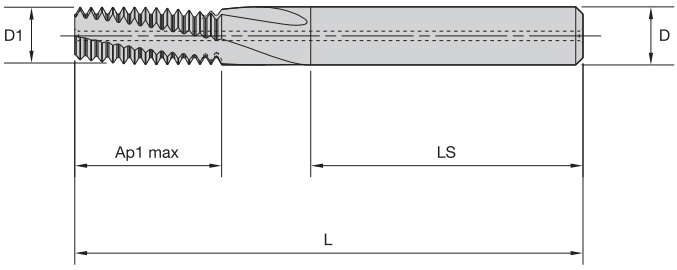
Coatings are designed for optimized tapping performance in specific materials.

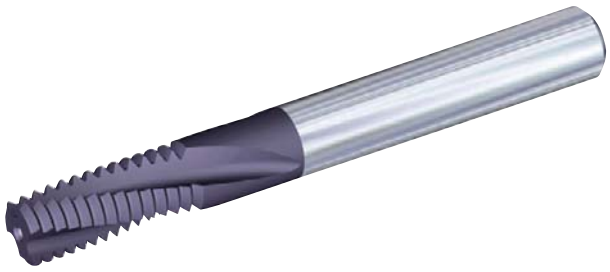
P	Steel
M	Stainless Steel
K	Cast Iron
N	Non-Ferrous
S	High-Temp Alloys
H	Hardened Materials

wear resistance ⟷ **toughness**

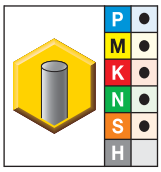
Grade

Coating	Grade Description		05	10	15	20	25	30	35	40	45
KCU32 	Coated carbide. PVD — Fine-grain carbide substrate with high hardness TiCN coating. Universal grade for thread milling most materials.	P		■	■	■					
		M		■	■	■					
		K		■	■	■					
		N			■	■	■				
		S		■	■	■					
KCU33 	Coated carbide. PVD — Carbide substrate with heat-resistant TiAlN coating. Universal grade for thread milling most materials.	P			■	■	■	■			
		M			■	■	■	■			
		K		■	■	■	■				
		N				■	■	■	■		
		S			■	■	■	■			
KCU36 	Coated carbide. PVD — two-layer coating with heat-resistant TiAlN base layer and low-friction MoS_2 top layer over carbide substrate. Use for thread milling most materials including high hardness materials.	P				■	■	■	■		
		M				■	■	■	■		
		K			■	■	■	■			
		N			■	■	■	■			
		S				■	■	■	■		
		H					■	■	■	■	





TM711 • Through Coolant • Metric and Metric Fine • Solid Carbide • Solid Carbide

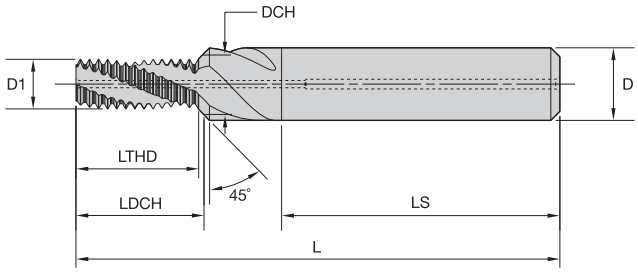


● first choice
○ alternate choice

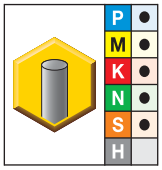
KCU33	D1 size	D1	Ap1 max	L	LS	D	cutting edges
TM711M030X050R2DHA	M3 X 0.5	2,4	–	42	28	4,0	3
TM711MF040X050R2DHA	M4 X 0.5	3,4	8,2	55	36	6,0	3
TM711M040X070R2DHA	M4 X 0.7	3,2	–	55	36	6,0	3
TM711MF050X050R2DHA	M5 X 0.5	4,3	10,2	55	36	6,0	3
TM711M050X080R2DHA	M5 X 0.8	4,0	–	55	36	6,0	3
TM711MF060X075R2DHA	M6 X 0.75	5,0	12,0	55	36	6,0	3
TM711M060X100R2DHA	M6 X 1	4,8	12,0	55	36	6,0	3
TM711MF080X075R2DHA	M8 X 0.75	6,0	16,8	63	36	6,0	3
TM711MF080X100R2DHA	M8 X 1.0	6,0	16,4	63	36	6,0	3
TM711M080X125R2DHA	M8 X 1.25	5,9	16,0	63	36	6,0	3
TM711MF100X100R2DHA	M10 X 1.0	8,0	20,5	70	36	8,0	3
TM711M100X150R2DHA	M10 X 1.5	8,0	20,2	70	36	8,0	3
TM711MF120X100R2DHA	M12 X 1.0	10,0	24,5	80	40	10,0	4
TM711MF120X150R2DHA	M12 X 1.5	10,0	24,7	80	40	10,0	4
TM711M120X175R2DHA	M12 X 1.75	10,0	25,3	80	40	10,0	4
TM711MF140X150R2DHA	M14 X 1.5	10,0	29,2	80	40	10,0	4
TM711M140X200R2DHA	M14 X 2.0	11,6	28,0	90	45	12,0	4
TM711MF160X150R2DHA	M16 X 1.5	12,0	32,2	90	45	12,0	4
TM711M160X200R2DHA	M16 X 2.0	12,0	32,9	90	45	12,0	4
TM711MF180X150R2DHA	M18 X 1.5	14,0	36,7	90	45	14,0	4
TM711M180X250R2DHA	M18 X 2.5	14,0	38,7	90	45	14,0	4
TM711MF200X150R2DHA	M20 X 1.5	14,0	41,2	90	45	14,0	4
TM711M200X250R2DHA	M20 X 2.5	14,0	41,2	90	45	14,0	4

Shank Tolerance

D	tolerance h6
6	+0, -0,008
8–10	+0, -0,009
12–18	+0, -0,011
20–30	+0, -0,013



TM721 • UNC • Through Coolant • Inch • Solid Carbide • Solid Carbide



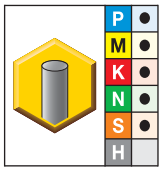
● first choice
○ alternate choice

KCU32	D1 size	D1	DCH	LTHD	LDCH	L	LS	D	cutting edges
TM721NC2500-20R2DHA	1/4-20	4,7	6,7	13,36	14,23	62	36	8,0	3
TM721NC3125-18R2DHA	5/16-18	6,2	8,3	16,26	17,19	74	40	10,0	3
TM721NC3750-16R2DHA	3/8-16	7,7	9,8	19,89	20,85	80	45	12,0	3
TM721NC4375-14R2DHA	7/16-14	9,0	11,4	22,72	23,79	80	45	12,0	3
TM721NC0500-13R2DHA	1/2-13	10,4	13,0	26,43	27,60	90	45	14,0	4
TM721NC5625-12R2DHA	9/16-12	11,8	14,6	30,75	31,99	100	48	16,0	4
TM721NC0625-11R2DHA	5/8-11	13,1	16,2	33,54	34,89	102	48	18,0	4

Shank Tolerance

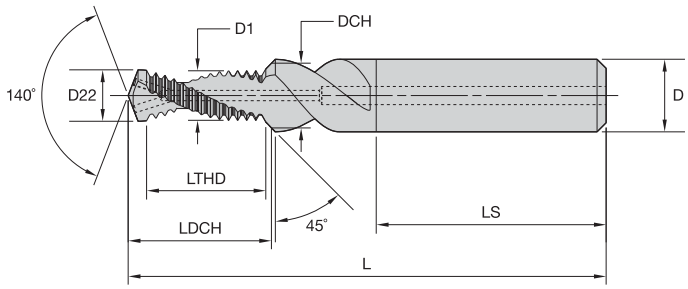
D	tolerance h6
6	+0, -0,008
8–10	+0, -0,009
12–18	+0, -0,011
20–30	+0, -0,013

TM721 • Through Coolant • Metric and Metric Fine • Solid Carbide • Solid Carbide

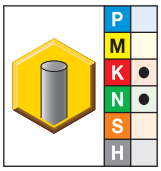


● first choice
○ alternate choice

KCU32	D1 size	D1	DCH	LTHD	LDCH	L	LS	D	cutting edges
TM721M060X100R2DHA	M6X1	4,8	6,3	12,52	13,19	62	36	8,0	3
TM721MF080X100R2DHA	M8X1	6,7	8,3	16,53	17,23	74	40	10,0	3
TM721M080X125R2DHA	M8X1.25	6,5	8,3	16,91	17,71	74	40	10,0	3
TM721M100X150R2DHA	M10X1.5	8,2	10,3	20,29	21,22	80	45	12,0	3
TM721MF120X125R2DHA	M12X1.25	10,4	12,3	24,43	25,24	90	45	14,0	4
TM721M120X175R2DHA	M12X1.75	9,9	12,3	25,42	26,48	90	45	14,0	4
TM721MF140X150R2DHA	M14X1.5	12,1	14,3	29,31	30,25	100	48	16,0	4
TM721M140X200R2DHA	M14X2	11,6	14,3	29,05	30,24	100	48	16,0	4
TM721M160X200R2DHA	M16X2	13,6	16,3	33,05	34,24	102	48	18,0	4



TM731 • Through Coolant • Metric and Metric Fine • Solid Carbide • Solid Carbide

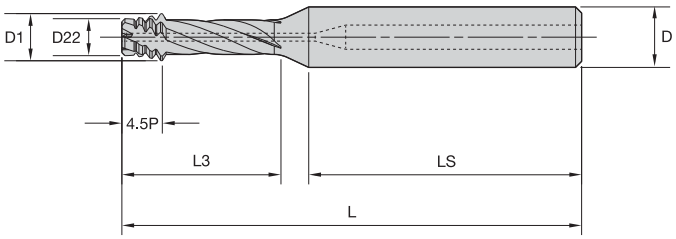


● first choice
○ alternate choice

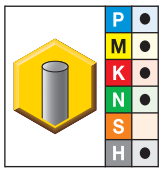
KCU32	D1 size	D22	D1	DCH	LTHD	LDCH	L	LS	D	cutting edges
TM731M040X070R2DHA	M4X0.7	3,3	3,2	4,3	7,74	9,59	49	36	6,0	2
TM731M050X080R2DHA	M5X0.8	4,2	4,0	5,3	9,65	11,82	55	36	6,0	2
TM731M060X100R2DHA	M6X1	5,0	4,8	6,3	12,06	14,69	62	36	8,0	2
TM731MF080X100R2DHA	M8X1	7,0	6,8	8,3	16,09	19,10	74	40	10,0	2
TM731M080X125R2DHA	M8X1.25	6,8	6,5	8,3	15,08	18,42	74	40	10,0	2
TM731M100X150R2DHA	M10X1.5	8,5	8,2	10,3	19,59	23,65	79	45	12,0	2
TM731M120X175R2DHA	M12X1.75	10,3	9,9	12,3	22,86	27,63	89	45	14,0	2
TM731M160X200R2DHA	M16X2	14,0	13,6	16,3	32,13	38,00	102	48	18,0	2

Shank Tolerance

D	tolerance h6
6	+0, -0,008
8–10	+0, -0,009
12–18	+0, -0,011
20–30	+0, -0,013



TM741 • UNC and UNF • Through Coolant • Right Hand • Inch • Solid Carbide

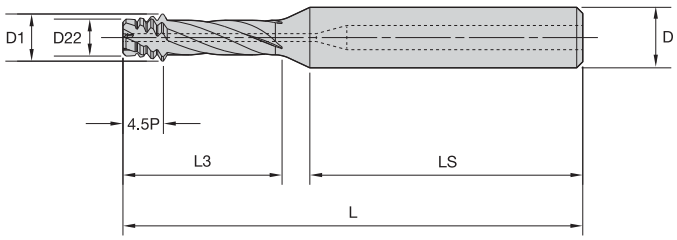


● first choice
○ alternate choice

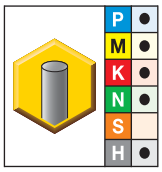
KCU36	D1 size	D1	D22	L3	L	LS	D	cutting edges
TM741NC2500-20R2DHA	1/4-20	4,64	3,34	17,0	60	36	8,0	3
TM741NF2500-28R2DHA	1/4-28	4,66	3,62	17,0	60	36	8,0	3
TM741NC3125-18R2DHA	5/16-18	5,64	4,12	21,9	76	40	10,0	4
TM741NF3125-24R2DHA	5/16-24	5,64	4,48	21,9	76	40	10,0	4
TM741NC3750-16R2DHA	3/8-16	7,16	5,42	26,3	76	40	10,0	4
TM741NF3750-24R2DHA	3/8-24	7,14	6,00	26,3	76	40	10,0	4
TM741NC0500-13R2DHA	1/2-13	10,08	7,95	33,4	86	45	12,0	4
TM741NF0500-20R2DHA	1/2-20	8,45	7,06	33,0	86	45	12,0	4
TM741NF0625-18R2DHA	5/8-18	12,38	10,83	42,0	98	48	16,0	4
TM741NC0750-10R2DHA	3/4-10	15,50	12,77	51,3	111	50	20,0	5
TM741NC0750-16R2DHA	3/4-16	15,38	13,65	51,3	111	50	20,0	5

Shank Tolerance

D	tolerance h6
6	+0, -0,008
8–10	+0, -0,009
12–18	+0, -0,011
20–30	+0, -0,013



TM741 • Through Coolant • Right Hand • Metric and Metric Fine • Solid Carbide



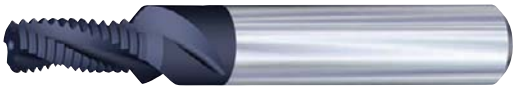
● first choice
○ alternate choice

KCU36	D1 size	D1	D22	L3	L	LS	D	cutting edges
TM741M060X100R2DHA	M6X1	4,51	3,41	16,5	60	36	8,0	3
TM741MF080X100R2DHA	M8X1	6,23	5,13	21,9	71	40	10,0	4
TM741M080X125R2DHA	M8X1.25	6,23	4,91	21,9	71	40	10,0	4
TM741MF100X100R2DHA	M10X1	6,23	5,13	21,9	71	40	10,0	4
TM741MF100X125R2DHA	M10X1.25	6,23	4,91	21,9	71	40	10,0	4
TM741M100X150R2DHA	M10X1.5	7,75	6,11	26,3	76	40	10,0	4
TM741MF120X150R2DHA	M12X1.5	7,75	6,11	26,3	76	40	10,0	4
TM741M120X175R2DHA	M12X1.75	9,16	7,21	32,4	86	45	12,0	4
TM741M140X200R2DHA	M14X2	11,08	8,91	41,0	98	48	16,0	4
TM741M160X200R2DHA	M16X2	11,08	8,91	41,0	98	48	16,0	4
TM741M180X250R2DHA	M18X2.5	14,38	11,71	51,3	111	50	20,0	5
TM741M200X250R2DHA	M20X2.5	14,38	11,71	51,3	111	50	20,0	5

Shank Tolerance

D	tolerance h6
6	+0, -0,008
8–10	+0, -0,009
12–18	+0, -0,011
20–30	+0, -0,013

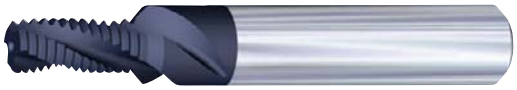
Thread Mill TM711



Thread Mill and Chamfer TM721

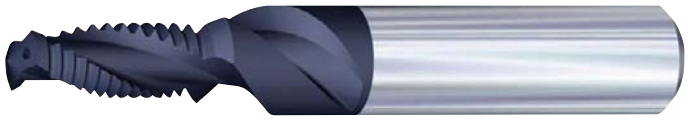
Material Group		Cutting Speed – vc Range – m/min			Feed/Tooth by Diameter			Cutting Speed – vc Range – m/min			Feed/Tooth by Diameter		
		min	Starting Value	max		<10mm	>10mm	min	Starting Value	max		<10mm	>10mm
P	1	90	**115**	150	mm/r	0,05	0,08	140	**185**	240	mm/r	0,06	0,10
	2	90	**115**	150	mm/r	0,05	0,08	140	**185**	240	mm/r	0,06	0,10
	3	40	**50**	70	mm/r	0,02	0,03	70	**90**	120	mm/r	0,03	0,04
	4	–	**–**	–	–	–	–	70	**90**	120	mm/r	0,03	0,04
	5	60	**80**	100	mm/r	0,04	0,06	70	**90**	120	mm/r	0,05	0,08
	6	–	**–**	–	–	–	–	–	**–**	–	–	–	–
M	1	60	**80**	100	mm/r	0,04	0,06	70	**90**	120	mm/r	0,05	0,08
	2	60	**80**	100	mm/r	0,04	0,06	70	**90**	120	mm/r	0,05	0,08
	3	–	**–**	–	–	–	–	–	**–**	–	–	–	–
K	1	120	**150**	200	mm/r	0,06	0,10	130	**170**	220	mm/r	0,06	0,11
	2	120	**150**	200	mm/r	0,06	0,10	130	**170**	220	mm/r	0,06	0,11
	3	90	**115**	150	mm/r	0,05	0,07	110	**140**	180	mm/r	0,05	0,07
N	1	250	**275**	300	mm/r	0,07	0,09	270	**300**	330	mm/r	0,08	0,16
	2	200	**225**	250	mm/r	0,05	0,06	270	**300**	330	mm/r	0,08	0,16
	3	170	**190**	210	mm/r	0,04	0,05	160	**175**	190	mm/r	0,08	0,16
	4	250	**275**	300	mm/r	0,07	0,09	270	**300**	330	mm/r	0,08	0,16
	5	270	**300**	330	mm/r	0,12	0,13	250	**275**	300	mm/r	0,11	0,20
	6	170	**190**	210	mm/r	0,05	0,06	90	**100**	110	mm/r	0,11	0,20
S	1	60	**80**	100	mm/r	0,04	0,06	70	**90**	120	mm/r	0,05	0,08
	2	50	**65**	80	mm/r	0,03	0,04	50	**60**	80	mm/r	0,03	0,05
	3	50	**65**	80	mm/r	0,03	0,04	50	**60**	80	mm/r	0,03	0,05
	4	50	**65**	80	mm/r	0,03	0,04	50	**60**	80	mm/r	0,03	0,05

Thread Mill TM711



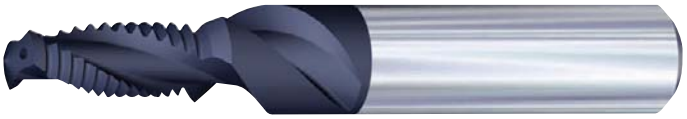
Thread Mill and Chamfer TM721

Material Group		Cutting Speed – vc Range – SFM			Feed/Tooth by Diameter			Cutting Speed – vc Range – SFM			Feed/Tooth by Diameter		
		min	Starting Value	max		<0.375"	>0.375"	min	Starting Value	max		<0.375"	>0.375"
P	1	300	**380**	490	IPR	0.002	0.003	460	**610**	790	IPR	0.002	0.004
	2	300	**380**	490	IPR	0.002	0.003	460	**610**	790	IPR	0.002	0.004
	3	130	**160**	230	IPR	0.001	0.001	230	**300**	390	IPR	0.001	0.001
	4	–	**–**	–	–	–	–	230	**300**	390	IPR	0.001	0.001
	5	200	**260**	330	IPR	0.002	0.002	230	**300**	390	IPR	0.002	0.003
	6	–	**–**	–	–	–	–	–	**–**	–	–	–	–
M	1	200	**260**	330	IPR	0.002	0.002	230	**300**	390	IPR	0.002	0.003
	2	200	**260**	330	IPR	0.002	0.002	230	**300**	390	IPR	0.002	0.003
	3	–	**–**	–	–	–	–	–	**–**	–	–	–	–
K	1	390	**490**	660	IPR	0.002	0.004	430	**560**	720	IPR	0.002	0.004
	2	390	**490**	660	IPR	0.002	0.004	430	**560**	720	IPR	0.002	0.004
	3	300	**380**	490	IPR	0.002	0.003	360	**460**	590	IPR	0.002	0.003
N	1	820	**900**	980	IPR	0.003	0.004	890	**980**	1080	IPR	0.003	0.006
	2	660	**740**	820	IPR	0.002	0.002	890	**980**	1080	IPR	0.003	0.006
	3	560	**620**	690	IPR	0.002	0.002	520	**570**	620	IPR	0.003	0.006
	4	820	**900**	980	IPR	0.003	0.004	890	**980**	1080	IPR	0.003	0.006
	5	890	**980**	1080	IPR	0.005	0.005	820	**900**	980	IPR	0.004	0.008
	6	560	**620**	690	IPR	0.002	0.002	300	**330**	360	IPR	0.004	0.008
S	1	200	**260**	330	IPR	0.002	0.002	230	**300**	390	IPR	0.002	0.003
	2	160	**210**	260	IPR	0.001	0.002	160	**200**	260	IPR	0.001	0.002
	3	160	**210**	260	IPR	0.001	0.002	160	**200**	260	IPR	0.001	0.002
	4	160	**210**	260	IPR	0.001	0.002	160	**200**	260	IPR	0.001	0.002



Drill, Chamfer, and Thread Mill TM731

Material Group		Cutting Speed — vc Range — m/min			Drilling Recommended Feed by Diameter				Milling Feed/Tooth by Diameter			
		min	Starting Value	max		<6mm	6–10mm	10–16mm		<6mm	6–10mm	10–16mm
K	1	130	**175**	230	mm/r	0,10	0,16	0,30	mm/r	0,05	0,07	0,10
N	2	270	**300**	330	mm/r	0,15	0,25	0,34	mm/r	0,06	0,08	0,12
	3	140	**150**	170	mm/r	0,15	0,25	0,34	mm/r	0,06	0,08	0,12
	4	270	**300**	330	mm/r	0,15	0,25	0,34	mm/r	0,06	0,08	0,12
	5	110	**120**	130	mm/r	0,12	0,20	0,32	mm/r	0,06	0,08	0,12

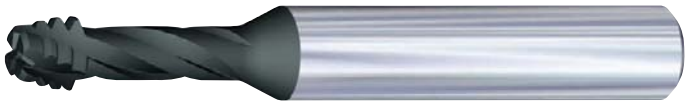


Drill, Chamfer, and Thread Mill TM731

Material Group		Cutting Speed — vc Range — SFM			Drilling Recommended Feed by Diameter				Milling Feed/Tooth by Diameter			
		min	Starting Value	max		<0.250"	0.250–0.375"	0.375–0.625"		<0.250"	0.250–0.375"	0.375–0.625"
K	1	430	**570**	750	IPR	0.004	0.006	0.012	IPR	0.002	0.003	0.004
N	2	890	**980**	1080	IPR	0.006	0.010	0.013	IPR	0.002	0.003	0.005
	3	460	**490**	560	IPR	0.006	0.010	0.013	IPR	0.002	0.003	0.005
	4	890	**980**	1080	IPR	0.006	0.010	0.013	IPR	0.002	0.003	0.005
	5	360	**390**	430	IPR	0.005	0.008	0.013	IPR	0.002	0.003	0.005

Mill, Chamfer, and Thread Mill TM741

Material Group		TM Style	Grade	Cutting Speed — vc Range — m/min			Feed/Tooth by Diameter		
				min	Starting Value	max		<10mm	>10mm
P	1	TM741 R	KCU36	170	225	290	mm/r	0,05	0,08
	2	TM741 R	KCU36	170	225	290	mm/r	0,05	0,08
	3	TM741 R	KCU36	120	150	200	mm/r	0,03	0,05
	4	TM741 R	KCU36	100	125	160	mm/r	0,03	0,05
	5	TM741 R	KCU36	120	150	200	mm/r	0,03	0,04
	6	TM741 R	KCU36	60	80	100	mm/r	0,03	0,04
M	1	TM741 R	KCU36	120	150	200	mm/r	0,03	0,04
	2	TM741 R	KCU36	120	150	200	mm/r	0,03	0,04
	3	TM741 R	KCU36	120	150	200	mm/r	0,03	0,04
K	1	TM741 R	KCU36	190	250	330	mm/r	0,06	0,10
	2	TM741 R	KCU36	190	250	330	mm/r	0,06	0,10
	3	TM741 R	KCU36	140	185	240	mm/r	0,04	0,07
N	1	–	–	–	–	–	–	–	–
	2	–	–	–	–	–	–	–	–
	3	TM741 R	KCU36	180	230	300	mm/r	0,06	0,07
	4	TM741 R	KCU36	210	275	360	mm/r	0,06	0,07
	5	–	–	–	–	–	–	–	–
	6	TM741 R	KCU36	210	275	360	mm/r	0,06	0,07
S	1	TM741 L	KCU36	120	150	200	mm/r	0,025	0,045
	2	TM741 L	KCU36	50	60	80	mm/r	0,015	0,025
	3	TM741 L	KCU36	50	60	80	mm/r	0,015	0,025
	4	TM741 L	KCU36	70	90	120	mm/r	0,025	0,035
H	1	TM741	KCU36	80	100	130	mm/r	0,030	0,050
	2	TM741	KCU36	80	100	130	mm/r	0,030	0,050
	3	TM741	KCU36	50	65	80	mm/r	0,020	0,030
	4	TM741	KCU36	50	65	80	mm/r	0,020	0,030



Mill, Chamfer, and Thread Mill TM741

Material Group		TM Style	Grade	Cutting Speed — vc Range — SFM			Feed/Tooth By Diameter		
				min	Starting Value	max		<0.375"	>0.375"
P	1	TM741 R	KCU36	560	740	950	IPR	0.002	0.003
	2	TM741 R	KCU36	560	740	950	IPR	0.002	0.003
	3	TM741 R	KCU36	390	490	660	IPR	0.001	0.002
	4	TM741 R	KCU36	330	410	520	IPR	0.001	0.002
	5	TM741 R	KCU36	390	490	660	IPR	0.001	0.002
	6	TM741 R	KCU36	200	260	330	IPR	0.001	0.002
M	1	TM741 R	KCU36	390	490	660	IPR	0.001	0.002
	2	TM741 R	KCU36	390	490	660	IPR	0.001	0.002
	3	TM741 R	KCU36	390	490	660	IPR	0.001	0.002
K	1	TM741 R	KCU36	620	820	1080	IPR	0.002	0.004
	2	TM741 R	KCU36	620	820	1080	IPR	0.002	0.004
	3	TM741 R	KCU36	460	610	790	IPR	0.002	0.003
N	1	–	–	–	–	–	–	–	–
	2	–	–	–	–	–	–	–	–
	3	TM741 R	KCU36	590	750	980	IPR	0.002	0.003
	4	TM741 R	KCU36	690	900	1180	IPR	0.002	0.003
	5	–	–	–	–	–	–	–	–
	6	TM741 R	KCU36	690	900	1180	IPR	0.002	0.003
S	1	TM741 L	KCU36	390	490	660	IPR	0.001	0.002
	2	TM741 L	KCU36	160	200	260	IPR	0.001	0.001
	3	TM741 L	KCU36	160	200	260	IPR	0.001	0.001
	4	TM741 L	KCU36	230	300	390	IPR	0.001	0.001
H	1	TM741	KCU36	260	330	430	IPR	0.001	0.002
	2	TM741	KCU36	260	330	430	IPR	0.001	0.002
	3	TM741	KCU36	160	210	260	IPR	0.001	0.001
	4	TM741	KCU36	160	210	260	IPR	0.001	0.001

NOTE: For thread depths over 2 x D up to 3 x D, reduce speed and feed by 25%.

Milling Methods

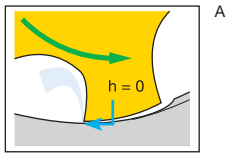
Climb Milling

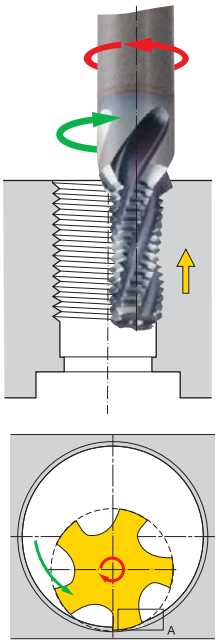
Properties:

- Tool rotation direction clockwise
- Tool moves counter-clockwise
- Pitch “upwards”

- Right-hand thread

Climb milling is always when the cutting edge goes out of the material with a chip thickness h = 0





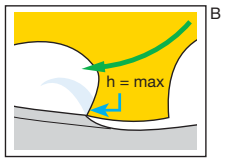
Conventional Milling

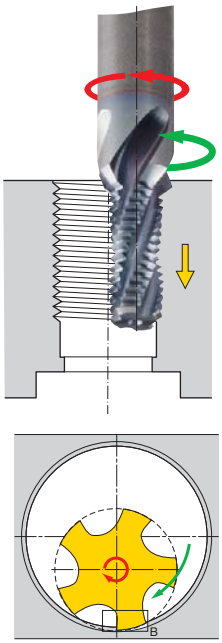
Properties:

- Tool rotation direction clockwise
- Tool moves clockwise
- Pitch “downwards”

- Right-hand thread

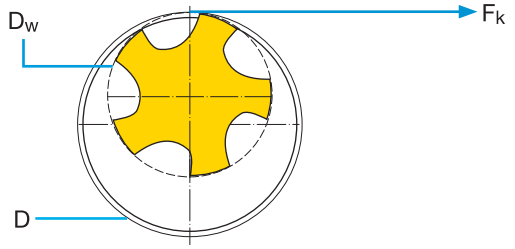
Conventional milling is always when the cutting edge goes out of the material with a chip thickness h = max





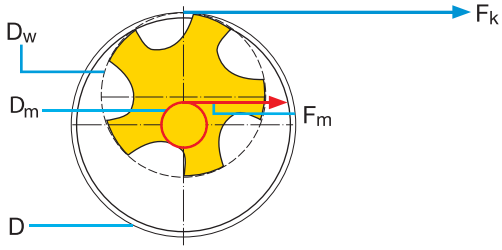
Counter Feed F_k

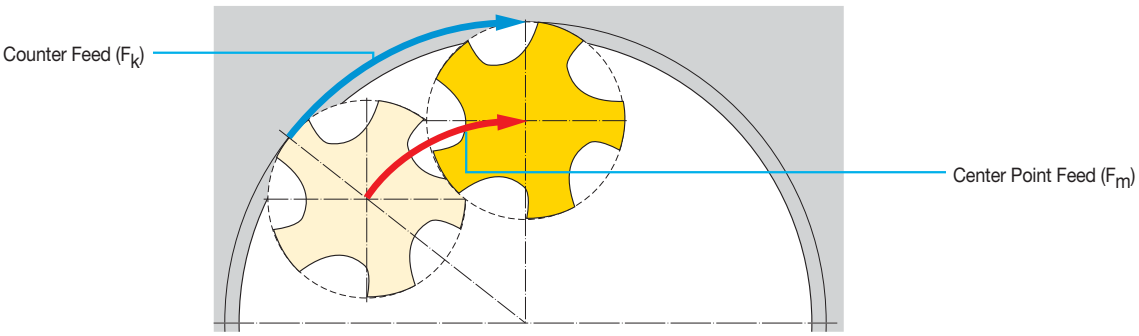
$$F_k = n \bullet f_z \bullet Z \text{ [mm/min]}$$



Center Point Feed F_m

$$F_m = \frac{F_k \bullet (D-Dw)}{D} \text{ [mm/min]}$$



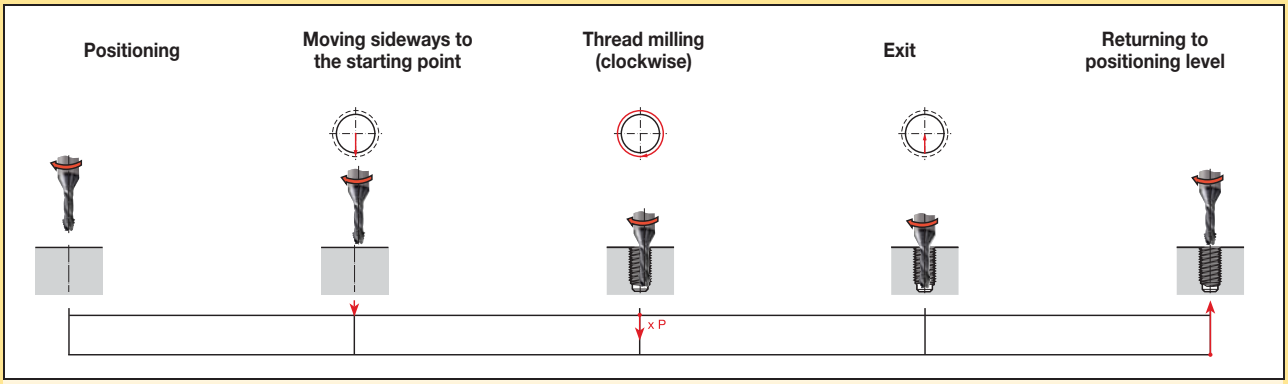


D_W = Tool diameter [mm]
n = RPM [min^{-1}]
f_Z = Feed per tooth [mm]
Z = Number of teeth on tool (radial)
D = Nominal diameter of thread = Diameter of external contour [mm]
D_m = Diameter of the center point (D-D_W) [mm]

Drill Thread Mill TM741 • Right Hand

Preparation	None
Process Principle	Milling thread and core hole, countersinking (conventional milling)

Cycle



Required Specification Values

Example

Size — M10-6H
Thread diameter D 10mm
Pitch P 1,5mm
Core hole diameter D_1 ..... 8,5mm

Material — Hard steel, 50 HRC
Grade — KCU36

Tool — TM741 Right Hand
Catalog number TM741M100X150R2DHA
Number of teeth Z 4
Tool diameter d_1 ..... 7,75mm*
Tool radius compensation k[1] 0,08mm**
Tool radius to be programmed[2] 3,795mm***
Thread depth b[3] 20mm
Cutting speed v_c ..... 100 m/min
Feed (milling) f_z ..... 0,04 mm/tooth
Number of turns [5] 17

$N = \frac{v_c \bullet 1000}{d_1 \bullet \pi}$ **S = 4109**

$v_f = f_z \bullet Z \bullet n$ **F = 657 (contour)**

$N = \frac{v_f \text{ contour} \bullet (D-d_1)}{D}$ **F = 148 (center point)**

*(measured on the cutting part) **(0.01 x D; adjust to application) ***(1/2 d_1 - k)

Program to DIN 66025 (conventional milling, on the contour, incremental)

Positioning the tool	N 10	G 54	G 90	G 00	X…	Y…	Z 1.500	S 4109	T01 [2]	M03
Incremental programming	N 20	G 91								
Moving sideways to the starting point	N 30	G 42	G 01	X 0	Y-5	F 657 (contour)		[F 148] [4] (center point)		
Thread milling	N 40	G 02		X 0	Y 0	Z-1.500	I 0	J 5.000		
Repeat thread milling	… [5]									
Exit	N 50	G 40	G 01	X 0	Y 5					
Retracting tool to positioning level	N 70	G 90	G 00	Z 2						

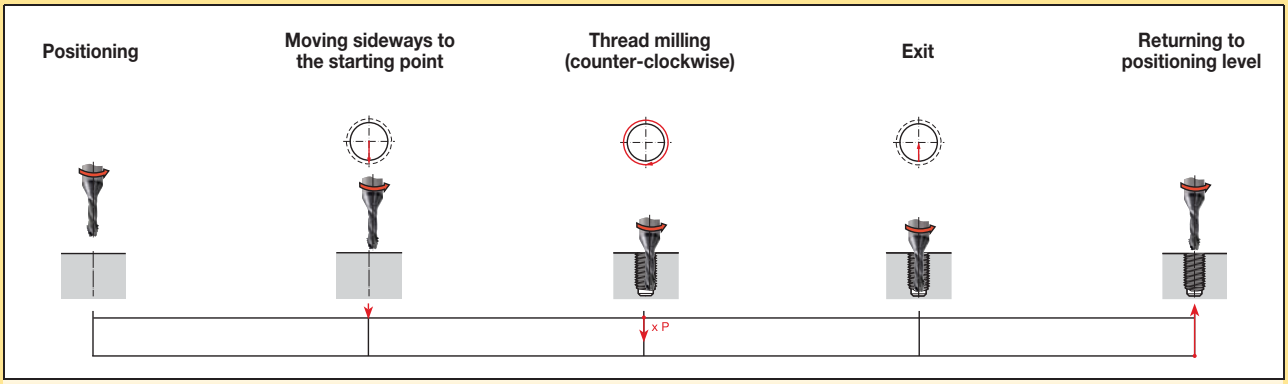
Cutting time t_h	51.6 seconds

NOTES:
[1] The cutter radius measured over the tooth crests of the threaded part must be reduced by the amount of the cutter radius compensation. This is necessary to achieve a depth of cut to the middle of the 6H/ISO2 nut tolerance. Please note, however, that this also depends on the radial deflection of the tool (tensile strength of the material, projecting length of the tool).
[2] The cutter radius to be programmed is normally included in the tool memory.
[3] The thread depth b must be divisible by the thread pitch P.
[4] The feed values in brackets must be used for controllers, which do not calculate the center point feed themselves.
[5] **Set N40 must be repeated with the number of threads.** Repetitions N = thread depth b/pitch P (rounded up to the nearest integer).

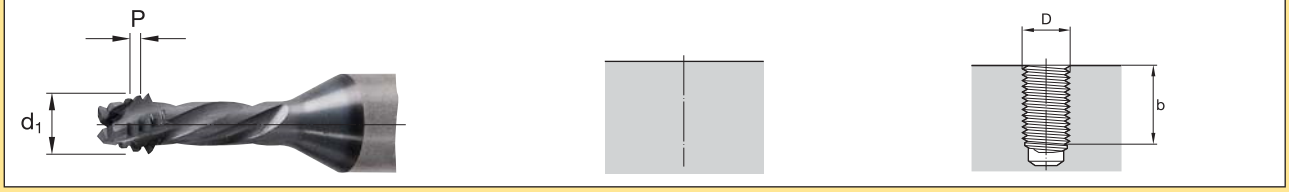
Drill Thread Mill TM741 • Left Hand

Preparation	None
Process Principle	Milling thread and core hole, countersinking (climb milling)

Cycle



Required Specification Values



Example

Size — M10-6H
Thread diameter D 10mm
Pitch P 1,5mm
Core hole diameter D_1 ..... 8,5mm

Material — TiAl6V4 titanium
Grade — KCU36

Tool — TM741 Left Hand
Catalog number TM741M100X150L2DHA
Number of teeth Z 4
Tool diameter d_1 ..... 7,75mm*
Tool radius compensation k[1] 0,08mm**
Tool radius to be programmed[2] 3,795mm***
Thread depth b[3] 20mm
Cutting speed v_c ..... 100 m/min
Feed (milling) f_z ..... 0,03 mm/tooth
Number of turns [5] 17

$N = \frac{V_c \bullet 1000}{d_1 \bullet \pi}$ **S = 4109**

$v_f = f_z \bullet Z \bullet n$ **F = 493 (contour)**

$v_f = \frac{v_f \text{ contour} \bullet (D - d_1)}{D}$ **F = 111 (center point)**

*(measured on the cutting part) **(0.01 x D) ***(1/2 d_1 - k)

Program to DIN 66025 (climb milling, on the contour, incremental)

Positioning the tool	N 10 G 54 G 90 G 00 X… Y… Z 1.500 S 4109 T01 [2] M04
Incremental programming	N 20 G 91
Moving sideways to the starting point	N 30 G 42 G 01 X 0 Y-5 F 493 (contour) [F 111] [4] (center point)
Thread milling	N 40 G 02 X 0 Y 0 Z-1.500 I 0 J 5.000
Repeat thread milling	… [5]
Exit	N 50 G 40 G 01 X 0 Y 5
Retracting tool to positioning level	N 70 G 90 G 00 Z 2

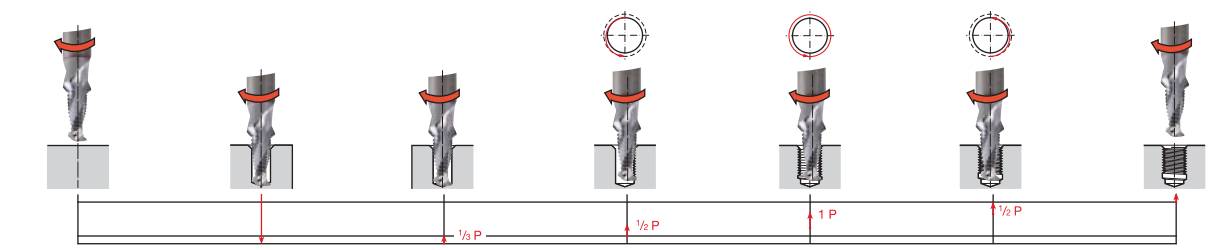
Cutting time t_h	**68.8 seconds**

NOTES:
[1] The cutter radius measured over the tooth crests of the threaded part must be reduced by the amount of the cutter radius compensation. This is necessary to achieve a depth of cut to the middle of the 6H/ISO2 nut tolerance. Please note, however, that this also depends on the radial deflection of the tool (tensile strength of the material, projecting length of the tool).
[2] The cutter radius to be programmed is normally included in the tool memory.
[3] The thread depth b must be divisible by the thread pitch P.
[4] The feed values in brackets must be used for controllers, which do not calculate the center point feed themselves.
[5] **Set N40 must be repeated with the number of threads.** Repetitions N = thread depth b/pitch P (rounded up to the nearest integer).

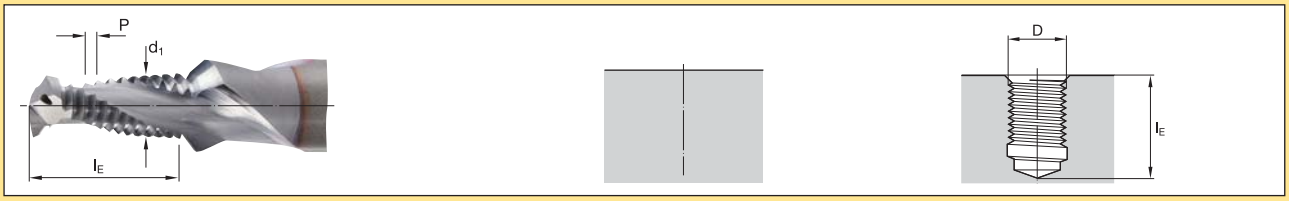
Drill Thread Mill TM731

Preparation	None
Process Principle	Drilling, countersinking, thread milling (climb milling)

Cycle



Required Specification Values



Example

Size — M10-6H
Thread diameter D 10mm
Pitch P 1,5mm
Core hole diameter D_1 ..... 8,5mm

Material — Gray cast iron
Grade — KCU32

Tool — TM731
Catalog number TM731M100X150R2DHA
Number of teeth Z 2
Tool diameter d_1 ..... 8,2mm*
Tool radius compensation k[1] 0,1mm**
Tool radius to be programmed[2] 4mm***
Dilling/Countersink depth l_E 19,11mm
Cutting speed v_c ..... 250 m/min
Feed (drilling, countersinking) f_b ..... 0,25 mm/U
Feed (milling) f_z ..... 0,1 mm/tooth

*(measured on the cutting part) **(0.01 x D) ***(1/2 d_1 - k)

$N = \frac{v_c \bullet 1000}{d_1 \bullet \pi}$ **S = 9709**

$v_b = f_b \bullet n$ **F = 2427 (drilling, countersinking)**

$v_f = f_z \bullet Z \bullet n$ **F = 1942 (contour)**

$v_f = \frac{v_f \text{ contour} \bullet (D - d_1)}{D}$ **F = 350 (center point)**

Program to DIN 66025 (climb milling, on the contour, incremental)

Positioning the tool	N 10 G 54 G 90 G 00 X… Y… Z 2 S 9709 T01 [2] M03
Drilling and countersinking	N 20 G 91 G 01 Z-21.110 F 2427 (drill, countersink)
Raise	N 30 G 01 Z 0.500
Moving sideways to the starting point	N 40 G 41 Y-4.250 F 971 (milling, 1/2 contour) [F 175] [3] (1/2 center point)
Run-in loop in arc	N 50 G 03 X 0 Y 9.250 Z 0.750 I 0 J 4.625
Thread milling	N 60 G 03 X 0 Y 0 Z 1.500 I 0 J -5.000
Run-out loop in arc	N 70 G 03 X 0 Y-9.250 Z 0.750 I 0 J- 4.625 F1942 [F 350] [3] (center point)
Exit	N 80 G 00 G 40 X 0 Y 4.250
Retracting tool to positioning level	N 90 G 90 Z 2

Cutting time t_h

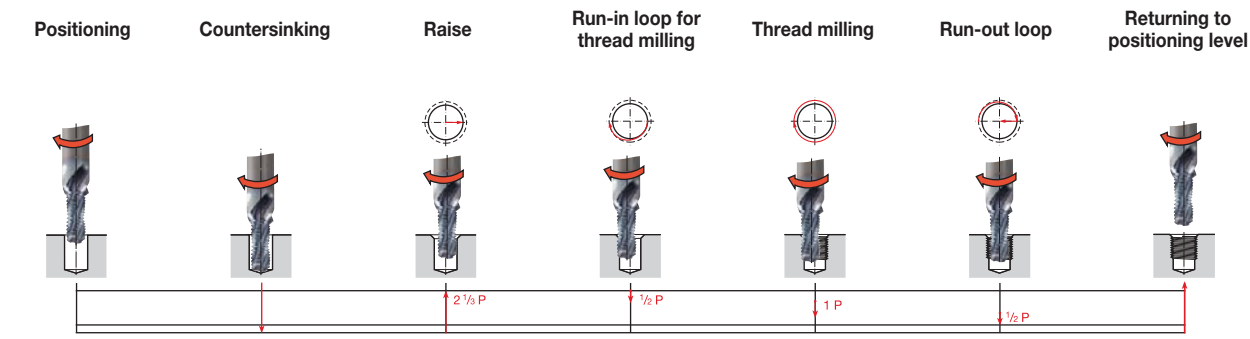
2.3 seconds

NOTES:
[1] The cutter radius measured over the tooth crests of the threaded part must be reduced by the amount of the cutter radius compensation. This is necessary to achieve a depth of cut to the middle of the 6H/ISO2 nut tolerance. Please note, however, that this also depends on the radial deflection of the tool (tensile strength of the material, projecting length of the tool).
[2] The cutter radius to be programmed is normally included in the tool memory.
[3] The feed values in brackets must be used for controllers, which do not calculate the center point feed themselves.

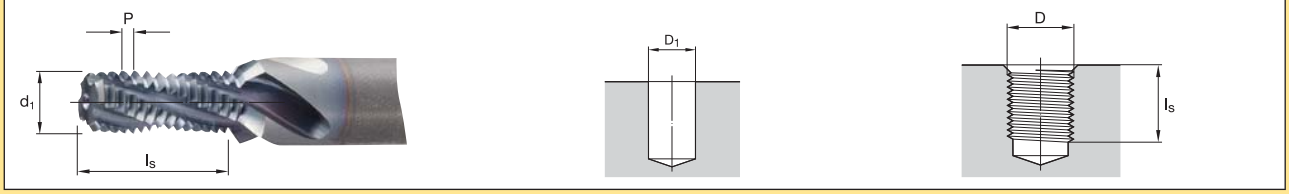
Thread Mill TM721

Preparation	Drilling of thread hole
Process Principle	Countersinking, thread milling (conventional milling)

Cycle



Required Specification Values



Example

Size — M10-6H
Thread diameter D 10mm
Pitch P 1,5mm
Core hole diameter D_1 ..... 8,5mm

Material — Cast aluminum
Grade — KCU32

Tool — TM721
Catalog number TM721M100X150R2DHA
Number of teeth Z 3
Tool diameter d_1 ..... 8,2mm*
Tool radius compensation k[1] 0,1mm**
Tool radius to be programmed[2] 4mm***
Countersink depth l_s 21,2mm
Cutting speed v_c ..... 250 m/min
Feed (countersinking) f_s ..... 0,3 mm/U
Feed (milling) f_z ..... 0,09 mm/tooth

*(measured on the cutting part) **(0.01 x D) ***(1/2 d_1 - k)

$$N = \frac{V_c \bullet 1000}{d_1 \bullet \pi} \quad S = 9709$$

$$v_s = f_s \bullet n \quad F = 2913 \text{ (countersinking)}$$

$$v_f = f_z \bullet Z \bullet n \quad F = 2622 \text{ (contour)}$$

$$v_f = \frac{v_f \text{ contour} \bullet (D-d_1)}{D} \quad F = 472 \text{ (center point)}$$

Program to DIN 66025 (conventional milling, on the contour, incremental)

Positioning the tool	N 10 G 54 G 90 G 00 X… Y… Z 2 S 9709 T01 [2] M03
Advancing tool to full thread depth	N 20 G 91 Z-21.200
Countersinking	N 30 G 01 Z-2 F 2913 (countersink)
Raise	N 40 G 00 Z 3.450
Moving sideways to the starting point	N 50 G 42 G01 X 4.250 F 1311 (milling, 1/2 contour) [F 236] [3] (milling,1/2 center point)
Run-in loop in arc	N 60 G 02 X-9.25 Y 0.000 Z-0.750 I-4.625 J 0
Thread milling	N 70 G 02 X 0 Y 0 Z-1.500 I 5 J 0.000 F2622 [F 472] [3] (center point)
Run-out loop in arc	N 80 G 02 X 9.25 Y 0.000 Z-0.750 I 4.625 J 0
Exit	N 90 G 40 G 01 X-4.25
Retracting tool to positioning level	N 100 G 90 G 00 Z 2

Cutting time t_h	**1.4 seconds**

NOTES:
[1] The cutter radius measured over the tooth crests of the threaded part must be reduced by the amount of the cutter radius compensation. This is necessary to achieve a depth of cut to the middle of the 6H/ISO2 nut tolerance. Please note, however, that this also depends on the radial deflection of the tool (tensile strength of the material, projecting length of the tool).
[2] The cutter radius to be programmed is normally included in the tool memory.
[3] The feed values in brackets must be used for controllers, which do not calculate the center point feed themselves.