THREAD MILL PROGRAMMING

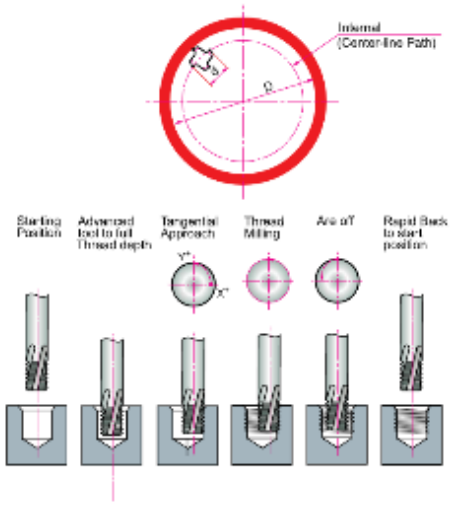
Programme data.

G Codes for Thread Milling

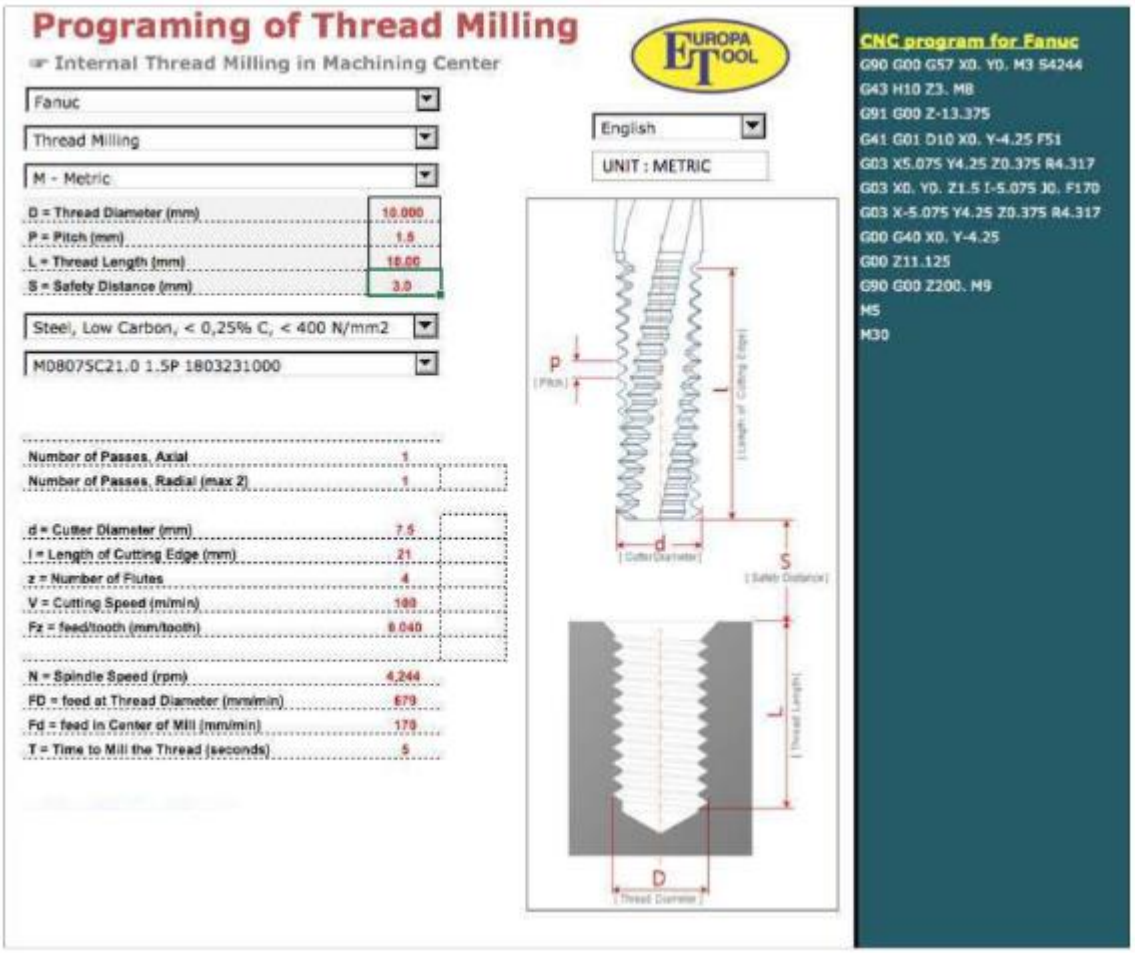
G00	Fast Feed Linear	G41	Cutter Radius Compensation Left	M08	Coolant On
G01	Linear Movement	G42	Cutter Radius Compensation Right	X	Horizontal Co-ordinate
G02	Circular/Helical Interpolation C.W.	G43	Tool Length Compensation Plus	Y	Horizontal Co-ordinate
G03	Circular/Helical Interpolation A.C.W.	G49	Tool Length Compensation Cancel	Z	Vertical Co-ordinate
G17	X, Y Plane(Vertical Machining)	G90	Absolute Command	I	X Co-ordinate to Center of Arc Travel
G18	Z, X Plane(Horizontal Machining)	G91	Incremental Command	J	Y Co-ordinate to Center of Arc Travel
G19	Y, Z Plane(Using 90 Head)	M03	Clockwise Rotation of Spindle	S	Spindle Speed R.P.M.
G40	Cutter Radius Compensation Cancel	M05	Spindle stop	F	Feed mm/min

CNC Internal Thread Milling

G90	G00	X···	Y···	Z5	S···
G91	G00	Z···(A3+2)			
Y··(A5)					
G41	G01	X···(A6)	F···		
G03	X···(A6)	Y···(A6)	Z···(A4)	I···(A6)	J0
G03	X0	Y0	Y··(A2)	I0	J···(A1)
G03	X···(A6)	Y···(A6)	Z···(A4)	I0	J···(A6)
G00	G40	X···(A6)	Y··(A5)		
G90	Z5				



Explanation of Parameters

A1 : 1/2 Nominal Thread Diameter 1/2D
A2 : Thread Pitch
A3 : Thread Depth
A4 : 1/4P(for climb milling and right-hand thread)
A5 : Beginning of Contour in Y 0.5xP
A6 : Arc Off (A1 - A5)

Download our thread mill programme generator.
Visit http://www.europatool.co.uk/threading/threadmills