3.5High-speed settings – CYCLE832

To simplify programming and clarify program structure, Sinumerik 840D includes CYCLE832, which contains the most important functions for freeform surface milling.
 CYCLE832 also makes it easier for the machine operator to influence the program.

Programming

CYCLE832(_TOL,_TOLM)	Programming the cycle
CYCLE832()	Abbreviated program call. Corresponds to selecting the "Machining" screen form "Deselection".
CYCLE832(0.01)	Abbreviated program call. To enter the tolerance value. The active G commands are not changed in the cycle.

Explanation of the parameters

_TOL	real	Tolerance on machining axes -> unit: mm/inches; degrees	
_TOLM 7	integer	Mode tolerance	
		Decimal place [2]	Input
		0	0 = Deselection 1 = Finishing (default)[1] 2 = Prefinishing 3 = Roughing
		1	0 = 1 =
		2	0 = TRAFOF (default)[1] 1 = TRAORI(1) 2 = TRAORI(2)
		3	0 = G64 1 = G641 2 = G642 (default)[1] 3 = G643 4 = G644
		4	0 FFWOF SOFT (default)[1] 1 FFWON SOFT 2 FFWOF BRISK
		5	0 = COMPOF 1 = COMPCAD (default)[1] 2 = COMPCURV 3 = B spline
		6	Reserved
		7	Reserved

[1] Setting can be changed by machine manufacturer.
[2] Sequence of parameters (CYCLE832(_TOL,76543210)