vhf camfacture AG ■ Im Marxle 3 ■ 72119 Ammerbuch ■ Tel. +49 (0)70 32 970 97-0 ■ Fax: -50 ■ www.vhf.eu

Parameters for Thread Cutters

General:
This carbide thread cutter is designed for cutting metric internal threads, but it can be used with certain restrictions also for external threads. The teeth of this tool have logarithmical relief grinding.

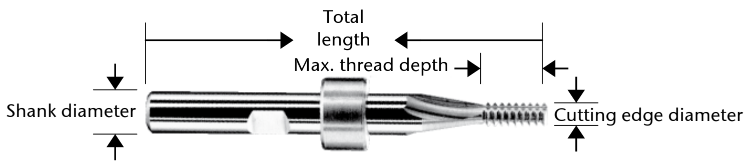
Fields of Application:
- Thread in drill holes in metal or plastics ready after one circulation
- Cutting threads into tapped blind holes
- For the production of normal threads

Hints for Machining:
- In any case, a core hole has to be drilled before cutting threads
- Drilling of the bore with an additional allowance of 0.05 to 0.10 mm in radius (0.10 to 0.20 mm in diameter). The inside of the thread is slightly topped in this procedure. Correct the tool radius inwards.
- When you cut threads in non-ferrous metals, you should work with liquid cooling – just as if you were milling.
- The thread cutter can be used only with a fully 3D capable controller (e.g. all vhf models from CNC 550 on).
- Enter the technical data (please turn over) of each tool that you use into the Tool Magazine of the production software Cenon (click on the button *Parameters* on the *Magazine* page of the *CAM Panel*) and set the tool type to *Thread Cutter*. For other software packages you have to carry out similar steps.

Thread cutting in only one operation cycle

May 2010

Thread Cutters



Please note...

In order to achieve best results with your new precision tool for a long time, you should set your tool parameters according to the following data. However, whether it is actually possible to work with these theoretically determined values for feed rate and rotational speed depends on the interaction of a number of factors. Thus we cannot assume liability for the calculated values. Among others, the following factors determine the machining process:
- additional cooling with compressed air or lubrication?
- power/maximum rotational speed of the spindle
- minimum/maximum feed rate of the CAM system
- controller features (look-ahead path calculation, etc.)
- stiffness of the machine.

Please turn over ⇨

Technical Data

Article No.	Cutting Edge Diameter (d)	Shank Diameter	Diameter Core Hole	Cutting Edge Length	Number of Teeth	Total Length	Thread Type	Pitch [mm]
GSF-M03	2.1 mm	3.0 mm	2.5 mm	4.5 mm	3	38 mm	M 3	0.50
GSF-M04	2.6 mm	3.0 mm	3.3 mm	6.3 mm	3	38 mm	M 4	0.70
GSF-M05	3.6 mm	4.0 mm	4.2 mm	6.3 mm	3	42 mm	M 5	0.80
GSF-M06	4.0 mm	6.0 mm	5.0 mm	9.0 mm	3	57 mm	M 6	1.00
GSF-M08	5.8 mm	6.0 mm	6.8 mm	9.0 mm	3	60 mm	M 8	1.25
GSF-M10	7.8 mm	8.0 mm	8.5 mm	12.0 mm	4	60 mm	M 10	1.50
GSF-M12	9.0 mm	10.0 mm	10.2 mm	13.5 mm	4	75 mm	M 12	1.75

No responsibility is taken for the correctness of this information!

Working Parameters

Material	Cutting Speed v_c [m/min]	Feed per Tooth f_z [mm]	Cooling
Steel	75–250	0.005–0.01 × d	Emulsion
Stainless Steel	40–75	0.002–0.05 × d	Emulsion
Cast Iron	40–120	0.005–0.01 × d	Air/Emulsion
Titanium	30–45	0.002–0.005 × d	Emulsion
Copper	100–200	0.005–0.012 × d	Emulsion
Brass	150–400	0.005–0.012 × d	Emulsion
Aluminium	200–300	0.007–0.012 × d	Lubricant/Emulsion
Plastics	200–300	0.007–0.015 × d	Compressed air

No responsibility is taken for the correctness of this information!

Formulas

Rotational Speed: $n\ [\text{rev/min}] = \dfrac{v_c\ [\text{m/min}] \times 1000}{\pi \times d\ [\text{mm}]}$

Feed Rate: $v_f\ [\text{mm/min}] = f_z\ [\text{mm}] \times z \times n\ [\text{rev/min}]$

Variablen:

n	rotational speed	v_f	feed rate
v_c	cutting speed	f_z	feed per tooth
d	cutting edge diameter	z	number of cutter teeth

Hints for the calculation:

Which values should be used for the calculation?
In general, you should approximately choose the average value of the given range for the cutting speed and the lower value for the feed per tooth. So you will work with rather high rotational speeds and low feed rates. Then you can approximate slowly to the optimum values for your given conditions. Please insert the rotational speed with which you actually work into the formula for calculating the feed rate in case it should deviate from the rotational speed which you have calculated before.

Sample calculation:
You want to mill aluminium with a thread cutter with a cutting edge diameter of 2.6 mm and the thread type M4.

Rotational Speed: $\dfrac{200\ \text{m/min} \times 1000}{3.14 \times 2.6\ \text{mm}} = 24{,}485\ \text{U/min}$

Feed rate: 0.007 mm × 2.6 mm × 3 × 24,485 U/min = 1,336.8 mm/min = 22.28 mm/s