Metric Single Form Thread Milling Guide

Single Form Threadmills are the most versatile threading tool due to their ability to mill multiple pitch sizes. Since they are used in a helical interpolation environment, specific machining parameters are needed to avoid deflection and breakage. These tools can be used successfully in materials ranging from Aluminum to Steels.

Speeds & Feeds calculations:

1. Determine the correct SFM and Chip Load (IPT) for the cutter and material
2. Calculate the Speed (RPM) and Linear Feed (IPM)
3. Adjust Linear Feed to account for helical interpolation of internal or external threads
4. Determine correct number of radial passes at full axial depth

Example: Tool #771624-C6 to machine an M3-0.50 internal thread in 17-4 stainless steel

1. From Speeds & Feeds chart (next page), SFM is 130 and Chip Load (IPT) is .00017
2. Calculate Speed (RPM) and Linear Feed (IPM)

   RPM = (SFM x 3.82) / Cutter Diameter
   = (130 x 3.82) / .091
   = 5457 RPM

   Linear Feed (IPM) = RPM x IPT x Number of Flutes
   = 5457 x .00017 x 4
   = 3.71 in/min

3. Adjust Linear Feed (use Table 1 to determine Major Thread Diameter)

   Adj Internal Feed = [(Major Thread Dia - Cutter Dia) / Major Thread Dia] x Linear Feed
   = [(.118 - .091) / .118] x 3.71
   = .85 in/min

   Adj External Feed = [(Major Thread Dia + Cutter Dia) / Major Thread Dia] x Linear Feed
   = [(.118 + .091 / .118] x 3.71
   = 6.57 in/min

4. Determine Number of Radial Passes using the Speeds & Feeds chart (next page).

   For steels with a hadness of 45-55 HRc, use 3-4 Radial Passes

   (Note: The number of passes should be based on the thread size of the tool, and not the machined part)

5. Conclusion

   In this example, the tool would run at 5457 RPM, .85 IPM and make 3-4 Radial Passes

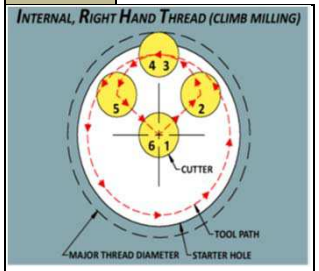
Setup & Use:

1. Check software and input proper feed values (Linear or Adjusted)
2. Choke up on tool
3. Minimize runout (consider entire system of spindle, collet, holders etc)
4. Minimize all vibration (consider tool holding, work holding, rpm "sweet spot" etc)
5. Break in tool by reducing feed rates by 25% on first 1-2 holes
6. Cutter should engage part using an arcing toolpath to avoid shock loading (see Table 2)
7. Climb mill for best finish and tool life (see Table 2)
8. Flush chips with coolant to avoid recutting

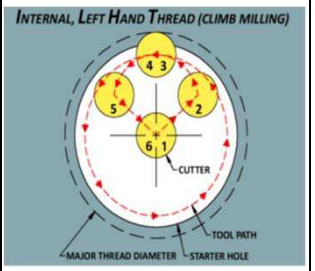
Table 1

Tool Thread Size	Major Thread Diameter	
	Millimeters	Inches
M1.6	1.50	0.059
M2	2.00	0.079
M2.5	2.50	0.098
M3	3.00	0.118
M3.5	3.50	0.138
M4	4.00	0.157
M5	5.00	0.197
M6	6.00	0.236
M8	8.00	0.315
M10	10.00	0.394
M12	12.00	0.472
M14	14.00	0.551
M15	15.00	0.591
M16	16.00	0.630
M17	17.00	0.669
M18	18.00	0.709
M20	20.00	0.787
M22	22.00	0.866
M24	24.00	0.945

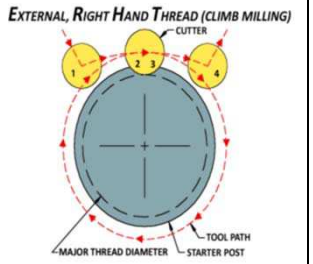
Table 2



Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while "Z" feeds up from bottom
Step 3-4: Cutter moves helically
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center

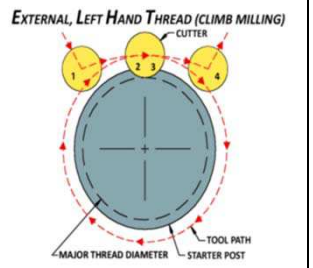


Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while "Z" feeds down from top
Step 3-4: Cutter moves helically
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center



Step 1-2: Cutter engages part with arcing tool path while "Z" feeds down from top
Step 2-3: Cutter moves helically
Step 3-4: Cutter exits part along arcing tool path while maintaining "Z" feed

---



Step 1-2: Cutter engages part with arcing tool path while "Z" feeds up from bottom
Step 2-3: Cutter moves helically
Step 3-4: Cutter exits part along arcing tool path while maintaining "Z" feed

Speeds & Feeds

Product Table: Thread Milling Cutters - Single Form - Metric - For Hardened Steels
Characteristics: 5x Reach Multiple
Series: 7727xx-C6

Product Notes:

Recommended Depths of Cut (Radial Passes) are based on the coarsest pitch by thread size. For finer pitches, the number of passes may be reduced by 1 pass.

General notes:

All posted speed and feed parameters are suggested starting values that may be increased given optimal setup conditions.

If you require additional information, Harvey Tool has a team of technical experts available to assist you through even the most challenging applications. Please contact us at **800-645-5609** or **Harveytech@harveyperformance.com**.

WARNING: Cutting tools may shatter under improper use. Government regulations require use of safety glasses and other appropriate safety equipment in the vicinity of use.

Material	Hardness (HRc)	SFM	Chip Load (IPT) By Cutter Diameter												Depth of Cut
			0.047	0.062	0.078	0.093	0.125	0.187	0.250	0.312	0.375	0.500	0.625	0.750	Radial Passes
Hardened Steels	46-55	130	.00007	.00009	.00012	.00014	.00019	.00029	.00038	.00064	.00077	.00102	.00128	.00153	3-4
	56-68	80	.00006	.00008	.00010	.00011	.00015	.00023	.00031	.00051	.00061	.00082	.00102	.00122	4-5