



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 #901210 to machine a 4-40 internal thread in 17-4 stainless steel

1. From Speeds & Feeds chart (next page), SFM is 150 and Chip Load (IPT) is .00011
2. Calculate Speed (RPM) and Linear Feed (IPM)

$$\begin{aligned} \text{RPM} &= (\text{SFM} \times 3.82) / \text{Cutter Diameter} \\ &= (150 \times 3.82) / .080 \\ &= 7162 \end{aligned}$$

$$\begin{aligned} \text{Linear Feed (IPM)} &= \text{RPM} \times \text{IPT} \times \text{Number of Flutes} \\ &= 7162 \times .00011 \times 2 \\ &= 1.57 \end{aligned}$$

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

$$\begin{aligned} \text{Adj Internal Feed} &= [(\text{Major Thread Dia} - \text{Cutter Dia}) / \text{Major Thread Dia}] \times \text{Linear Feed} \\ &= [(.112 - .080) / .112] \times 1.57 \\ &= .45 \end{aligned}$$

$$\begin{aligned} \text{Adj External Feed} &= [(\text{Major Thread Dia} + \text{Cutter Dia}) / \text{Major Thread Dia}] \times \text{Linear Feed} \\ &= [(.112 + .080) / .112] \times 3.15 \\ &= 2.69 \end{aligned}$$

4. Determine Number of Radial Passes using Table 1

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

For Easy Machinability = 2 Radial Pass at full Axial Depth
 For Moderate Machinability = 3 Radial Passes at full Axial Depth
 For Difficult Machinability = 4 Radial Passes at full Axial Depth

Definitions:

Easy Machinability materials include Non-Ferrous alloys and Leaded Steels
 Moderate Machinability materials include 200/300/400 Stainless Steels and Steels up to 35 Rc
 Difficult Machinability materials include Inconel, Titanium and Steels 36-45 Rc

5. Conclusion

In this example, the tool would run at 7162 RPM, .45 IPM and make 3 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	Number of Radial Passes*		
		Easy Machinability	Moderate Machinability	Difficult Machinability
00	0.047	2	3	4
0	0.060	2	3	4
1	0.073	2	3	4
2	0.086	2	3	3
3	0.099	2	3	3
4	0.112	2	3	4
5	0.125	2	3	3
6	0.138	2	3	4
8	0.164	2	2	3
10	0.190	2	3	4
12	0.216	2	2	3
1/4	0.250	2	2	3
5/16	0.312	2	2	3
3/8	0.375	2	2	3
7/16	0.437	2	2	3
1/2	0.500	2	2	3
9/16	0.562	2	2	3
5/8	0.625	2	2	3
3/4	0.750	2	2	3
7/8	0.875	2	2	3
1	1.000	2	3	4

* Number of Radial Passes are based on the coarsest pitch by thread size. For finer pitches, the number of passes may be reduced by 1 pass.

Table 2
<p>INTERNAL, RIGHT HAND THREAD (CLIMB MILLING)</p> <p>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</p>
<p>INTERNAL, LEFT HAND THREAD (CLIMB MILLING)</p> <p>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</p>
<p>EXTERNAL, RIGHT HAND THREAD (CLIMB MILLING)</p> <p>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</p>
<p>EXTERNAL, LEFT HAND THREAD (CLIMB MILLING)</p> <p>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</p>

