HARVEY TOOL

Tri-Form Threadmilling Guide for Hardened and Difficult Materials

Hardened steels and other tough exotic alloys can pose a big challenge when machining threads. A tri-form thread mill can help reduce tool deflection and machining errors in these materials. Their left hand cut, left hand spiral design allows the use of the tools in climb milling situations, while eliminating the need to arc-in when beginning to mill a thread.

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 #896428-C6 to machine a 8-32 internal thread in steel hardened to 52 HRC.

1. From Speeds & Feeds chart (next page), SFM is 130 and Chip Load (IPT) is .00026.
2. Calculate Speed (RPM) and Linear Feed (IPM)

   RPM = (SFM x 3.82) / Cutter Diameter
   = (130 x 3.82) / .126
   = 3941

   Linear Feed (IPM) = RPM x IPT x Number of Flutes
   = 3941 x .00023 x 3
   = 2.72

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
   = [(.164 - .126) / .164] x 2.72
   = .63

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

   For steels with a hardness of 46-55 HRc, use 3-4 Radial Passes.

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

5. Conclusion

   In this example, the tool would run at 3941 RPM, 0.63 IPM and make 3-4 Radial Passes

Table 1

Thread Size	Major Thread Diameter
0	0.060
1	0.073
2	0.086
3	0.099
4	0.112
5	0.125
6	0.138
8	0.164
10	0.190
12	0.216
1/4	0.250
5/16	0.312
3/8	0.375
7/16	0.437
1/2	0.500
9/16	0.562
5/8	0.625
3/4	0.750
7/8	0.875
1	1.000

Setup & Use:
1. Check software and input proper feed values (Linear or Adjusted)
2. Use code M04 for left-hand spindle rotation
3. Choke up on tool
4. Minimize runout (consider entire system of spindle, collet, holders etc)
5. Minimize all vibration (consider tool holding, work holding, rpm "sweet spot" etc)
6. Break in tool by reducing feed rates by 25% on first 1-2 holes
7. Climb mill from the top of the hole for best finish and tool life
8. Flush chips with coolant to avoid recutting

HARVEY TOOL

Speeds & Feeds

Product Table: Thread Milling Cutters - Tri-Form - UN Threads
Characteristics: Medium Reach
Series: 8964xx-C6

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 Carbon Steel, Stainless Steel, Tool Steel	38-45	180	.00010	.00013	.00016	.00019	.00026	.00038	.00051	.00085	.00102	.00136	.00170	.00204	3-4
	46-55	130	.00008	.00011	.00014	.00017	.00023	.00034	.00045	.00075	.00090	.00120	.00150	.00180	3-4
	56-68	80	.00007	.00009	.00011	.00013	.00018	.00027	.00036	.00060	.00072	.00096	.00120	.00144	4-5
Titanium Alloys		150	.00012	.00016	.00021	.00025	.00033	.00049	.00066	.00096	.00116	.00154	.00193	.00231	3-4
High Temp Alloys Inconel, Hastelloy, Waspalloy, Monel, Nimonic, Haynes, Discoloy, Incoloy		70	.00011	.00015	.00019	.00022	.00030	.00045	.00060	.00100	.00120	.00160	.00200	.00240	3-4