HARVEY TOOL

Single Form Threadmilling Guide for Hardened Steels

Threading in hardened steel is a very challenging application. 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.

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 #930330-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 .00016.
2. Calculate Speed (RPM) and Linear Feed (IPM)

   RPM = (SFM x 3.82) / Cutter Diameter
   = (130 x 3.82) / .120
   = 4138

   Linear Feed (IPM) = RPM x IPT x Number of Flutes
   = 4138 x .00016 x 4
   = 2.65

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 - .120) / .164] x 2.65
   = .71

   Adj External Feed = [(Major Thread Dia + Cutter Dia) / Major Thread Dia] x Linear Feed
   = [(.164 + .120) / .164] x 2.65
   = 4.59

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 4138 RPM, .71 IPM and make 3-4 Radial Passes

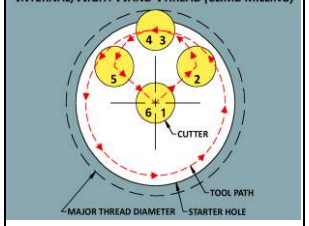
Setup & Use:
1. Check software and input proper feed values (Linear or Adjusted)
2. Choke up on tool
3. Minimize runout (consider entire system of spindle, collet, holders etc)
4. Minimize all vibration (consider tool holding, work holding, rpm "sweet spot" etc)
5. Break in tool by reducing feed rates 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

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

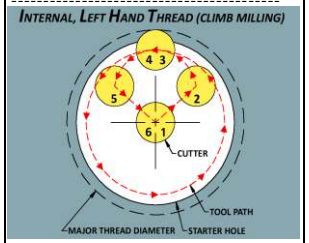
Table 2

INTERNAL, RIGHT HAND THREAD (CLIMB MILLING)



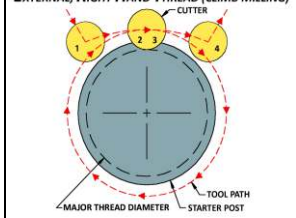
Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while "Z" feeds up from bottom
Step 3-4: Cutter moves helically
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center

INTERNAL, LEFT HAND THREAD (CLIMB MILLING)



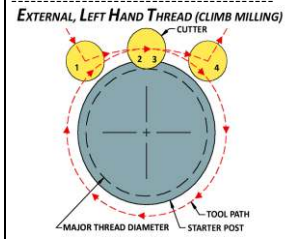
Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while "Z" feeds down from top
Step 3-4: Cutter moves helically
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center

EXTERNAL, RIGHT HAND THREAD (CLIMB MILLING)



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

EXTERNAL, LEFT HAND THREAD (CLIMB MILLING)



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

HARVEY TOOL

Speeds & Feeds

Product Table: Thread Milling Cutters - Single Form for Hardened Steels
Characteristics: Extra Long Reach
Series: 9303xx-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	.00006	.00008	.00010	.00012	.00016	.00024	.00032	.00052	.00063	.00084	.00105	.00126	3-4
	56-68	80	.00005	.00006	.00008	.00009	.00013	.00019	.00025	.00042	.00050	.00067	.00084	.00101	4-5