HARVEY TOOL

Multi-form Threadmilling Guide

Series: 8367xx-C6

Multi-Form Thread Mills are a fast way to threadmill a part. Since they use the entire length of cut 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 up to and including Hardened 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 #836764-C6 to machine a 3/8-16 internal thread in 56 - 60 Rc

1. From Speeds & Feeds chart (next page), SFM is 350 and Chip Load (IPT) is .00044"
2. Calculate Speed (RPM) and Linear Feed (IPM)

   RPM = (SFM x 3.82) / Cutter Diameter
   = (350 x 3.82) / .285
   = 4691

   Linear Feed (IPM) = RPM x IPT x Number of Flutes
   = 4691 x .00044 x 4
   = 8.2

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
   = [(.375 - .285) / .375] x 8.2
   = 1.9

   Adj External Feed = [(Major Thread Dia + Cutter Dia) / Major Thread Dia] x Linear Feed
   = [(.375 + .285) / .375] x 8.2
   = 14.4

4. Determine Number of descending Radial Passes (next page)

   Total stock being removed = (Major Thread Dia - Minor Thread Dia) / 2
   = (.375 - .307) / 2
   = .034

   1st pass is 21%, 2nd pass is 18%, 3rd pass is 16%, 4th pass is 14%,
   5th pass is 12%, 6th pass is 10%, 7th pass is 9%, 8th pass is a spring pass

5. Conclusion

   In this example, the tool would run at 4691 RPM, 1.9 IPM and make 8 Radial Passes

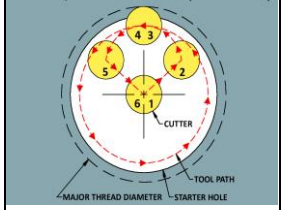
Setup & Use:
1. Check software and input proper feed values (Linear or Adjusted)
2. Choke up on tool
3. Minimize runout (consider entire system of spindle, collet, holders etc)
4. Minimize all vibration (consider tool holding, work holding, rpm "sweet spot" etc)
5. To break in the tool, reduce feed rates by 75% on the on the first one to two 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
0-80	0.060
1-64	0.073
2-56	0.086
3-48	0.099
4-40	0.112
5-40	0.125
5-44	0.125
6-32	0.138
8-32	0.164
8-36	0.164
10-24	0.190
10-28	0.190
10-32	0.190
12-24	0.216
12-28	0.216
1/4-20	0.250
1/4-28	0.250
5/16-18	0.312
5/16-24	0.312
3/8-16	0.375
3/8-24	0.375
7/16-14	0.437
7/16-20	0.437
1/2-13	0.500
1/2-20	0.500
9/16-12	0.562
9/16-18	0.562
5/8-11	0.625
5/8-18	0.625
3/4-10	0.750
3/4-12	0.750
3/4-16	0.750
7/8-9	0.875
7/8-14	0.875
1-8	1.000
1-12	1.000
1-14	1.000

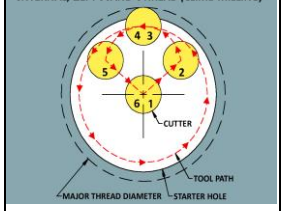
Table 2

INTERNAL, RIGHT HAND THREAD (CLIMB MILLING)



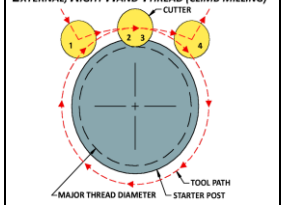
Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while "Z" feeds up from bottom
Step 3-4: Cutter moves helically one rotation
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center

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 one rotation
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center

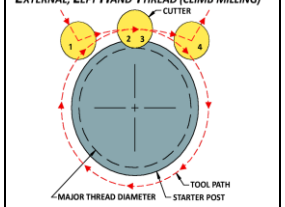
EXTERNAL, RIGHT HAND THREAD (CLIMB MILLING)



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

HARVEY TOOL

Speeds & Feeds

Product Table: Thread Milling Cutters - Multi-Form - UN Threads - For Hardened Steels
Series: 8367xx-C6

Product Notes:

Recommended Depths of Cut (Radial Passes) are based on course of fine pitches. For course pitch, work towards more radial passes. For fine pitch, work towards less radial passes.
Each successive stepover will increase tool engagement, it is recommended to have descending radial passes.

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@harveyperfromance.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												Radial Passes
			0.047	0.062	0.078	0.093	0.125	0.187	0.250	0.312	0.375	0.500	0.625	0.750	
Hardened Steels	46-55	600	.00008	.00011	.00014	.00017	.00023	.00034	.00045	.00075	.00090	.00120	.00150	.00180	7-8
	56-60	350	.00007	.00010	.00012	.00015	.00020	.00029	.00039	.00065	.00078	.00104	.00130	.00156	7-9
	61-68	175	.00007	.00009	.00011	.00013	.00018	.00027	.00036	.00060	.00072	.00096	.00120	.00144	7-9