HARVEY TOOL

Speeds & Feeds

Product Notes:

Drill Thread Mills are a fast way to create a threaded hole in 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 non-ferrous materials

Drilling Notes:
Tools are designed to full plunge, if you are having a difficulties then a pecking cycle is recommended to avoid chip packing and breakage.
• For Non-Ferrous materials, the initial peck depth should be 1-2x Diameter with each subsequent peck at .50-1x Diameter.

Chamfering Notes:
Depending on the type/depth of the hole, chamfering can be done during or after the drilling operation. One chamfer is between the end of the threads and the shank portion and can be used in through hole operations or blind holes with an unspecified depth. The second chamfer is on the drill head, between the intial point angle and the thread relief portion. This chamfer can be used for blind holes with a specifed depth. Use the threading chip loads when using the tip chamfer.

Due to a varying diameter, an <u>Effective Cutter Diameter</u> is needed for Chip Load selection and RPM calculation when using the tip chamfer:
Effective Cutter Diameter = (Major Diameter + Minor Diameter)/2.
Or consider the actual diameter along the angle that is engaged with the workpiece.
Depth of Cut is shown as number of Passes with each pass resulting in a <u>descending</u> stepover

Speeds & Feeds calculations - Threading:
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 #820676 to machine a 7/16-20 internal thread in 6061 Aluminum

1. From Speeds & Feeds chart (next page), SFM is 900 and Chip Load (IPT) is .00147
2. Calculate Speed (RPM) and Linear Feed (IPM)

RPM = (SFM x 3.82) / Threading Portion Diameter
= (900 x 3.82) / .335
= 10263

Linear Feed (IPM) = RPM x IPT x Number of Flutes
= 10263 x .00147 x 3
= 45.3

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
= [(.4375 - .335) / .4375] x 45.3
= 10.6

Adj External Feed = [(Major Thread Dia + Cutter Dia) / Major Thread Dia] x Linear Feed
= [(.4375 + .335) / .4375] x 45.3
= 80

4. Determine Number of Radial Passes using Table 1
5. Conclusion

In this example, the tool would run at 10263 RPM, 10.6 IPM and make 2 Radial Passes

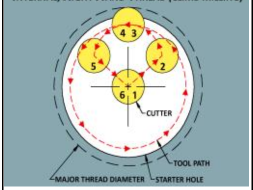
Setup & Use:
1. Check software and input proper feed values (Linear or Adjusted)
2. Choke up on tool
3. Minimize runout (consider entire system of spindle, collet, holders etc)
4. Minimize all vibration (consider tool holding, work holding, rpm "sweet spot" etc)
5. To break in the tool, reduce feed rates 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	Number of Radial Passes
0-80	0.060	3
1-64	0.073	3
2-56	0.086	3
3-48	0.099	3
4-40	0.112	3
5-40	0.125	3
5-44	0.125	3
6-32	0.138	3
8-32	0.164	3
8-36	0.164	2
10-24	0.190	3
10-28	0.190	2
10-32	0.190	2
12-24	0.216	3
12-28	0.216	2
1/4-20	0.250	3
1/4-28	0.250	2
5/16-18	0.312	3
5/16-24	0.312	2
3/8-16	0.375	3
3/8-24	0.375	2
7/16-14	0.437	3
7/16-20	0.437	2
1/2-13	0.500	3
1/2-20	0.500	2
9/16-12	0.562	3
9/16-18	0.562	2
5/8-11	0.625	3
5/8-18	0.625	2
3/4-10	0.750	3
3/4-12	0.750	2
3/4-16	0.750	2
7/8-9	0.875	3
7/8-14	0.875	2
1-8	1.000	3
1-12	1.000	3
1-14	1.000	3
METRIC		
M4.5 x .75	.177 (4.5mm)	2
M5 X .8	.197 (5mm)	2
M6 X .75	.236 (6mm)	2
M6 X 1	.236 (6mm)	2
M8 X .75	.315 (8mm)	2
M8 X 1.25	.315 (8mm)	3
M10 X 1	.394 (10mm)	2
M10 X 1.5	.394 (10mm)	3
M12 X 1	.472 (12mm)	2
M12 X 1.75	.472 (12mm)	3
M14 X 1.5	.551 (14mm)	2
M14 X 2.0	.551 (14mm)	3
M16 X 1	.630 (16mm)	2
M16 X 2.0	.630 (16mm)	3
M18 X 1.5	.709 (18mm)	2
M18 X 2	.709 (18mm)	3
M20 X 2.5	.787 (20mm)	3

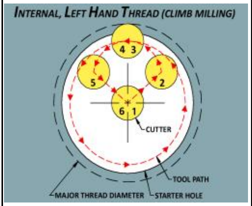
Table 2

INTERNAL, RIGHT HAND THREAD (CLIMB MILLING)



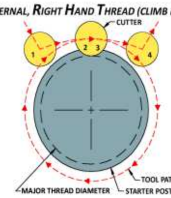
Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while <u>"Z" feeds up from bottom</u>
Step 3-4: Cutter moves helically one rotation
Step 4-5: Cutter exits part along arcing tool path while maintaining "Z" feed
Step 5-6: Cutter returns to center

INTERNAL, LEFT HAND THREAD (CLIMB MILLING)



Step 1-2: Cutter moves into position
Step 2-3: Cutter engages part with arcing tool path while <u>"Z" feeds down from top</u>
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

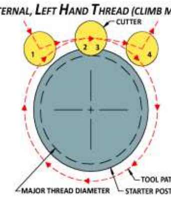
EXTERNAL, RIGHT HAND THREAD (CLIMB MILLING)



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

MATERIAL	Hardness: ≤ 28 Rc (≤ 271 HBn)													
	SFM	Operation	Chip Load (IPT) By Cutter Diameter											
			0.047	0.062	0.078	0.093	0.125	0.187	0.250	0.312	0.375	0.500	0.625	0.750
ALUMINUM ALLOYS Casting (2xx, 5xx, 7xx, 8xx)	450-700	Drilling	.00024	.00032	.00040	.00048	.00064	.00168	.00225	.00343	.00413	.00550	.00688	.00825
Wrought (1xxx, 2xxx, 3xxx, 5xxx, 6xxx, 7xxx, 8xxx)	600-800	Threading	.00012	.00016	.00020	.00024	.00032	.00067	.00090	.00137	.00165	.00220	.00275	.00330
Casting - 3%-5% Si (3xx, A3xx, C3xx, 4xx, A4xx, B4xx)	450-700	Drilling	.00022	.00029	.00036	.00043	.00058	.00151	.00203	.00309	.00371	.00495	.00619	.00743
Casting - 5%-8% Si (3xx, A3xx, C3xx, 4xx, A4xx, B4xx)	420-600													
Casting - 8%-12% Si (3xx, A3xx, C3xx, 4xx, A4xx, B4xx)	390-500													
Casting - 12%-16% Si (3xx, A3xx, C3xx, 4xx, A4xx, B4xx)	350-450	Theading	.00011	.00014	.00018	.00021	.00029	.00061	.00081	.00124	.00149	.00198	.00248	.00297
Wrought - 5%-8% Si (4xxx)	600-800													
Wrought - 8%-12% Si (4xxx)	450-650													
MAGNESIUM ALLOYS	900-1200	Drilling	.00024	.00032	.00040	.00048	.00064	.00168	.00225	.00343	.00413	.00550	.00688	.00825
ZINC ALLOYS	480-700	Threading	.00012	.00016	.00020	.00024	.00032	.00067	.00090	.00137	.00165	.00220	.00275	.00330
COPPER ALLOYS High Coppers - 90%+ (C1xxxx)	170-300	Drilling	.00023	.00030	.00038	.00046	.00061	.00137	.00183	.00288	.00346	.00461	.00576	.00692
Brass (Copper Zinc alloys, C2xxxx, C3xxxx, C4xxxx, C66400-C69800)	375-625													
Phosphor Bronzes (Copper Tin alloys, C5xxxx)	170-400													
Aluminum Bronzes (Copper Aluminum alloys, C60600-C64200)	375-550	Threading	.00012	.00015	.00019	.00023	.00031	.00055	.00073	.00115	.00138	.00184	.00231	.00277
Silicon Bronzes (Copper Silicon alloys, C64700-C66100)	375-625													
Copper Nickels, Nickel Silvers (Copper Nickel alloys, C7xxxx)	170-400													
Cast Copper Alloys (C83300-C86200, C86400-C87900, C9200-C95800, C97300-C97800, C99400-C99700)	400-700													

HARVEY TOOL

Speeds & Feeds

Product Table: Combination Drill/Thread Mills
Series: 8206xx

Product Notes:

Drilling Notes:

Tools are designed to full plunge, if you are having a difficulties then a pecking cycle is recommended to avoid chip packing and breakage.

- For Non-Ferrous materials, the initial peck depth should be 1-2x Diameter with each subsequent peck at .50-1x Diameter.

Chamfering Notes:

Depending on the type/depth of the hole, chamfering can be done during or after the drilling operation. One chamfer is between the end of the threads and the shank portion and can be used in through hole operations or blind holes with an unspecified depth. The second chamfer is on the drill head, between the intial point angle and the thread relief portion. This chamfer can be used for blind holes with a specifed depth. Use the threading chip loads when using the tip chamfer.

Due to a varying diameter, an Effective Cutter Diameter is needed for Chip Load selection and RPM calculation when using the tip chamfer:

Effective Cutter Diameter = (Major Diameter + Minor Diameter)/2.

Or consider the actual diameter along the angle that is engaged with the workpiece.

Depth of Cut is shown as number of Passes with each pass resulting in a descending stepover

SFM Notes:

Drilling SFMs are on the lower end of the range listed while threading SFMs are on the higher end.

Please note:

All posted speed and feed parameters are suggested starting values that may be increased given optimal setup conditions. SFMs reflect uncoated cutters and may be increased 10%-15% if coated.

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 **tech@harveytool.com**.