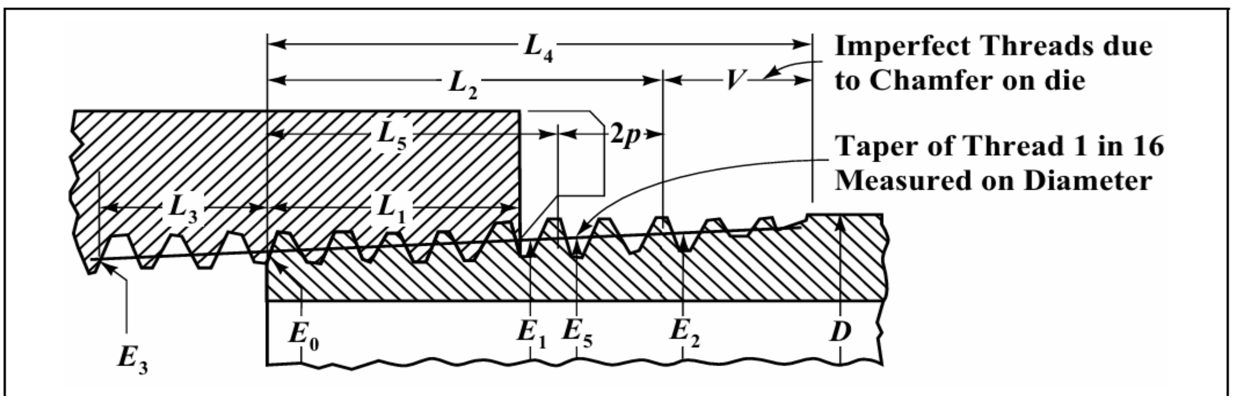
# NPT Thread Milling Info and Guide

Here is a reference guide I put together with all the required information for thread milling an NPT thread.

**Table 4. Basic Dimensions, American National Standard Taper Pipe Threads, NPT**
*ANSI/ASME B1.20.1-1983 (R1992)*




Some information on NPT threads.

L1 = hand tightening length

L3 = Wrenching Engagement length

E3 = Major Diameter at bottom of thread

A general rule is that it will take 3 to 5 turns before the thread is hand tight and then a further 1 to 3 turns to be “Air Tight”. Air tight is between L1 and L3, tightening the thread past L3 is not in specification.

A thread mill can cut out the profile with greater accuracy and far less cutting forces. The functional length of thread is L1+L3 and should not be engaged beyond this point, therefore the tool does not need to go as deep as a tap would. It would be technically sufficient to go to L1+L3 but for an added margin of safety I suggest to go an extra turn.

Here is a table I put together with the relevant information.

| Thread Mill NPT | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| Nominal Pipe Size | Pipe Dia | L1-Hand Tight | TPI | Pitch | L3-Wrench | E3 Major | L1+L3 | ThreadMill | MillDia | Bore Dia |
| 1/16" | 0.3125 | 0.16 | 27 | 0.0370 | 0.1111 | 0.2939 | 0.2711 | 0.3081 | 0.2916 | 0.2416 |
| 1/8" | 0.405 | 0.1615 | 27 | 0.0370 | 0.1111 | 0.3862 | 0.2726 | 0.3096 | 0.3839 | 0.3339 |
| 1/4" | 0.54 | 0.2278 | 18 | 0.0556 | 0.1667 | 0.5114 | 0.3945 | 0.4501 | 0.5079 | 0.4329 |
| 3/8" | 0.675 | 0.24 | 18 | 0.0556 | 0.1667 | 0.6460 | 0.4067 | 0.4623 | 0.6426 | 0.5676 |
| 1/2" | 0.84 | 0.32 | 14 | 0.0714 | 0.2143 | 0.8022 | 0.5343 | 0.6057 | 0.7977 | 0.7013 |
| 3/4" | 1.05 | 0.339 | 14 | 0.0714 | 0.2143 | 1.0114 | 0.5533 | 0.6247 | 1.0070 | 0.9105 |
| 1" | 1.315 | 0.4 | 11.5 | 0.0870 | 0.2609 | 1.2896 | 0.6609 | 0.7479 | 1.2841 | 1.1667 |
| 1.25" | 1.66 | 0.42 | 11.5 | 0.0870 | 0.2609 | 1.6104 | 0.6809 | 0.7679 | 1.6050 | 1.4876 |
| 1.5" | 1.9 | 0.42 | 11.5 | 0.0870 | 0.2609 | 1.8493 | 0.6809 | 0.7679 | 1.8439 | 1.7265 |
| 2" | 2.375 | 0.436 | 11.5 | 0.0870 | 0.2609 | 2.3223 | 0.6969 | 0.7839 | 2.3168 | 2.1995 |
| 2.5" | 2.875 | 0.682 | 8 | 0.1250 | 0.25 | 2.8039 | 0.9320 | 1.0570 | 2.7961 | 2.6195 |
| 3" | 3.5 | 0.766 | 8 | 0.1250 | 0.25 | 3.4250 | 1.0160 | 1.1410 | 3.4172 | 3.2406 |
| 3.5" | 4 | 0.821 | 8 | 0.1250 | 0.25 | 3.9219 | 1.0710 | 1.1960 | 3.9141 | 3.7375 |
| 4" | 4.5 | 0.844 | 8 | 0.1250 | 0.25 | 4.4188 | 1.0940 | 1.2190 | 4.4109 | 4.2344 |
| 5" | 5.563 | 0.937 | 8 | 0.1250 | 0.25 | 5.4751 | 1.1870 | 1.3120 | 5.4673 | 5.2907 |
| 6" | 6.625 | 0.958 | 8 | 0.1250 | 0.25 | 6.5305 | 1.2080 | 1.3330 | 6.5227 | 6.3461 |

The bore of the thread can simply be drilled straight as the threading tool itself will profile the taper. It is wise to check that the tool is indeed a “full profile” cutter, but almost all NPT thread mills come in this form.

The tool goes to the “Mill Depth” and then interpolates a single rotation of the pitch upwards, the thread is then done.

Details to note is that the tool will often state the major diameter at its tip. So, lets say for example I want to do a 1/8-27 thread and my tool states a tip diameter of 0.291”. I would drop to a depth of 0.3096” in the centre of the hole, I would then interpolate out to a diameter of 0.3839” or in the program it would be a radius of (0.3839-0.291)/2 = 0.0465”. Once I do a full turn at this radius while also raising at a pitch of 0.037” I then interpolate back to the centre and retract out of the bore.

Almost all of the CAD software’s out there will support thread milling, but almost no one specifies what these dimensions should be, hence me writing this guide.

How to improve success.

Thankfully the amount of depth and engagement on NPT thread milling is very small, so it's quite unlikely you will break the tool.

Thread milling is the best choice for NPT and in some cases is a must where a tap cannot be used, a very good example of this are ports for actuators. With the hole being blind it is very important that the chips can be extracted, this is the biggest causes for tool failure and wear. A coolant through thread mill will help push the chips out.

A general rule of 0.0015 IPT for every 0.25" of cutter diameter should get through most materials without issue and that is a single pass.

Check out my other guides for general information.