

THREAD FORMING TAPS DATA

THREAD FORMING TAP ENTRY LENGTHS:

Entry taper length is measured on the full diameter of the thread forming lobes and is the axial distance from the entry diameter position to the theoretical intersection of tap major diameter and entry taper angle.

Whenever entry taper length is specified in terms of number of threads, this length is measured in number of pitches (p).

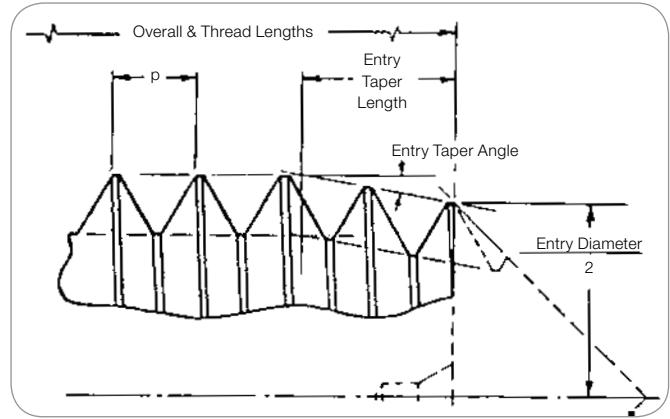
BOTTOMING LENGTH = 1-1/2 to 2-1/2 PITCHES

PLUG LENGTH = 3 to 5 PITCHES

The chamfer on **BOTTOM** taps is approximately 2 threads long and requires a drilled hole depth 3-4 pitches beyond the full thread required. When a controlled maximum chamfer shorter than 2 threads is required, an additional charge will apply. *We will not guarantee the performance of taps with the shorter chamfer.*

Entry diameter, measured at the thread crest nearest the front of the tap, is an appropriate amount smaller than the diameter of the hole drilled for tapping. See below for tap/drill size formulas, and formulas to determine maximum and minimum drill hole sizes for appropriate percent of thread.

TAPPING SPEEDS: Form taps operate most efficiently at spindle speeds 1-1/2 to 2 times faster than those recommended for conventional cutting taps, especially in softer materials and/or with fine pitch form taps. As higher



speeds are attained, adequate lubrication is essential for prolonged tap life and thread quality.

LUBRICATION: Since it is more important to 'lubricate' the cold-forming tap than to 'cool' the tap, form taps should be used with conventional lubricating cutting oils or EP (extreme pressure) rated oil...soluble oils and similar coolants are not recommended.

PRE-TAPPED HOLE SIZE: cold forming taps require a larger pre-tapped hole size than conventional cutting taps. To insure a properly tapped (cold formed) hole, adhere to the following:

FORMULA FOR TAP / DRILL SIZES FOR DECIMAL / INCH FORM TAPS:

$$\text{HOLE SIZE} = \text{Basic Tap O.D.} - \left(\frac{0.0068 \times \% \text{ of Thread}^*}{\text{Threads per Inch}} \right)$$

For example:

To determine drill size for a 1/4-20 thread forming tap at 65% of thread: $0.250 - \left(\frac{0.0068 \times 65}{20} \right) = 0.2279$

* Use whole number for % of thread...for 65%, use 65 (not 0.65).

FORMULA FOR TAP / DRILL SIZES FOR METRIC FORM TAPS:

$$\text{HOLE SIZE (mm)} = \text{Basic Tap O.D. (mm)} - \left(\frac{\% \text{ of Thread} \times \text{mm Pitch}}{147.06} \right)$$

* Use whole number for % of thread...for 65%, use 65 (not .65).

There is no true method of predicting percent of thread that will be obtained when tapping with forming taps due to the many variables involved. As a starting point, however, 55% for maximum drill size and 75% for minimum drill size can be used as a guide. Any desired percent of thread can be approximated by using drill sizes in between. To determine theoretical maximum and minimum drill sizes (for average operating conditions), see formulas below.

For UNIFIED INCH Threads:

Max. Drill Size = Basic Major Diameter - $\frac{3}{8N}$
Min. Drill Size = Basic Major Diameter - $\frac{1}{2N}$

N = T.P.I. (Threads per Inch)

For 60° Metric Threads:

Max. Drill Size = Basic Major Diameter - 0.375P
Min. Drill Size = Basic Major Diameter - 0.5P

P = Pitch

Note: For Basic Major Diameter and Pitch, use millimeter value to obtain drill size in mm. To convert mm to inch value, divide by 25.4:

$$\frac{\text{mm Value}}{25.4} = \text{Inch Value}$$