NORTH AMERICAN TOOL

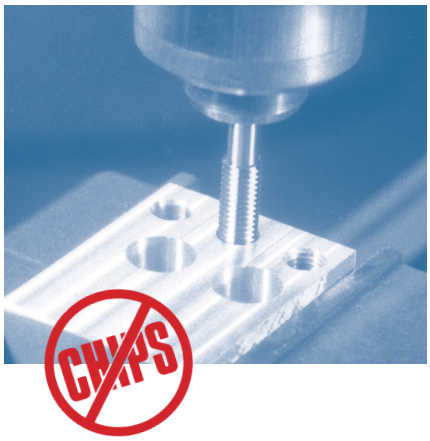
Tru-Flo
Thread Forming Taps

- Improved Threads
- Faster Speeds
- Increased Production
- Tapping with No Chips
- Longer Tap Life
- Efficient Design
- Ideal for Blind Holes
- Optimum Performance
- Fast Shipment

- Improved thread quality and strength due to fluteless design of tap. Allows greater fastener strength in threaded product.
- Tapping speeds can be increased as much as double that of conventional cutting taps = more tapped holes per run.
- Thread forming eliminates costly and time-consuming chip clean up and disposal and produces cleaner internal threads.
- Improved tap life, even in abrasive materials.
- Lubrication grooves allow forced passage of air and lubricant when tapping.
- Simplifies tapping of problematic blind bottoming holes where adequate chip clearance is impossible.
- Available with a variety of performance-enhancing surface treatments to optimize tap life and cutting performance.
- Popular special sizes shipped in 24 Hours! North American Tool provides the industry's fastest shipment of popular special taps, dies and gages.

Multiple leads, special chamfers, and spiral lobes are available.
TRU-FLO™ thread forming taps are also available in SOLID CARBIDE.
Call for details and pricing.
See reverse side for Engineering Data.

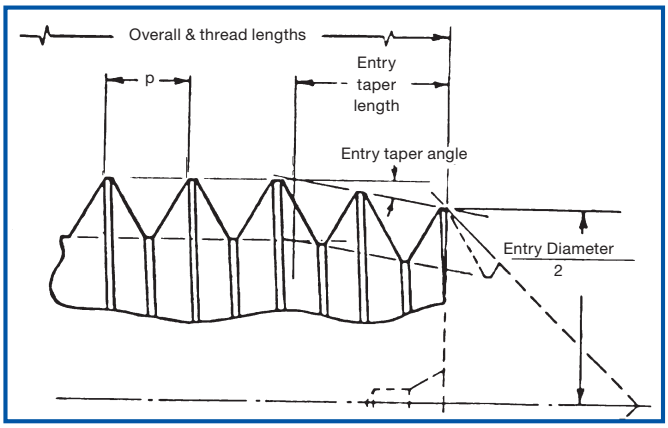
TRU-FLO Thread Forming Taps



North American Tool Corporation
215 Elmwood Avenue
PO Box 116
South Beloit, IL 61080-0116
toll free: 800-872-8277
phone: 815-389-2300
fax: 800-872-3299
sales@natool.com
www.natool.com

TRU-FLO™ ENGINEERING 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 Tru-Flo 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: TRU-FLO 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



TRU-FLO 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, TRU-FLO 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: TRU-FLO 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 TRU-FLO TAPS:

$$\text{HOLE SIZE} = \text{Basic Tap O.D.} - \left(\frac{.0068 \times \%\ \text{of Thread}^*}{\text{Threads per Inch}}\right)$$

For example:
To determine drill size for a 1/4-20 thread forming tap at 65% of thread: $.250 - \left(\frac{.0068 \times 65}{20}\right) = .2279$
* Use whole number for % of thread...for 65%, use 65 (not .65).

FORMULA FOR TAP/DRILL SIZES FOR METRIC TRU-FLO 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:

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