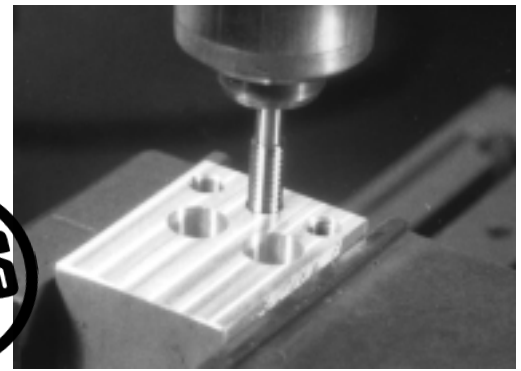


TRU-FLO[®]

Thread Forming Taps



POPULAR SPECIAL SIZES

SIZE	SIZE MM
#0	M1.5, M1.6
#1	M1.8
#2	M2, M2.2
#3	M2.3, M2.5, M2.6
#4	-----
#5	M3
#6	M3.5
#8	M4
#10	M4.5, M5
#12	M5.5
1/4	M6
5/16	M7, M8
3/8	M9, M10
7/16	M11
1/2	M12
9/16	M14
5/8	M16
11/16	M18
3/4	-----
13/16	M20
7/8	M22
15/16	M24
1"	M25

• **IMPROVED THREADS**

Improved thread quality and strength due to fluteless design of tap; allows greater fastener strength in threaded product.

• **FASTER SPEEDS = INCREASED PRODUCTION**

Tapping speeds can be increased as much as double that of conventional cutting taps...more tapped holes per hour.

• **NO CHIPS**

Thread forming (over thread cutting) eliminates costly and time-consuming chip clean-up and disposal.

• **LONGER TAP LIFE**

Improved tap life, even in abrasive materials...no cutting edges to get dull.

• **EFFICIENT DESIGN**

Lubrication grooves allow forced passage of air and lubricant when tapping.

• **IDEAL FOR BLIND HOLES**

Simplifies tapping of problematic blind bottoming holes where adequate chip clearance is impossible with chip-producing cutting taps.

• **OPTIMUM PERFORMANCE**

Available with a variety of surface treatments to optimize tap life and cutting performance.

• **FAST SHIPMENT**

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 chamfer, and spiral lobes are available.

"TRU-FLO" Thread Forming Taps are available in SOLID CARBIDE. Call for pricing.

TECHNICAL DATA

THREAD FORMING TAP ENTRY LENGTHS

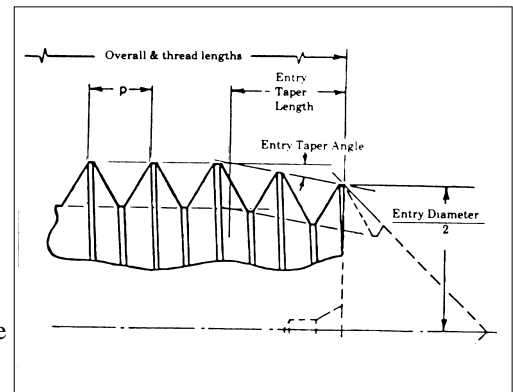
Entry taper length is measured on the full diameter of the thread forming lobes and is the axial distance from the entry diameter position to the theoretical intersection of tap major diameter and entry taper angle.

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

BOTTOMING LENGTH = 1 to 1-1/2 PITCHES

PLUG LENGTH = 3 to 5 PITCHES

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 next page for tap/drill size formulas, and formulas to determine maximum and minimum drill hole sizes for appropriate percent of thread.



north american tool

ISO 9001 REGISTERED

SPECIAL
THREAD TOOL
PRODUCTS

215 Elmwood Ave. • South Beloit, IL 61080 U.S.A.

CUSTOMER SERVICE

Phone: 800.872.8277 • 815.389.2300

Fax: 800.872.3299 • 815.389.2952

E-Mail: natool@att.net

Web: www.natool.com

TRU-FLO[®]

THREAD FORMING TAPS

TECHNICAL DATA

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 cutting conventional taps. To insure a properly tapped (cold formed) hole, adhere to the following formulae:

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

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

Example: 1/4-20 TAP / 65% of Thread...HOLE SIZE = .250 - $\left(\frac{.0068 \times 65}{20} \right)$

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

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

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

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:

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

north american tool | SPECIAL
ISO 9001 REGISTERED | THREAD TOOL
PRODUCTS

215 Elmwood Ave. • South Beloit, IL 61080 U.S.A.

CUSTOMER SERVICE
Phone: 800.872.8277 • 815.389.2300
Fax: 800.872.3299 • 815.389.2952
E-Mail: natool@att.net
Web: www.natool.com