

Programing Information

Experience has shown that a tap will cut the best quality threads when allowed to act as its own lead screw, feeding precisely on pitch to exactly the required depth of thread. The machine must be prevented from forcing the tap to do anything else or the thread quality will suffer or the tap may break. The tension stroke in the NUMERTAP® provides the freedom required to cut the best quality threads with the least risk of breakage.

NUMERTAP® Systems can be used on any suitable machine tool which has a reversing spindle. Effective choices of spindle speeds and feeds for particular tapping requirements can be made by the following the sample calculations: Tapping a 3/4 –10 thread, 1" deep in mild steel:

TAPPING A 3/4 THREAD 1" DEEP IN MILD STEEL:

RPM = (12 x SFM)/(3.82 X SFM)/D where D is the tap diameter.

Use the chart on the previous page to find the tapping speed in SFM.

RPM = (3.82 x .50)/.75 – 255 RPM

Tap Feed Rate = Pitch x RPM = .100 x 255 = 255 = 25.5 inches/minute.

Feed in and out at the same feed rate.

When using a conventional tapping cycle, optimum performance is insured by slightly underfeeding the tap, normally by 2% - 10%. This forces the tension stroke of the tapping head to be used and eliminates any effect of the machine tool. The Z axis feed distance must be reduced by this same percentage. The axial float in the NUMERTAP takes up the difference between the required thread depth and the programmed depth. If a 10% underfeed was selected, the calculations would be as follows:

Program Feed Rate = .90 x on pitch feed rate = .90 x 25.5 = 22.95"/minute.

Spindle Z-Axis Travel = .90 x required depth = .90 x 1.000 = .900".

Axial Float used = 1.000 - .900 = .100"

Do not allow the Axial Float used to equal the tension stroke length (.56).

If the tapping cycle is controlled by a "canned" program which calculates its own feed rates from an input of pitch and speed. The desired underfeed can be obtained by deliberately entering a reduced value of pitch or an increased value of threads per inch as follows:

Actual TPI = 10 (pitch = 1/10 = .100")

Input TPI = 11 (pitch = 1/11 = .091")

Program Feed Rate = .091 x 255 = 23.2"/minute

Program Feed Depth = .91 x 1.000 = .910"

If the CNC machine has a slow spindle reversal, the program must compensate for any drive system inertia by including a dwell not long enough for the spindle to come to a full stop when the tap has reached full thread depth. If the spindle has not stopped when the program calls for it to feed out, the tap could break or be pulled out of the adapter, or the threads could be ruined. These problems can be rectified using a program similar to the following:

Spindle clockwise

Feed to depth (incorporating underfeed)

Spindle stop

When using a synchronous tapping cycle, the drive system inertia may create the undesirable effect of tap elongation, or thread distortion. The best answer to this problem is the use of tension only tapping heads.