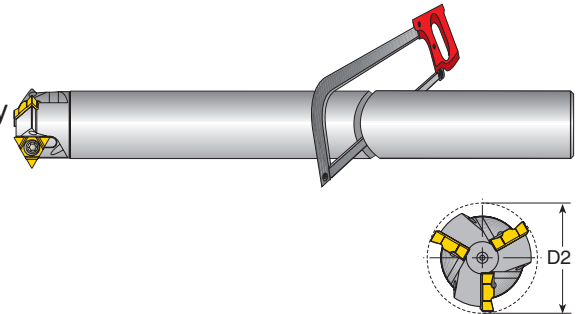


## Tip 1

**To achieve highest stability and accuracy, use a thread milling tool with the largest possible diameter and minimum overhang**

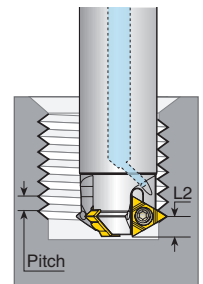
- In long threads, it is very important to use the tool with the largest cutting diameter to minimize tool vibration.
- Using a longer, weaker tool than required reduces stability and has a negative effect on overall performance.
- In the case of cylindrical shank tools, shorten the tool length when possible by cutting the shank.



## Tip 2

**Use a single-point insert to reduce tool load**

- Use of long tools, especially with multi-tooth inserts, often causes higher load and chatter.
- This problem can be solved by reducing the feed and increasing the number of passes. However, this type of solution increases the machining time.
- By using single-point inserts, you can dramatically reduce the load and increase the feed, resulting in overall reduction in machining time.
- Single-point inserts provide an excellent solution for threading hard materials.

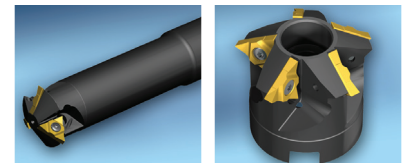


## Tip 3

**For fast machining of deep threads use a multi-flute thread milling tool**

- Feed (F) is a function of the number of flutes, where  

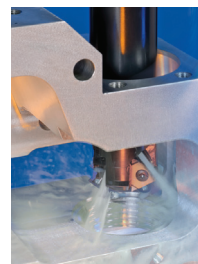
$$F = N \times z \times f$$
  - N - Rotational Velocity [R.P.M.]
  - z - No. of Cutting Edges
  - f - Feed per Tooth per Rotation [mm/rev]
- More flutes (z) means faster feed (F) !



## Tip 4

**Use cooling thru when threading deep holes**

- External cooling does not reach the cutting corner in deep holes.
- When cooling thru is directed to the cutting point, the result is:
  - Better tool life
  - Better chip evacuation
  - Better surface finish



Visit our new Vargus website at [www.vargus.com](http://www.vargus.com)

Download the complete TMSD catalog and the latest version of TM Gen

