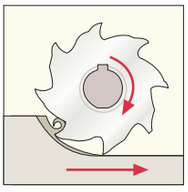
VARDEX Tip 9
Making Threading Easy
Thread Milling Tips
July 2006

Tip 1 — Use climb milling operation for Thread Milling machining

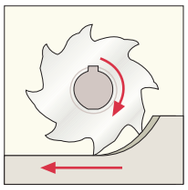
Conventional Milling - Feed movement opposite tool rotation



Drawbacks:

- Width of chip starts from zero and increase therefore tool edge rubbing generates much more friction and heat
- Tool edge rubbing requires more power
- Chip evacuation - Chips fall in front of cutter and spoiled the surface finish
- Rubbing and falling chips cause faster wear and shorten tool life
- Tool force tends to lift up workpiece

Climb Milling - Feed movement and tool rotation same direction



Advantages:

- Width of chip starts from maximum and decrease, no rubbing
- Less power required
- Better chips evacuation - chips flow behind the tool, excellent surface finish
- Less wear and better tool life
- Tool push the workpiece down to the table

Tip 2 — Take 2 passes to complete thread when thread milling hard materials

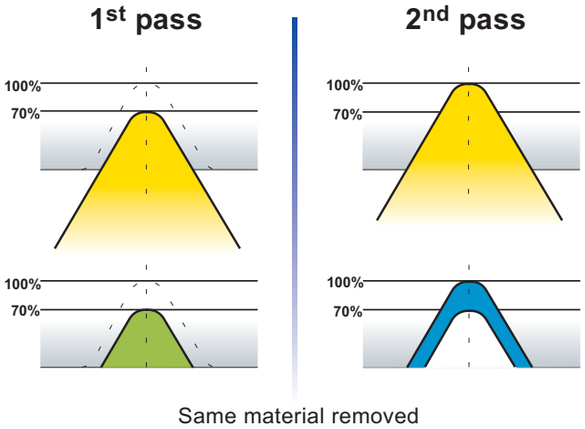
Preferred method:

1st pass - 70% of profile depth

2nd pass - remaning 30%

Thus, the same amount of material is removed in each pass:

- 2nd pass should be deep enough so that material is cut, not just pressed.
- TM Gen allows you to choose 1 or 2 passes according to material type



Same material removed

Tip 3 — Beware, in circular movement the feed rate at the cutting edge (F_1) is faster than at the center (F_2)

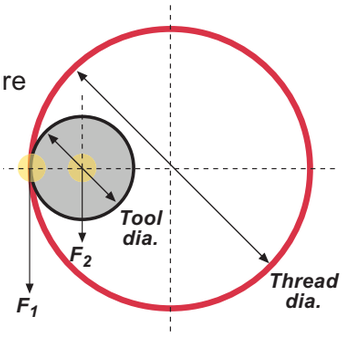
$F_1 = N \times z \times f$

N - Rotation Velocity (R.P.M.)
z - No. of Cutting Edges
f - Feed per Tooth per Rotation (mm/rev)

- The formula for tool feed rate (mm/min) caculates the feed on the **cutting edge** however, most CNC controllers require the feed at the **tool center**. Therefore you should calculate the feed at the tool center
- TM Gen displays both F_1 and F_2 automatically, and based on your controller, inserts the appropriate one into the CNC program

Example: M6 x 1
Tool cutting diameter 4.8 mm.
F_1= 238(mm/min)
F_2= 47(mm/min)

Conclusion: Be sure to use the correct feed rate in your application!



For tool selection and cutting data in Thread Turning applications use our TT Gen.
For the best Thread Milling CNC Programming, use VARDEX TM Gen software utilities.
For free copies, go to www.vargus.com

All issues of the VARDEX Tips are available on the VARGUS website: **www.vargus.com**