Recommended Speeds (SFM) For Threading With Horizon Carbide Inserts							
Material Group	**Material Specs.**	**C23**	**123**	**156**	**310**	**323 & 323F**	**356**
Aluminum	2024, 6061, 7075	Maximum SFM	Maximum SFM	Maximum SFM	---	---	---
Copper, Brass, Bronze	Most Alloys	200-600	300-800	---	---	---	---
Low Carbon Steels	1018, 1025, 1117, 12L14	---	300-500	400-650	200-300	300-600	450-750
Medium Carbon Steels	1045, 1070, 1144,	---	300-450	350-600	200-300	300-500	450-700
Alloy Steels	4130, 4140, 8620	---	250-450	350-550	200-300	300-500	400-650
Alloy Steels 28 Rc+	4150, 4340, 52100	---	250-400	300-500	200-275	300-450	350-600
Stainless Steels	303, 304, 410, 416	---	225-400	250-350	200-300	250-450	250-500
Stainless Steels	316, 316L, 422,17-4PH	---	200-375	200-300	150-275	225-400	225-450
Titanium - CP	Commercially Pure	150-300	200-400	---	200-300	250-500	---
Titanium - Alloys	6AL-4V, 5AL-2.5SN	100-200	100-225	---	100-250	150-275	---
Nickel / Cobalt Alloys	Monel, Invar, Kovar	100-250	100-300	---	100-300	100-300	---
High Temp Alloys	Inconel, Hastelloy, A286	75-150	100-150	---	75-200	100-200	---
Cast Iron 150-325 BHN	Class 20, 30, 35, 40	200-300	200-600	---	100-300	250-600	---
Cast Iron 375-450 BHN	Class 50, 55, 60	150-250	150-450	---	100-250	200-500	---
Alloy / Ductile Iron	60-40-18, 80-55-06	100-250	150-400	200-400	100-300	250-450	300-500

Threading Guidelines For CNC Lathes

1. *Start near the top of the SFM range for the material being threaded.*

Higher *SFM* reduces Built-up Edge, the major cause of poor tool life in threading. For Harder or more Abrasive Materials start in the Middle of the SFM range. Once setup is complete *SFM* can be adjusted for optimum tool life. Tough Micrograin Grade *310* reduces insert chipping from edge build-up at lower *SFM* and on parts under 1" Diameter.

Formulas to calculate *Revolutions Per Minute* (RPM) and *Surface Feet Per Minute* (SFM) :

***RPM* = *SFM* x 12 *Divided by Part Diameter* x 3.1416**
SFM* = 3.1416 x *Part Diameter, Divided by* 12 x *RPM

2. *Use the* G76 *Threading Cycle with Fanuc, Yasnac and similar CNC controls.*

If ***P1, P2, P3, & P4*** parameters are available, use ***P1*** for most applications. ***G76 - P1*** removes equal amounts of material with every pass. ***P2*** alternates between front & back cutting edges and should only be used on 6 tpi & coarser threads.

3. *Set Depth of Cut for the First Pass at 20% to 30% of the Thread Height* Per Side.

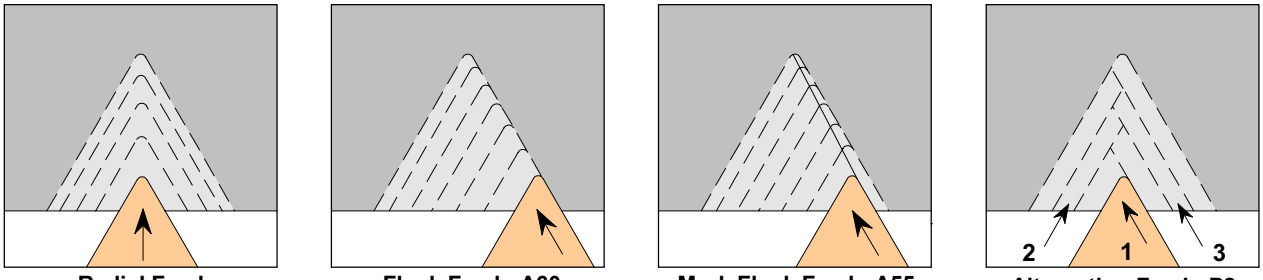
G76 controls the depth of cut for the remaining passes. To find the *DOC* for the 1st Pass multiply the *PITCH* by *.6* to get the approximate *THREAD HEIGHT*. Multiply *THREAD HEIGHT* by *20 - 30%* to get the *DEPTH* of the 1st Pass. Use less than *20%* when threading hard materials or larger thread pitches. *Don't use "Spring Passes" under .002 DOC Per Side!*

Example for calculating the *DOC* for the first pass on a 1/2-20 thread:

1" ÷ 20 = .050 PITCH), (.050 x .6 = .030 (THREAD HEIGHT), .030 x .3 = .009 (DEPTH OF 1st PASS)

4. *Use the A55 Parameter to set the Infeed Angle.*

Most CNC Lathe Controls offer a choice of ***A60, A55, A30*** & ***A29***. ***A60*** is the default setting. ***A55*** is equivalent to setting the compound feed on a manual lathe to 27.5°. In most applications ***A55*** will double or triple tool life over ***A60***. See below:



Radial Feed | **Flank Feed - A60** | **Mod. Flank Feed - A55** | **Alternating Feed - P2**