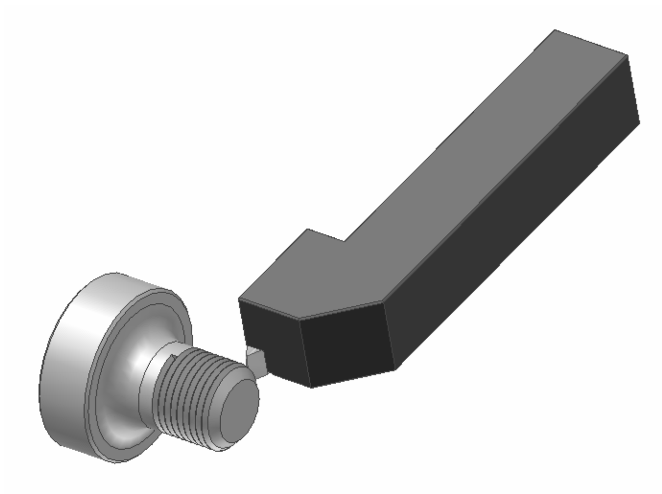
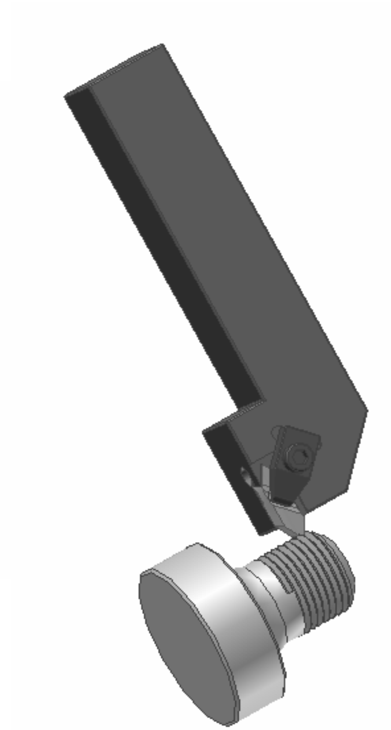
CNC Applications

Threading on Turning Centers





Conventional vs. CNC Threading

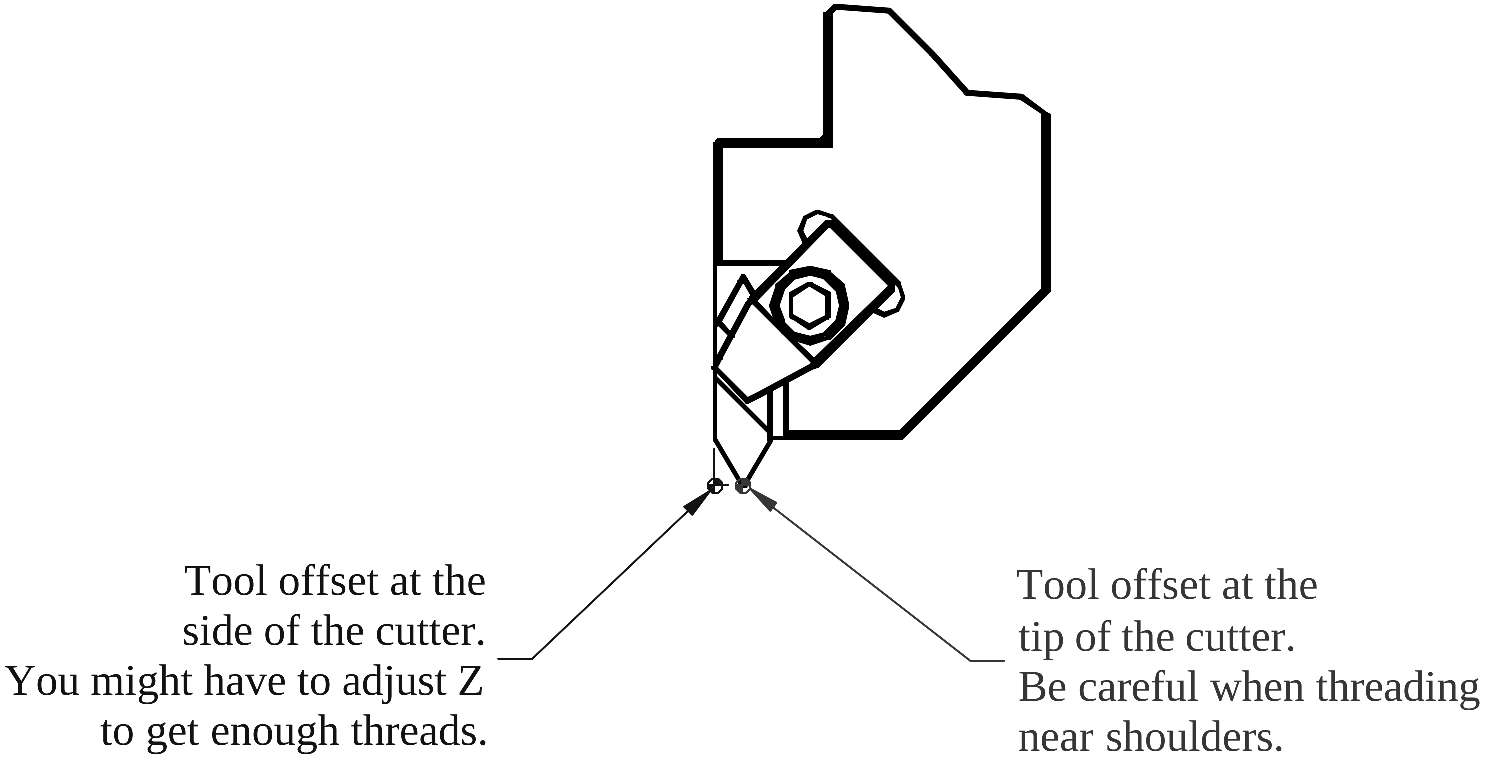
Conventional	CNC
Accuracy is dependant on the lead screw and gears.	Accuracy is dependant on the Z axis ballscrew and the electronics.
Manually synchronize multiple cuts with a threading dial	Electronically synchronize multiple cuts.
Cutting speed is limited by the operator’s ability to engage the half nuts.	Cutting speed is limited by how accurately the machine can synchronize the feed with the spindle RPM.
Takes about 10 minutes for an experienced operator to thread a ¾-16 UNF 2A 1” long.	Takes less than 1 minute for a good turning center to thread a ¾-16 UNF 2A 1” long.

Notes on CNC Threading

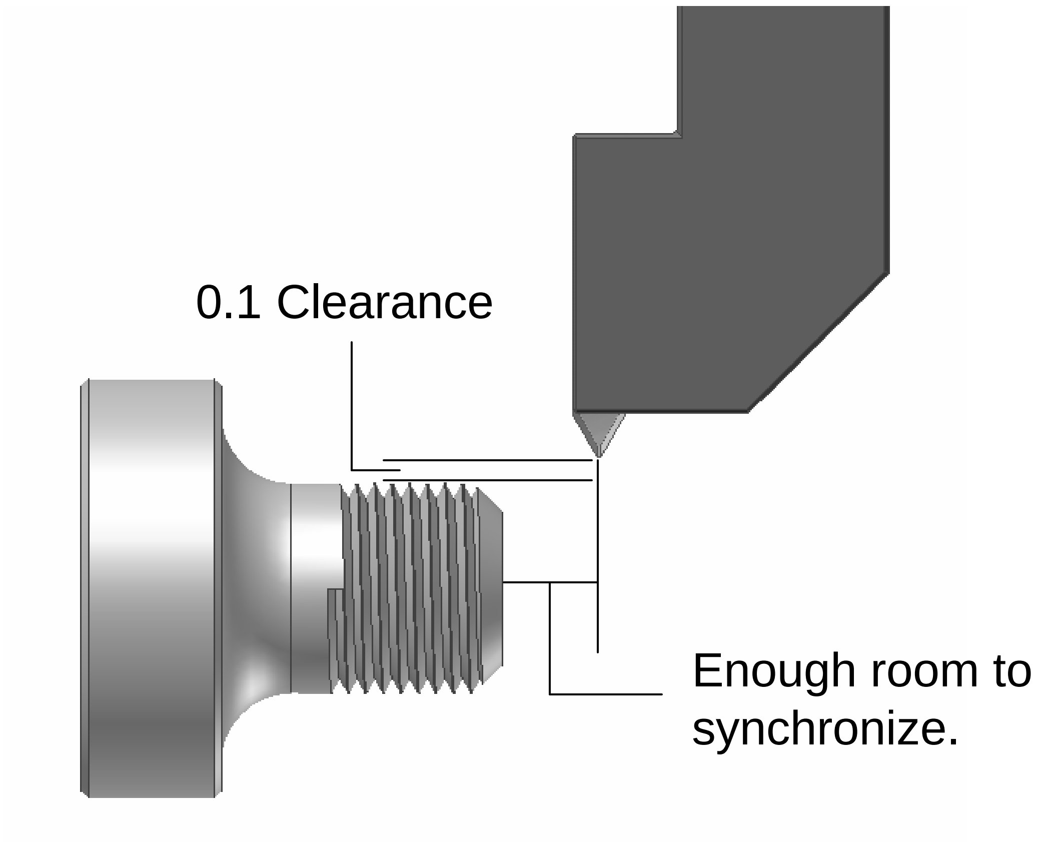
- Threading on a turning center is much faster than conventional threading because:
  - The machine can synchronize the feed and spindle RPM much faster than a person can engage the half nuts.
  - Faster synchronization means higher cutting speeds are used on CNC equipment resulting in faster production, better thread finishes, and more accurate threads.
  - The high rapid traverse rates re-position the cutter for subsequent cuts much more quickly than a person can.
- A CNC machine can cut any thread – English or Metric – without special equipment.

Threading Tool Offsets

You can set the tool offset for a threading tool at the tip or at the side. The X value is the same in either case, only the Z value differs. As noted below, setting at the side helps prevent running into shoulders but may not have enough threads, while setting at the tip gives the correct thread length with increased risk of hitting a shoulder. Be aware of the method being used.



Insuring Thread Accuracy

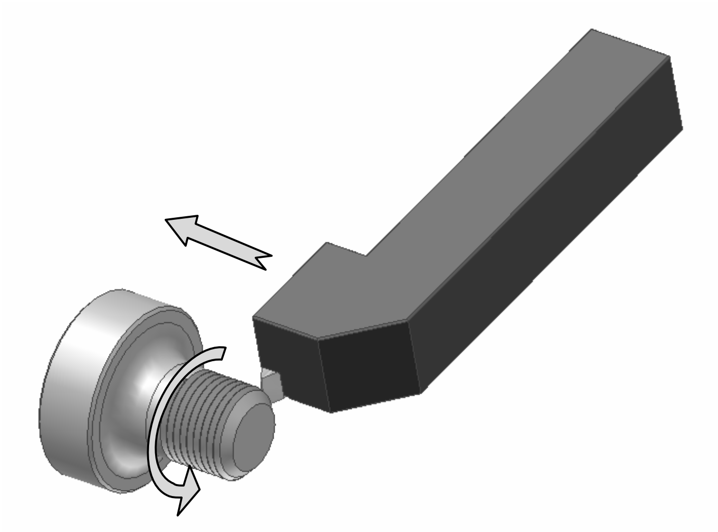


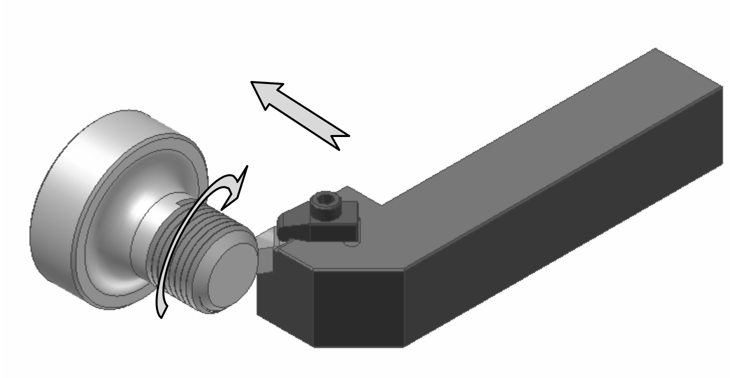
Be sure you have enough room to move around a live center or tailstock if one is being used!

While CNC machines are fast, they are not infallible. You must have enough clearance between the end of the part and the cutter's start point for the feed motors to synchronize with the spindle. Most machine manuals have a formula for this distance which depends on RPM and thread pitch.

If you cut a thread that has the correct pitch diameter but still won't fit a GO gage, increase this distance.

Right Hand or Left Hand?





Right Hand Thread – spindle is going forward (M3) and the cut is towards the headstock.

Left Hand Thread – spindle is going reverse (M4) and the cut is towards the headstock.

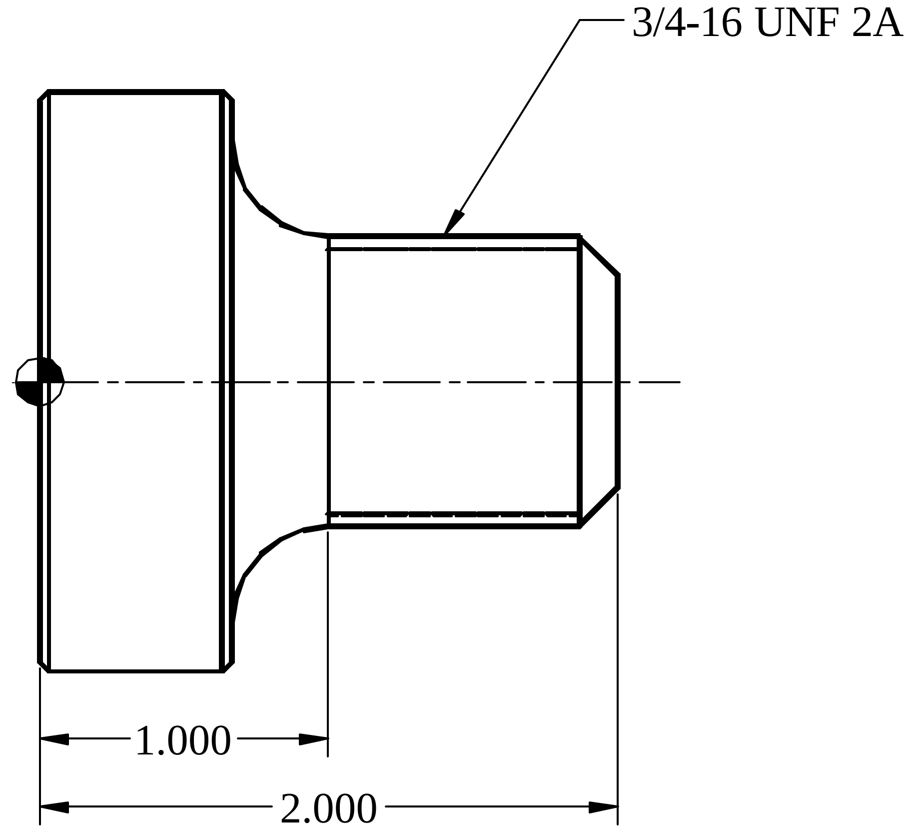
To complicate things further, we can reverse the spindle rotations, cutter hands, and cut directions shown above and end up with the same thing. However, the two pictures shown above are the most common methods of threading, so be sure you understand them.

G Codes for Threading

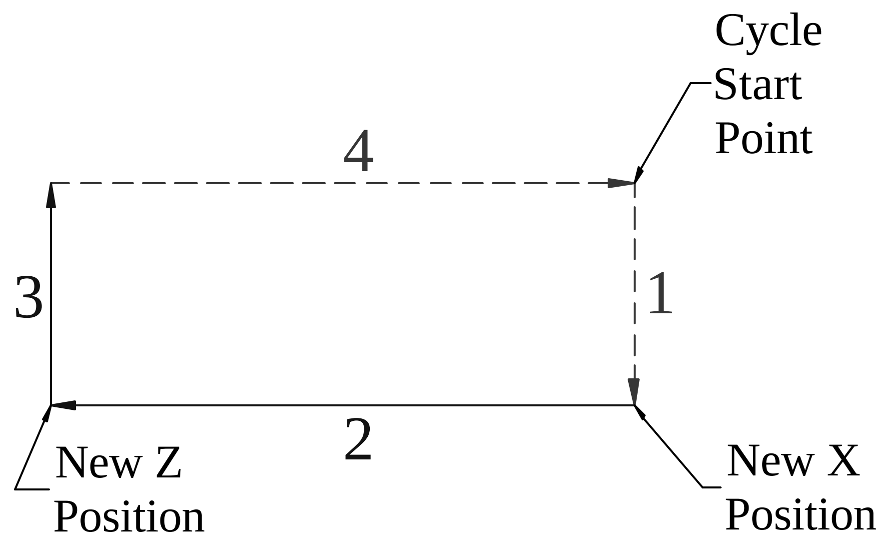
G Code	Application
G32	Requires four blocks per cut, mostly obsolete now.
G92	Works similarly to the G90 turning cycle with one block per cut required.
G76	The whole thread is cut with one block. This is the most common form of threading.

A Threading Example

We'll do this program twice – once with G92 and again with G76. In both cases, we'll assume that the profile is already turned, and we will do the threading at 400fpm.



The G92
Rectangular Threading Cycle



G92 works the same as G90 except for the synchronization between spindle and cutter to create the threads. Start the cutter at the Cycle Start Point. The cutter will also end up at the cycle start point at the end of the cycle. Each block looks like this:

```
G92 Xnewx Znewz Flead
```

Note that lead is actual pitch calculated as 1/tpi for single start threads. Also, many turning centers use E instead of F on threading cycles. Know your machine!

Follow Planning and Programming Steps (1-3)

1. Examine the drawing. We have to find some data from the *Machinery's Handbook* for a ¾-16 UNF 2A thread:
   - Major Diameter Range: 0.7391-0.7485"
   - Minor Diameter (Maximum): 0.674"
   - Lead = Pitch = 1/tpi = 1/16 = 0.0625"
2. How will we hold the raw material – in a collet chuck.
3. Decide what cutters to use – given a tough, coated carbide threading insert, and the cutting speed is also given (400fpm). We have to calculate the RPM since CSS should not be used when threading:

$$N = RPM = \frac{12 \times V}{p \times D} = \frac{12 \times 400}{p \times 0.75} = 2037$$

Follow Planning and Programming Steps (4-5)

4. Write down the exact sequence of operations:
   A. Rapid position the cutter in Z 0.25” away from the face.
   B. Rapid position the cutter 0.1” away from the part in X (radial).
   C. Based on the Machinery’s Handbook data, we’ll assume the blank is 0.745” diameter to start, and we’ll take 6 passes at the following X values:

0.725	0.705
0.690	0.680
0.677	0.674

   D. Program end.
5. Convert the sequence of operations to a program:

Program Start

Make the Threading Passes

Program End

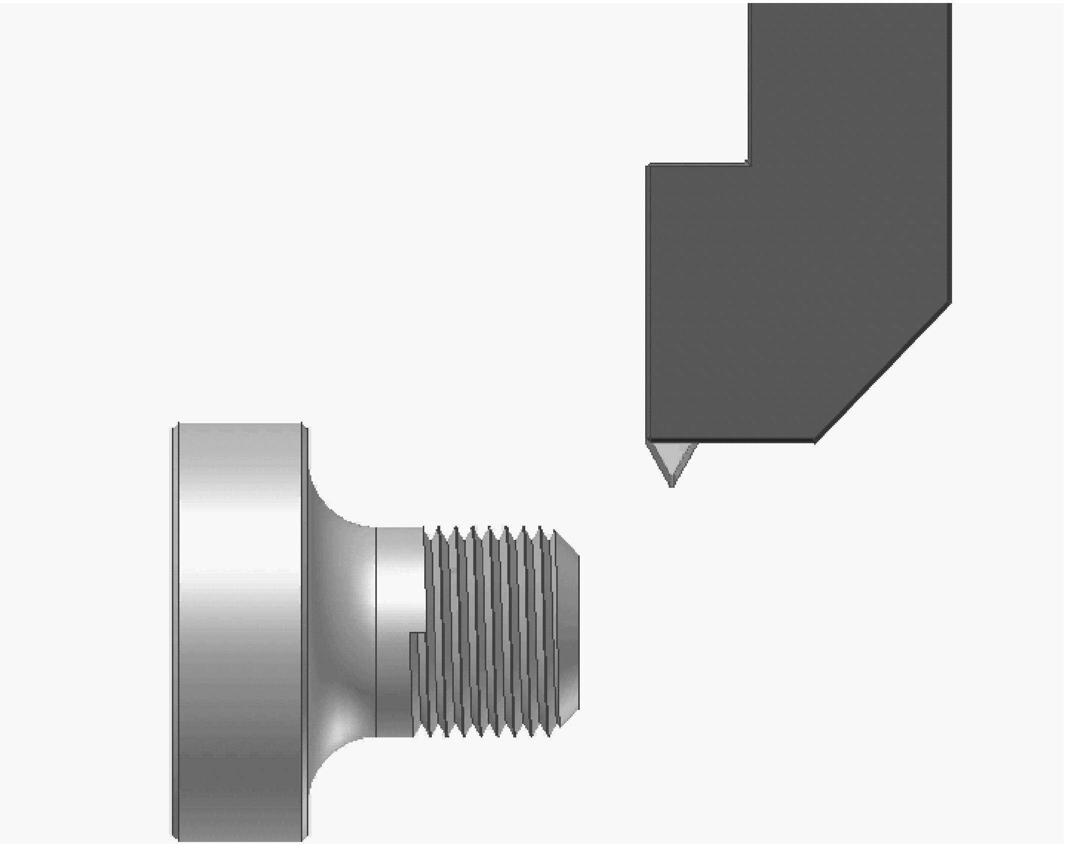
Note: on our machine, the threading tool offset is taken from the side, not the point, of the cutter, so our threads will be somewhat short which we can adjust for in the program if we need to.

The Program with G92

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the threading tool
G54	
G97 S2037 M3	Set the RPM, forward direction.
G0 Z2.25	Rapid 0.25 away from part in Z.
X0.945 M8	Rapid 0.1” radial distance from the part in X, coolant on.
G92 X0.720 Z1.0 F0.0625	First threading cycle cut removing 0.020” from diameter.
X0.705	Second threading cut, G92 is still active.
X0.690	
X0.680	
X0.677	
X0.674	Final threading cut, just a light pass.
M9	
M5	
G28 U0	Program End
G28 W0	
M30	
%	

Threading Animation

Select this link to start the animation.



Note: the animation does not show the cutter moving to the start point or to home after machining the thread. It only shows the G92 cycle blocks.

Again, this is just an animation. The machine would cut the thread much faster than the animation shows.

G76 Threading Cycle

Cutting threads is so common, the CNC designers have created the G76 cycle to cut the entire thread in one pass. The format looks like this:

`G76 X``rootx `**`Z`**`endz `**`I`**`taper `**`K`**`height `**`D`**`pass1 `**`F`**`lead `**`A`**`angle`

Where: rootx = minor diameter of the thread (required)

endz = the ending Z value of the thread (required)

taper = amount of taper when cutting a tapered thread (optional)

height = radial height of the thread (required)

pass1 = depth of the first pass (Note, most machines do not allow a decimal point on D, so an integer must be used.) (required)

lead = pitch for a single start thread which is 1/tpi (required)

angle = angle to enter the thread (optional)

General Comments about G76

- If you leave I off, the cycle produces a straight thread which is most common.
- If you leave A off, the cutter feeds straight in (see the next two slides for a more detailed description of A).
- The cycle works for both ID and OD threads based on the cycle start point and the values of X and Z.
- The cycle automatically decides how many passes to take depending on the value of K and D. Each pass is smaller than the previous pass.
- Some machines have more control over the number of passes and the depth of the final pass. Know your machine!

The K and D Codes

K is the height of the thread and is easily calculated with the following formula.

$$K = Thread\ Height = \frac{Major\ Diameter - Minor\ Diameter}{2}$$

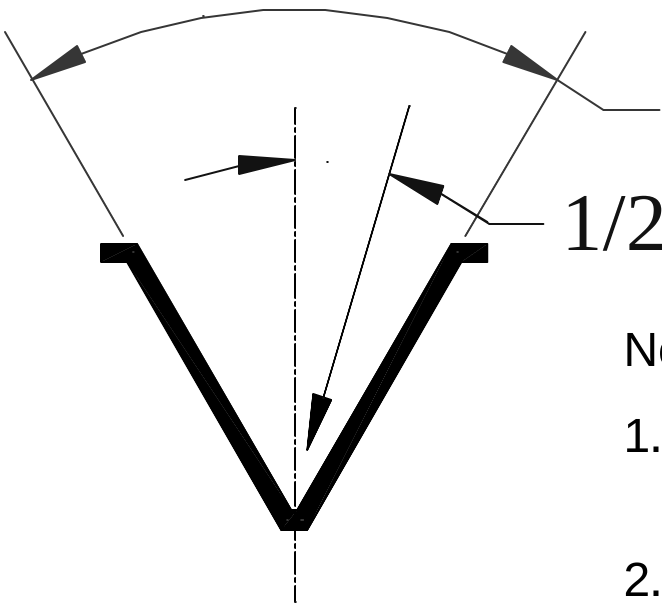
When looking in the Machinery's Handbook for thread specifications, remember that the major diameter for an external thread is given as a range, so use the high side of the range. For our ¾-16 UNF 2A:

$$K = Thread\ Height = \frac{0.748 - 0.674}{2} = 0.037$$

D is the depth of the first pass, usually in integer form. If you want a 0.012" deep first pass, set D to 0120. Remember the resolution of most machines is 0.0001", so 0120=0.0120".

Deciphering the A Code

As shown in the diagram below, a thread has an included angle. The most common angles are 60º for both metric and inch V threads and 29º for ACME threads. By changing the value of A, we can change the infeed angle of the threading cutter. The infeed angle is always ½ the value of A specified in the G76 cycle.

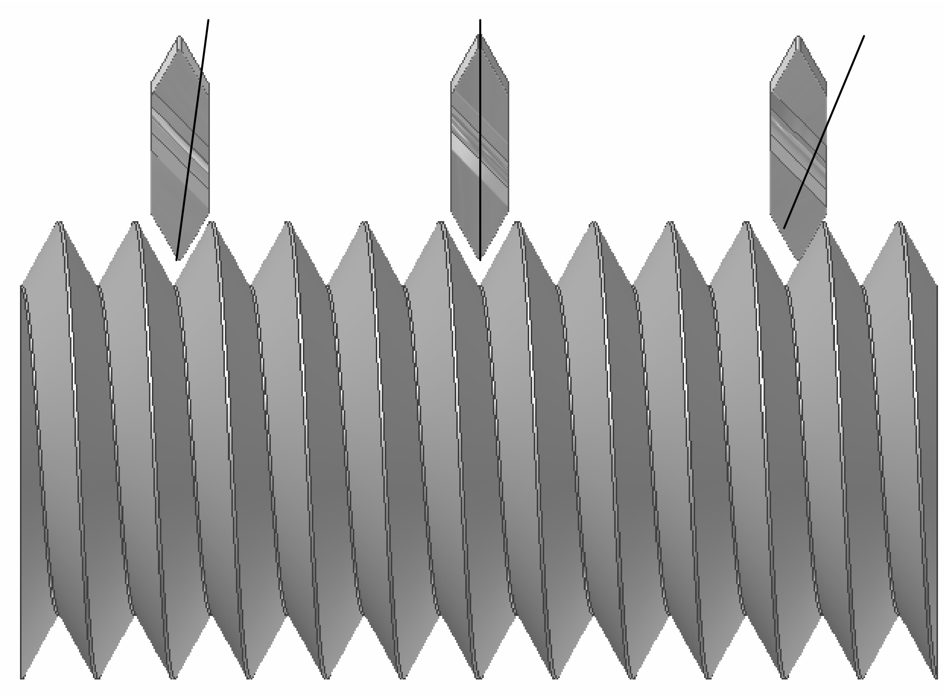


Notes:

1. Some machines will not accept a decimal point on A!
2. Some machines limit A to common thread angles, others allow any value.

Threading Feed Angle

A20 – the machine actually feeds at 10º. Most cutting takes place on the leading flank, but some takes place on the trailing flank. A good compromise since it is fairly easy on the cutter, leaves a good finish, and tends to minimize chatter.



A60 – the machine actually feeds at 30º, or down the trailing flank. All cutting takes place on the insert's leading edge, which is easiest on the cutter. The trailing flank usually has a poor finish. This is how most conventional (manual) threading is done.

A0 – the default. The cutter feeds straight in, and the insert cuts equally on both flanks. This is hard on the cutter, but both flanks usually have a good finish.

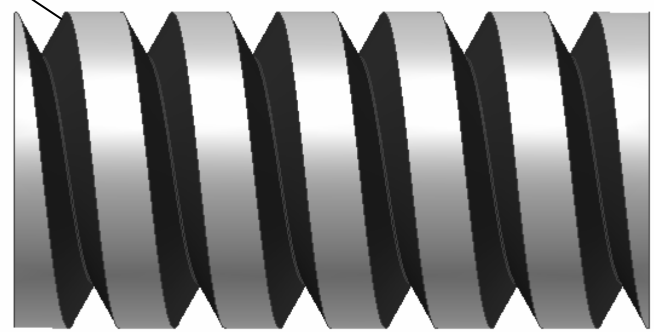
The Program with G76

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the threading tool
G54	
G97 S2037 M3	Set the RPM, forward direction.
G0 Z2.25	Rapid 0.25 away from part in Z.
X0.945 M8	Rapid 0.1” radial distance from the part.
G76 X0.674 Z1.0 D0120 K0.037 A20 F0.0625	Cuts the entire thread in one block!
M9	
M5	Program End
G28 U0	
G28 W0	
M30	
%	

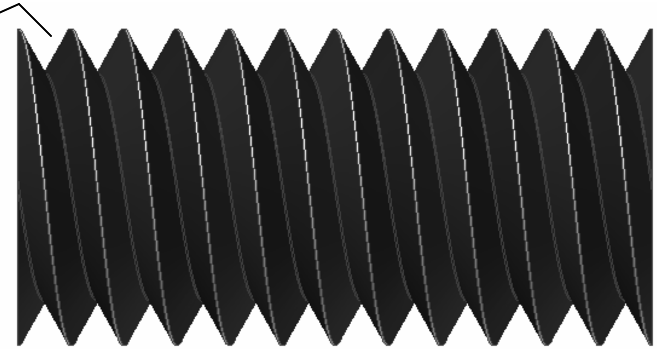
Cutting Multiple Lead Threads

For single start threads (the most common), the lead is equal to the pitch. Or, in threading terms, the amount of advancement for one turn is equal to the distance between the threads.

Occasionally, we have to cut multiple start threads where the lead is an even multiple of the pitch. For example, to cut a ¾-16 double lead thread, we would cut a ¾-8 thread half-way deep like this:



Then, we move the start point of the threading cycle over by the pitch (0.0625”) and cut another thread in between those we just cut like this:



Cutting a Double Lead Thread with G76

Program Codes	Action
%	Program Start
O999	
G20 G40 G99	
G28 U0	
G28 W0	
T0505	Load the threading tool
G54	
G97 S2037 M3	Set the RPM, forward direction.
G0 Z2.25	Rapid 0.25 away from part in Z.
X0.945 M8	Rapid 0.1” radial distance from the part.
G76 X0.674 Z1.0 D0120 K0.037 A20 F0.125	Cuts the first thread, note the lead.
G0 Z2.3125	Re-position the starting point.
G76 X0.674 Z1.0 D0120 K0.037 A20 F0.125	Cut the second thread.
M9	
M5	
G28 U0	Program End
G28 W0	
M30	
%	