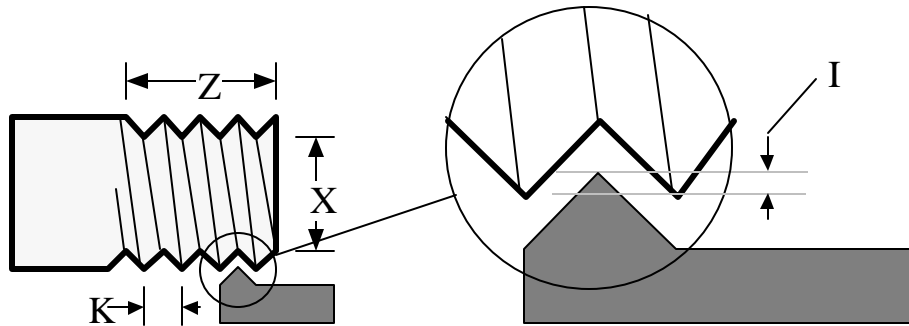


The format is: **G33X_nZ_nI_nK_nA_nC_nPO**



- X** The X axis location (as a radius) of the final pass of the cycle
In G72 mode this is the final pass as a diameter
- Z** The Z axis location of the end of the thread
- I** The starting incremental amount of material to be removed after the first pass
This is to be defined as the diameter removal in diameter mode
- K** The lead of the thread, amount per revolution, .2" max. For larger see G35
- A** used for tapered threading, it specifies the amount the X axis will move over the length of a tapered thread
- C** Causes the infeed to be at an angle, the default is 29
- P** used when you want the tool to keep traveling forward while it pulls out of the work. This will leave no undercut
- O** Including the letter O makes a single pass at the finished depth

Notes:

Diameter or radius mode

The use of the threading cycles is the same for either diameter (G72) or radius (G73) mode. Only the the values of X will be different. The values will correspond to the mode.

Starting position in Z

The tool in most cases will be started at least .1" away from the start of the thread to allow the slide to get up to speed before it makes contact with the material. This number will vary depending on the spindle speed and the pitch of the thread. The courser the thread and faster the spindle speed, the farther away you will need to start. Under worst case conditions the slide can get up to full threading speed in about 1/2 revolution of the ball screw. In most cases this does not mater, however if you are threading from an undercut and the tool has very little room to ramp up to speed, this is very important. You will have to slow the spindle down until the thread gauge goes on.

Starting position in X

The tool should be positioned to take the first pass. The farther away you start the tool, the more passes will be needed. In production runs it pays to experiment a little for the best results and speed.

G33 Threading

Depth of each pass: I

The control will start with removing the amount given as I. Then the control will automatically reduce the depth of the cut as the tool gets deeper. This is a fixed procedure that cannot be changed, it keeps the amount of material removed constant. Start the tool so that it takes a full cut on the first pass.

Retraction position between passes:

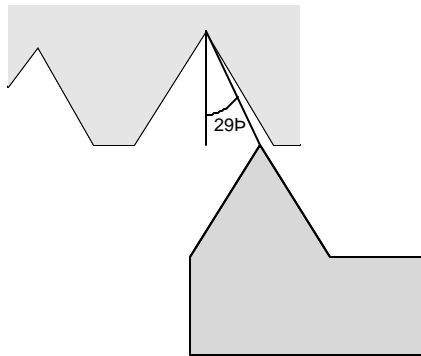
The tool will back away from the starting position plus 3 times the amount of I. Even as the tool gets deeper into the material it will always retract to the same point.

Pullout position in Z when using P option:

The tool will start to pull out at the location given in Z. It will travel beyond Z the same amount as it has to travel in X to reach the retraction position.

Angle infeed C option

If C is included in the G33 command the tool will feed in at an angle. This defaults to 29°. The maximum angle is 30° (based on standard 60° tool geometry) the min is 0°. If you wanted the tool to angle in at 27°, add C27 to the threading cycle command.



The single pass option O can be used for a cleanup pass:

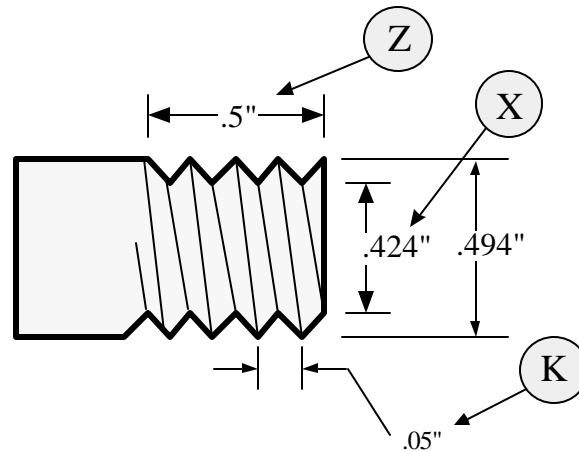
When a single pass is needed write the same threading pass as used for cutting the thread. Just add a O to the command. Be sure to start the thread at the same point and at the same spindle speed. This option can be used with all variations of the threading command.

End of cycle position:

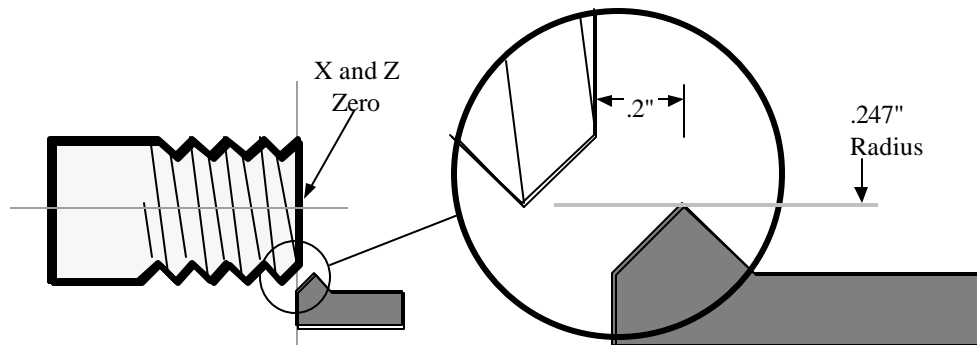
At the end of the threading cycle the tool will return to the starting point.

Threading example

Example: A straight thread, Male, 20 pitch, Minor $\varnothing = .424''$, Major $\varnothing = .494''$, length of thread = $.5''$, and there is no undercut.



For this part the X zero is at the center of the part. The Z zero is at the face of the part.

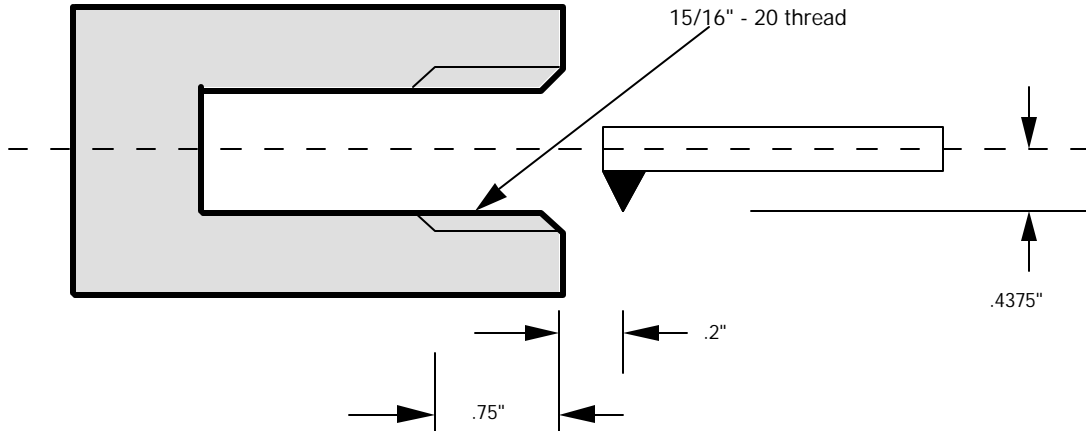


The locations shown above are given in radiuses. The starting location in X is the major diameter, less the amount of the first pass. In this case the major diameter is $.494''$, half of this is $.247''$. Less $.01$ to be taken for the first pass: $.237$

For the above example the program commands are (written in radius mode, G73):

T4F200	Call the threading tool and position it at tool offset
X0Z.2	The offset puts the tool at the part center, $.2''$ from the face in Z
X.237	Positions the tool at the starting radius to cut the thread
G33X.212Z-.5I.012K.05PC	Cut thread

Internal threading example



In the above example we will be cutting a 15/16-20 internal thread. (written in radius mode, G73):

T8 (Internal threading tool)

X.4375Z.2

X.4425

G33X.46875Z-.75I.01K.05C

Call threading tool into position

Give a value to the tool call location

Move tool out to take a .005" cut on the first pass

Threading cycle for internal thread

Threading, Single Pass

The format is: **G33XnZnInKnAnCnPO**

The single pass cycle is the same format as the regular cycle except it has the addition of the letter O in the command. The cycle will automatically move the tool to the finish depth in X and perform a single pass, then pull out to 3 times I in the direction from where the tool was before the command was started. When doing a single pass cycle to clean up a thread that has already been cut, copy the exact same code that you had in the original threading cycle and start the tool in the same spot as before.

It is possible to do single pass threading with the G33 command. The tool will follow the same lead that has already been cut as long as:

- the part has not moved in the collet
- the cycle is started in the same location in Z and X.
- the spindle speed has not changed
- If P or C has been used in the first threading cycle be sure to include it with the single pass

This is useful for:

Deburring: To deburr you can cut the thread and then use the turning tool to top the thread and chamfer the entry and exit of the thread. Then reposition the threading tool and take a clean up pass.

Control: The single pass technique can be used to have greater control over the amount to be removed with each pass.

G33 Threading

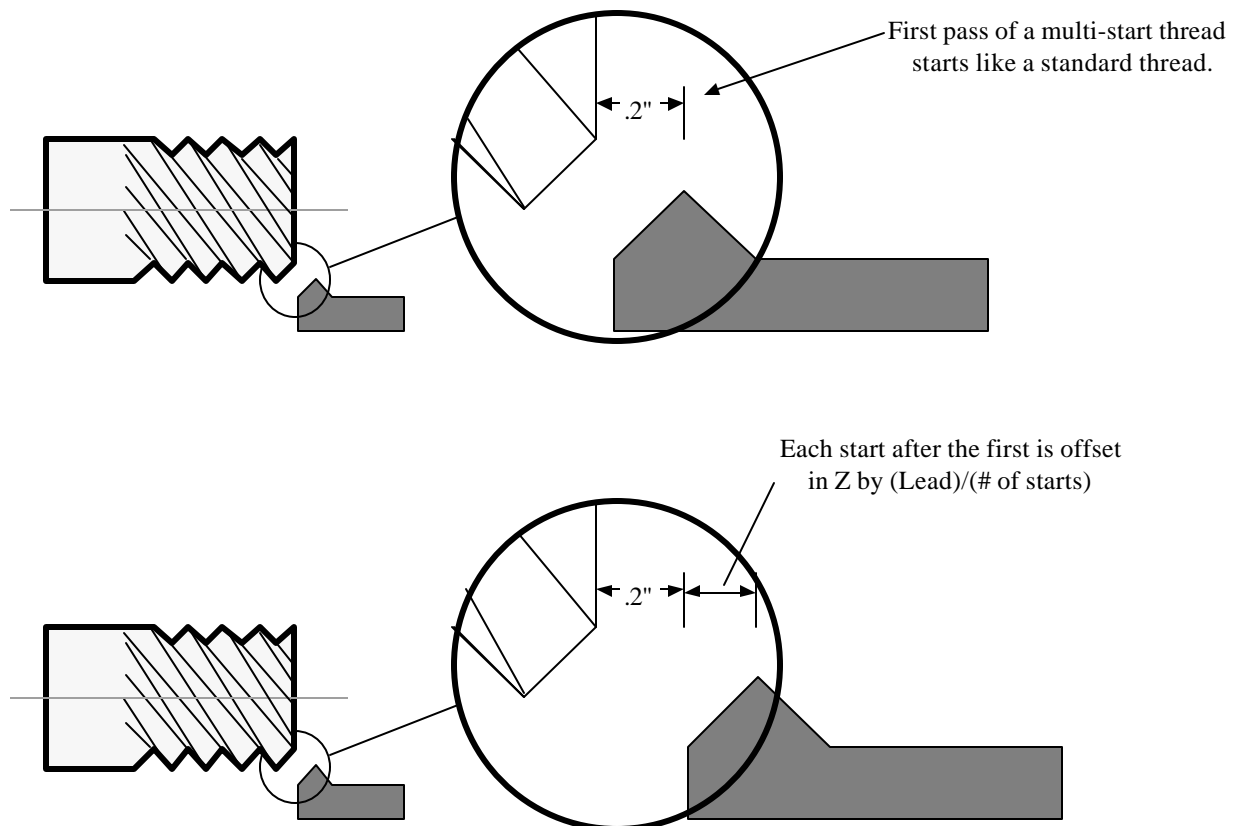
As an example we will go back and take a cleanup pass on the first external example. We will start the tool in Z at the same position as we did with the first cycle, .2". The X starting location will be the same diameter.

T4F300	Call the threading tool and position it at tool offset
X0Z.2	The offset puts the tool at the part center, .2" from the face in Z
X.237	Positions the tool at the starting radius to cut the thread
G33X.212Z-.5I.008K.05PCO	Cut a single pass thread

Threading, Multi Start

Multi start threading can be done by using the regular G33 command. Each start has to be its own G33 command, ie. a three start thread would require three G33 commands. The differences between each of the commands would be:

Value for K: This value would be multiplied by the number of starts. As an example if we had a three start 20 pitch thread, 1 start value for K is .05". With the three start the value is three times that, .150".



Value of Z: The starting position of Z is offset for each start by (pitch)/(number of starts). IE. if we have a three start thread the first start is at the normal .2" from the face of the material. The second start is offset by $.2" + (.15/3) = .25$. The third start is offset by $.25 + (.15/3) = .3$

G33 Threading

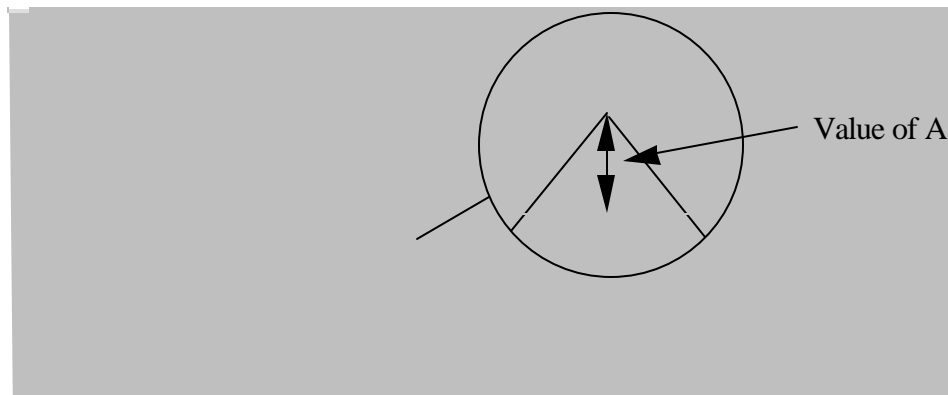
Code for the previous example:

```
G90G94 F300  
T4  
X0Z.2  
X.247  
G33X.212Z-.5I.01K.15P  
Z.25  
G33X.212Z-.5I.01K.15P  
Z.3  
G33X.212Z-.5I.01K.15P
```

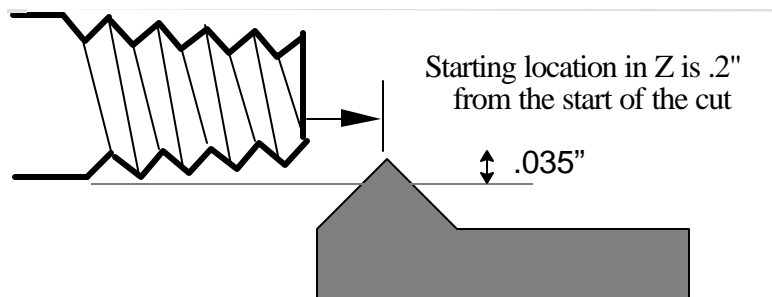
Threading, Tapered

G33X_nZ_nI_nK_nA_nP

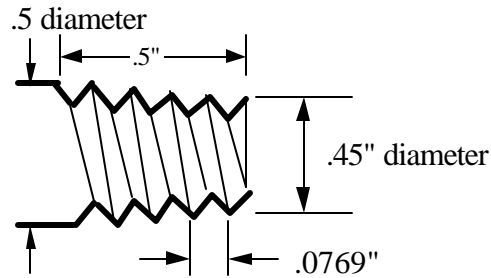
Tapered threading is done with the G33 command that includes an “A”. This is the amount traveled in X over the distance traveled in Z.



Remember that A is the amount over the total distance traveled, this has to include the .2” used to get up to speed.



Example



G90G94F300G72

T5

X0Z.2

X.415

G33 X.365 Z-.5 I.00 5K.0769 A.035

Move to start corrected for taper (remember .2 in Z)

Taper thread

Extra Coarse Feeds (IPR)

G35 - IPR feeds up to 1"

G35F2 - IPR feed up to 2"

Format:

Start Mode - G35 or G35F2 must be on a line by itself. The following line must be a G92XnZn, where XnZn are the current working coordinates.

Cancel course mode - G36. The tool must first be returned to the location where the G35 was first used. Then put G36 on a line by itself. The following line must again have a G92XnZn, where XnZn are the current work coordinates.

The normal maximum feed rate in IPR is .2" per revolution of the spindle. The code would be G95F.2. At this feed rate the spindle rpm is limited to 1500 rpm. because the maximum feed rate of the slide is 300 ipm (1500 x .2 = 300).

Feed rates higher than .2" are available in the G35 mode. Multi-start threads can often require long leads. an 8 pitch 4 start acme thread requires a .5" lead for each of the four starts.

G35 alone enables feeds up to 1.000" per revolution

G35F2 enables feeds up to 2.000" per revolution

Resolution limits are imposed with the use of these codes:

.00005" resolution in normal feed mode

.00025" resolution in G35 mode

.0005" resolution in G35F2 mode

Example

In the following example we will show a four start thread with a course pitch:

T4(Threading tool)

X0Z1

X.5Z.2

G35

G92X.5Z.2

G33X.45Z-.75I.01K.4C

X.5Z.3

G33X.45Z-.75I.01K.4C

X.5Z.4

G33X.45Z-.75I.01K.4C

X.5Z.5

G33X.45Z-.75I.01K.4C

X.5Z.2

G36

G92X.5Z.2

G00Z2