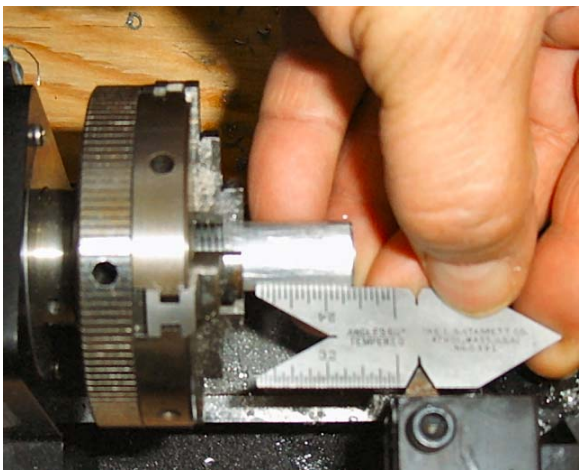
Single Point Threading on a Desktop CNC Lathe.

A small desktop CNC lathe can be used to create precise threads. The threads can be any of the standard US unified series, or the common metric threads. In addition, nearly any combination of screw diameter and thread pitch can be quickly programmed and successfully cut. Machinery's Handbook is an excellent source for all the details of thread design dimensions.

For all modern V cross-section screws, a tool with a 60-degree ground tip is required. This tool is mounted and aligned within a ½ degree or so, with a turned diameter in the lathe chuck, using a threading gage, such as Starrett part No. C391 to align the threading tool.



The goal for the expert thread turner is to mount and set his tool, then run a perfect first part. Few are able to accomplish this due to that fact that a thread is measured at the thread pitch, rather than the minor diameter. The tool tip cuts at the minor diameter, and so the process is always one of cut, measure, adjust, re-cut, re-measure, re-cut, etc, until the thread is finally the right size. Not all low cost CNC controllers permit re-cutting a thread. DeskCNC is one that does.

The tools used to measure the thread pitch are either a thread micrometer, which is relatively expensive, or the 3-wire method, which uses a standard micrometer and 3 steel wires to measure the pitch diameter. Thread wire sets are very inexpensive and include full instructions on how to measure a thread pitch.

Thread micrometers have interchangeable anvils that are selected for a series of thread pitch sizes. After the anvils are mounted to the micrometer sockets, a precision, thread pitch standard is used to set the zero reading of the micrometer. After this the micrometer can be used to directly measure the thread pitch.

FA SMITH, April 10, 2005

Single Point Threading on a Desktop CNC Lathe.

Once the hurdles of tool setting and alignment, and thread measurement are past, the hurdle of creating a CNC program to cut a perfect thread is next. On desktop CNC lathes (and even larger ones), it is common to have an acme or V threaded lead screw. The screws repeat very well as long as they are always used in the same direction. This is different from CNC lathes equipped with zero backlash ball screws. On the ball screw equipped machines; any number of complex routines can be used to turn a screw thread. Without that advantage however, the number of techniques that will yield a perfect and predictable thread are reduced. A technique, similar to that used on a manual lathe with a compound, yields very accurate and repeatable CNC turned screw threads.

The secret to the success is to take many small cuts, following along the back flank of the thread, but never actually cutting that side with the tip of the threading tool. This type of cutting action always opposes the direction of the forces on the lead screw and the tool will never be pulled into the workpiece.

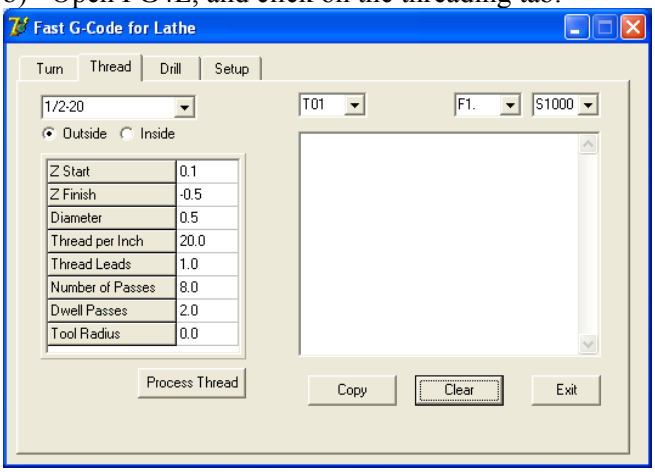
Three other factors to creating a good thread:

1) Know the total depth to cut so the pitch diameter turns out the right size.
2) Take many equal sized cuts. This does not mean take the same depth of cut for each pass, but rather take the same cross sectional area of material
3) Correctly adjust for tool tip radius. A radiused-threading tool is often used, especially with Carbide inserts. The total depth of cut must be adjusted when a radiused tool is used.

All these factors are incorporated into the **Fast G-code for Lathe** (FG4L) program. This program automatically creates a CNC program that accounts for all the above factors and helps the CNC turner to make a perfect thread, the first time.

Here is the process to create the thread:

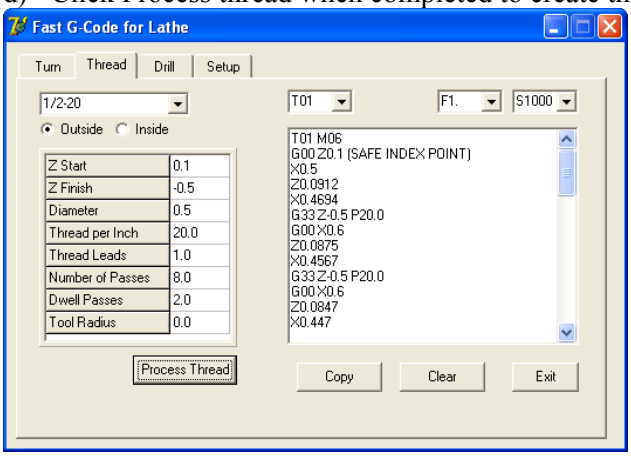
a) With a part mounted in your lathe chuck or collet, turn the major diameter of the thread. For unified series threads, such as ½-13, this diameter is .500 in.
b) Open FG4L, and click on the threading tab.



c) Select the desired thread from the drop down list, select ½-20, set the various parameters to match your thread requirements. The standard threads are auto filled and you can then override both diameter and threads per inch.

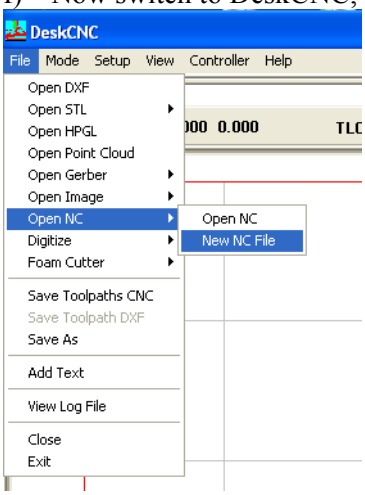
Single Point Threading on a Desktop CNC Lathe.

d) Click Process thread when completed to create the G-code

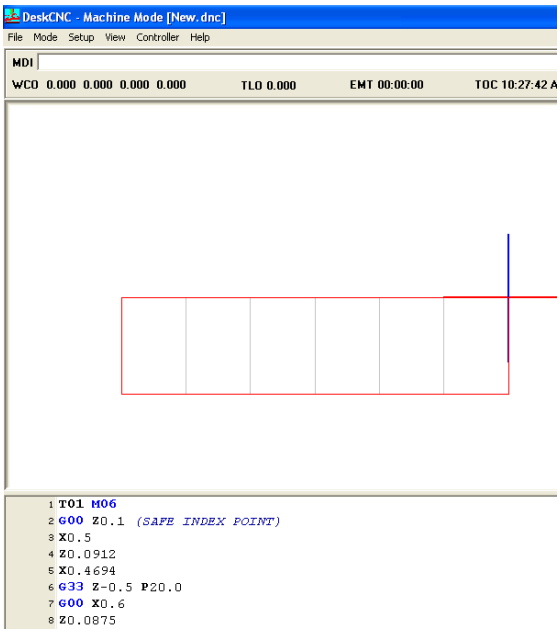


e) Click Copy to write the program to the Windows Clipboard

f) Now switch to DeskCNC, File-Open NC-New NC File

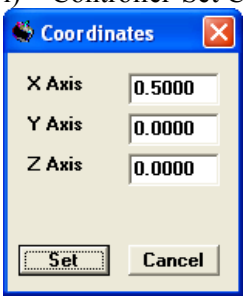


g) Click in the program box at the bottom of the screen and paste the code into the box with Ctrl-V

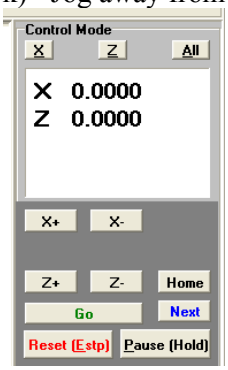


Single Point Threading on a Desktop CNC Lathe.

h) Touch the threading tool to the part diameter, by using the .001 jog and a piece of paper.
i) Controller-Set Coordinates, and enter the X axis value



j) Jog the tool away from the part in X, and jog over to the front of the part. Use a magnifier to set the center of the tool exactly even with the front of the part and use the same command to set the Z=0.
k) Jog away from the part and run the CNC program by clicking the Go button



The spindle must be running to cut the thread. You can adjust the variable speed while the cut is running to improve the chip formation. If cutting metals, be sure to use a thick cutting oil which will not be wiped off by multiple passes of the threading tool. Oily tapping fluids such as Cool-tool or Tapmatic are suitable for thread cutting. Water based coolants and Tapmagic are too thin and generally are not suitable for thread cutting. For threading ferrous materials with high-speed steel tools, heavy weight sulpherized oils (dark brown) may be needed to get a satisfactory surface finish.