



Threading and Spindle Speeds/MASSO Conversational Threading Program

Understanding How to Calculate a Spindle Speed that Will Not Exceed the Capabilities of Our Machine

1. The larger the number of threads per inch, the smaller the actual thread size. Whereas with metric threads, a larger pitch value is going to be a larger thread.

Example: 16 TPI is equal to .0625/rev with a thread height of .0313", whereas 20 TPI is equal to .050/rev with a thread height of .0245".

NOTE: The largest thread size that we recommend for our machine is 16 TPI (1.5 mm) or smaller.

Larger threads than 16 TPI can be cut with multiple passes at a smaller depth of cut (.001" (.025 mm)) and lower spindle speeds, but it will take a lot of time and many passes! Not recommended.

2. When you are doing a turning operation, you have the ability to adjust both the feed rate and the spindle speed.

When you are cutting threads, the feed rate is dictated by the thread pitch, or lead (the number of threads per inch or mm). Therefore, the only value that you can adjust is the spindle speed. The higher the spindle speed, the faster the feed rate must go to match the thread pitch.

3. The maximum feed rate on our lathe is 32 In/Min.
 - a. The higher the feed rate, the lower the torque output of the stepper motor.
 - b. This means that for courser threads where the thread height is larger, you will want to use lower RPM.
4. For courser threads you will also want to use the low speed/high torque pulley setting on our headstock spindle. This setting has a maximum speed of 1350 RPM. This is the pulley setting that you should use for cutting course threads. However, this will limit your maximum RPM for any other operations, such as turning, drilling, or grooving.

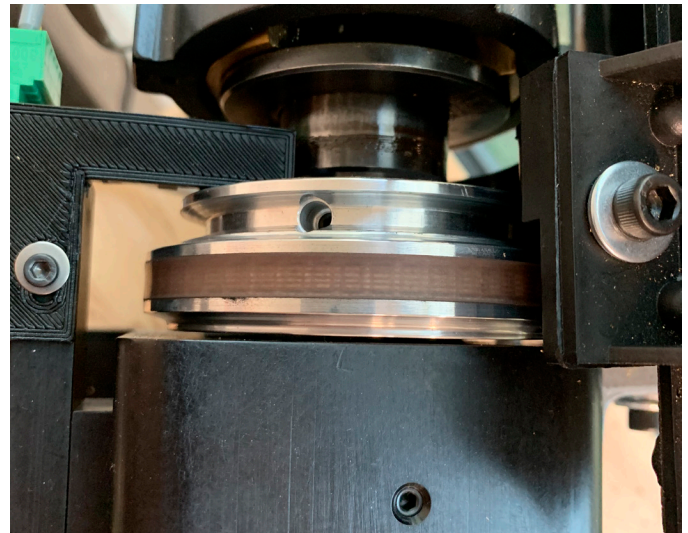


FIGURE 1—The V-belt is positioned in the high-torque, lower-speed pulley groove.

5. The high speed pulley setting on our headstock has a maximum speed of 2800 RPM. This pulley setting is suited for finer threads and all other operations.

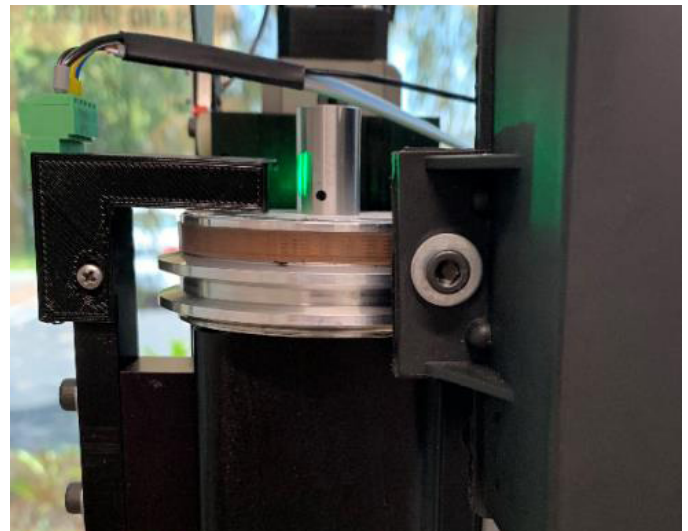


FIGURE 2—The V-belt is positioned in the high-speed, upper pulley groove.

How to Calculate Feeds in “Inches per Minute from Inches per Rev”

When you are cutting threads, you are giving the machine a feed rate that is in Inch/Rev (or mm/Rev) which is equal to the pitch of the thread.

Example: A (20) Thread Per Inch thread has a pitch that is equal $1.0"/20 = .050"/\text{rev}$.

An 8 x 1.25 mm thread has a pitch of 1.25 mm/rev.

In order to cut the 20 TPI thread, I am going to use a spindle speed of 300RPM.

To calculate what my feed in “In/Min” will be, I take the RPM and multiply it by the Feed/Rev.

$300\text{RPM} \times .050 \text{ In/Rev} = 15 \text{ In/Min}$

NOTE: The 15 In/Min is at the midrange of my maximum stepper motor feed rate (32 In/Min) and also at the approximate midrange of the stepper motor’s torque level.

If I want more torque, then I would lower the RPM, which in turn lowers the feed rate in “In/Min.”

Example: $200\text{RPM} \times .050" = 10 \text{ In/Min}$

Now, if I should pick a faster spindle speed, I could exceed the maximum feed rate of the machine (32 In/Min) and the stepper motors will just stall out when they attempt to move at that speed.

Example: $700 \text{ RPM} \times .050 = 35 \text{ In/Min}$

This spindle speed is not going to work!

MASSO Conversational Threading Software

Link to the MASSO Lathe Conversational Wizard: <https://masso.com.au/masso-documentation/?section=lathe-conversational-wizards>

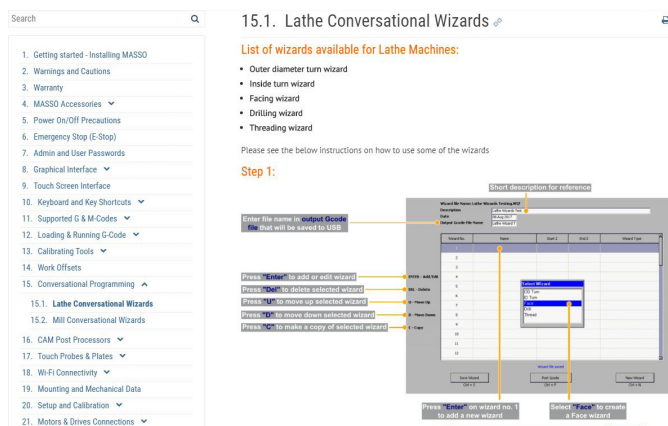


FIGURE 3—The MASSO Lathe Conversational Wizard web page.

NOTE: The MASSO link above covers all of the basic information. We have been working with MASSO to fine tune their software for our machine and our needs. The information that follows covers those changes, settings, and special notes for our Sherline machines.

Whether you are using Cad-Cam software or not. We recommend that you use the conversational program for your threading cycles. If you are using Cad-Cam software,

cut and paste the MASSO conversational program into your main program.

We are going to set up the conversation page to cut a $1/2"$ – 20 thread. The picture below shows the main page for threading with all of the setting boxes filled.

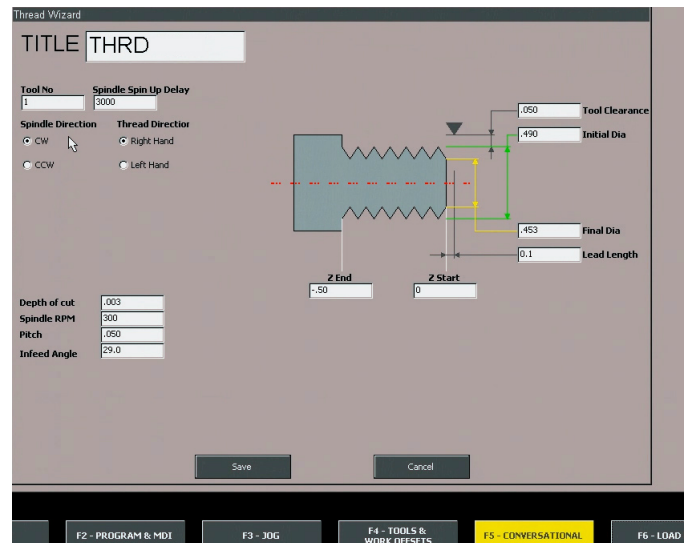


FIGURE 4

In the upper left corner, you have the following settings:



FIGURE 5

Title: Input (6) characters that define the operation

Tool No: Input the tool number for your threading tool.

Spindle Spin Up Delay: Input the number of seconds that you want the machine to wait while your spindle speed adjusts to a lower RPM change for your threading cycle. A value of “1000” is equal to 1 second.

Spindle Direction: For our machines this will always be “CW” (clockwise rotation).

Thread Direction: Choose Right Hand or Left Hand thread.

Note for Left hand threads: Our Spindle does not turn CCW. In order to cut left hand threads on our machine, you will need to start at the end of the thread (or shoulder) and feed out to the front of the part.

In the lower left corner, you have the following settings:

Depth of cut	.003
Spindle RPM	300
Pitch	.050
Infeed Angle	29.0

FIGURE 6

Depth of cut: This is how deep the cutter will advance into the material on each pass. Depths of .001" - .005" can be set here. We recommend .002" or .003" for most threads

Spindle RPM: Calculating this value was covered previously. We recommend "300" as an average spindle speed for most threads in most materials.

NOTE: With the software upgrade of 6/2020 we have been able to cut fine pitch threads at an RPM OF 600. You will need to play around and see what the limit is for any given thread. We still recommend 300 RPM as an average spindle speed for most threads and materials.

Pitch: Enter the pitch (or lead) of your thread here. For our 1/2"-20 thread the value will be .050.

Infeed Angle: This was added for our machines. You can choose either 0.0 degrees or 29.0 degrees.

NOTE: 0.0 and 29.0 are the only two angles that the software will accept.

If you choose 0.0 degrees, the threading tool will advance straight into the part at 0.0 degrees on each pass. This means that the cutter will be cutting on both sides of the 60-degree angle. It also means that with each increased depth of cut, the actual area of the cutter that is cutting the part will increase. The pattern will look like the picture below.

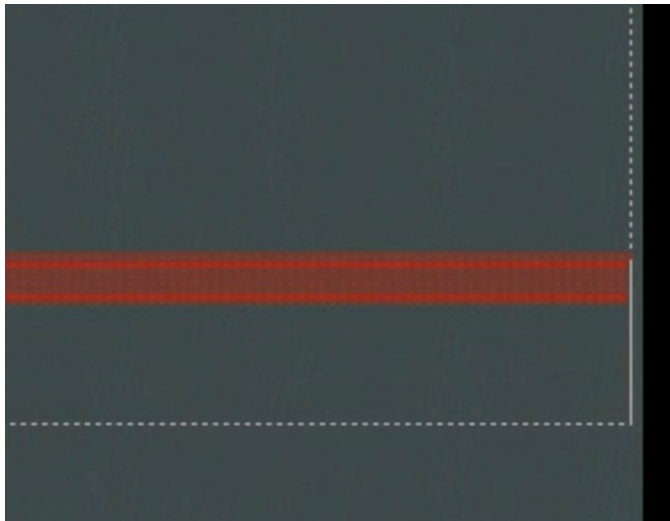


FIGURE 7— 0.0 degrees thread-cutting pattern.

If you choose 29.0 degrees, the threading tool will advance into the part at 29 degrees. By advancing into the part at 29 degrees, the lead edge of the cutter will cut the majority of the thread on each pass. The surface area of the cutter that is actually cutting the thread will increase with each pass. However, the overall surface area used will be less than if you had chosen 0.0-degrees. The pattern will look like the picture below.

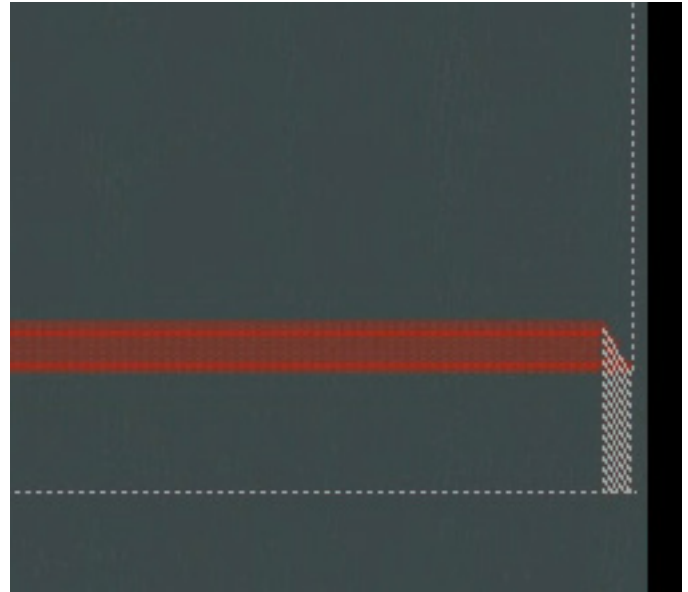


FIGURE 8— 29.0 degrees thread-cutting pattern

On the main screen are the following settings:

Tool Clearance	.050
Initial Dia	.490
Final Dia	.453
Lead Length	0.1
Z End	-.50
Z Start	0

FIGURE 9

Tool Clearance: Enter the distance above the thread major diameter that you want the tool to move to after each cutting pass. We input .050".

Initial Diameter: This should be the "Major Diameter" of the thread. We input .490".

Final Diameter: This should be the "Minor Diameter" of the thread. We input .453".

Lead Length: This amount represents how far in front of the part you would like to start the threading cycle. This gives the computer some advance space to get both the spindle speed and the feed rate in sync before the cutter makes contact with the part. We generally choose .100" to .200" when there is space available.

Z Start: This is the Z-axis position for the start of the thread. We input Z 0.0.

Z End: This is the Z-axis position for the end of the thread. We input -.50.

NOTE: If we were cutting a left-hand thread, our Z start will still be Z 0.0". Our Z finish will be Z -.500". The software reverses the direction of the threading cut when you input "Left Hand." Our Lead Length will be .100" (to cut past the front of the part). (see Figures 10 and 11).

Tool No	Spindle Spin Up Delay
1	1000
Spindle Direction	Thread Direction
<input checked="" type="radio"/> CW	<input type="radio"/> Right Hand
<input type="radio"/> CCW	<input checked="" type="radio"/> Left Hand
Depth of cut	.002
Spindle RPM	300
Pitch	.050
Infeed Angle	29.0

FIGURE 10

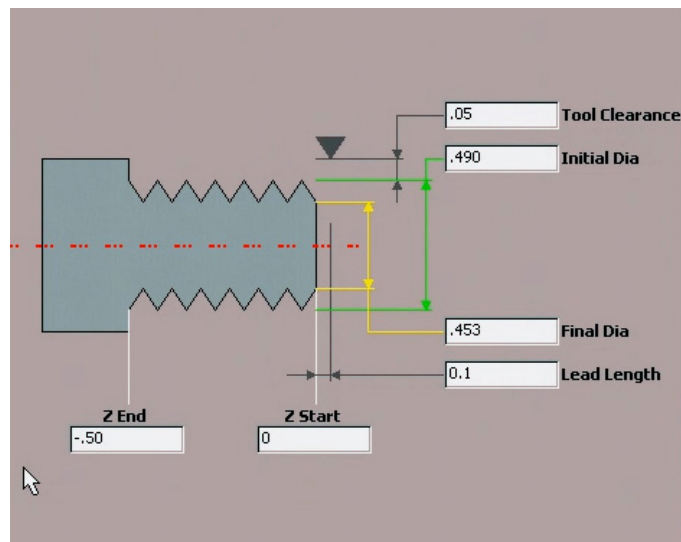
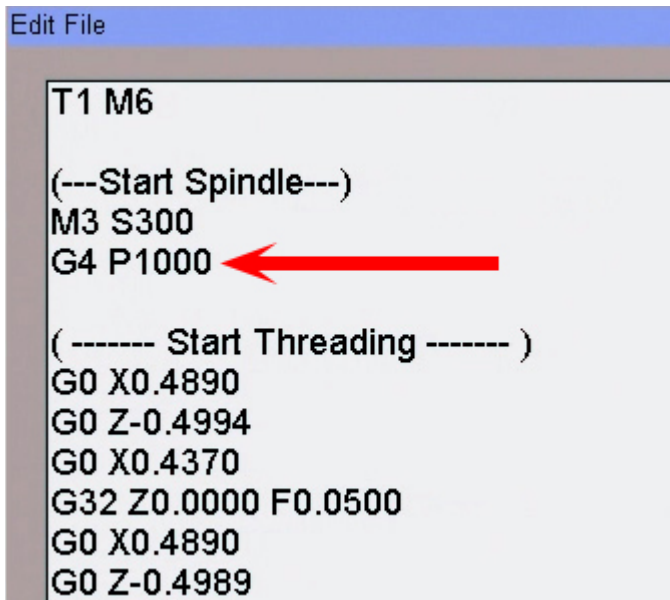


FIGURE 11

Some Additional Notes on the Threading Cycle

1. With the MASSO software and the Optical Encoder, the spindle speed and the feed rate are always in sync. If there is an RPM drop or gain during the threading cycle, the feed rate will decrease or increase with the RPM change. This will prohibit bad threads and cross-threading.
2. On finer pitch threads, use an Infeed Angle setting of 0.0. The surface area of the threading tool that will be engaged with the part will not be sufficient enough to warrant cutting with the lead edge of the tool.
3. On courser pitch threads use the Infeed Angle of 29.0.
4. With many other threading cycles, the tool will ramp into and out of the threaded area at an angle. This leads to incomplete threads at the beginning and end of the threading pass. It also means that you must compensate for this ramping distance when your thread is going up to a shoulder. With the MASSO software there is some ramping at the beginning of the thread and there is about one thread length of bad thread at the end of the threading pass. This gives you almost a complete thread all the way to the shoulder of a part.
5. If you are doing a turning operation or other operation that is using a higher spindle speed, you will need to put a value in the "Spindle Spin Up Delay." When you do this, the Wizard will put a G4 P1000 (or whatever seconds you want in the P value) in directly after the M3 block (see Figure 12).



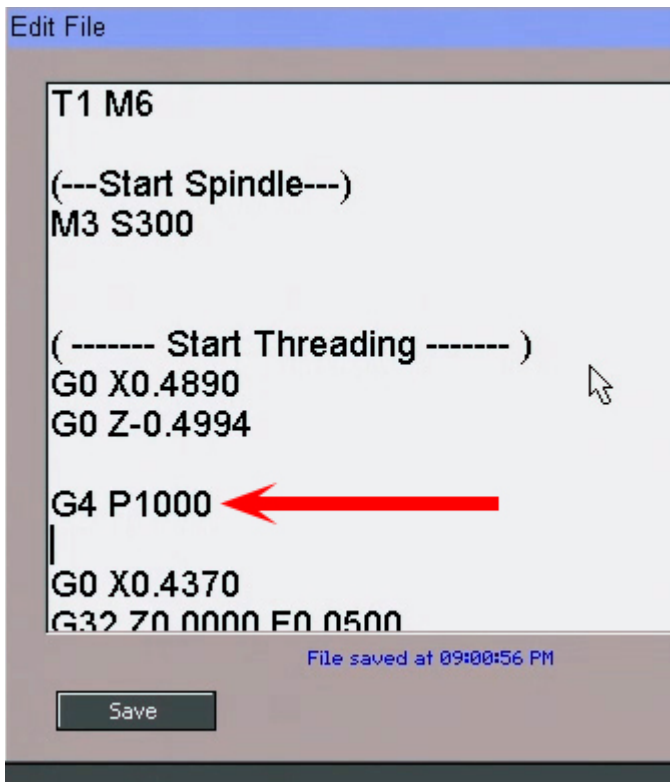
```

Edit File
T1 M6
(---Start Spindle---)
M3 S300
G4 P1000
( ----- Start Threading ----- )
G0 X0.4890
G0 Z-0.4994
G0 X0.4370
G32 Z0.0000 F0.0500
G0 X0.4890
G0 Z-0.4989

```

FIGURE 12—The Spindle Spin Up Delay is generated.

Unfortunately, the spindle speed change doesn't happen until the machine makes a move. Because of this, you will need to cut and paste this block down lower in the program for it to work effectively (see Figure 13).



```

Edit File
T1 M6
(---Start Spindle---)
M3 S300
( ----- Start Threading ----- )
G0 X0.4890
G0 Z-0.4994
G4 P1000
G0 X0.4370
G32 Z0.0000 F0.0500

```

File saved at 09:00:56 PM

Save

FIGURE 13—Cut and paste the Spindle Spin Up Delay.

The larger the difference is between the two spindle speeds, the longer it will take for the controller to adjust the spindle to the new speed. You may need to try a longer time period.

If you are starting the program with the spindle off, the threading cycle may move in the Z-axis with the spindle off by a distance equal to your clearance amount, but it should not cut the part when the spindle is not moving. It

may begin the threading cut when the RPM is lower than programmed. However, because the controller is taking readings from the optical encoder, the feed rate will be adjusted automatically to match the current spindle RPM. Generally, in this situation, the RPM range will increase to the programmed speed as the threading cycle is cutting the first thread. Since the first threading pass is generally not more than a witness line on the part this "RPM to feed" adjustment will not be detrimental to the quality or accuracy of the finished thread.

Sample Program

This program is for a 10-32 thread

(---Program units: inch---)

G90 G20

(---MASSO - Threading Wizard---)

T1 M6

(---Start Spindle---)

M3 S300.000

(----- Start Threading -----)

G0 X0.183 Z0.050

G32 Z-0.500 F0.031

G0 X0.238

G0 Z0.050

G0 X0.178 Z0.050

G32 Z-0.500 F0.031

G0 X0.238

G0 Z0.050

G0 X0.173 Z0.050

G32 Z-0.500 F0.031

G0 X0.238

G0 Z0.050

G0 X0.168 Z0.050

G32 Z-0.500 F0.031

G0 X0.238

G0 Z0.050

G0 X0.163 Z0.050

G32 Z-0.500 F0.031

G0 X0.238

G0 Z0.050

G0 X0.159 Z0.050

G32 Z-0.500 F0.031

G0 X0.238

G0 Z0.050

G0 G53 Z0

G0 G53 X0

(---Stop Spindle---)

M5

(---END OF PROGRAM---)

M30

Low-Speed, High-Torque Pulley settings

1. The headstock pulley has a high range and a low range. The high range is good for most operations. The lower

range has slower maximum RPM with higher torque. The low-speed belt setting is designed for heavy cuts in hard material and also for courser thread cutting.

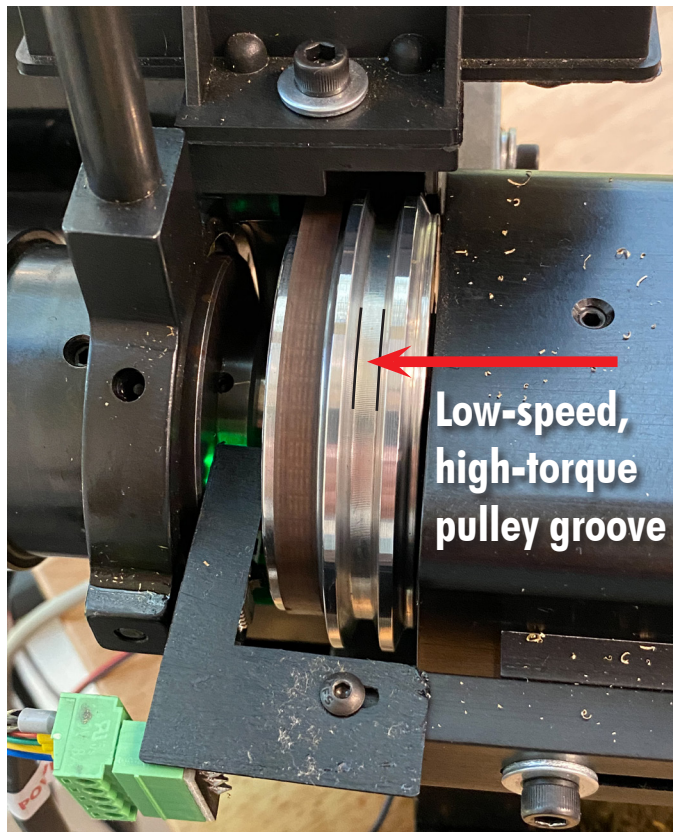


FIGURE 14—Use the pulley groove closest to the headstock for low-speed, high-torque settings.

2. If you use this pulley position, the maximum RPM of the spindle will be 1350 RPM, whereas the maximum RPM in the high-speed position is 2800-3100 RPM. You will also need to change your settings for your spindle on the F1 Setup page so the control can sync the spindle speed and the Z-axis feed for thread cutting.
3. To do this click on the F1 Setup page. Then double click on “Spindle” in the function settings.

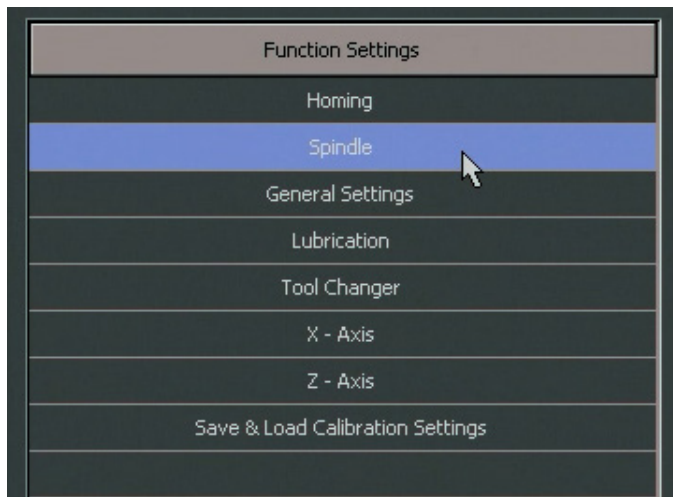


FIGURE 15

4. The “Spindle RPM at 10 volts” will be set at 2800-3100 for the high-speed pulley setting.

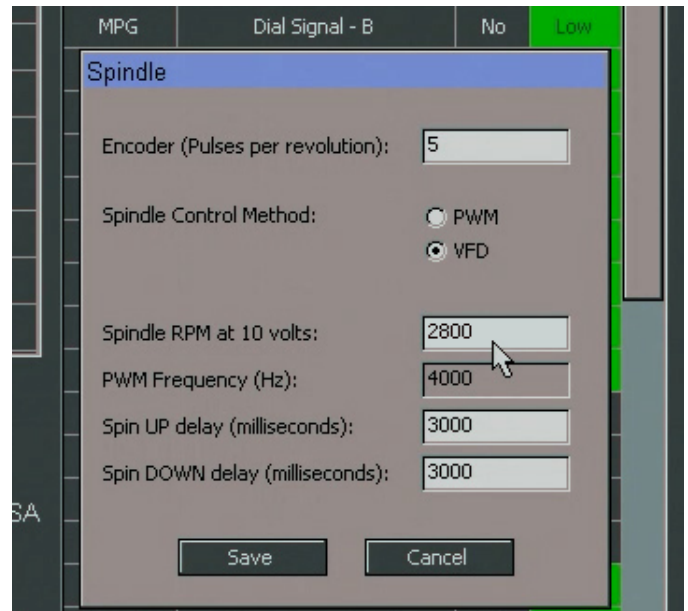


FIGURE 16

5. You will need to change this setting to 1350 and then click on Save.

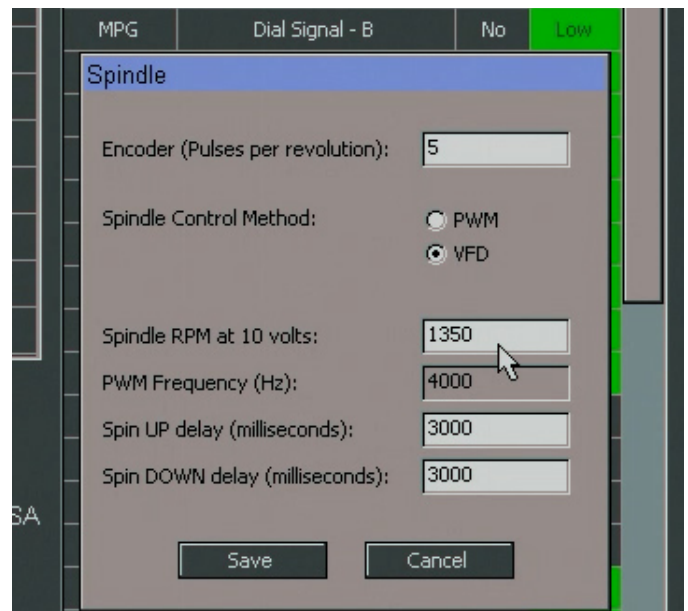


FIGURE 17—Change the Spindle RPM setting from 3100 to 1350 for the low-speed, high-torque setting.

Now you are set to cut threads using the low-speed/high-torque pulley.

Thank you,
Sherline Products