TB170 (Rev 3) – Thread Repair (Thread Recut) on a Centroid CNC control

Overview
This document describes how to pick up and recut an existing thread on the Centroid lathe CNC control. The example here assumes external threads, that are cut from right to left. However, the same techniques can be used on internal threads and left-to-right threads as well.

Thread recut procedure in steps:

1.) Load the part
2.) Index the spindle
3.) Program the thread in Intercon
4.) Touch off on a good section of the existing thread
5.) Teach-in and then reset the "Thread Face" position in the Intercon thread program

Thread recut procedure in words: The operator simply has to rotate the spindle to an index location and then jog the tool to align it with an existing thread. At that point the operator cursors to the "Thread Face" coordinate in the threading canned cycle and then press the Teach "Z" key to teach in the touch off position. Then simply add a distance divisable by the thread lead to that value so the thread starts at the beginning of the threaded part or just beyond it.

Follow these steps

1.) Load the part: Load the damaged threaded part into the chuck and dial it in so that the threaded workpiece is concentric with the spindle and runs true.

2.) Index the Spindle: Identify and mark the spindle Index location. This is accomplished by having the control show us where the spindle index pulse is located as the operator slowly rotates the spindle by hand. Once the index location is identified, mark the spindle location so it is easy to reposition the spindle back to the index location. (Note: Once the spindle index position has been identified and permanently marked on the machine the steps below describing how to identify the index will not have to be repeated for any future thread repair.)

a.) Depress Estop. Go to the PID screen, Press "F1" Setup, "F3" Config, type in password 137 press enter, "F4" PID.

b.) Now, slowly rotate the spindle by hand (put the gear box in neutral) and once every revolution of the spindle the index position will appear beside the spindle axis label (usually the 5th or 6th axis label and spindle axis label is typically labeled "N") The spindle index location is identified by an asterisk, this asterisk will appear beside the axis label like so: N*
Now, slowly turn the chuck by hand until the asterisk "*" appears on the screen. If you passed by it turn back and forth slowly until the "*" stays on the screen beside the spindle axis label.

c.) Now mark this spindle location in some clear and impervious manner. And press ESC several times to return to the main menu.

3.) Program the Thread using Intercon Threading Canned Cycle.

a.) In Intercon Select "F5 (Thread)" and enter all the thread information as usual except the "Thread Face" and "Clearance" value.

```
Intercon Lathe v3.12                        Current Part: Photo Thread 2.lth

 Operation              End              N0002 Thread Cycle
  #   Type      X(D)      Z      Tool
0001  Header                             Thread Type        : External
0002  Thread    1.9703   0.3098  02      Designation        : 1 7/8-16 UN
0003  End Prog  1.9703   0.3098  02      Class              : 3A
                                         Clearance      Z:      0.0000 INC
                                         Thread Face    Z:      0.0000
                                         Ending         Z:     -1.5000
                                         Minimum Cut Depth  :   0.0010
                                         First Cut Depth    :   0.0100
                                         Tool Number        : 0202
                                         Spindle Speed      :     600 RPM
                                         Finish Pass Amt.   :   0.0020
                                         Num Spring Passes  :       3
                                         Pre/Post Cycle Pos.  : None


                                         Press "Details" for more.




        Tool                  Help   Math   Details  Graph  Teach   Accept
                                     Help                   Mode
         F2                    F5     F6       F7      F8     F9      F10
```

Press "F7" (Details) and complete the thread data entry.

```
Intercon Lathe v3.12                          Current Part: Photo Thread 2.lth
 Operation              End              N0002 Thread Cycle
  #   Type        X(D)        Z     Tool
0001  Header                                Thread Type        : External
0002  Thread      1.9703   0.0000  02      Designation        : 1 7/8-16 UN
0003  End Prog    1.9703   0.0000  02      Class              : 3A
                                           Thread Angle       :    59.0000°
                                           Threads / Inch     :    16.0000
                                           Thread Lead        :     0.0625
                                           Major Diameter     :     1.8703
                                           Minor Diameter     :     1.8005
                                           Chamfer Amount     :     0.0000
                                           Taper Amount       :     0.0000
                                           Taper Angle        :     0.0000°
                                           Clearance       Z:     0.0000 INC


                                           Press "Main" to return.



                                     Math                         Teach
           Tool              Help    Help    Main    Graph        Mode    Accept
                                                                   
            F2               F5       F6      F7      F8           F9       F10
```
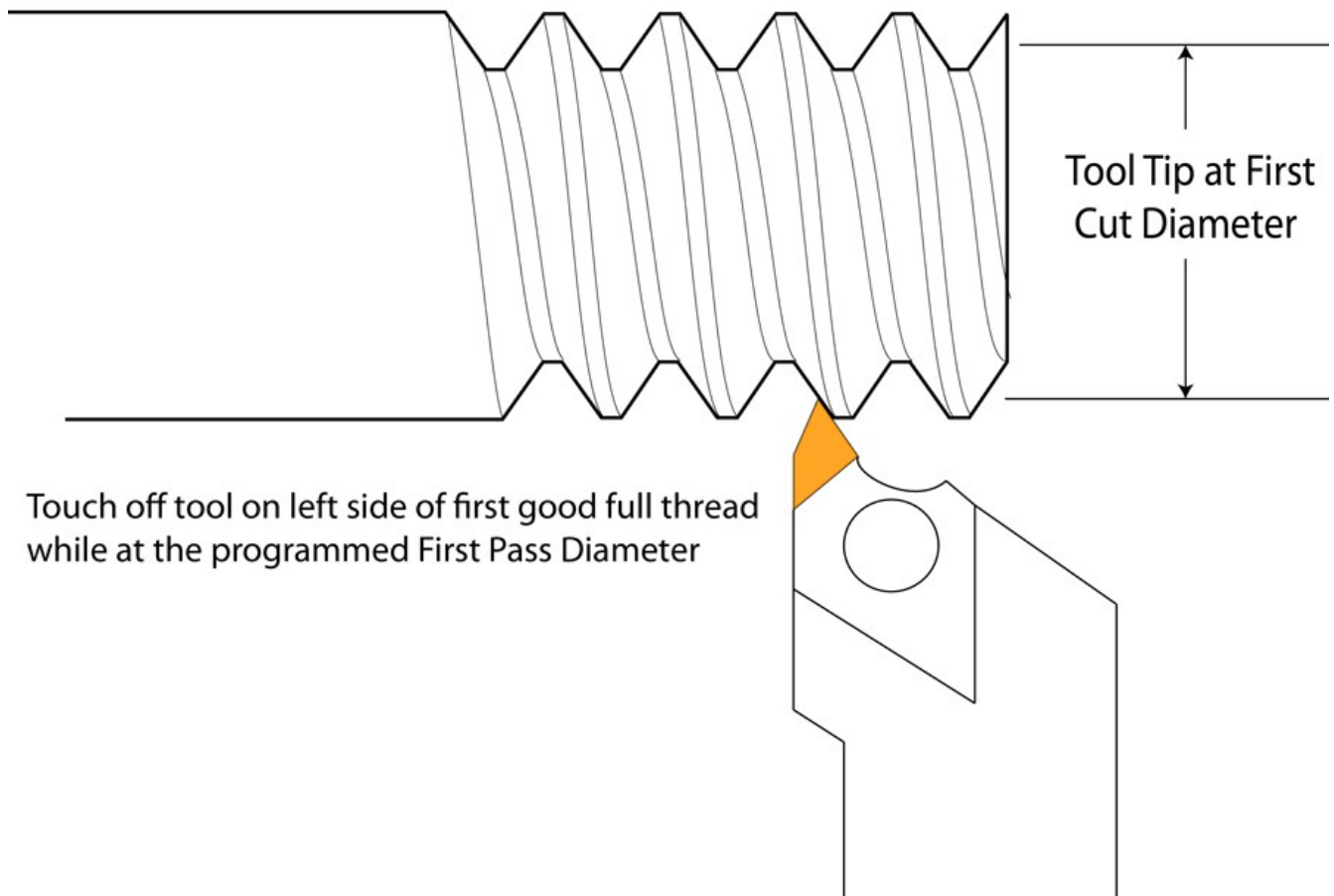
b.) After you've entered the thread data, figure out what the First Cut (pass) <u>Diameter</u> will be for the programmed First Cut Depth in the threading cycle, We will use this First Cut Diameter to line the tool up with the existing thread!

The first pass diameter of the threading cycle will be made at the Major Diameter minus (2x the First Cut Depth). The First Cut Depth is determined by the operator. In our case we chose .01" for a First Cut Depth.
For our 1 7/8 -16 external thread the Major Diameter = 1.8703" and we decided to program the First Cut Depth at: .010" so Intercon will make the first cut at this diameter: 1.8703 – (2*.01") = 1.8503" as the First Cut Diameter for this threading cycle. Make a note of this First Cut Diameter as its used in Step 4 below.

**4.) Touch off the Threading tool to a Location of the Existing Thread that is not Damaged.**

a.) Set Z and X offsets for the threading tool and any other tools you will be using as usual.

b.) Set the Z-axis part zero at the end of the part or at your customary location.

c.) If the offsets for the threading tool are not already active (as displayed in the status box in the upper right corner of the screen) then activate them using the MDI prompt. For example, to activate tool offsets for tool press F3 for MDI and enter the command: T0505 then press CYCLE START. Press ESC to leave the MDI prompt.

d.) Turn the spindle around to the index pulse position using the spindle index mark that you located earlier in Step 2.

e.) Touch off the tool on the left flank of the first full good thread:

Position the threading tool in the middle (in Z) of a good thread, carefully jogging the tool tip between the crests of the first two fully formed threads, now jog X to the first cut (first pass that the threading cycle will make as programmed in Intercon) diameter.   Slowly and incrementally jog the tool tip right (Z+) until it just touches the left flank of the first full good thread. Note the Z axis position on the DRO.

Tool Tip at First Cut Diameter

Touch off tool on left side of first good full thread while at the programmed First Pass Diameter

5.) Set "Starting Z" Position in the Intercon Program using Teach.

a.) In Intercon Press "F9" (Teach Mode) to display the Teach Mode DRO on the Thread Cycle Screen.

b.) Highlight the "Thread Face" value in the threading operation.

c.) Teach in the Z touch off position. Press "Z" on the keyboard to copy the Z position from the Teach DRO to the Thread Face field in Intercon.
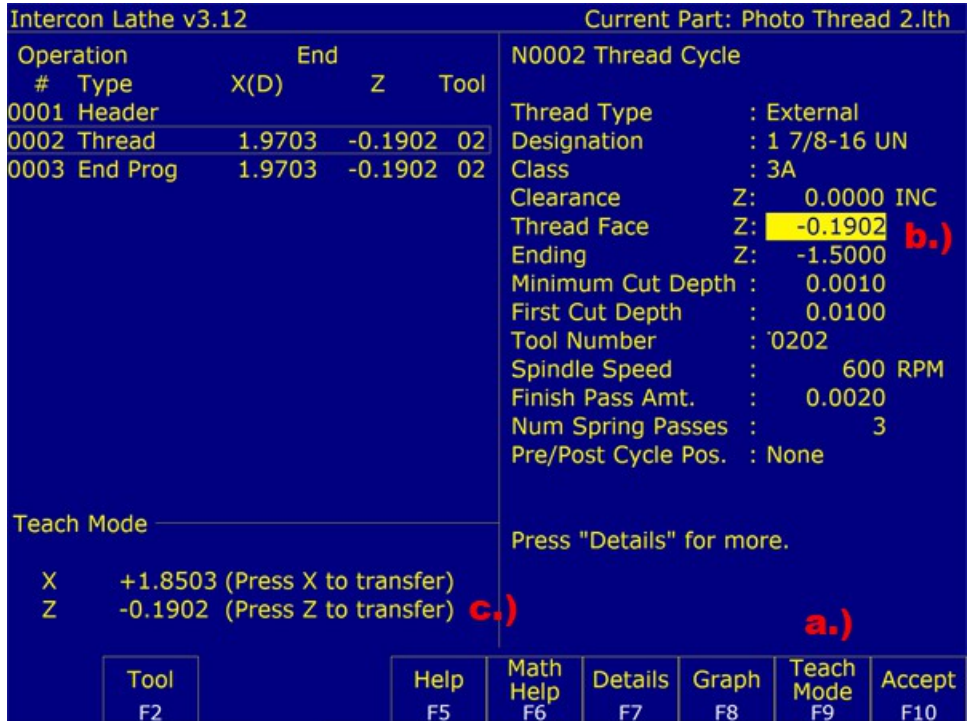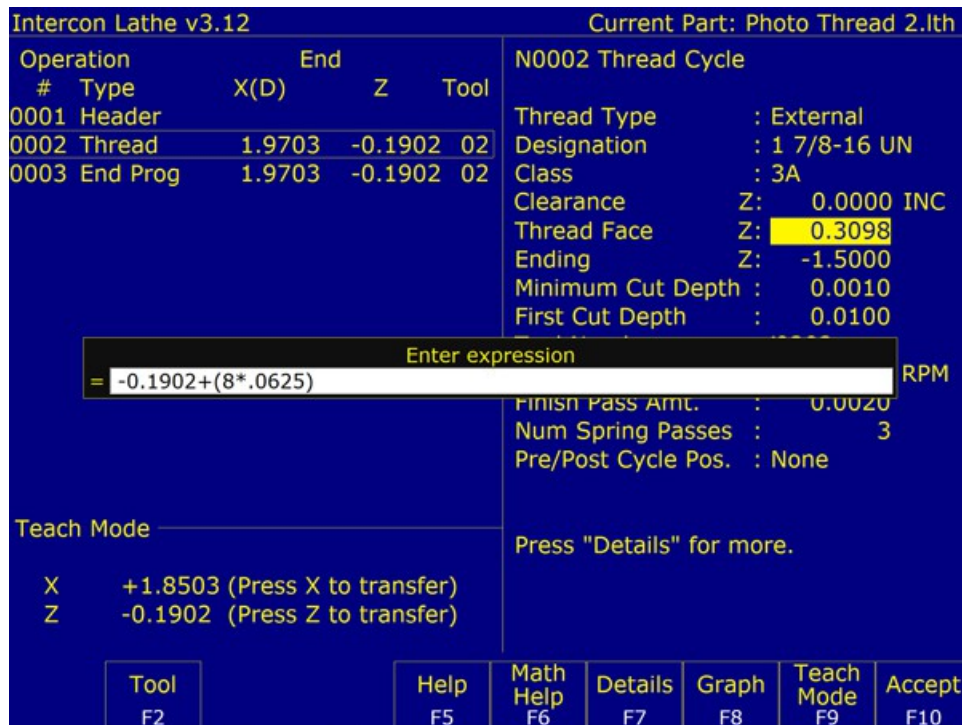

You now have a thread start position that is synchronized with the existing thread. The only problem is that it is already on the thread, and you probably don't want to start threading at the touch off position, you want to start at the beginning of the thread or better yet further out in air before the thread starts. So let's start some whole number of thread turns out to the right. Typically you would add enough thread distances that the threading cycle will start to thread in air beyond the beginning of the part. The proper starting Z position is also dependant on the Thread Infeed Angle. There are two cases when figuring out a proper starting Z position, one where the Thread Infeed Angle is = 0 (equal front back loading on the threading tool) and one where the thread infeed angle is not 0 (common value is 59 degrees, Thread Angle – 1 degree). Don't know which one to use? The simpilest is using a Thread Infeed Angle of 0 (equal tool loading front to back) Thread Infeed Angle of zero simple runs the tool down the middle of the threads, for thread recutting this typically is fine since the tool usually isn't doing a lot of work, just cleaning up a damaged area.

d.) To calculate a Starting Z value with a Thread Infeed Angle of 0: press Enter to modify the Thread Face value and add several times <u>the thread lead</u> to the value you got from the Teach DRO. In our example we need about 1/2" from the touch off position to start the thread clear of the part. 1/2" is an even multiple of our thread lead, exactly 8 threads. So adding .5" to the touch off position of -.1902 yields a Thread Face value of .3098"

(For more complicated thread pitch calculations you can use Intercon's expression calculator: With the cursor on the Thread Face Z value press enter and then type "=" and now cursor to the right of the touch off value -.1902 using the "->" right arrow key, and add a distance to the Thread Face Value that is an even multiple of the thread pitch. In our example the pitch is 16 thread per inch with yields a distance of .0625"

per thread. (1"/16=.0625") So, we just eyeball the number of thread distances to add, in our case its about 8 threads to be beyond the actual start of the threads. So, we will add 8 times .0625" to the Taught-in Thread Face Z value.   -.1902 touch off position +(8 threads*.0625 thread lead)= .3098" is our real Thread Face Z value.

```
Intercon Lathe v3.12                        Current Part: Photo Thread 2.lth
Operation              End              N0002 Thread Cycle
 #   Type        X(D)      Z    Tool
0001 Header                             Thread Type        : External
0002 Thread      1.9703  -0.1902  02    Designation        : 1 7/8-16 UN
0003 End Prog    1.9703  -0.1902  02    Class              : 3A
                                        Clearance       Z:    0.0000 INC
                                        Thread Face     Z:    0.3098
                                        Ending          Z:   -1.5000
                                        Minimum Cut Depth :   0.0010
                                        First Cut Depth   :   0.0100
                  Enter expression
       = -0.1902+(8*.0625)                                          RPM
                                        Finish Pass Amt.   :   0.0020
                                        Num Spring Passes  :      3
                                        Pre/Post Cycle Pos. : None

Teach Mode
                                        Press "Details" for more.
   X    +1.8503 (Press X to transfer)
   Z    -0.1902 (Press Z to transfer)

           Tool              Help   Math   Details  Graph  Teach  Accept
                                    Help                   Mode
            F2                F5     F6      F7      F8     F9     F10
```

e.) To calculate a Starting Z value with a Thread Infeed Angle of 59 degrees: Perform the calculation in the step above <u>and add</u> the value that results from this calculation: TAN(Thread Infeed Angle/2)*First Cut Depth. In our case TAN (59°/2)X.01"= .00566". So, add .00566" to .3098" (calculated in part d. above) for a new starting Z value of .3098"+.00566"= 0.31546"

f.) Now set the <u>Ending Z value</u> to some reasonable position at the end of the thread. If the thread ends in a relief groove, then you can stop the cut anywhere in the groove. If the thread ends in a chamfer out, then you should error on the side of chamfering out early. In any case, Ending Z does not need to be synchronized the way Starting Z does.

6.) Post the Intercon program and run the program to recut the threads just like any other job.

Notes:

1.) Special Conditions: Sometimes it is not always easy to touch the tool off a thread at the First Cut (pass) Depth. You may need to jog the tool deeper into the thread (a smaller diameter) in order to reliably touch off a good thread flank. In this case, to determine the Thread Face value in Intercon, you will have to adjust the resulting deeper touch off Thread Face value by following the Thread Infeed Angle back out to the Major diameter.
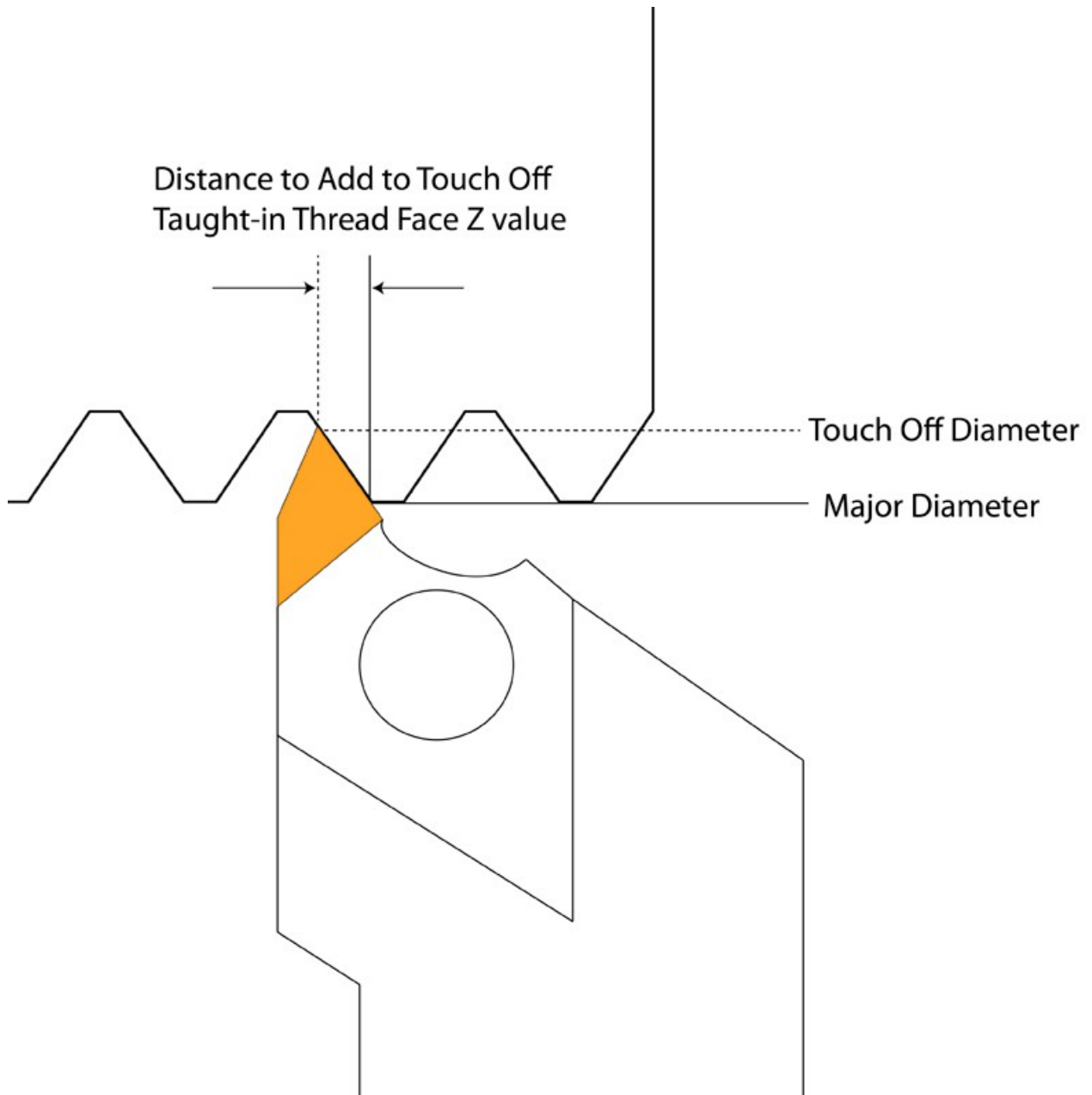
To calculate the proper Starting Z while deeper in the thread use this formula:
((Major Diameter/2) – (Tool Touch Off Diameter/2))*TAN(1/2 Thread Infeed angle) = Amount to add to Starting Z value

((MD)/2 - (TTOD)/2) * TAN (½ Thread Infeed Angle) = Amount to add to Starting Z in Intercon


For example, we are using a Thread Infeed Angle of 59 Degrees, and we programmed the First Cut to be at a diameter of 3.860", with a Major Diameter of 3.880", but we have to go in to a 3.790" diameter to locate the thread. Calculate the Amount to add to the Thread Face Z value in Intercon.

(3.880"/2) – (3.790"/2) * TAN (½ of 59 degrees) =
1.94– 1.895 * TAN 29.5 =
.045 * .56577 = .02546"

Now add 0.02546" to the Thread Face Z position that you got off the Teach DRO.
This is the Thread Face Z value

If you don't have a calculator handy you can do this calculation with Intercon's built in expression calculator.
Right after you touch off and teach in the "Starting Z Value"
While the cursor is on the Starting Z value, Press Enter to modify it and then type "=" and Press "->" right arrow key to move the cursor to the right of the value, now type after the Z value "+ Tan(29.5) * .090/2 and press enter. (.090" in this example is the MD-TTOD)
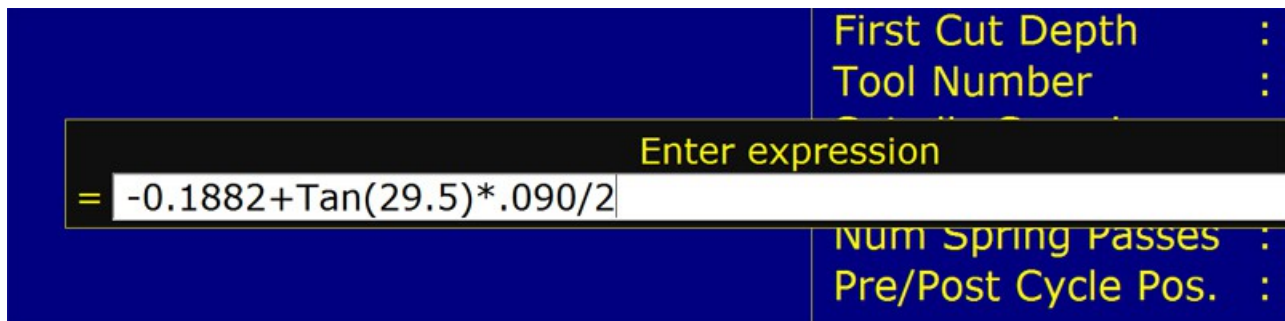
Now to start the threading cycle before the actual threads add some thread distance to start the thread recut cycle before the actual threads. Follow the same procedure as in Step #5.) above.

2.) Clearance Amount.   In our example we did not use the Clearance amount feature in Intercon since we programmed our Z face value out away from the actual Z face so the thread would start in air out beyond the threads.   If you need to add more clearance room so that the threading cycle starts further away from the actual thread (usually to give the spindle time to get up to speed). Simply enter a distance that is a multiple of the Thread Lead for the Clearance value. Our example Lead was 16 threads per inch so any distance amount that is divisible by .0625 (1/16) would be a proper clearance amount. Clearance amount is Incremental from the Starting Z face value.

3.) "Thread Angle" Intercon is referring to the Cutting Compound Angle also called Thread Infeed Angle For example: a common 60 degree thread, the "Thread Angle" value in Intercon is commonly set to 59 degrees (a degree less than the thread angle). This loads the front of the cutter more than the back. A 0 value for "Thread Angle" produces even loading of the threading tool (front and back of insert is loaded the same) and cuts in the middle of the threads. Setting the "Thread angle" to 59 loads the cutting insert to one side, this is called a Compound Infeed Angle and makes for a better smoother cut.

Intercon (and G76) can do both types and all "infeed" angles in between.

4.) Intercon will use the "First Cut Depth" and the "Minimum Cut depth" determined by the operator (material and tooling dependent) to make automatic multiple passes from the Major Diameter to the Minor Diameter.

5.) Find Marker Pulse on some older CNC controls with AC servo motors: In the PID screen press "F8" Drive to get the marker pulse to show.