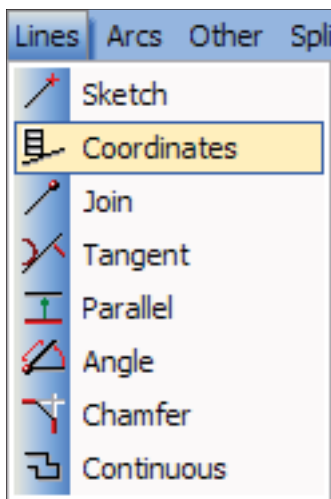


FAQ #40: How do I make an ID thread with a lathe?

Q: I need to make an ID (Inside Diameter) thread with BobCAD-CAM on my lathe. How would I do this?

A: This is a simple one. The only change that is required from an OD (Outside Diameter) thread is that the tool orientation be switched from an OD direction to an ID direction. That's really all there is to it! See the following example for an illustration.

This exercise is to show the user the basics of using the threading feature. It will be a simple 2"-13 UNC ID screw thread. It will use the default 1/8 .003R 60° threading tool, and is drawn and programmed in diameter mode.

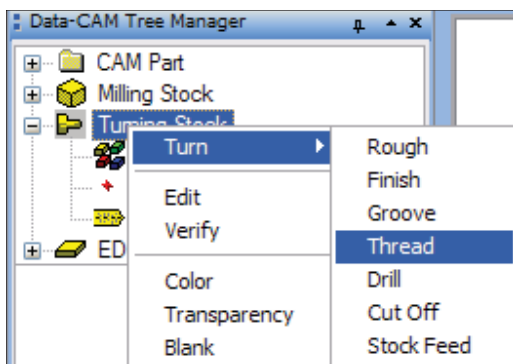


Step 1: Click on **Line** and then **Coordinates**. In the **Data Entry** tab of the **Data-CAM Tree Manager**, enter in these values:

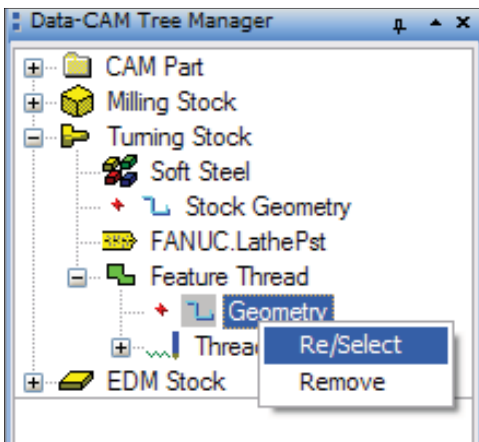
Start Z: 0.
Start X: 2.
Start Y: 0.
End Z: -1.
End X: 2.
End Y: 0.



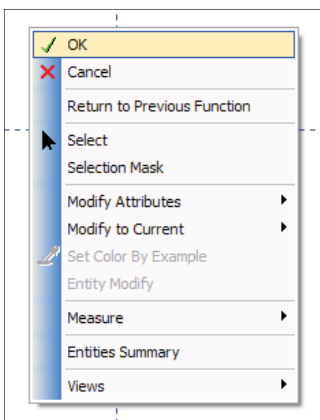
Step 2: Click on the **CAM Tree** tab of the **Data-CAM Tree Manager**. Right-click on **Turning Stock**, choose **Turn**, and then **Thread** from the menu. A new **Feature Thread** feature will be added to the tree.



Step 3: Right-click on the **Thread** item under the new feature and choose **Re/Select**. Hover the mouse cursor over the left end of the line so that it highlights. Hold down the Shift key on the keyboard and left click. The line will have become selected and show two arrows on it, one at the front and one in the back.

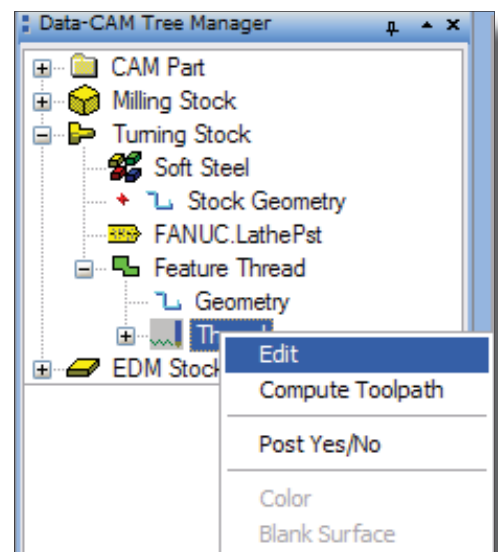


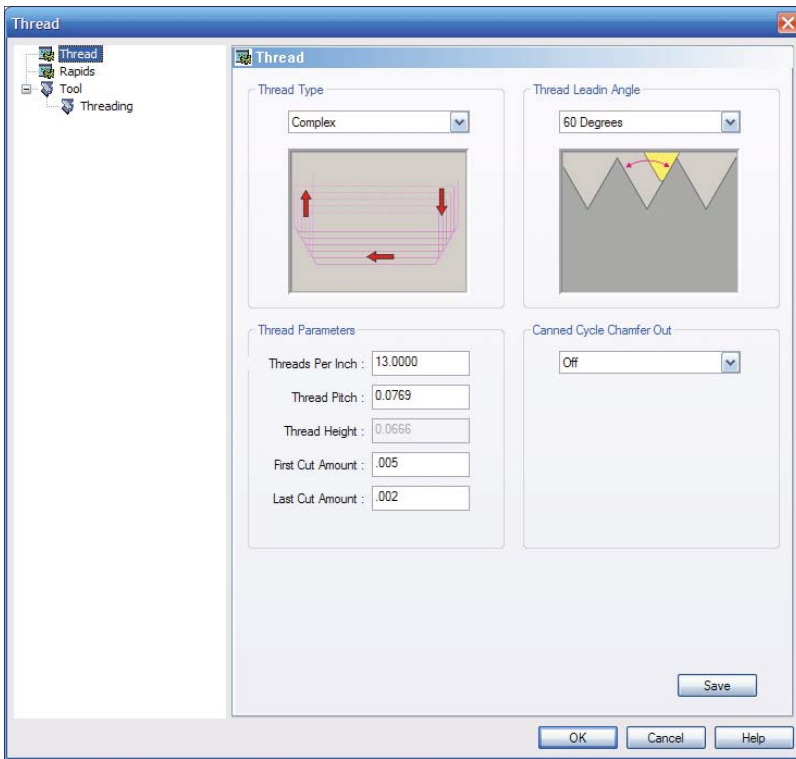
Note: Threading operations require the geometry to be only ONE single line, selected in the direction of thread cut. By hovering the cursor over the left end of the line while holding the shift key down before clicking on it, the software will highlight the line from right to left.



Right-click on the **Workspace** somewhere and choose **OK**. The geometry will be set in the feature and will no longer have the red dot next to it.

Step 4: Right-click on the **Thread** item in the new feature. Choose **Edit**.





The first item in the tree to the left of the box that will pop-up is called **Thread**. In the right-hand pane, set the **Thread Type** to **Complex**. This will ensure that the posted NC code will use the post processor's complex thread canned cycle, normally a G76 cycle on most controls.

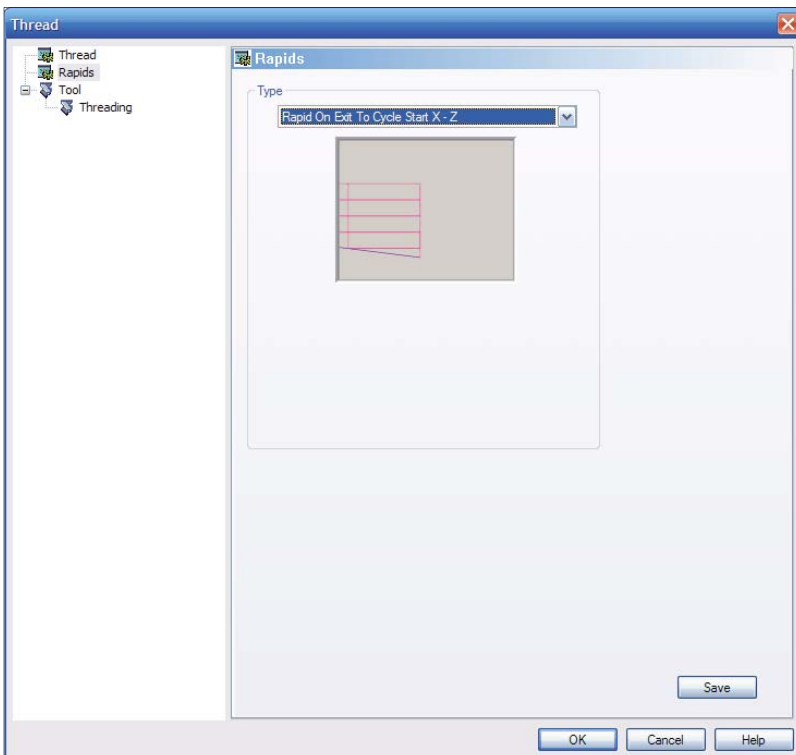
Set the **Thread Leadin Angle** to **60 Degrees**. This is the most common leadin angle for threading so it is the default in the feature.

Since this is an ID thread for a screw there will normally be a wall to the left of the thread, so set the **Thread Chamfer** to **Off**. If this were a pipe thread, there would normally be a chamfer on the end, and in that case, it would need to be set to **On**.

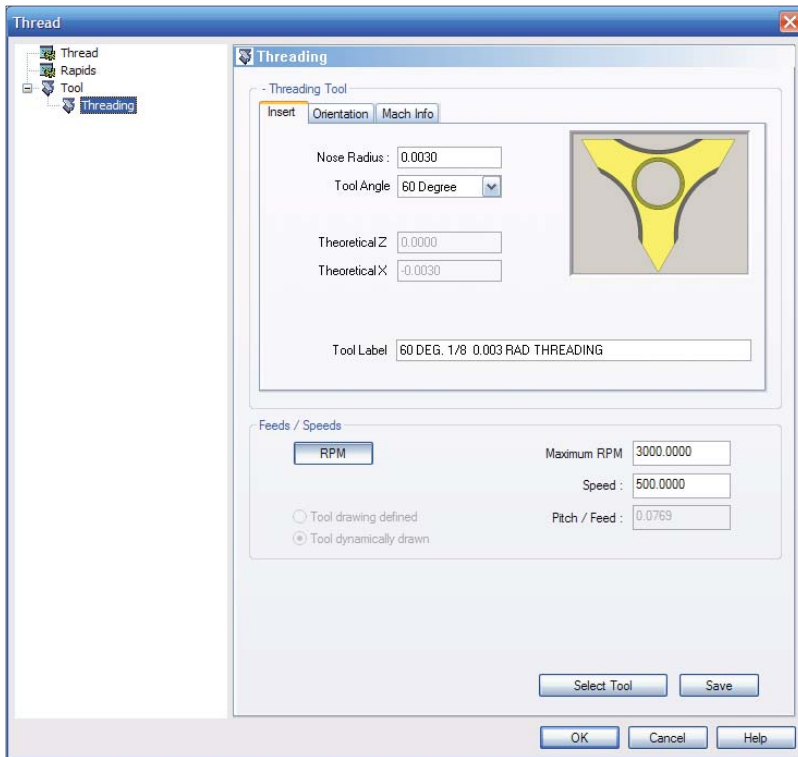
Under **Thread Parameters** is most of the data used to cut the thread. Enter in these values:

Threads Per Inch: 13
First Cut Amount: .005
Last Cut Amount: .002

*Note: The **Threads Per Inch** and the **Thread Pitch** settings work from one another. When one of them is changed, the other changes to match. For example, if the **Thread Pitch** were set to .05, the **Threads Per Inch** parameter would automatically change to 20, and vice-versa.*

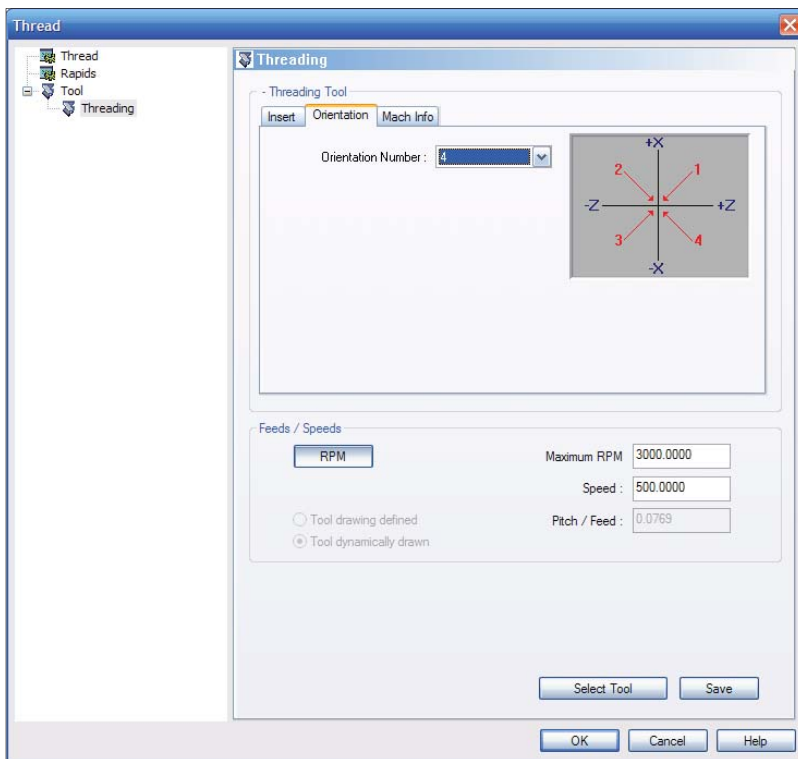


The next item in the tree on the left is **Rapids**. Click on it, then set the drop-down box in the right pane to **Rapid on Exit to Cycle Start X - Z**. For most operations the default is **Rapid on Exit To Tool Home X - Z**, but since this is an ID thread cut, there's really no reason to have to return it to tool home as the tool change on the machine will perform the same action. The cycle start point will be upper-right corner of the generated toolpath.



Under **Tool** in the tree there will be one item, **Threading**. Click on it to access the tooling parameters. The default tool is a 1/8" 60 degree threading tool with a .003" radius, and this is the tool to use for this thread.

Note: The form the thread will take will be dependent on the tool loaded into the machine. If the thread is required to have a square form, as in an ACME thread, for example, the proper tool must be loaded into the machine. BobCAD-CAM calculates threading passes according to the angle on the flank of the thread, not its form.



Next, click on **Orientation** and choose **4** from the drop-down box if it isn't chosen already. Number 4 is the default for ID threading operations in BobCAD-CAM. To cut an OD thread, simply switch the orientation to an OD direction and the operation will automatically reverse direction.

Step 5: The setup for the threading operation is complete, so click **OK**. Right-click on the **Thread** item in the feature and click **Compute**. BobCAD-CAM will have generated the toolpath for the thread.