

SOLIDWORKS – Add More Sizes to Thread Command

Learn how to add more thread sizes in Thread command

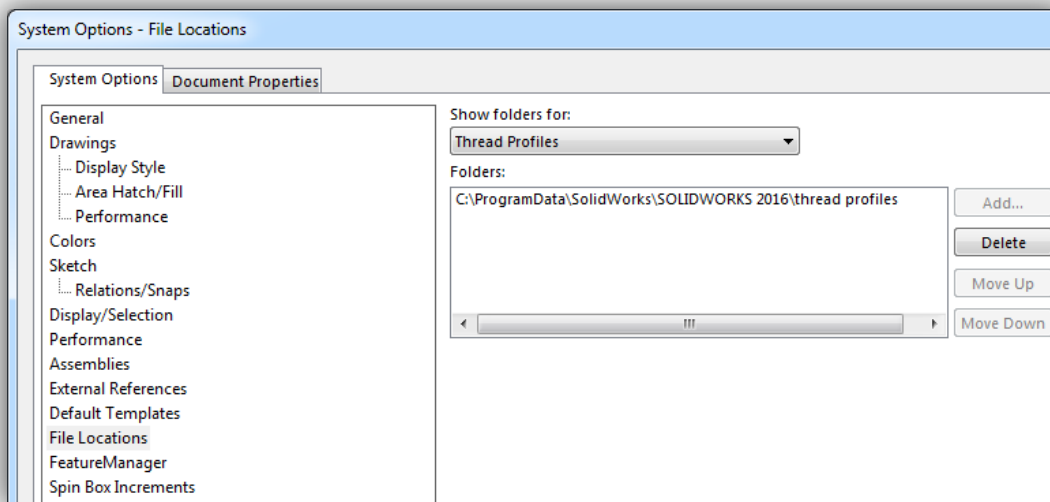
Starting with SOLIDWORKS 2016 you have a command for creating threads, no more manually creating a helix and a sweep. What you may find is that not all “standard” sizes are present, much less any custom threads.

Let’s see what steps are needed to add the ones you want:

- 📁 Find where the thread profiles are located
- 📁 Open the correct one (metric, inch, etc)
- 📁 Add needed size(s)

Thread Profile Location:

- 📁 In SOLIDWORKS browse to Tools, Options, System Options, File Locations, then pick Thread Profiles.

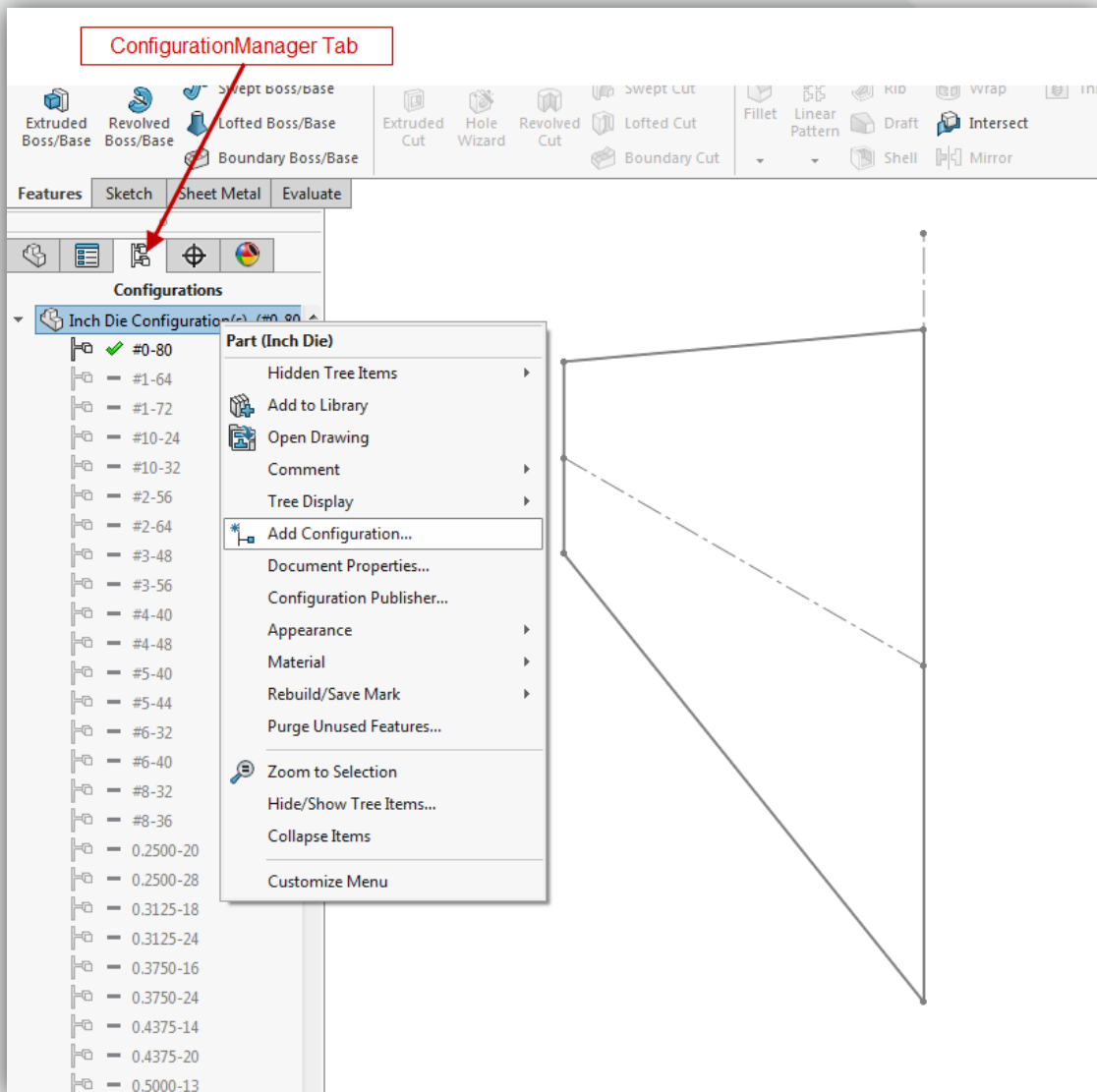


Browse to that location and open the correct file:

- 📁 Make sure you put new sizes in the right file so you can find and use them

Add needed size(s):

- 🟢 Open the right file (inch/metric/tap/die)
- 🟢 Click on the ConfigurationManager Tab
- 🟢 RMB the file name and choose Add Configuration, the size and TPI
- 🟢 Edit the sketch dimensions for that configuration to reflect the new size



- 🟢 New size will now be available for selection in the Thread command
- 🟢 Save thread profile folder on network drive and have other users change that location in their settings to share new sizes with all