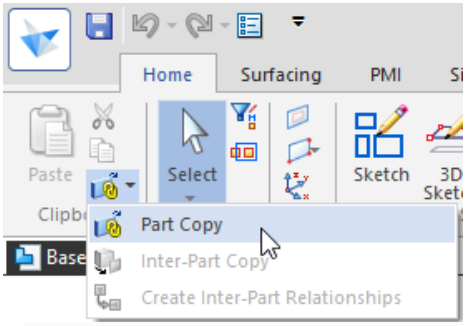
Creating single-body Part file of an Assembly

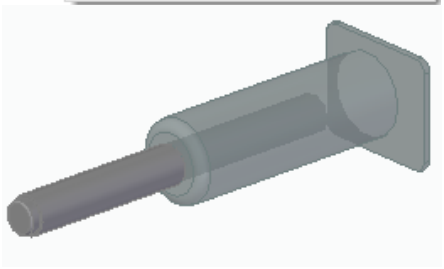
This Tech Tip looks at how to create a part file of an assembly that is formed of a single body feature.

The advantage of converting an assembly into a single body part file is that you can send the part file out to customers, without giving away your IP, but you could also use it as a replacement of a sub-assembly, thus reducing the amount of face processing required, which increase performance.



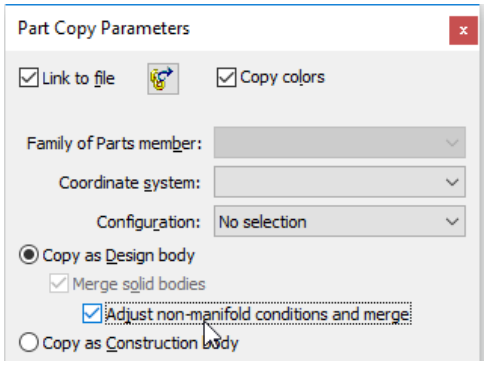
The first step in the process is to create a new part file and ensure you are in the Ordered mode, as this allows the assembly to be linked (and updated) to the part file. It is possible to do this in Synchronous, but the link to the assembly will be lost.

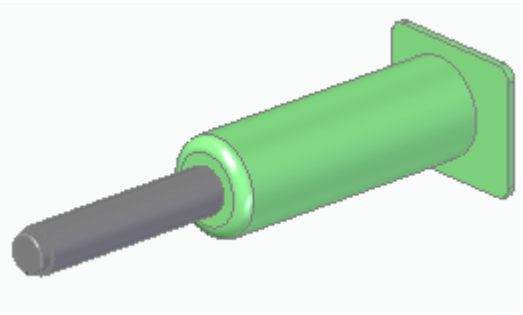
From the Home tab, click on the "Part Copy" command and select the assembly. It will then open up the Part Copy options.

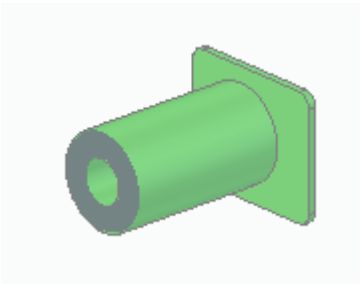


Leaving all the settings as defaults will result in the assembly being copied similar to the assembly. Take this simplified version of a car jack, you will notice that the individual parts are individually distinguishable. This will be a result of there being slight gaps between parts, or parts just touching (also known as non-manifold bodies), and is obviously

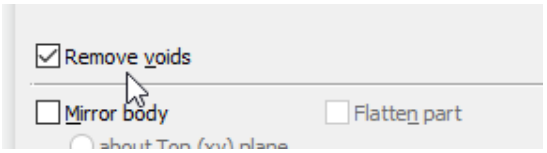
If we change the settings by turning on the "Adjust non-manifold conditions and merge" option, the result is that it adds a little overlap of material, thus joining the bodies. If we use the clipping planes though, you will see that some of the internal geometry is still visible.







The final setting available is the "Remove voids" option, just below half way down the options. When we turn this on, any internal geometry is also removed.



The part file can now be exported to send to customers without losing your IP, but can be useful for them to build their assembly around.