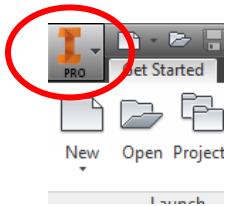


Inventor to 3D Printer File Type

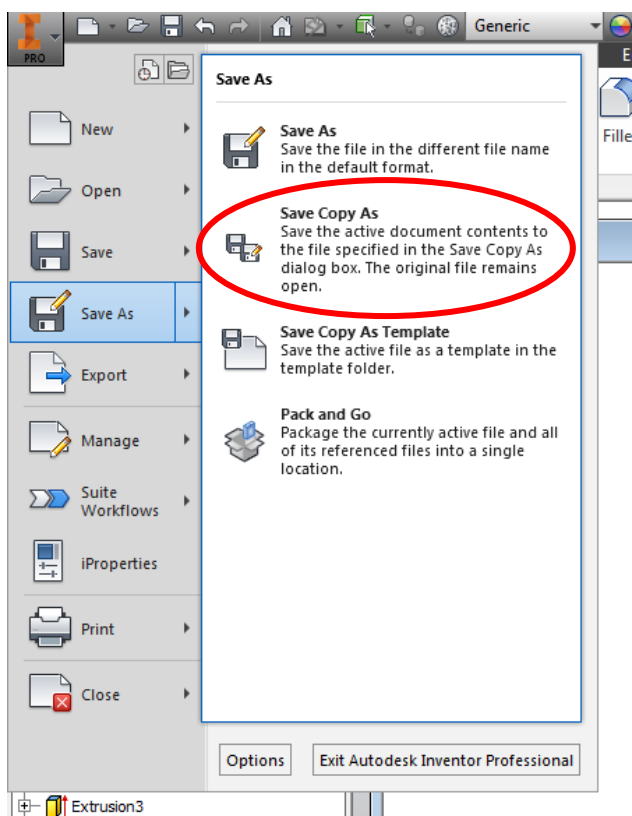
How to convert an Autodesk Inventor part file (.ipt) to a file for the 3D printer (.stl).

In order to print a part on the 3D printer you must convert your Autodesk Inventor file to a .stl file type but there are a few steps you must take to make sure you have the correct units when doing this conversion. If these steps are not followed your part will not be the right size and most often will be too big for the work space of the 3D printer.

You want to start by clicking on the inventor **I** icon in the upper left hand corner.

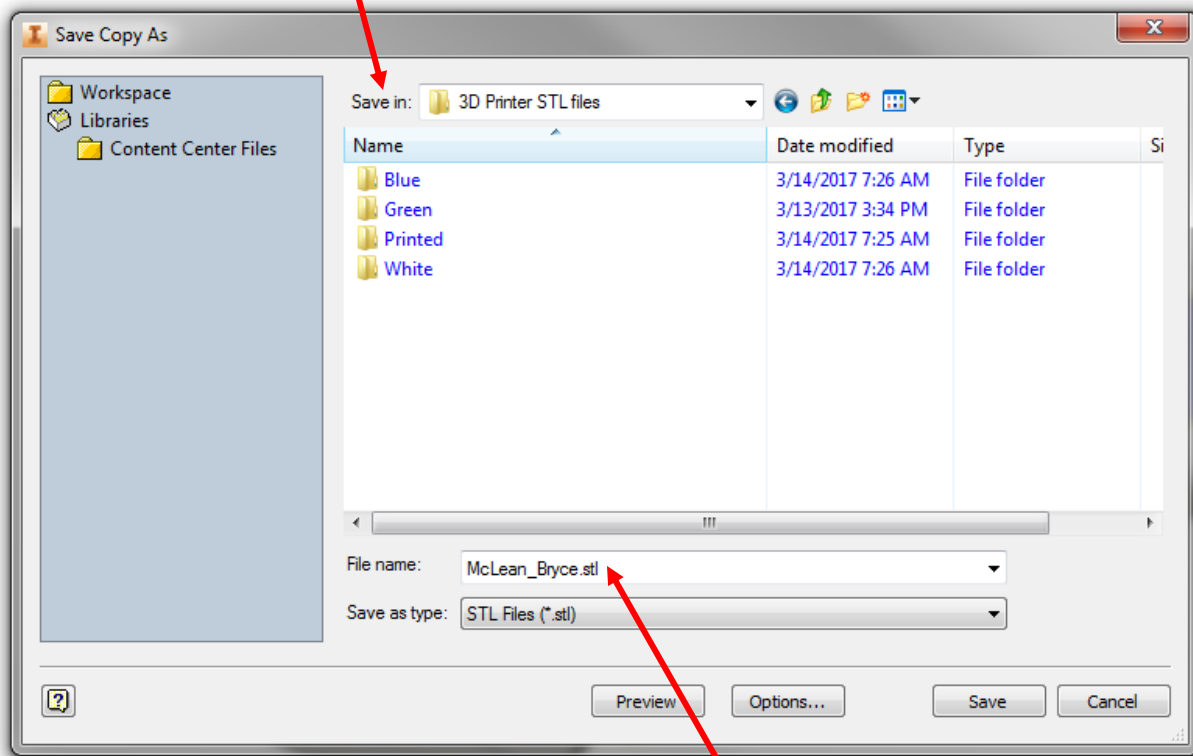


Hover over Save As until you get the choice of Save Copy As. Then click on Save Copy As.



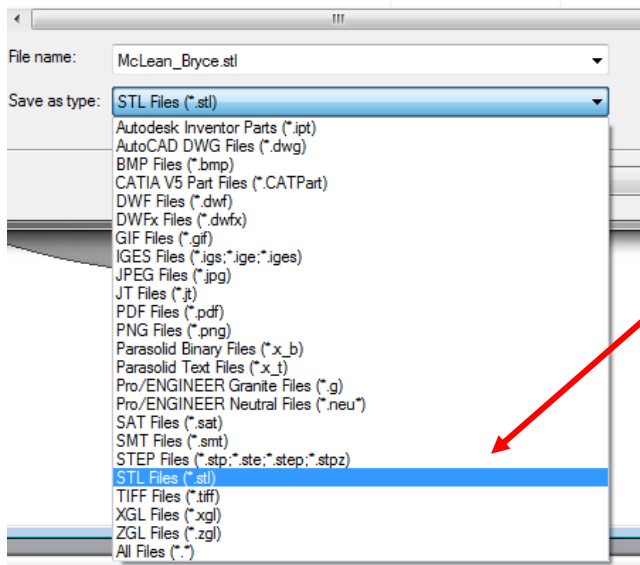
The pop up window below should come up.

Change the file destination, Save in: , to the folder designated by your instructor.



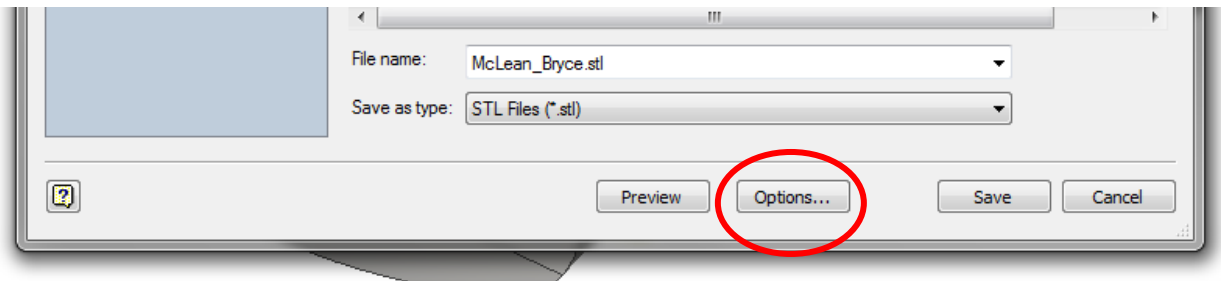
Change the file name to be your last name then your first name.

Change the Save as type to STL files by clicking on the drop down and selecting STL Files.

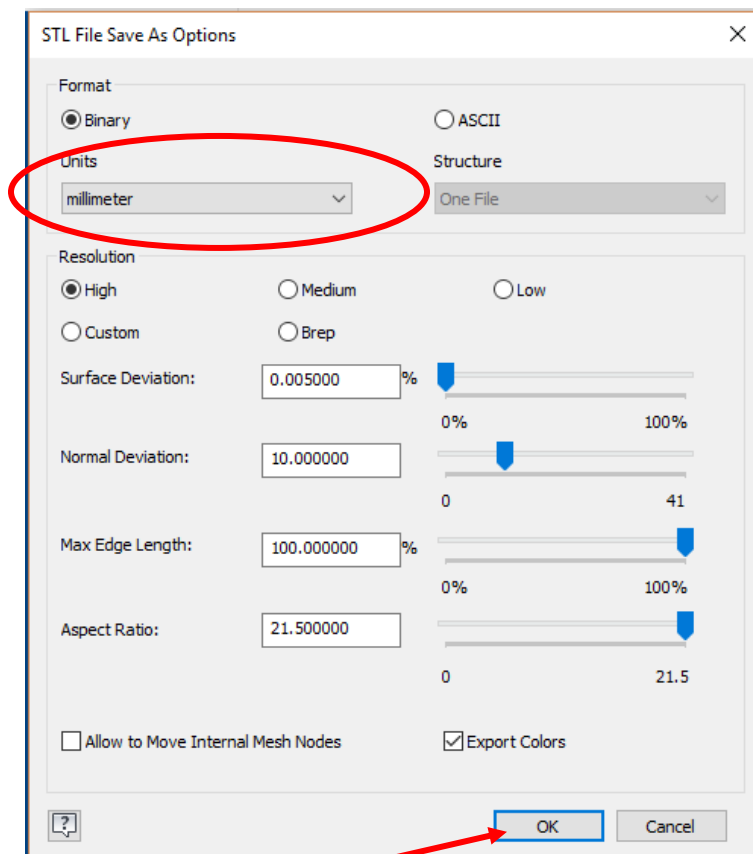


Do not click save yet!

Click the Options... button on the bottom of the pop up window.



Make sure the Units in the upper left hand side of the STL File Save As Options pop up window is set to **millimeters**.



Click OK on the STL File Save As Options window.

Then click Save on the Save Copy As window.

You have now properly saved your file as an STL that can be printed on the 3D printer.