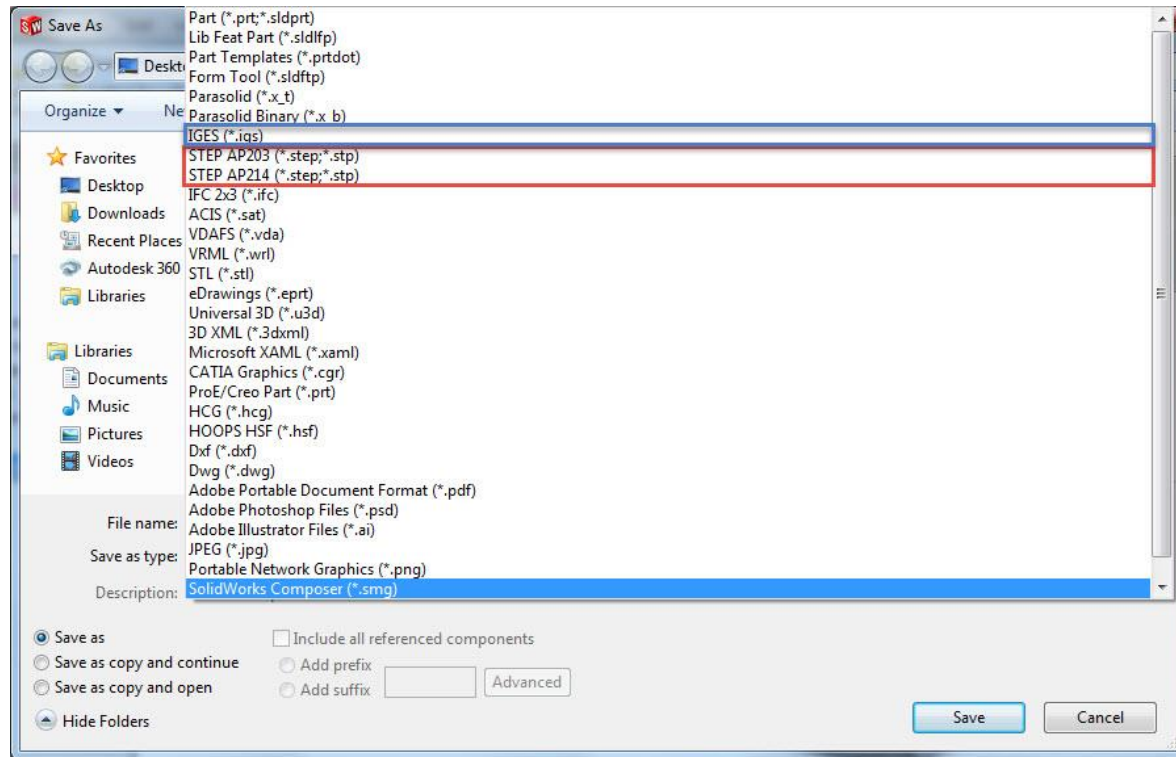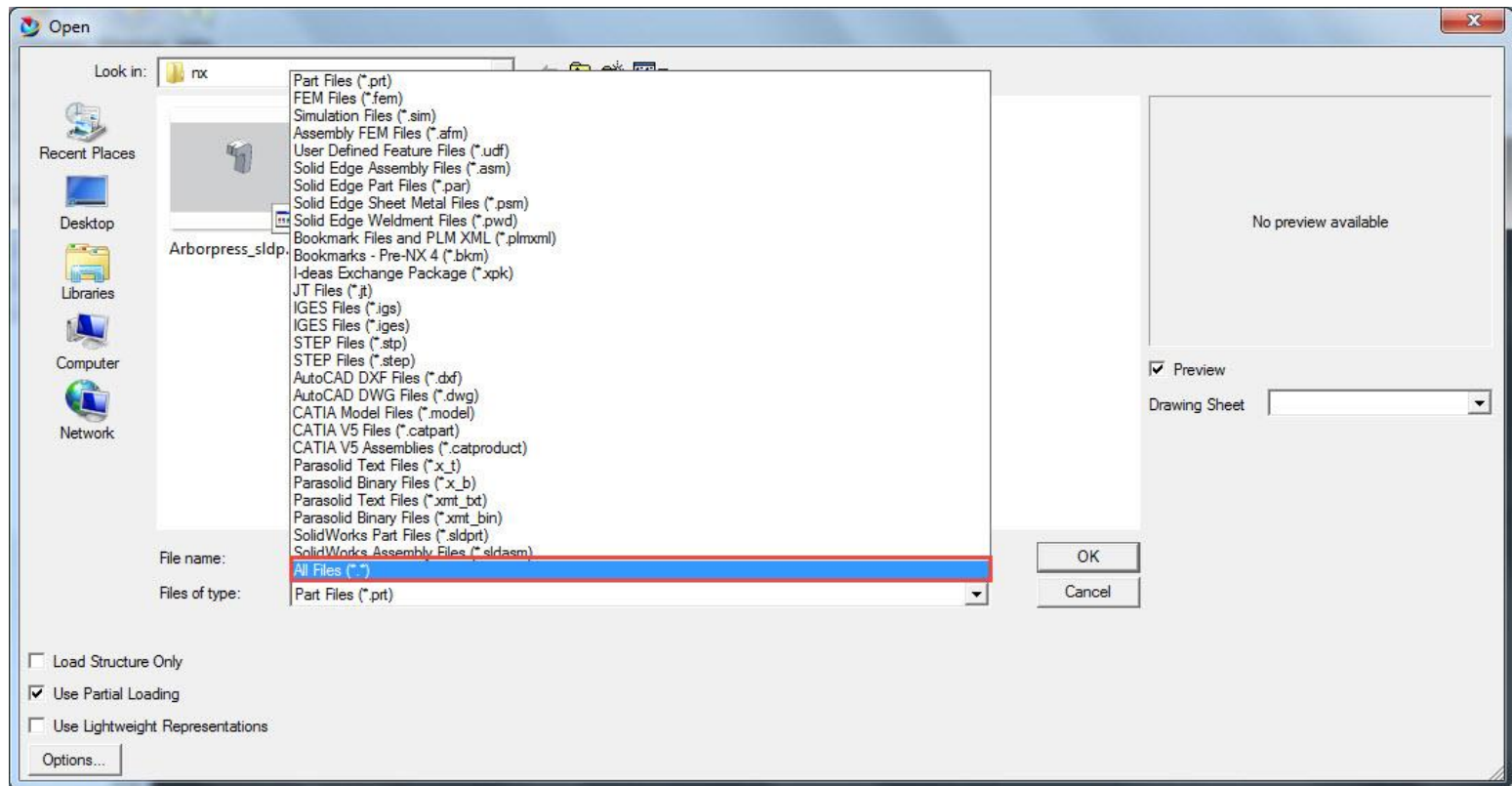# How to import model from solidworks to Nx

Armin Yousefi Kanani

Exporting:

First design your model in solidwork and save it as Step or IGES format

Importing

In NX >> File >> open >> Then change file type to All files and select solidwork file. Then click on ok. After opening your file you need to save as it to make NX format file (XXX.prt) file