Using ‘STEP’ wall specifications

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1. Get a STEP file from engineering (e.g. from Alex Creely). Should have extension ‘.stp’
2. Open FreeCAD 0.19 (available on my old office PC on 3/30/2022)
3. choose File/Open. Navigate to proper folder, find the .stp file, and open it.
4. An image of the object should appear. Put cursor in the window in which the object is displayed and select ALL (ctrl-A)
5. Click on File … Export
6. Choose file format: STL Mesh (\*.stl \*.ast)
7. Define a filename without any extension and SAVE. Note (5/3/22): for the first engineered stp file from Tom (about 13 MB), this takes several minutes. There is a progress bar at the lower right. It takes 90+% of memory on the laptop.
8. The file should appear in your folder as having file format “3D Object”. It should be 4-5 times bigger than the original .stp file.
9. Copy the .stp file you just created to the ‘runs’ directory at NERSC, using e.g. SecureFX. Stl files are binary
10. Go to the runs directory python … import read\_stl\_write\_triangles. Run that code, you will be prompted for the name of the .stl file and the output triangles.txt file.
11. The triangles.txt file can then be used as input to e.g. group\_go\_931.py.

If you want to visualize the .stl file, you open it with ParaView. Note: the filename might or might not appear in the usual list of files. but if you manually type in the filename, it seems to work.