FREECAD to BRLCAD

1. Make a model in FREECAD and save the file.
2. Select the object and go to export in file tab and export as .stp at the required place.



1. Open cmd and change the directory to the bin directory where the BRL-CAD has been built.
2. Use the step-g command to change from .stp to .g file. The syntax is as follows:

step-g -o <full output name> <full file name>



1. Open BRL-CAD MGED.
2. Open the .g file.
3. Browse and find the .g file to be imported and open it.
4. Draw the object.

1. Render the object using rt command.

