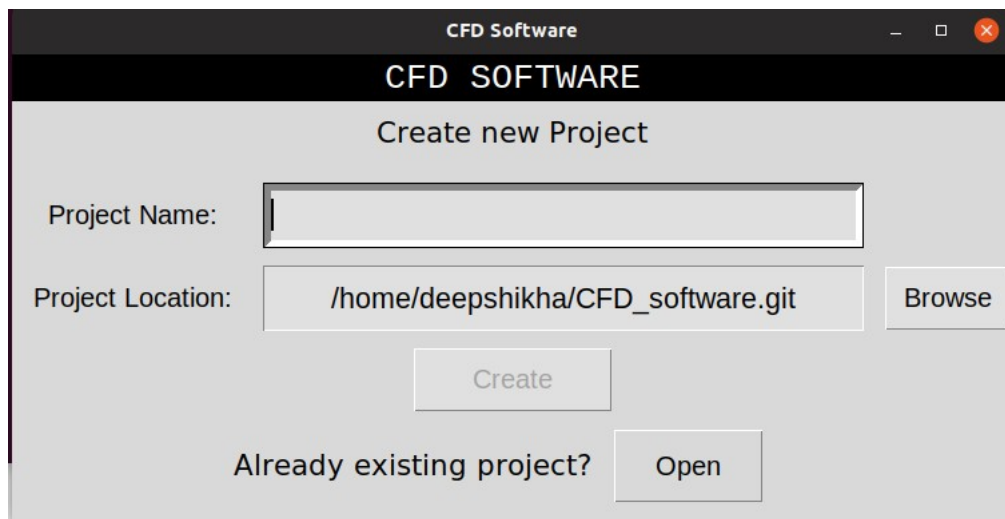# User Manual

CFD Software

Group-5

Kriti Mehta(B17087), Kanika Gupta(B17047), Khushi(B17048),Deepshikha(B17120)

1. Application can be opened by double-click on icon.(Application Launcher)

2. A main window along with error window gets opened. User is asked to create a new project or open an existing project. User can browse the location for the project directory.


3. If a new project is made, on creation, three sub-directories, namely, preprocessing, solver and post-processing are made. A window opens consisting of 3 tabs for 3 different processes. By default the pre-processing tab is open, but user can change which stage to start from. A log window is there in the right which creates a log with timestamp for each action taken.
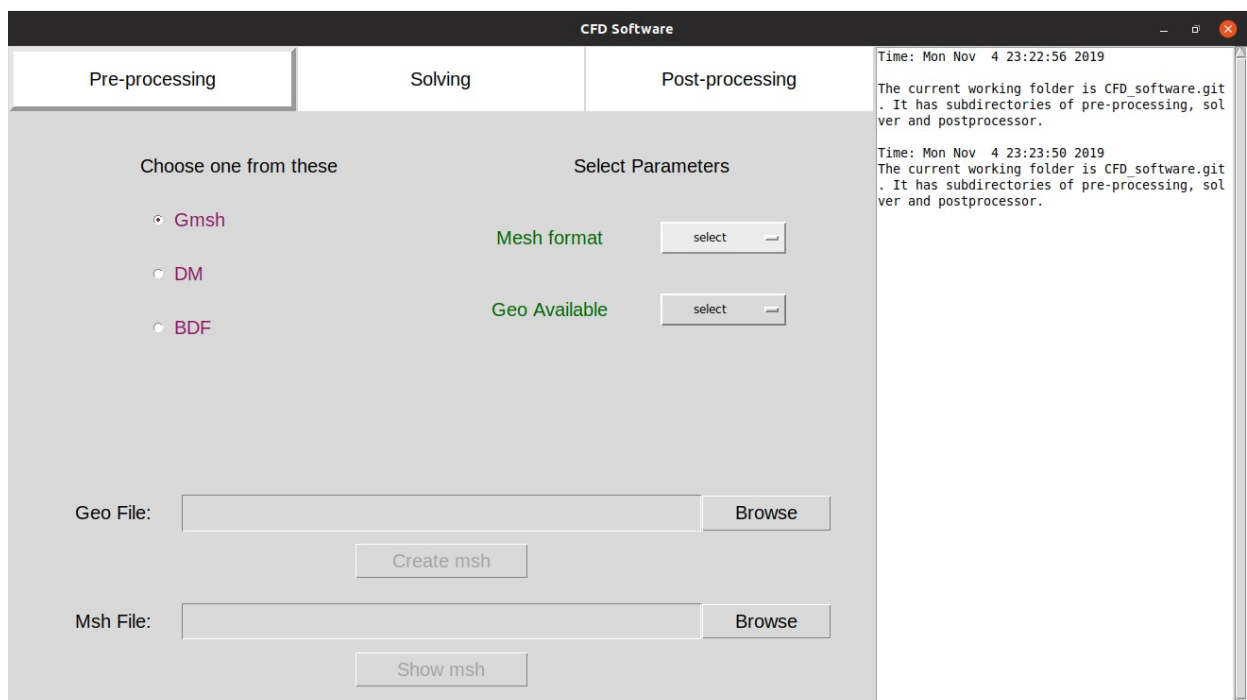
4. In pre-processing stage, user can browse the .geo file and convert it to .msh file using gmsh software. User can see the mesh created using "show mesh" button.
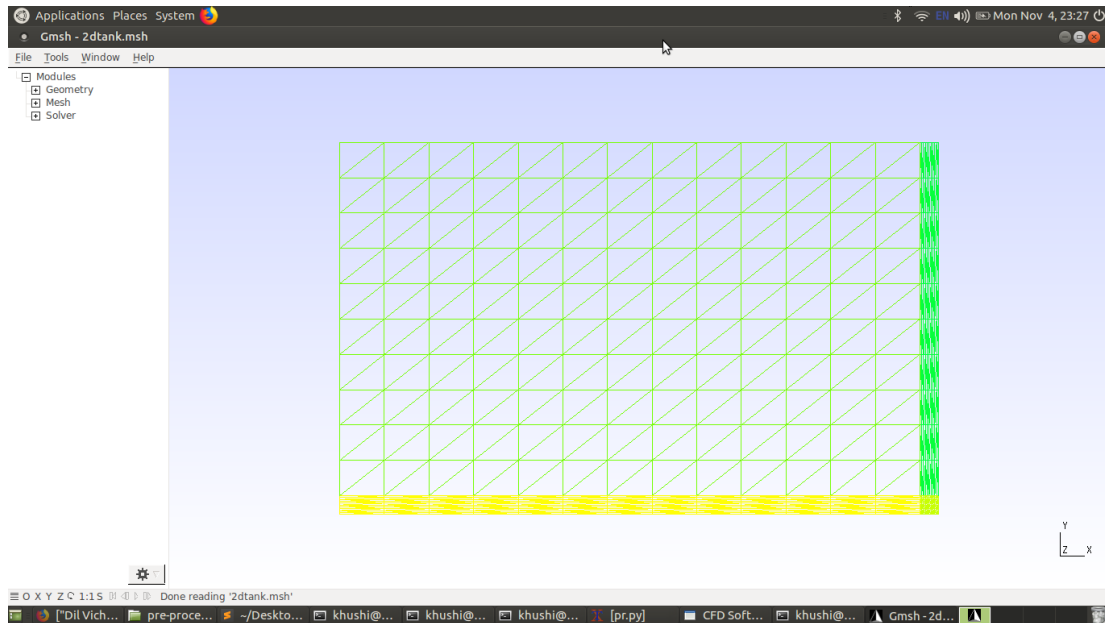

5. In the solver stage, user can either use the .msh file generated in the previous stage, or browse some already .msh existing file.
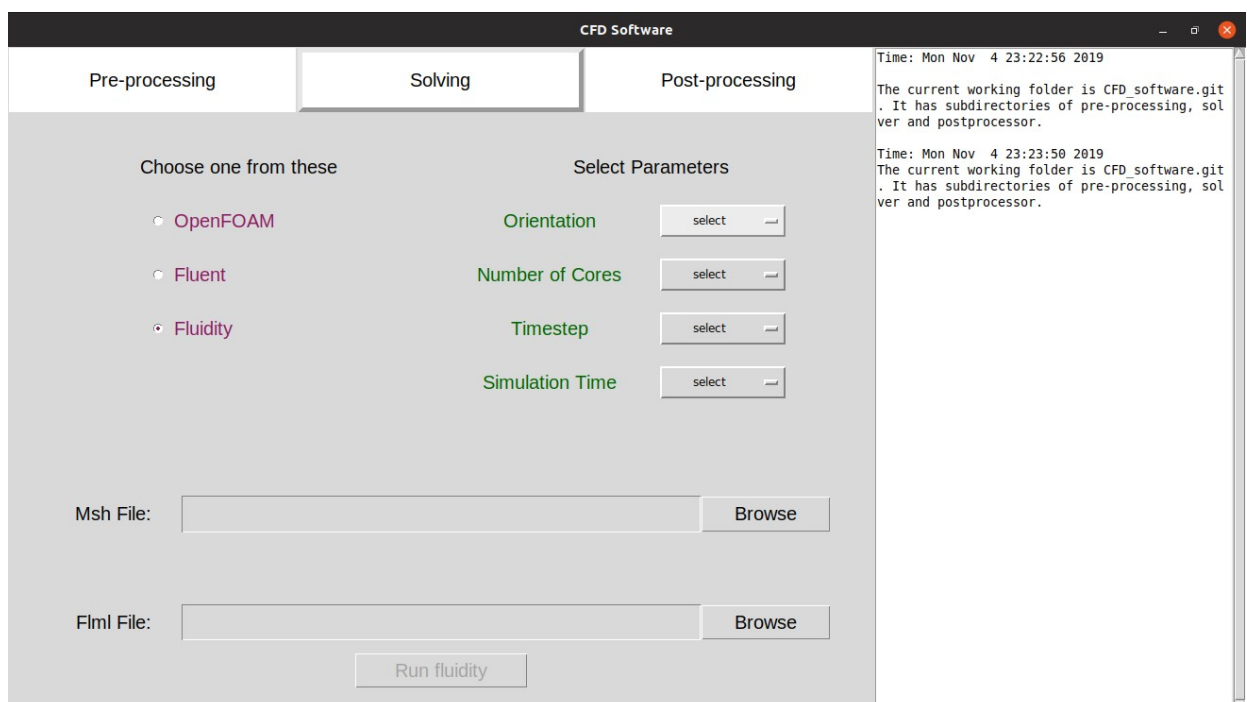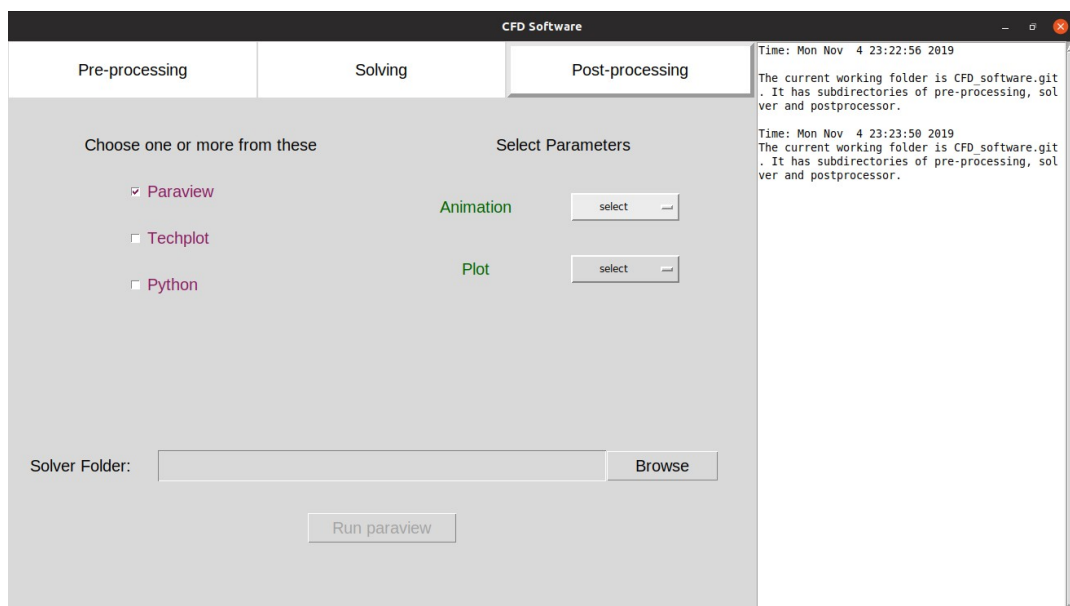
If chosen fluidity: User also needs to browse .flml file. Many files with .detector and .vtu files are generated in post-processing folder after this process.

If chosen openFOAM: User needs to browse for ControlDict file. User gets 3D mesh and graphs as output in paraview GUI.
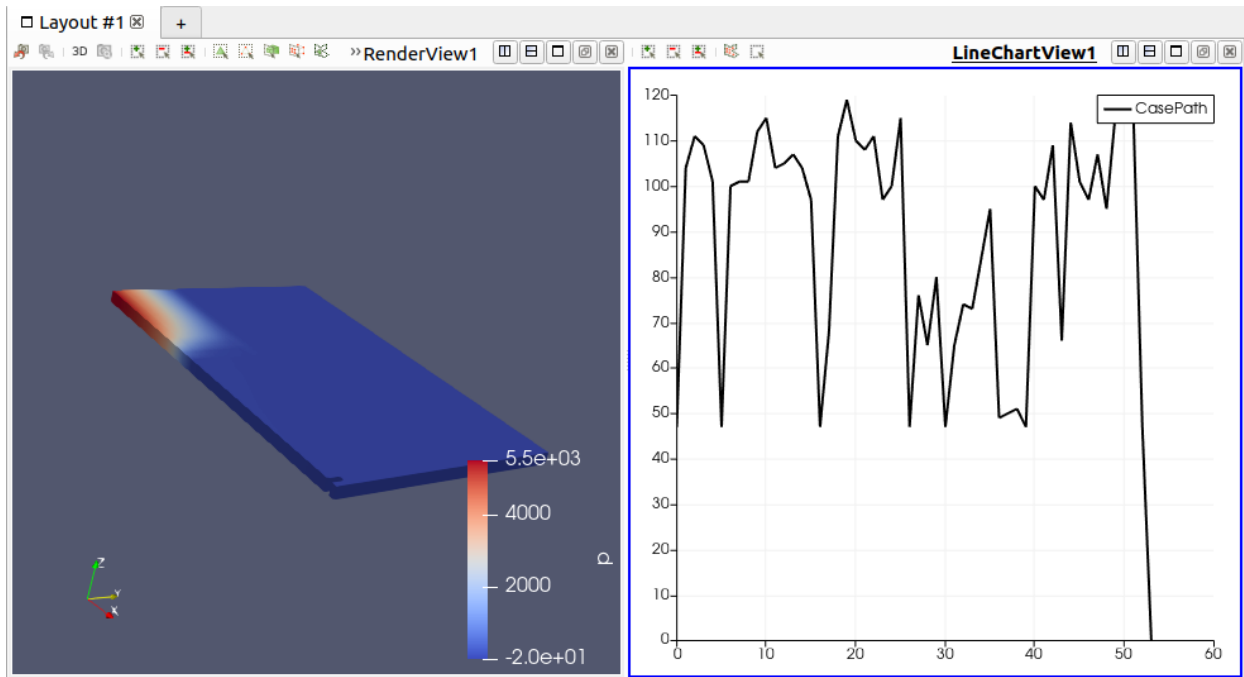
6. In the post-processing stage:

If chosen Python: "Plot_data.py" python script is run at the backend to generate output graphs.

If chosen Paraview: Paraview software gets opened and user need to manually select some options to generate output graphs.