FOSSEE, IIT Bombay

OpenFOAM GUI Project

Date: 25–02–2026

Streamlining Open-Source CFD: A Comparative Study of GUI-based vs. Script-based OpenFOAM Workflows

Medehal Pavan Sai Tarun[1], Mr. Vedant Dubey[2], Mr. Diptangshu Dey[3]

[1]*VIT University, Bhopal*

[2]*National Institute of Technology, Raipur*

[3]*National Institute of Technology, Durgapur*

Tested OpenFOAM Cases:

1. **Cavity Case:**
   This Cavity case is defined with the pisoFoam solver, as defined in the controlDict, which defines a transient incompressible flow simulation. The simulation is performed from 0 to 0.5 s with a constant time step of 0.005 s, and the solution is written every 20-timesteps. The mesh is defined explicitly in the constant/polyMesh directory, and the presence of blockMeshDict in the system directory indicates that the mesh is generated with blockMesh. The boundary definition defines three patches: movingWall (wall), fixedWalls (wall), and frontAndBack (empty), which again confirms that it is a two-dimensional simulation. The case defines the velocity (U) and pressure (p) fields, as well as turbulence-related fields (k, omega, nut). The turbulence is modeled with a RAS method using the k-ω SST model, as defined in the turbulenceProperties dictionary, with turbulence turned on.
   File Location:
   https://drive.google.com/file/d/1myARhVxFxrAQCrgOGzYROjdjQIdrwELn/view?usp=sharing

2. **PitzDaily Case:**
   This PitzDaily case employs the simpleFoam solver, as defined in the controlDict, which defines a steady-state simulation of incompressible flow. The simulation begins at time 0 and proceeds until an end time of 2000, with a pseudo-time step (deltaT) of 1, with output every 100 iterations. The mesh is stored in constant/polyMesh, and the presence of blockMeshDict in the system directory verifies that the mesh is created by blockMesh. The initial conditions include velocity (U), pressure (p), and several turbulence-related variables (k, epsilon, omega, nut, nuTilda), as found in the 0 directory. The simulation employs turbulence modeling with simulationType set to RAS, and the turbulence model chosen is k-epsilon, as defined in turbulenceProperties. The boundary file contains patches of the empty type, which defines that the case is set up for a two-dimensional simulation.
   File Location:
   https://drive.google.com/file/d/1sqfmTBt7IRJmrlmIgxAnJ8iVDVmiH0qM/view?usp=sharing

3. **BackwardStep_Case:**

   This case is setup to run with the simpleFoam solver, as defined in the controlDict, which gives a steady-state incompressible flow simulation. The simulation starts from time 0 and proceeds up to an end time of 1000, with a pseudo–time step size (deltaT) of 1, and solution data written at intervals specified in the control dictionary. The mesh is present in the constant/polyMesh directory, and the inclusion of blockMeshDict in the system folder confirms that the mesh is generated using blockMesh. The 0 directory contains the primary flow variables velocity (U) and pressure (p), along with turbulence-related fields such as k, epsilon, and nut. According to turbulenceProperties, the case uses a RAS turbulence modeling approach with the k–ε model enabled. The boundary configuration includes patches marked as empty, which indicates that the case is set up as a two-dimensional simulation.

   File Location:

   https://drive.google.com/file/d/1JFlXhWEj1Ku2rxDmrt0jzVH2zEkLunG0/view?usp=sharing

4. **2D_Channel_Case:**

   This case is setup to run with the simpleFoam solver, as defined in the controlDict, which gives a steady-state incompressible flow simulation. The simulation is initialized at time 0 and run until an end time defined in the control dictionary, with a fixed pseudo–time step (deltaT) of 1, and results written at the specified write interval. The computational mesh is available in the constant/polyMesh directory, and the presence of blockMeshDict in the system folder confirms that the mesh is generated using blockMesh. The 0 directory contains the flow field variables velocity (U) and pressure (p), along with turbulence-related fields k, epsilon, and nut. As defined in turbulenceProperties, the case employs a RAS turbulence modeling approach with the k–ε turbulence model enabled. The boundary file includes patches defined as empty, indicating that the case is configured as a two-dimensional channel flow simulation.

   File Location:

   https://drive.google.com/file/d/1zwT4-hPoj-eZo-5EHVqKxP1Lz7PIizg6/view?usp=sharing

5. **Mach_6_Case:**

This complicated case is configured to simulate high – speed compressible flow, with the sonicFoam solver, as defined in the controlDict. The solver selection option verifies that the problem is transient and compressible, appropriate for supersonic/hypersonic flow conditions. The simulation begins at time 0 and progresses in time with a fixed time step specified in the control dictionary, with output written at regular intervals. The grid for the simulation is specified in the constant/polyMesh directory, and the presence of blockMeshDict in the system directory indicates that the grid is created with blockMesh. The 0 directory holds variables related to compressible flow, such as velocity (U), pressure (p), temperature (T), and density (rho), which are explicitly stated in the given files. The thermophysicalProperties file verifies that a thermodynamic approach is used for compressible flow simulation, and turbulence is specified as per the turbulenceProperties file (as explicitly stated in the case).

File Location:

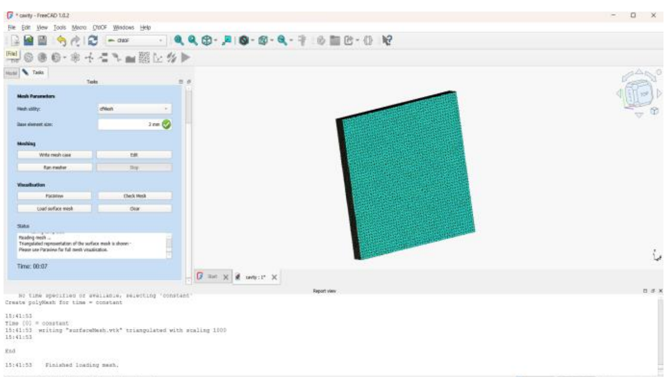
https://drive.google.com/file/d/14XiNor5STB_iPpy5ViXh_JBifYgcjOhi/view?usp=sharing

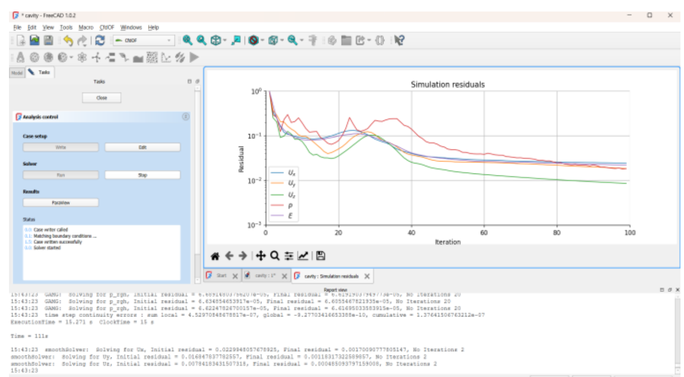
FreeCad_CfdOF_Cases:

I replicated the cases simulated in OpenFOAM and in addition to that I have also simulated the Airfoil, Plane, Projectile, Simplexace and tesla-4680-battery-cooling unit.

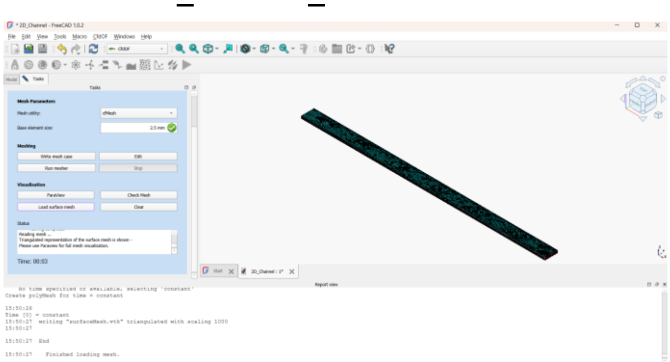
Following are the simulation results:

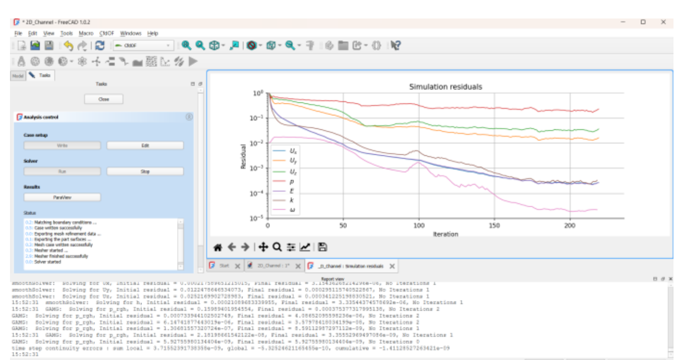
1. **Cavity_Case:**



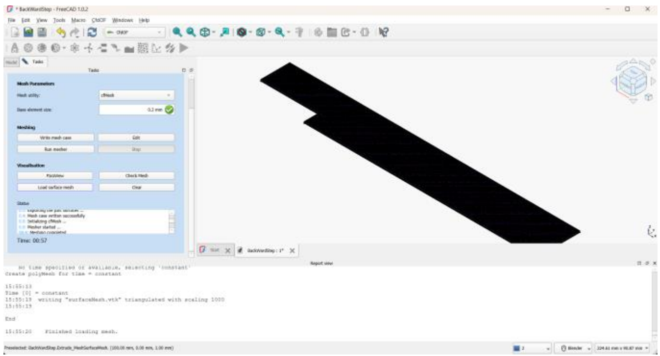


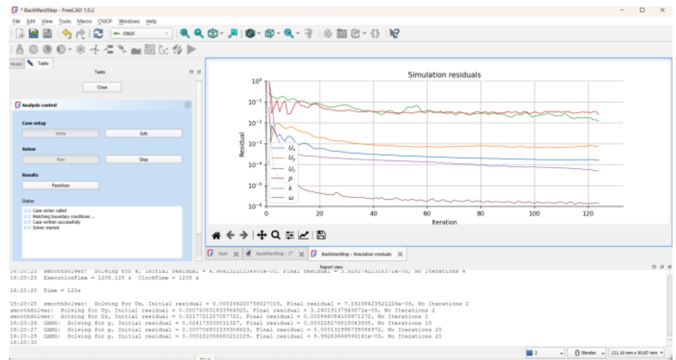
2. **2D_Channel_Case:**



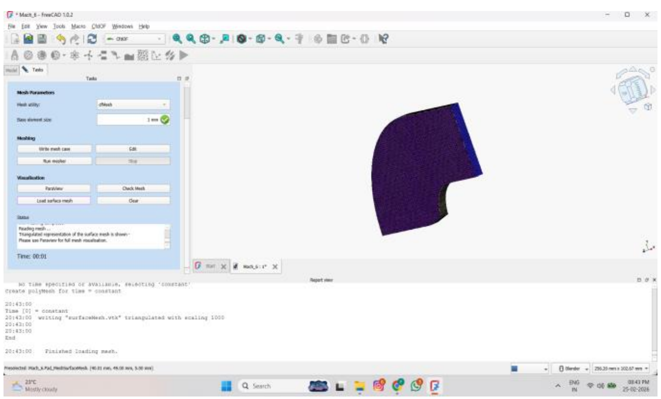


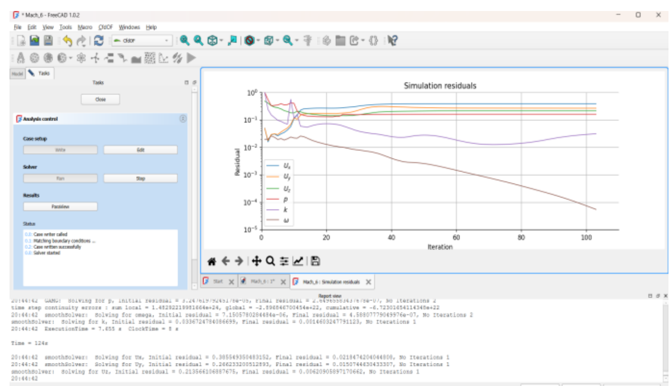
3. BackwardStep_Case:



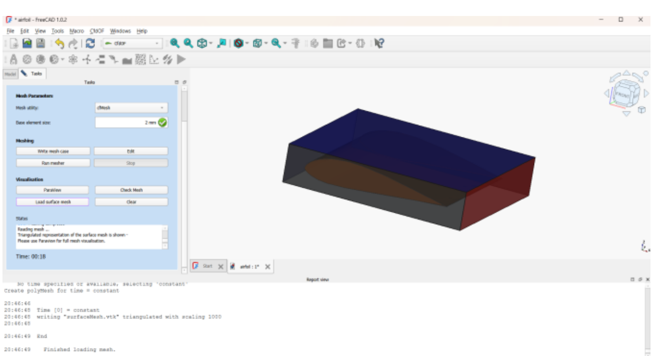


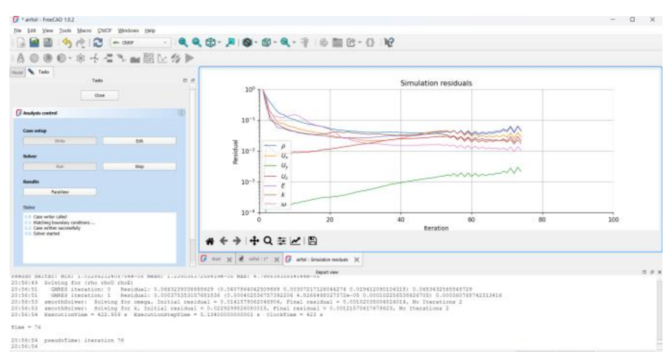
4. Mach_6_Case:



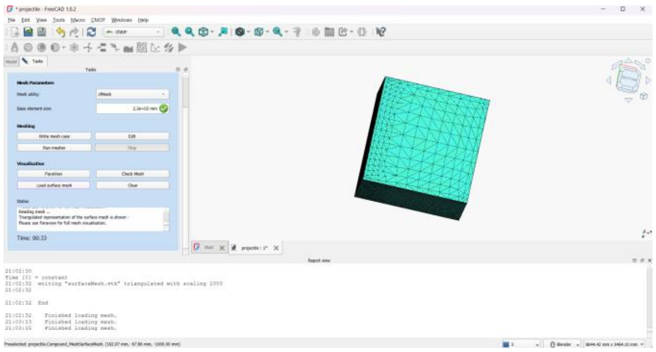


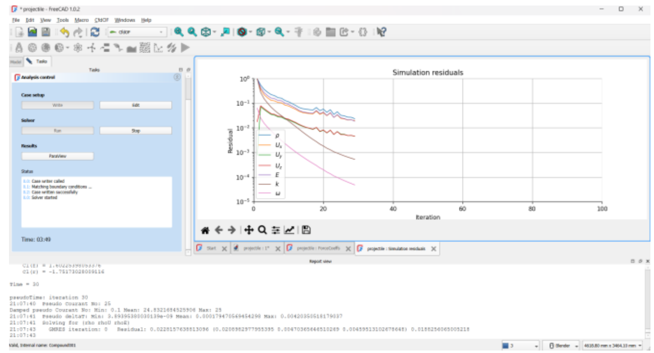
5. Airfoil_Case:



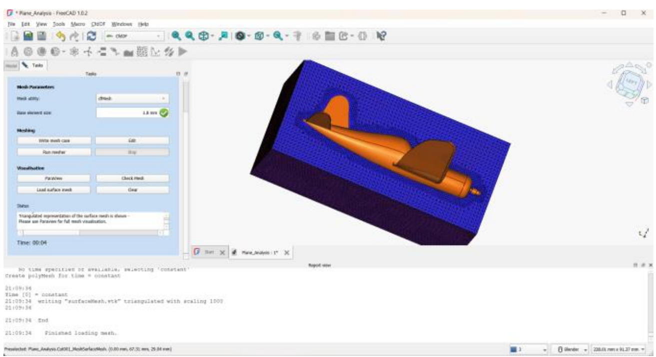


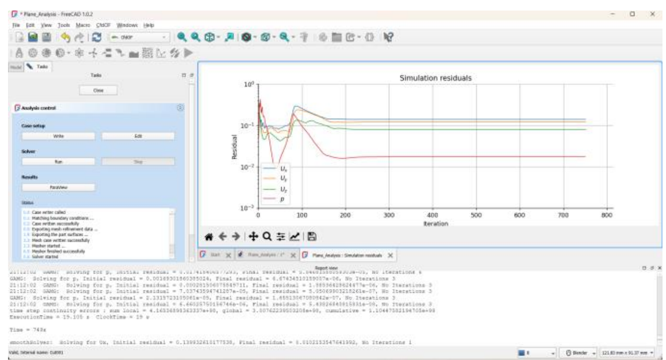
6. **Projectile:**





7. **Plane_Analysis_Case:**





CfdOf_Cases Location:

All Cases Location:

https://drive.google.com/drive/folders/1DCB7jGUhs_vMS0H0KXPH7Imx4GntPMT6?usp=sharing

https://drive.google.com/drive/folders/1DSXQaGnyuspibFt6Rsotv4Jjr7ElZ_gH?usp=sharing