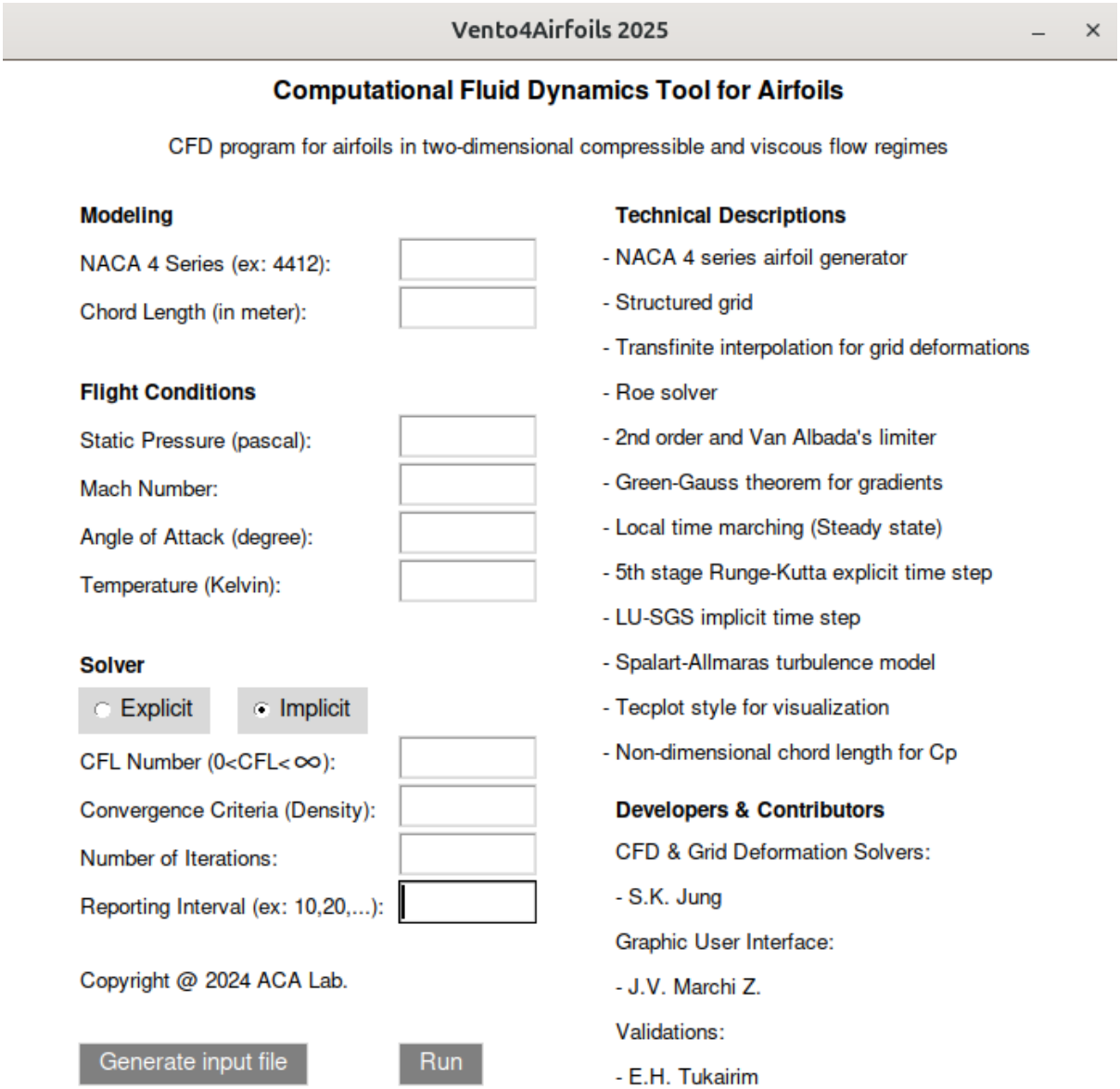
A Tutorial for **Vento4Airfoil 2025**

Prof. SungKi Jung, Ph.D.

Universidade Federal do ABC,
Aerospace Engineering, Centro de Engenharia, Modelagem e Ciências Sociais Aplicadas,
Rua Arcturus 03, 09606-070, São Bernardo do Campo, São Paulo, Brazil.
E-mail: sungki.jung@ufabc.edu.br

March 2025



1 Introduction

This program is fully tested on Ubuntu 24.04.2. The technical descriptions for this program are briefly shown in the Graphic User Interface windows. To download this program, access url: `https://github.com/sungki-jung/CFD4Airfoils`, then download the following folders andnd files"

- folders: bin, db, outputs
- files: GUI

2 Start the Program

Once you downloaded the folders and files, put the files and folders into any place in your computer. Then, find out **GUI** file as shown in fig. 1.

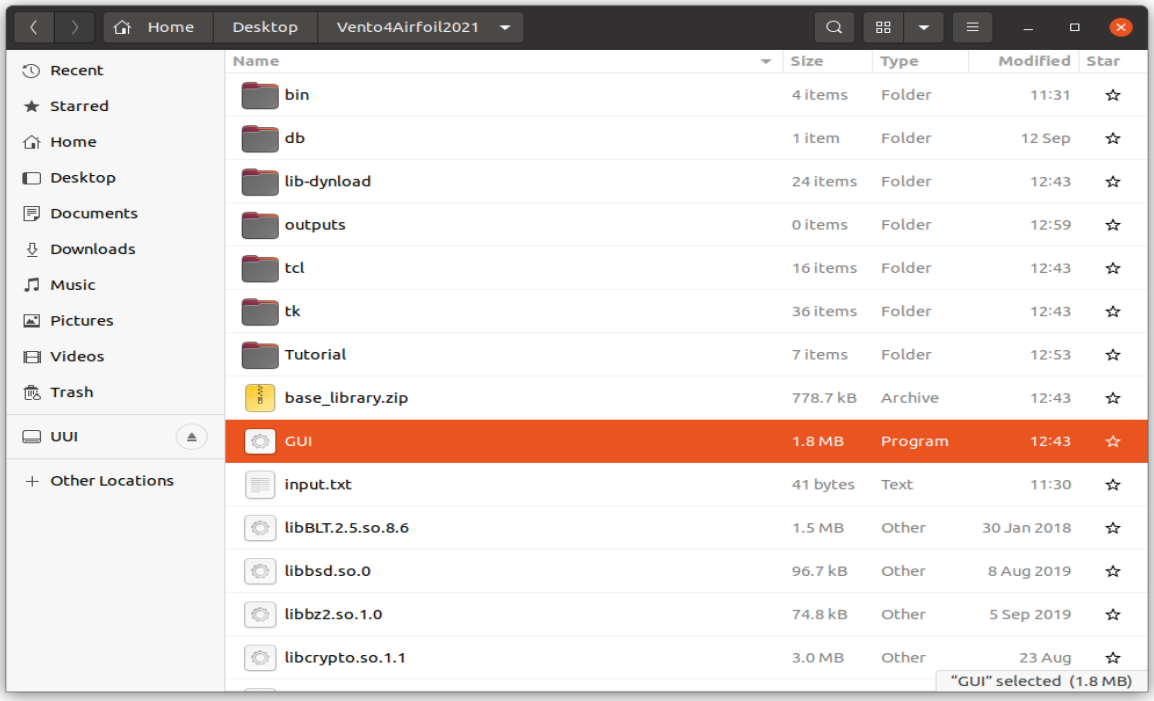


Figure 1: Program files and folders.

Open the terminal in **Vento4Airfoil2025** folder as shown in fig. 2.

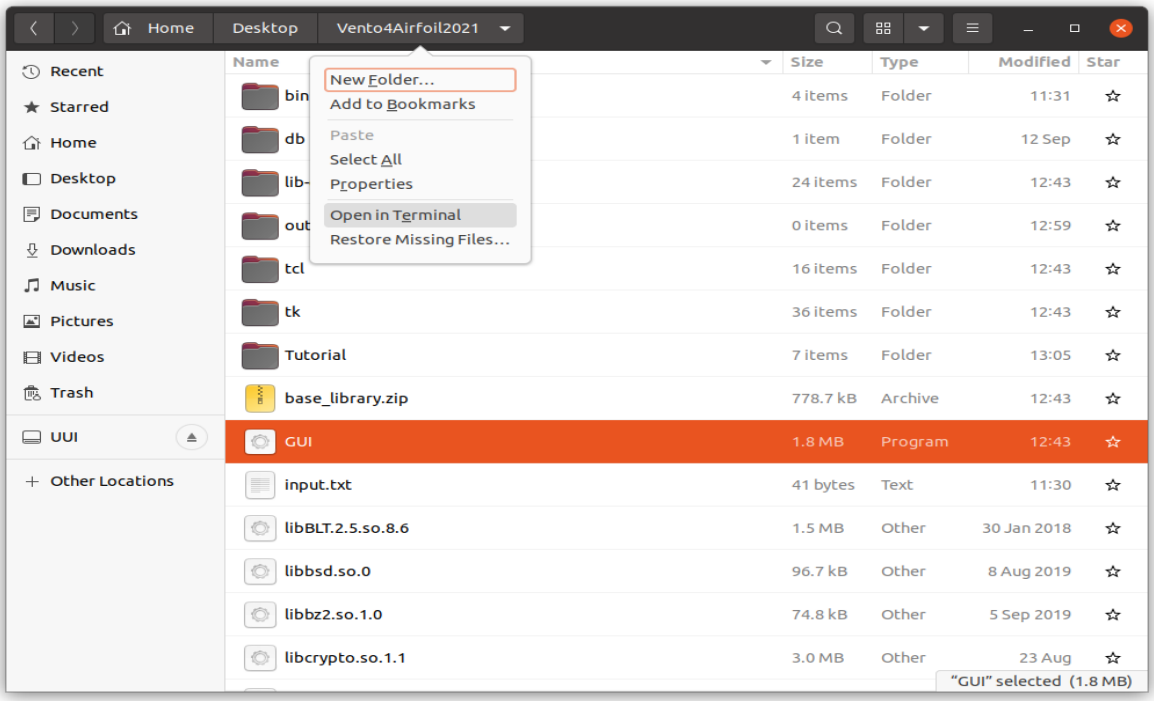


Figure 2: Terminal in **Vento4Airfoil2025** folder.

The terminal windows are shown in fig. 3. Execute the **GUI** file (type ./GUI on the terminal as shown in fig. 3 then hit "enter" on your keyboard).

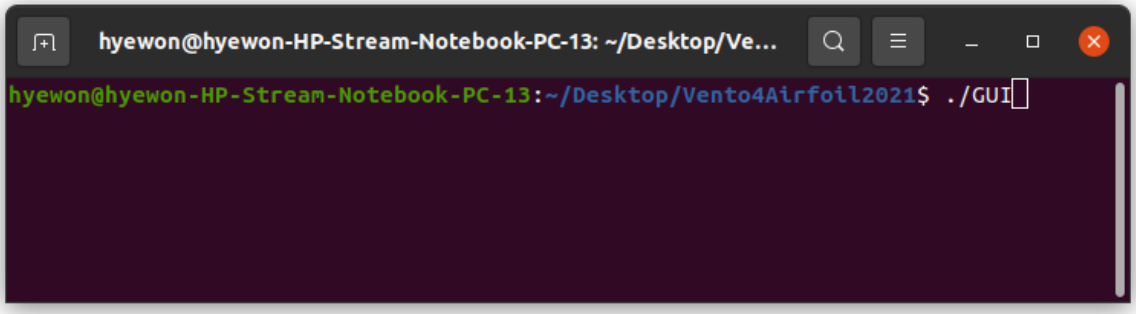


Figure 3: Terminal window.

3 GUI for Vento4Airfoil 2025

Insert all parameters as shown in fig. 4.

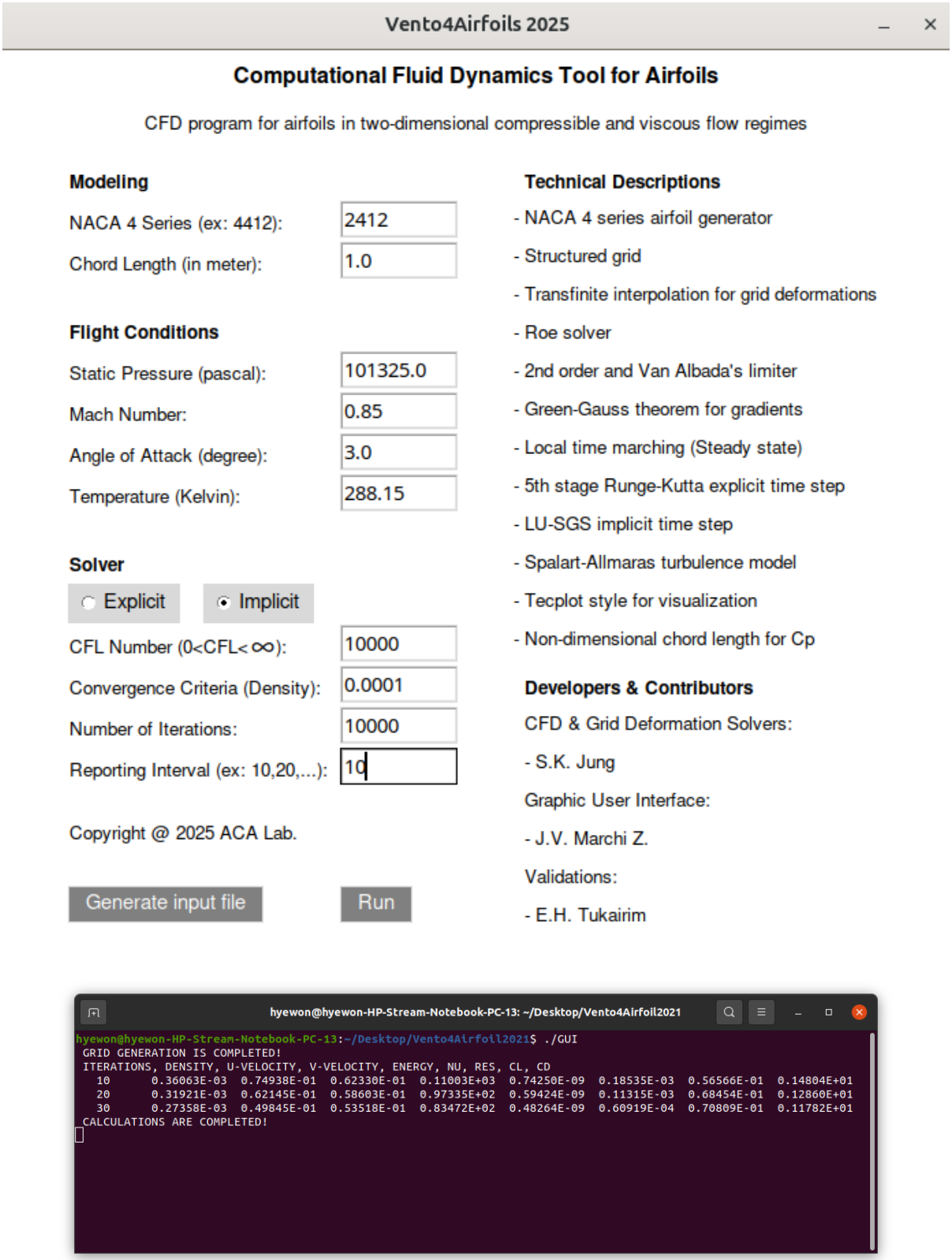


Figure 4: Example for all parameters and Run.

Click an icon, **Generate input file**, then, click an icon, **Run**. Residuals are shown in the terminal windows (fig. 4).

4 Post-processing for Vento4Airfoil 2025

Once all calculations are completed, open **output** folder in **Vento4Airfoil2021** folder as shown in fig. 5. Extensions for each file in **output** folder denote the characteristics of outputs.

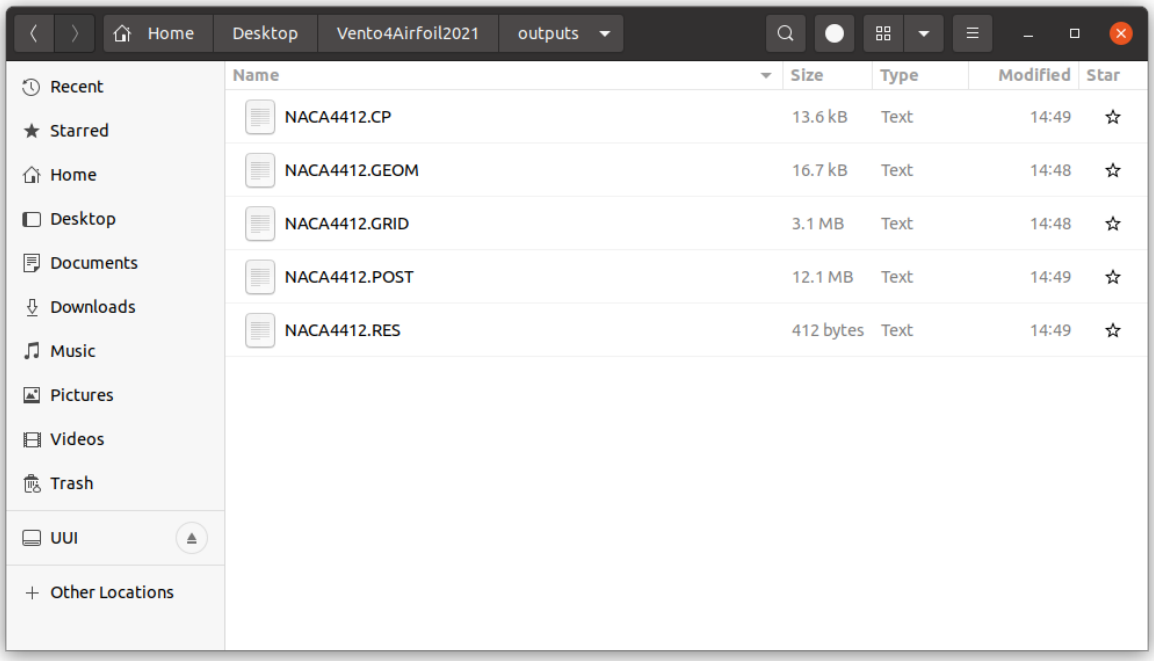


Figure 5: Outputs.

.CP: Pressure Coefficient
.GEOM: Coordinate values of NACA 4 series airfoil
.GRID: Grid around NACA 4 series airfoil
.RES: Residuals during CFD calculations
.POST: Post-processing
.AIR: Primitive variables of airflow

4.1 Preparation for Visualization

For visualization of CFD simulations, install a visualization tool, **Paraview** that is a free open-source program, on your computer. To install **Paraview**, see url: `https://jungs-odds-and-ends.blogspot.com/2021/05/paraview-for-windows.html` or you can just type "sudo apt install paraview" on the terminal of Ubuntu to install the Paraview program.

4.2 Visualization Using Paraview

Once **Paraview** is installed, open **Paraview** program (type paraview on the terminal). Then follow figs. 6 to 8. Note that .POST file includes all information for CFD visualization. .Post file was written in Tecplot format.

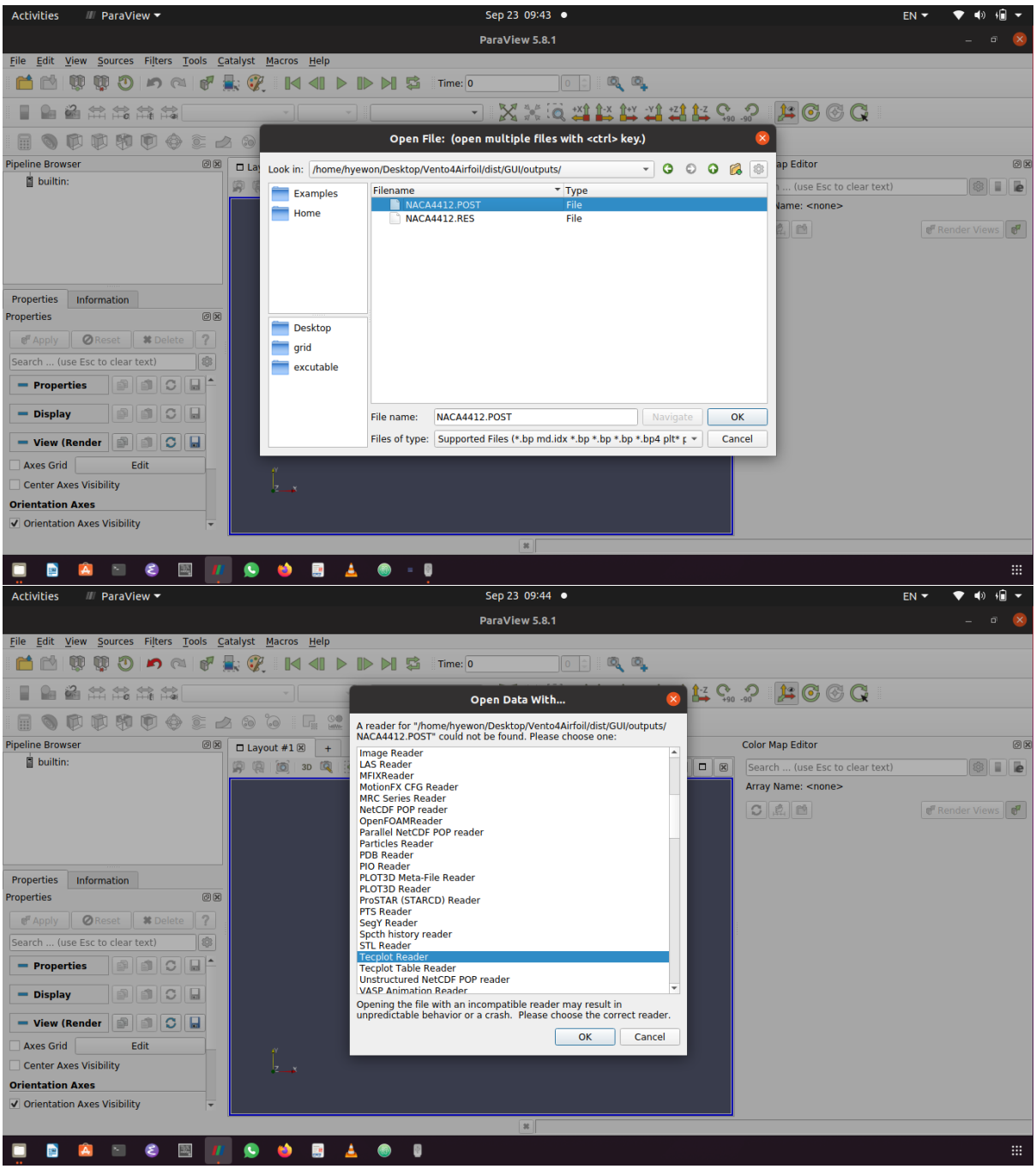


Figure 6: Select .POST file and choose Tecplot Reader in Paraview.

After you check the Tecplot format as shown in fig. 6, click **Apply** shown in fig. 7.

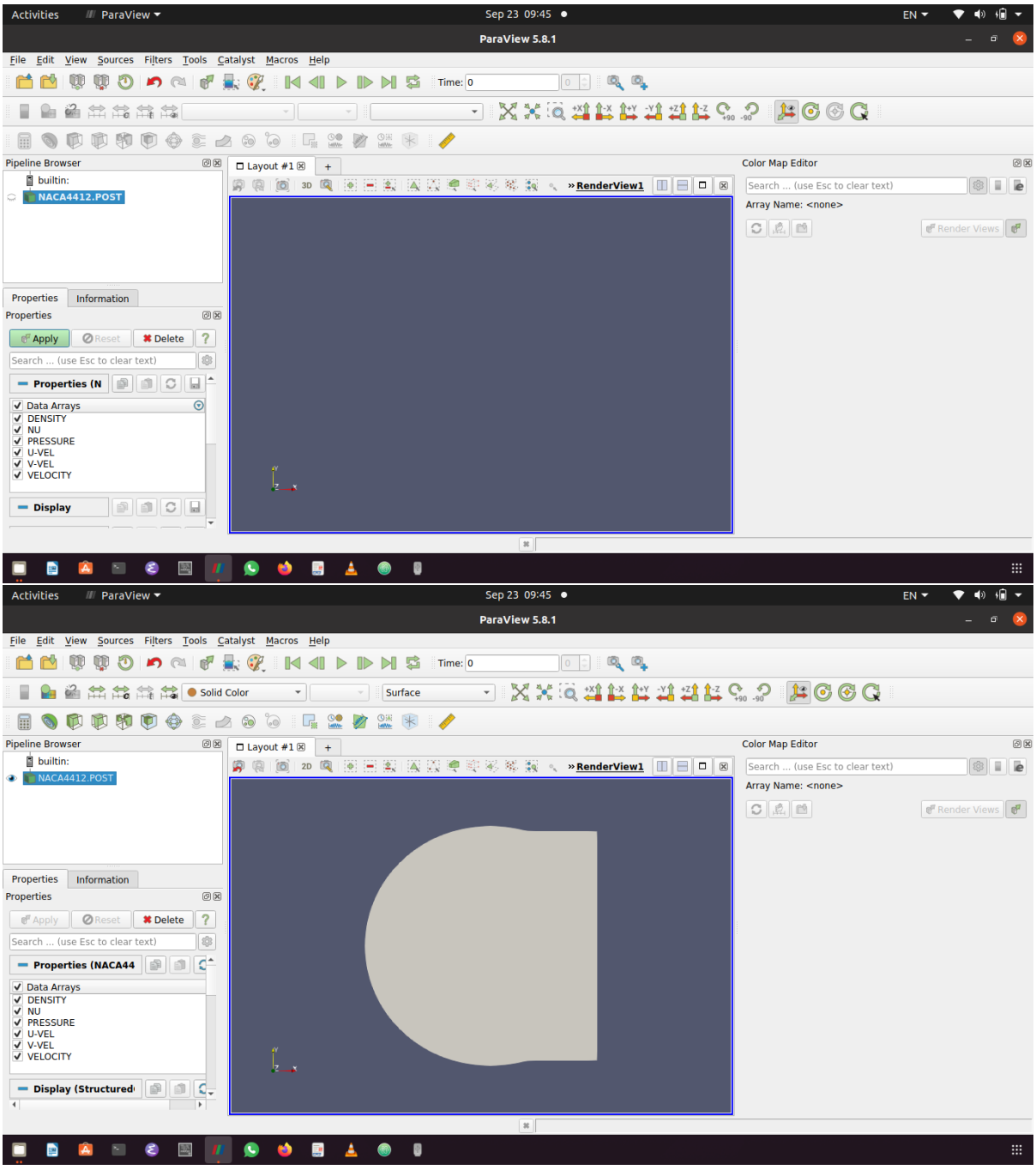


Figure 7: Apply and visualizations.

Lastly, change some options to visualize the simulation results.

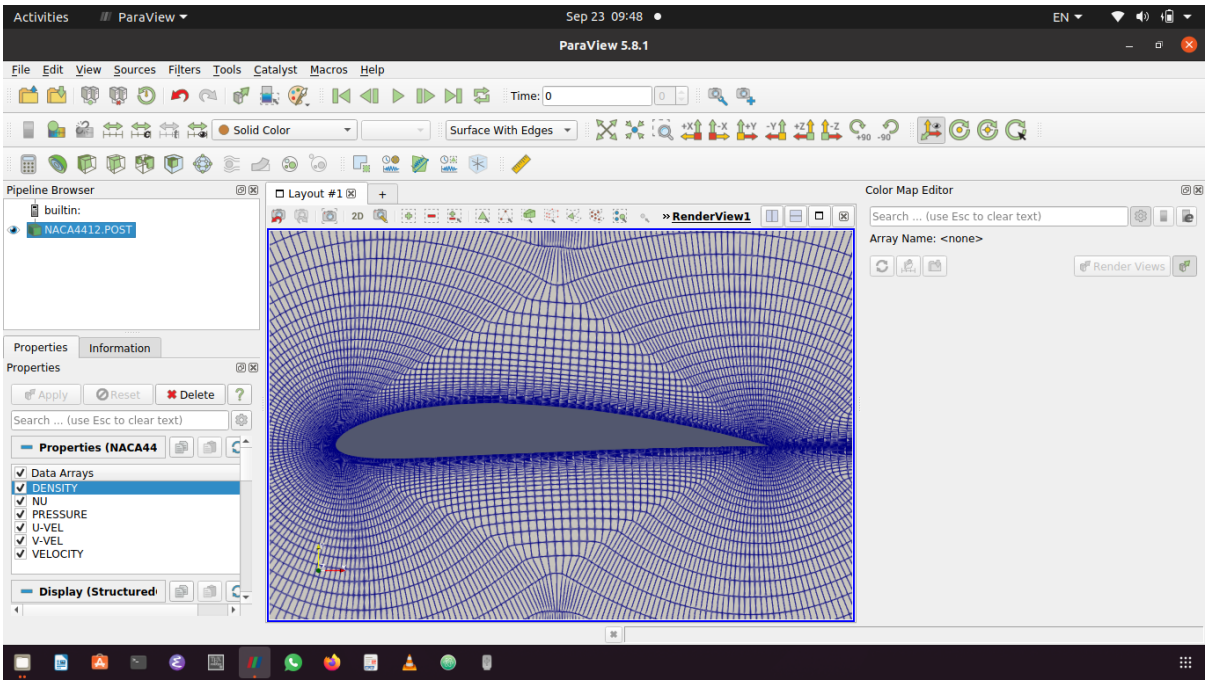


Figure 8: Options for visualizations.