*Proceedings of the National Conference on Computational Modeling of Fluid Dynamics Problems (CMFDP-2019)*
*NIT Warangal, India – Jan 18-20, 2019*
*Paper No. XXX (The number assigned when the papaer is accepted )*

# CFD Analysis Of Sedan Car: A Case Study

Anand Sasidharan[a] , Arun Radhakrishnan[a] , K J Thrissul[a] , Adarsh k[a] , Priyabrata Adhikary[a]
[a]*Mechanical Engineering, NewHorizon College of Engineering, Bengaluru-560103, Karanataka, India*

*Corresponding author Email: ____arun246007@gmail.com____________________________

Received: Date? Accepted: Date?

The use of fluid flow simulations (or Computational Fluid Dynamics, CFD) in such a CAD-embedded context is obviously very attractive as it can not only accelerate the design process, but it can make these processes more predictable and reliable, against a background of increasing design complexity and dependence on external development partners

## 1. Introduction

Aerodynamics is something that changes not only how fast a vehicle goes, but also how efficient it is. In simple words, the lesser the force the vehicle has to exert against the atmosphere to reach speeds, the faster and more efficient the vehicle will be.[1-3]

.

## 2. Methodology Adopted

The systematic search for the best solution for a design is the objective of most CFD simulations and CFD software packages. The main criterion for flow and heat transfer simulation as an integral part of a PLM concept is efficient turnaround of high quality CFD solutions, from geometry changes to resultant engineering interpretation in order to keep pace with design changes.[4-5]

In this case, the governing system of equations can be written as follows:

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot (\rho\, \mathbf{u}) \tag{1}$$

$$\rho\left(\frac{\partial}{\partial t} + \mathbf{u} \cdot \nabla\right)\mathbf{u} = -\nabla P + \rho\mathbf{g} + \frac{1}{c}\mathbf{J} \times \mathbf{B} \tag{2}$$

$$\rho\left(\frac{\partial}{\partial t} + \mathbf{u} \cdot \nabla\right)e = -P\nabla \cdot \mathbf{u} + \rho\mathbf{u} \cdot \mathbf{g} + \frac{1}{\sigma}\mathbf{J}^2 \tag{3}$$

## 3. Theory and Calculation

For external problems such as flow over an aircraft or building, the parameters of the external incoming flow (so-called "ambient" conditions) must be defined. Namely the velocity, pressure, temperature, fluid mixture composition and turbulence parameters must be specified. Evidently, during the calculation they can be partly violated at the flow boundary lying downstream of the model.[6-7]

## 4. Result and Discussion:

**Case-1:** A Box shape and no smooth edges, so the friction with the air is too much which affects its performance and this in turn affects the mileage of the car. The FloEFD of wagonR is attached below.

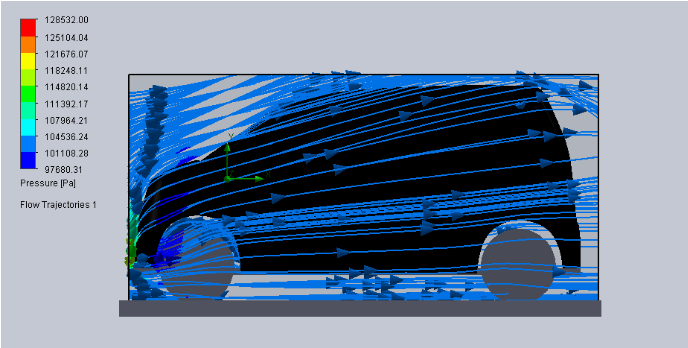


Fig. 1. Pressure [Pa] – CFD Analysis (FloEFD)

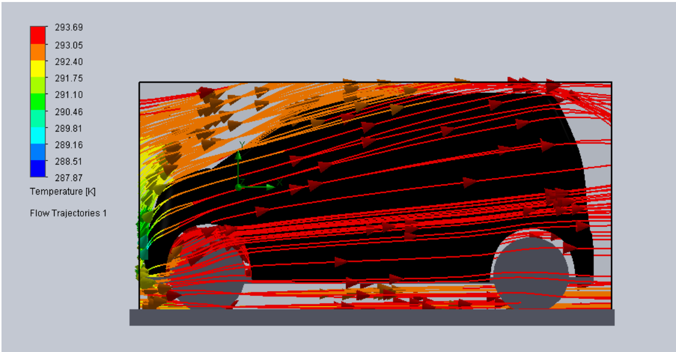


Fig. 2. Temperature [k] – CFD Analysis (FloEFD)

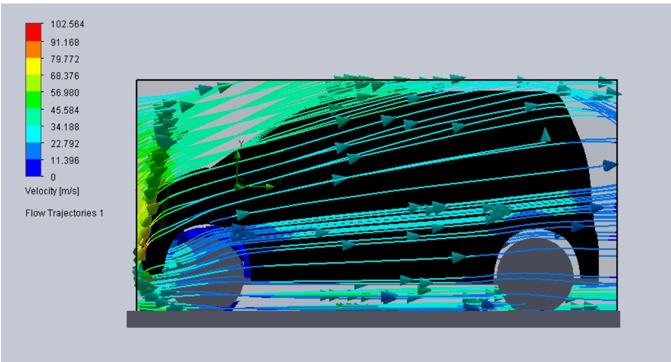


Fig. 3. Velocity [m/s] – CFD Analysis (FloEFD)

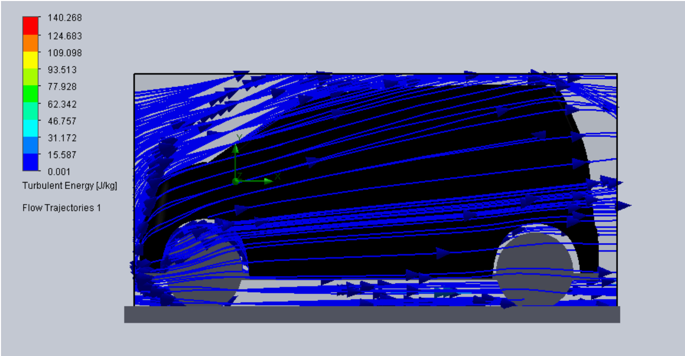


Fig. 4. Turbulent Energy [J/kg] – CFD Analysis (FloEFD)

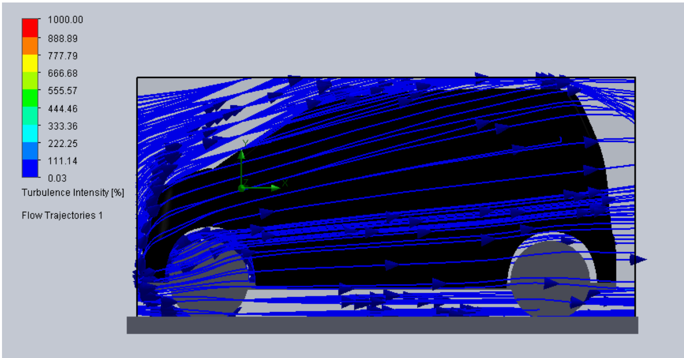


Fig. 5. Turbulence Intensity [%]– CFD Analysis (FloEFD)

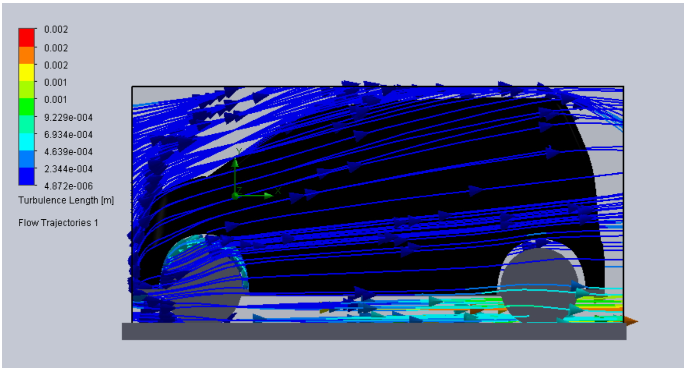


Fig. 6. Turbulence Length [m] – CFD Analysis (FloEFD)

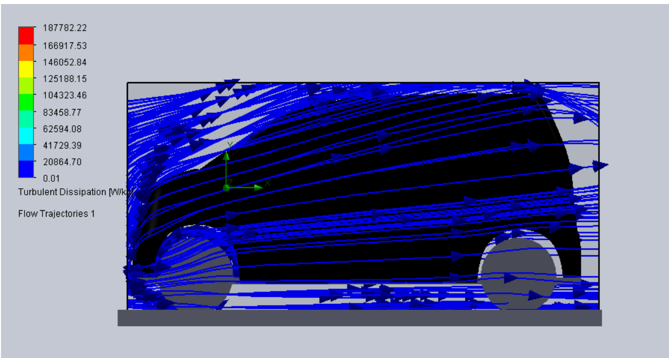


Fig. 7. Turbulent Dissipation [W/kg] – CFD Analysis (FloEFD)

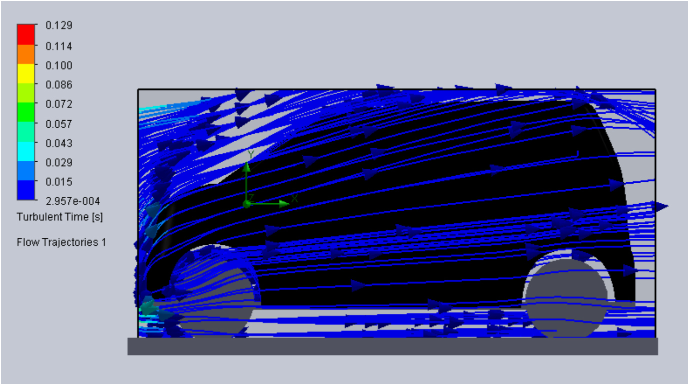


Fig. 8. Turbulent Time [s] – CFD Analysis (FloEFD)

**Case-2:** A smooth shape and smooth edges, so the friction with the air is too low which iproves its performance and this in turn improves the mileage of the car. The FloEFD of Swift is attached below.

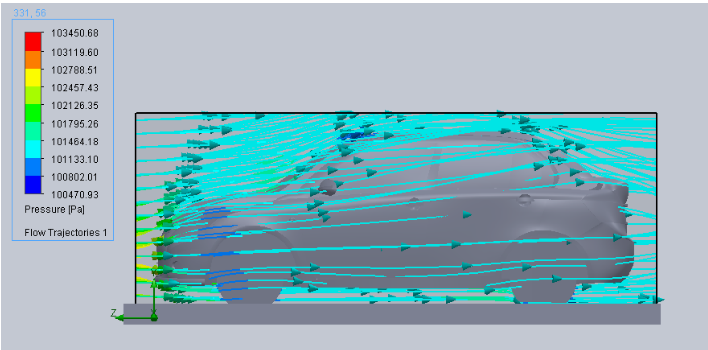


Fig. 9. Pressure [Pa] – CFD Analysis (FloEFD)

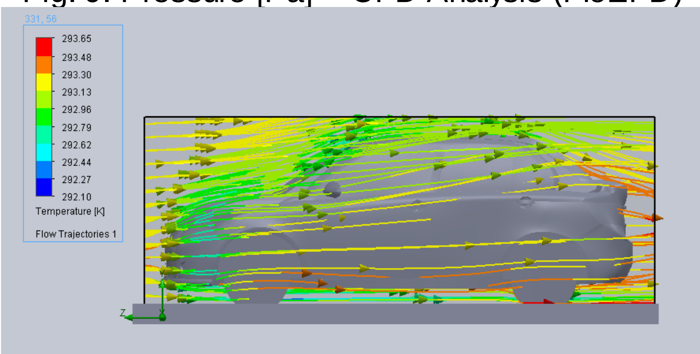


Fig. 10. Temprature [K] – CFD Analysis (FloEFD)

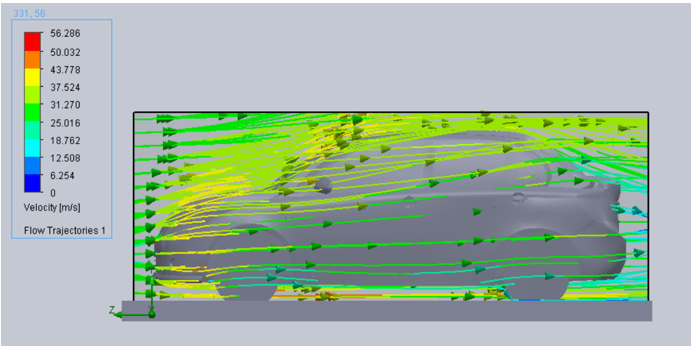

Fig. 11. Velocity [m/s] – CFD Analysis (FloEFD)

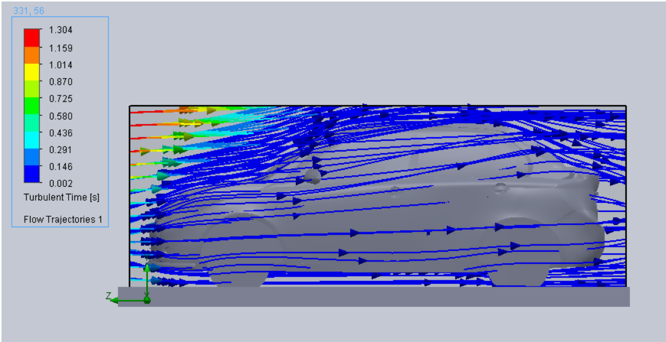


Fig. 12. Turbulence Time [s] – CFD Analysis (FloEFD)

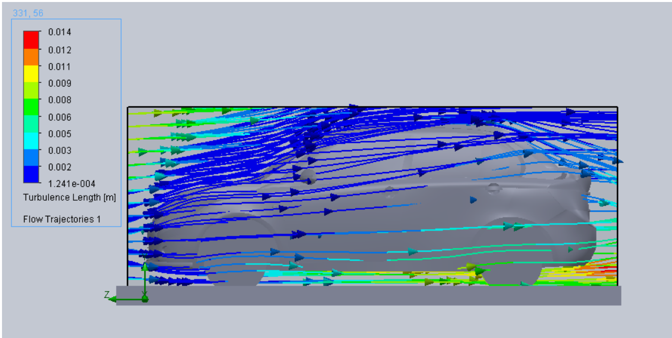


Fig. 13. Turbulence Length [m] – CFD Analysis (FloEFD)

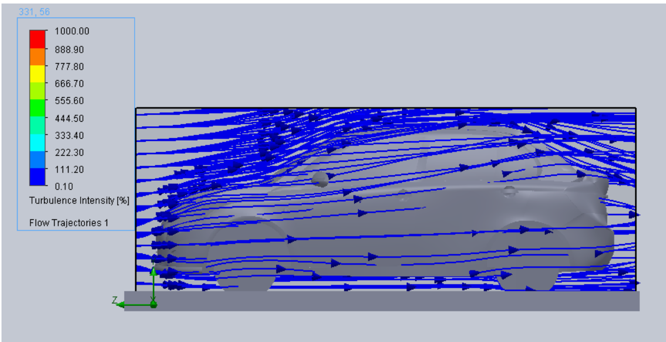


Fig. 14. Turbulence Intensity [%]– CFD Analysis (FloEFD)

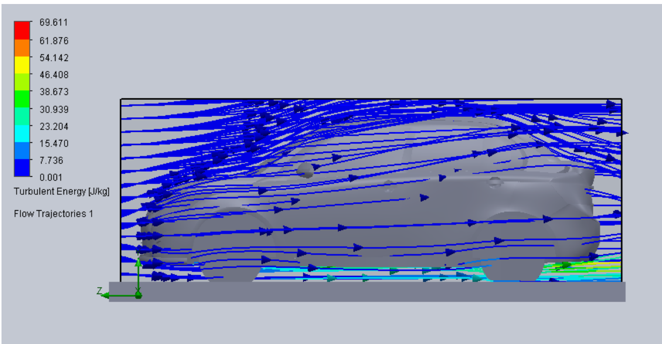


Fig. 15. Turbulent Energy [J/kg] – CFD Analysis (FloEFD)

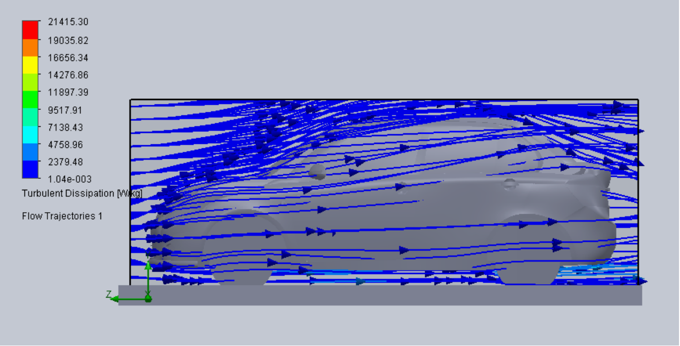


Fig. 16. Turbulent Dissipation [W/kg] – CFD Analysis (FloEFD)

## 5. Conclusion

The general purpose CAD-embedded CFD solver in the FloEFD software package has been benchmarked against a wide range of CFD turbulence cases as used here and its two equation modified $k$-$\varepsilon$ turbulence model with its unique two-scale wall functions approach and immersed boundary Cartesian meshes leads to good predictions for spatial laminar, turbulent, and transitional flows over a range of compressible and anisotropic flows.


### Acknowledgements

The authors wish to thank N.H.C.E. Bangalore, M.G.P.L. Bangalore and S.W.R.E., Jadavpur University , Kolkata for the valuable technical support.


### References


1. P.Adhikary;Design and CFD analysis for pump impeller;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P25

2. P.Adhikary;CFD analysis of concept car for improvement of aerodynamic design;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P20

3. P.Adhikary;Design and CFD analysis of centrifugal pump;2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P11

4. P.Adhikary;12V DC pico hydro for hilly rural electrification: performance analysis by C.F.D. (Best Poster Award);2nd RSTC (WR), Science Congress 2017 (WBSSTC 2018), Paper ID: EG-P1

5. P.Adhikary;Micro Hydropower Generation from Rural Drinking W.T.P.: C.F.D. Analysis & It's Validation;FMFP 2016, International Conference, MNNIT Allahabad

6. P.Adhikary;C.F.D. ANALYSIS OF 12V 10W DC MICRO HYDRO TURBINE: A CASE STUDY;NCETPFS 2016, National Conference, Jadavpur University, Kolkata

7. P.Adhikary;Design and development of Arduino controlled self balancing duct cleaning robot: a case study;International Conference on "Emerging Technologies for Sustainable and Intelligent HVAC &R Systems" (ICHVACR 2018);Pages:121-124