Computation of High-Lift Aerodynamics for Multi-Element Aerofoils at Landing and subsonic Flight Conditions using OpenFOAM

Naveen.K.M, naveenmtech07@gmail.com	National Aerospace Laboratories (CSIR), Bangalore-560017, India
Vidyadhar Y. Mudkavi, vm@ctfd.cmmacs.ernet.in	National Aerospace Laboratories (CSIR), Bangalore-560017, India

Abstract

The OpenSOURCE CFD (Computational Fluid Dynamics) software package OpenFOAM is used in this work to evaluate the Aerodynamics co-efficients of a two dimensional Multi-Element Aerofoils for a Regional Transport Aircraft. OpenFOAM is evaluated against results obtained from the commercial CFD program Fluent and an in-house solver JUMBO for CFD analysis.

There has been a major boost in the volume of air traffic in Asian region in general and in India in particular. Geographical and population distribution suggests that there is a need for a new regional transport aircraft of 70 to 90 seat capacity. However, most airports that can serve such aircraft have short runways, typically 1 km to 1.5 km in length. This places a design constraint on the aircraft as a whole. Particularly, there arises a need for design of a suitable high-lift system **[1, 2].**

In this study, we consider CFD analysis of high-lift configuration based on a newly designed aerofoil, NLF-7025, meant for sustaining natural laminar flow in cruise (Airfoil numbering is internal to our design team). The configuration consists of double slotted flap system at the trailing edge and a drooped nose at the leading edge **(Figure 1).**

The primary objective of the present study is to establish basic methodology of CFD analysis using OpenFOAM so as to employ the same for determining optimal flap location by performing analysis for different flap locations.

The second objective is to evaluate an OpenSOURCE CFD tool OpenFOAM **[3]** as viable tool for design inputs, because many Aerospace industries investing a lot of money each year in commercial CFD solvers. The availability of cheap hardware makes it possible to do simulations on a large number of CPUs, which requires many expensive software licenses. There is thus a need for a high quality CFD tool that is cheap, and OpenFOAM is the first tool to meet those demands. OpenFOAM has many of the features that are available in the commercial CFD codes, and due to the OpenSource distribution under the GPL licence it can be used at no cost and offers choice of different solvers and options for turbulence modeling and provides a variety of Finite volume solvers and supports both structured and unstructured grids. One aspect of OpenFOAM that is different than FLUENT is that more options are available for customizing fields, where as FLUENT can only specify the velocity, pressure, k and ϵ (for k-ϵ model), OpenFOAM additionally allows specification of the Reynolds stresses,etc. Since it also provides access to source code, custom changes can be made by user to address specific problems.

In the current study, basic validation studies were carried out on standard aerofoil cases using OpenFOAM and compared with Commercial CFD tool FLUENT and JUMBO, an in-house code **[4]**. Next, computations were carried out for the multi-Element aerofoil with drooped nose and the results were compared with FLUENT and JUMBO **(Figure 2)**. A multi-block grid was employed for OpenFOAM, FLUENT and in-house code with 1,56,000 cells. The flow conditions consisted of M=0.2 and Reynolds number of 6 million. The results are in good agreement with each other. For the OpenFOAM and FLUENT solution, Spalart-Allmaras turbulence model was employed assuming fully turbulent flow. However, the flow was assumed incompressible, SimpleFOAM and turbFOAM solvers were employed in OpenFOAM. For the in-house code, Baldwin-Lomax model was used.

Further studies were carried out in order to make a qualitative assessment of the flow features such as possible flow separation, flow nature through the gaps and so on. Comparisons of pressure fields and velocity fields are shown in **Figures 3** and **4** respectively. In general, there is a good qualitative agreement with OpenFOAM and FLUENT. Pressure co-efficient distribution on the main element and flaps is shown in **Figure 5**. Once again, the results are in good agreement except for possible differences in suction peak values. In general, OpenFOAM solutions reached convergence in half as many iterations when compared to FLUENT. This suggests that OpenFOAM is a viable option for CFD analysis due to its free-to-use OpenSOURCE Numerical simulation software with extensive CFD and multi-physics capabilities without paying for license and support, including massively parallel computers.

Keywords: CFD, OpenFOAM, FLUENT, High-lift Systems, Multi-Element Aerofoils, Flaps.

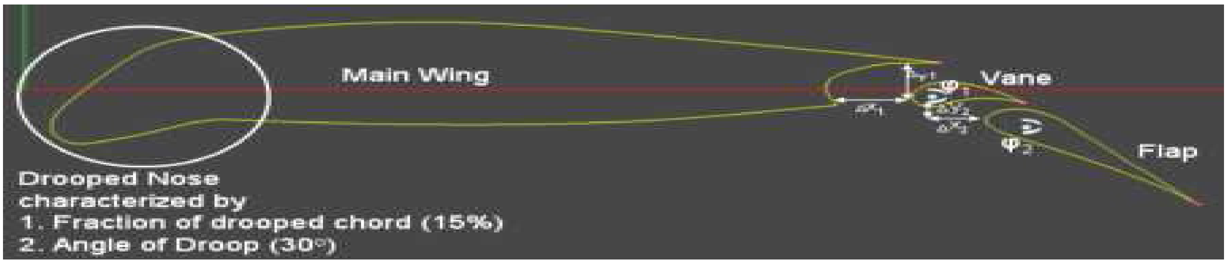
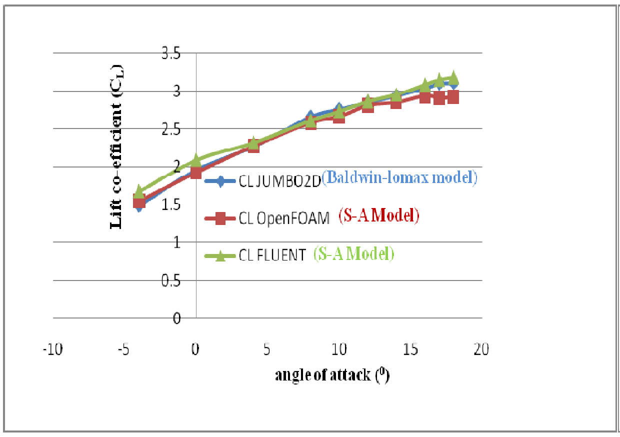
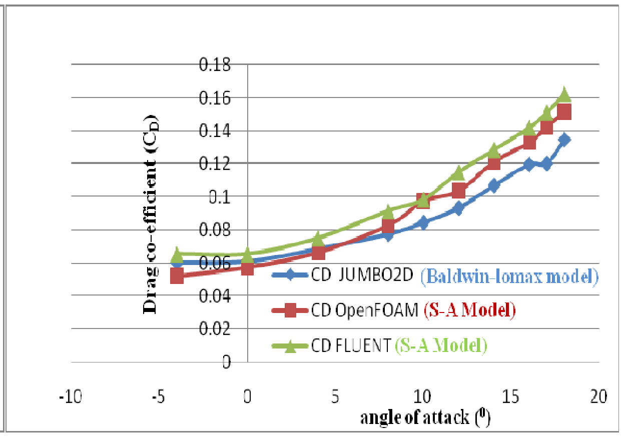


Figure 1: Double slotted Multi-Element aerofoil geometry. Typical landing configuration consisting of flap deflection, δ_F=35^0. Basic aerofoil consits of newly designed NLF 7025.

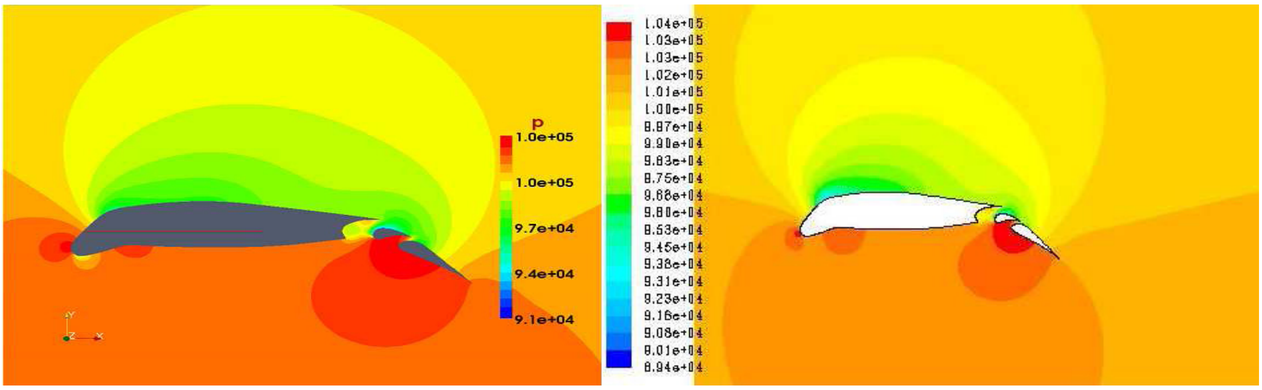


(a) Lift co-efficient (C_L) vs. angle of attack (α)

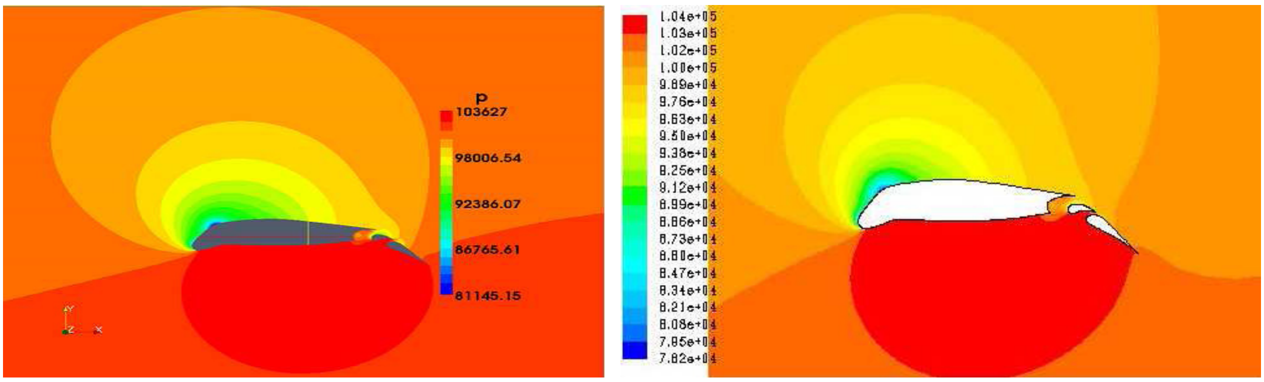


(b) Drag co-efficient (C_D) vs. angle of attack (α)

Figure 2: Comparison of Aerodynamic co-efficients using OpenFOAM, FLUENT and JUMBO

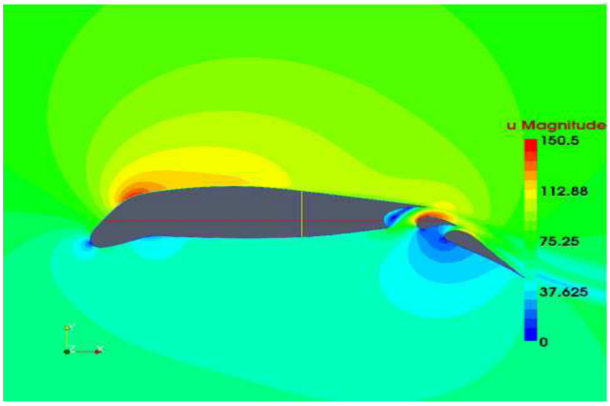


(a) **OpenFOAM**, α=0^0 (b) **FLUENT**, α=0^0

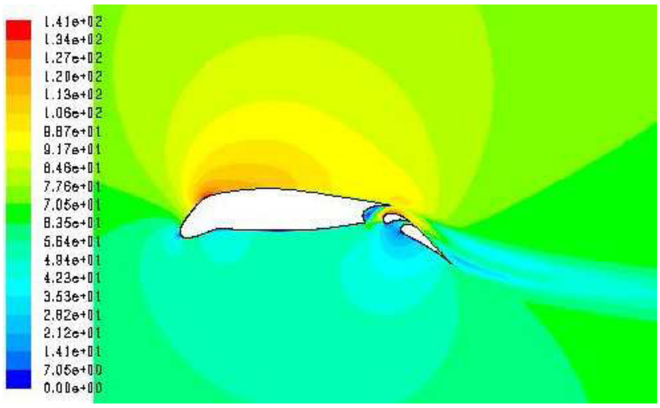


(a) **OpenFOAM**, α=16^0 (b) **FLUENT**, α=16^0

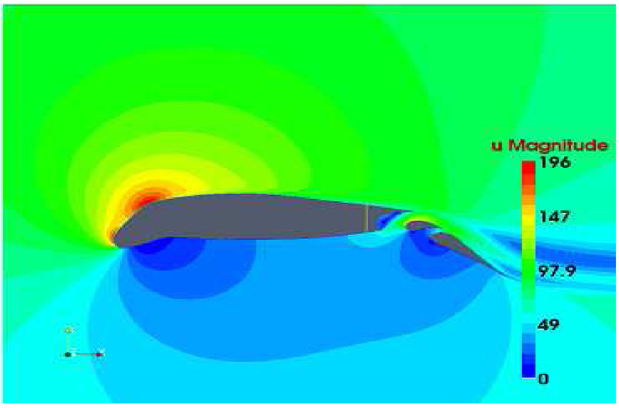
Figure 3: Qualitative comparison of pressure distributions between OpenFOAM and FLUENT



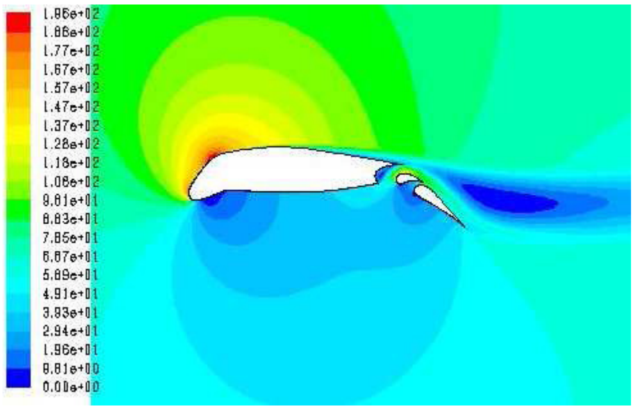
(a) **OpenFOAM**, α=0^0



(b) **FLUENT**, α=0^0

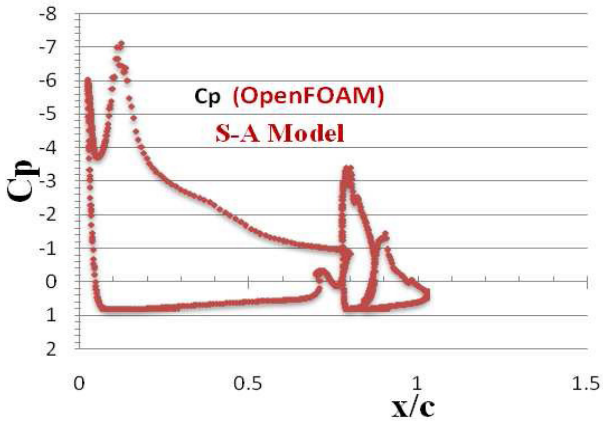


(a) **OpenFOAM**, α=16^0

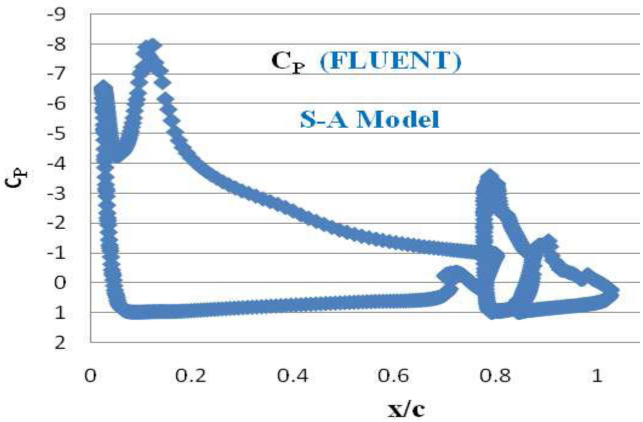


(b) **FLUENT**, α=16^0

Figure 4: Qualitative comparison of velocity distributions between OpenFOAM and FLUENT



(a) **OpenFOAM**, Pressure co-efficient (Cp) vs. x/c



(b) **FLUENT**, Pressure co-efficient (Cp) vs. x/c

Figure 5: Pressure co-efficient distribution over main element and flaps

References

[1] Peter K. C. Rudolph, *"High-Lift Systems on Commercial Subsonic Airliners"*. NASA CR 4746, 1996.

[2] M. A. Averardo, M. de Leo, V. Russo, AGARD-CP-515, *High-lift aerodynamics, optimization of a high-lift system for an advanced aircraft. 1992.*

[3] http://www.opencfd.co.uk/openfoam.

[4] Manish K. Singh, K. Dhanalakshmi and J. S. Mathur. "*Computation of Two-Dimensional Flow Past Multi-Element Airfoils"*. Project Document PD CF 0902. National Aerospace Labs., Bangalore, India. 2009.