COBEM 2023
27TH INTERNATIONAL CONGRESS OF MECHANICAL ENGENEERING

ABCM
Brazilian Society of Mechanical Sciences and Engineering
27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0272
VALIDATION AND VERIFICATION OF AN OPEN-SOURCE CFD CODE FOR THE AERODYNAMICS SOLUTION OF AIR AND GROUND VEHICLES

Guilherme Espíndola da Silva
Odenir de Almeida
CPAERO - Aerodynamic Research Center, Federal University of Uberlândia, School of Mechanical Engineering, BR 05, Km 78, Bloco 1DCG, Campus do Glória, Uberlândia, Minas Gerais, 38400-902, Brazil
guilhermeespindola123@ufu.br
odenir.almeida@ufu.br

Abstract. *This work aims to present data in order to compare two Computational Fluid Dynamics (CFD) software: the commercial ANSYS Fluent and the open-source OpenFOAM, using experimental test data as a reference. This research seeks to validate OpenFOAM as a CFD tool for incompressible, steady-state external flow models. The study focuses on the aerodynamics of the Ahmed body with a 25° tilt angle and an air velocity of 40 m/s, resulting in a Reynolds number of* 2.859×10^6 *(based on the length of the body, which is* 1.044 *m). The focus of this work is built upon turbulence models, computational time, the influence of the mesh element count, and how all these configurations can affect the prediction of drag coefficient and velocity profiles. The adopted methodology consists of two steps, with the first being a mesh test in both software, varying the number of elements from 2 to 5 million. The second step investigates the influence of turbulence models: SST k-Omega, Realizable k-Epsilon, and Spalart-Allmaras. The results showed similar trends in velocity profiles but diverged with the Realizable k-Epsilon turbulence model. OpenFOAM achieved good accuracy with the SST k-Omega model, showing an error of 2.02% compared to ANSYS Fluent's 3.35% . However, OpenFOAM was four times slower. Furthermore, the use of the Spalart-Allmaras turbulence model resulted in an unfeasible error of over 50% .*

Keywords: *CFD, OpenFOAM, Turbulence Model, Open Source, Validation.*

1. INTRODUCTION

Ahmed body is a simplified car model originally developed for time-averaged vehicle wakes investigation since its form makes it possible to generate the main flow characteristics of real vehicles without its geometrical details. Aerodynamic studies on the Ahmed body reveal significant information about how the air flow is altered due to several vehicle aspects for instance its shape, size, and inclination angle at the rear of the vehicle. This allows for designers to perform adjustments in order to optimize and improve the aerodynamic aspects of the cars and therefore, their efficiency.

Although the vehicle wake structure is essentially unsteady, the inflow time average is capable of displaying the macrostructure responsible for the pressure drag generation on the rear region of the vehicle, which represents 85% of the total drag (Ahmed *et al.*, 1984). The research conducted by Ahmed in 1984 was able to characterize the regions of recirculation, flow separation and longitudinal vortex on the vehicle wake. Through the well stablished flow characterization, the Ahmed body emerged as a reference to qualitative flow comparisons between experimental and numerical methods. In virtue of the great number of published references, substantial experimental database and time averaged wake investigation possibility, the Ahmed body is one of the most widely bluff bodies used for theoretical automotives studies and computational fluid dynamics (CFD) software validation.

Significant breakthroughs have been made since the emergence of the CFD in 1960 whereas theoretical or in practice of numerical simulations. The numerical algorithms have evolved from finite differences and first order finite volumes to high order methods, namely compact finite differences and finite elements methods. These high order methods endorse more precise and efficient simulations. Furthermore, the high-performance computation has become more available, facilitating the numerical simulation of more complex problems, and providing better resolution. In addition, parallelization and workload distribution techniques have also contributed to the larger scale complex simulations.

Therefore, the present paper aim to present simulations data in order to validate an open source CFD software for the aerodynamic solution of air and ground vehicles. This paper proposes the validation of OpenFOAM, a freely distributed source code CFD software capable of managing external, incompressible, or steady flows. In order to generate a solid database that enables the validation, ANSYS Fluent computational simulations results were used, as well as experimental tests results taken from literature.

The chosen approach for this paper has its foundation on the turbulence models, computational time usage, mesh elements number and the influence of these configurations on the drag coefficient prediction and velocity profiles. The

Ahmed body was selected to the aerodynamic investigation due to its favorable characteristics, as mentioned previously. The model characteristics used to this study are: 25° of inclination, air velocity of 40 m/s, body length of 1.044 m and the respective Reynolds Number of 2.859×10^6.

2. METHODOLOGY

2.1 Numerical codes and methods

OpenFOAM is a computational fluid dynamics software, free and open source, developed by OpenFOAM Foundation. The code is essentially written in C++ and presents a large variety of numerical schemes, methods, and turbulence models. The turbulence methods available range from RANS (Reynolds-Averaged Navier-Stokes) to RANS/LES (Large Eddy Simulation) (HRL) and LES. It is also possible to solve all the scales using the Direct Numerical Simulation (DNS). ANSYS Fluent is a commercial CFD software developed by ANSYS Inc. ANSYS Fluent is broadly used in academic research and industrial applications all around the world, and it is also considered the major CFD software available. Both softwares operate through the finite volume method for the Navier-Stokes numerical solution and encompass farther relevant fluid dynamics equations.

The simulations were conducted based on the RANS approach. This method consists of a mathematical model used in CFD based on the time average fluid dynamics equations, considering that the variables of interest, namely the fluid pressure and velocity, can be divided into an average component and a turbulent one. The RANS model assumes that the turbulent component is suitably described with the assistance of a turbulence model. Therefore, the present paper incorporates three turbulence models: k-Omega SST, k-Epsilon Realizable and Spalart-Allmaras.

In order to guarantee a correspondence between softwares, the pressure-velocity coupling method SIMPLE was used, which stands for a semi-implicit method for equations related to pressure. This method pursued a steady-state solution with the assistance of underrelaxing factors between iterations. Therefore, the under-relaxing of the equation promotes the diagonal dominance by increasing the influence of the proprietary cell terms (Robertson *et al.*, 2015). Further details of the complementary data for the simulations are arranged in Table 1 such as turbulence boundary conditions, over-relaxation factors and reference values. Moreover, discretization methods selected for this study are presented in Table 2 and Table 3.

The computer configurations where the simulations were run are: Inter(R) Core(TM) i5-9400F CPU @ 2.90 GHz, which presents 6 cores per socket, 24 Gb of 2666 MHz RAM. All simulations were run in parallel with 6 processors cores. The operating system for the OpenFOAM simulations was Ubuntu-Linux (22.04 LTS) whereas ANSYS Fluent (2022 R1) was run on Windows 10.

Table 1. Summary of settings used in simulations.

Discretization		Value	Unit
Turbulence properties	Turbulent kinetic energy - k	0.24	m^2/s^2
	Specific turbulence dissipation - ω	1644	1/s
	Turbulence dissipation - ε	35.51	m^2/s^3
	Turbulence intensity - Tu	1	%
	Eddy viscosity ratio - μ_t/μ	10	-
Relaxation factors	Pressure - P	0.3	-
	Turbulent kinetic energy - k	0.8	-
	Specific dissipation rate - ω	0.8	-
	Turbulence dissipation rate - ε	0.8	-
	Modified turbulent viscosity - nut	0.8	-
Reference values	Half of front area	0.0575	m^2
	Density	1.225	kg/m^3
	Length	1.044	m
	Pressure	0	Pa
	Velocity	40	m/s
	Dynamic viscosity	$1.789e^{-5}$	kg/(ms)
	Kinematic viscosity	$1.46e^{-5}$	m^2/s

Table 2. Summary of discretization and modeling methods used in OpenFOAM.

Discretization		Schemes
Time schemes		Steady-state
Spatial discretization	Gradient	Least squares
	Divergence	Linear upwind
	Laplacian	Linear
	Interpolation	Linear
	Surface normal gradient	Limited 1
Pressure-velocity coupling		SIMPLE
Wall functions	Turbulent kinetic energy - k	kqRWallFunction
	Specific turbulence dissipation - ω	omegaWallFunction
	Turbulence dissipation - ε	epsilonWallFunction
	Turbulent viscosity - μ_t	nutUSpaldingWallFunction

Table 3. Summary of discretization and modeling methods used in ANSYS Fluent.

Discretization		Schemes
Time schemes		Steady
Solver		Pressure-based
Spatial discretization	Gradient	Least Squares Cell Based
	Divergence	Second order Upwind
Pressure-velocity coupling		SIMPLE

2.2 Computational domain, grids and boundary conditions

The schematic drawing of a conventional Ahmed body with a tilt angle of 25° is displayed in Figure 1, with the main measurements presented in millimeters.

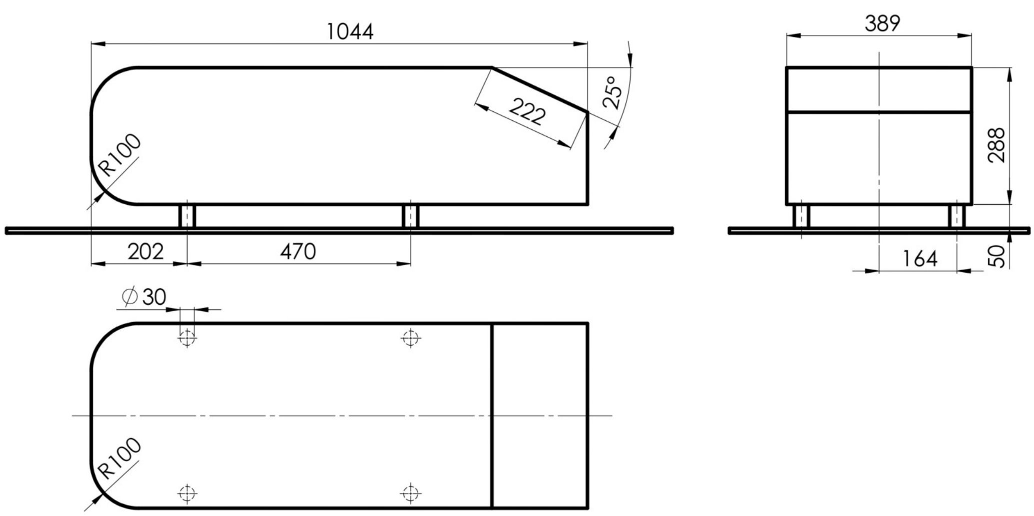


Figure 1. Ahmed body schematic drawing considering its main dimensions.

The computational domain and the boundary conditions applied in this study are illustrated in Figure 2. The domain dimensions are presented in dimensionless form, where the symbol L is defined as the length of the Ahmed body, which is 1.044 m. Aiming to reduce the computational demand, this research explored the use of symmetry boundary conditions applying a half model. This is possible considering the half body simulation sustains the representation of some the main flow characteristics observed in experiments with the whole Ahmed body.

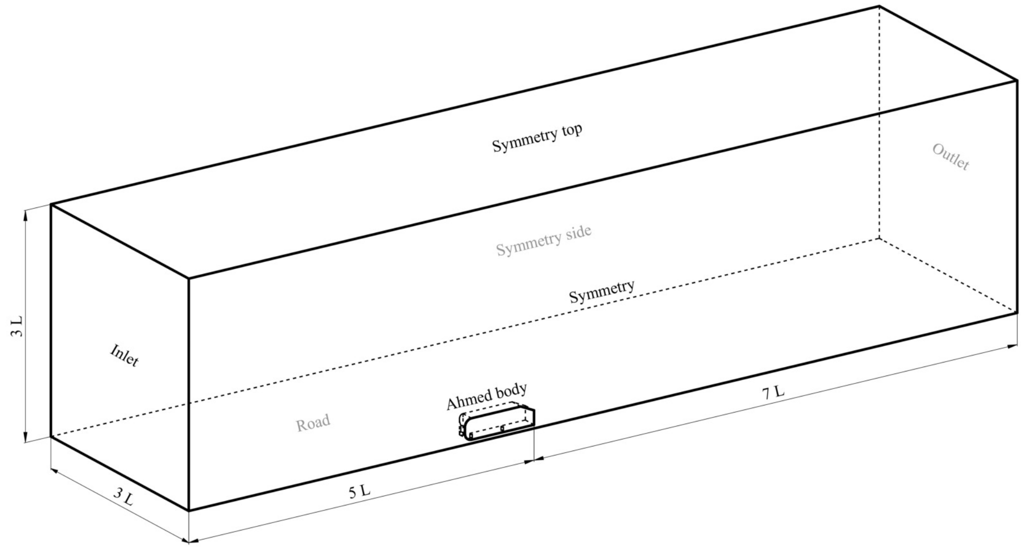


Figure 2. Computational Domain and boundary conditions of Ahmed body numerical simulation.

The chosen grid for the turbulence influence study consists of approximately 5 million elements, a number suitable with the computational time alongside with the precision required for the results. Furthermore, with respect to the viscous sublayer, it is required that the first cell in the grid reaches $y^+ \approx 1$ in order to solve the sublayer gradients properly. This is consistent with low Reynolds turbulence models with k-Omega SST. When it comes to establishing a correspondence between turbulence models with high Reynolds numbers such as k-Epsilon, which requires a wall function, the k-Epsilon Realizable model is adopted. This model incorporates an enhanced wall treatment (EWT), which is a wall function unresponsive to y^+. The grids created for this study present a maximum y^+ of 2.329422.

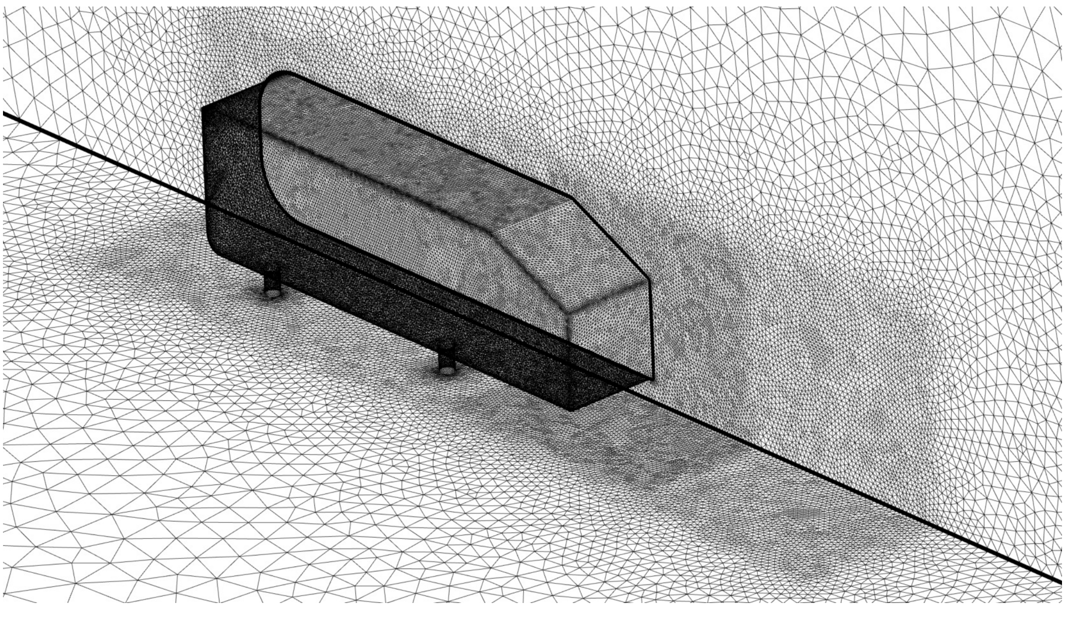


Figure 3. Surface mesh of Ahmed body with Refinement boxes.

Amid the many criteria used to evaluate the mesh quality, the chosen one for this study is the orthogonal quality represented as a range of values comprehended in the interval between 0 and 1, with 0 being a highly distorted grid and

1 being neatly orthogonal. A favorable orthogonal mesh quality is expected around 1 for a good mesh. The values for orthogonal mesh quality of this study were a minimum of 0.02721, the maximum of 0.99878 and the average 0.78632.

The mesh construction was made by prism layers fitted into the Ahmed Body boundary-layer and tetrahedral cells in other parts as shown in Figure 3. The local refinement was designed to diminish the computational cost and increase the number of cells in specific regions. This technique was applied to the region around the Ahmed body due to the intensified variation of flow properties in this area. Therefore, the generated mesh classifies as an unstructured grid.

3. RESULTS

3.1 Grid dependence and processing time

The two aspects that characterize a successful simulation are convergence and grid independence. The only way to eliminate errors due to coarseness of a grid is to perform a grid dependence study, which is a procedure of successive refinement of an initially coarse grid until certain key results do not change (Versteeg and Malalasekera, 2007). Thereby, the grid dependence test was performed on both softwares. Initially, this process aims to identify a convergence in the results as the number of elements increases. However, the time taken to complete each simulation, namely processing time, is also a relevant aspect of evaluation, since it represents a proportional relationship with the number of elements. Accordingly, it is beneficial to identify the situation in which the results display a grid independence as well as the minimum processing time.

Furthermore, the Table 4 and Table 5 exhibit data from the simulations in both softwares, demonstrating the relationship between the drag coefficient (Cd) depending on the number of grid elements. In order to compare the processing time between both softwares, a maximum of 6000 iterations number was stablished. Moreover, the convergence throughout the simulations was guaranteed with a maximum residue of $10e^{-5}$ for all variables.

Table 4. Mesh test for ANSYS Fluent, using SST k-Omega with 6000 iterations.

Number of elements	Cd	Processing time	
		Seconds	Hours
2437843	0.3146	18416.68	5.12
3584401	0.3021	26948.70	7.49
4308435	0.3099	34456.24	9.57
5065622	0.3096	44932.05	12.48
6622739	0.3078	53211.94	14.78

Table 5. Mesh test for OpenFOAM, using SST k-Omega with 6000 iterations.

Number of elements	Cd	Processing time	
		Seconds	Hours
2437843	0.3440	71802.60	19.98
3584401	0.3145	101088.00	28.08
4308435	0.3278	129588.47	36.00
5065622	0.2930	147550.00	40.99
6622739	0.3265	282893.03	78.58

The following investigations of this study required a definition of a reference mesh in order to examine the influence of the variation of turbulence models in the results. Nonetheless, a reference mesh demands a region of independence from the number of elements and the minimum computational cost. Therefore, based on the displayed results, the grid that better served the requirements is the one with approximately 5 million elements.

Moreover, the gathered data subsequently the grid dependence test from both softwares showed that the OpenFOAM simulations presented a processing time 4 times longer than the simulations run on ANSYS Fluent. Since, all the configuration parameters were kept equal in both softwares, this is an important result despite the fact that it is difficult to verify the reasons for that.

3.2 Influence of turbulence model

This section purpose relies on the presentation of the turbulence models influence on the drag coefficient prediction and the velocity profiles on the rear part of the Ahmed body. Furthermore, a relevant comparison parameter is considered through the evaluation of the 6000 iterations processing time.

In search of a solid validation, the present study explores the experimental data as an additional comparison procedure. The experimental velocity profiles data were taken from the ERCOFTAC (European Research Community On Flow, Turbulence And Combustion) database, entitled as "Flow Around a Simplified Car Body (Ahmed Body), experiments by (Lienhart *et al.*, 2002). However, the data used as reference for the drag coefficient was taken from (Meile *et al.*, 2011) study. All the data used as reference present a Reynolds Number of 2.859×10^6 and inclination angle of 25°.

Therefore, Table 6 and Table 7 provide data from the ANSYS Fluent and OpenFOAM simulations, respectively. These tables present the drag coefficient (Cd) as well as the processing time depending on the turbulence model. Thus, the percentage error between experimental and computational methods is displayed. The mathematical model used for the error calculation is $Error = (D - S)/D \times 100$; D = experimental data and S = simulation results.

Figure 4 presents graphs illustrating the variation of the drag coefficient as a function of the number of iterations for different simulations conducted, each associated with distinct software and turbulence models. The drag coefficient is a relevant measure in this study, and the information derived from these graphs pertains to the convergence of the simulations. It can be stated that a CFD simulation reaches a state of convergence when the fluctuation of the variable of interest is reduced to a level below the established tolerance. Visually, this convergence becomes evident when the analyzed curve tends to stabilize in a constant pattern. Upon examining Figure 4 (e) and (f), it is noticeable that the variable of interest tends to assume atypical values. This characterizes a situation of divergence in the simulations. Based on this, we can conclude that simulations employing the Realizable k-Epsilon turbulence model exhibited divergence in both utilized software. Due to this observation, the absence of using this turbulence model was chosen for future analyses.

Table 6. Influence of the turbulence model using ANSYS Fluent, based on 6000 iterations.

Turbulence Model	Cd	$Error$	Processing time	
			Seconds	Hours
SST k-Omega	0.310	3.35	44932.05	12.48
Spalart-Allmaras	0.316	5.80	30935.57	8.59
Experiment	0.299			

Table 7. Influence of the turbulence model using OpenFOAM, based on 6000 iterations.

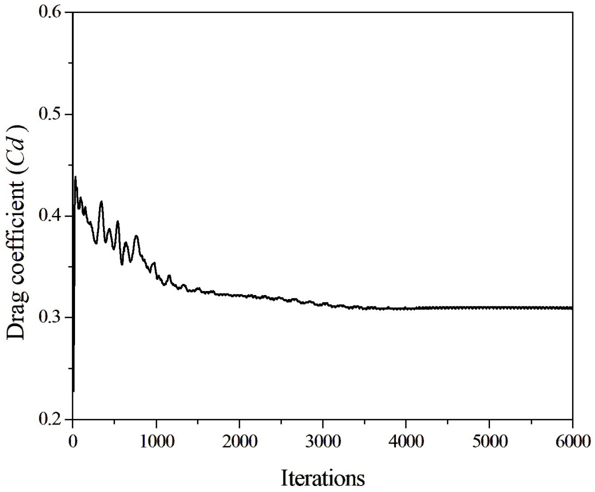
Turbulence Model	Cd	$Error$	Processing time	
			Seconds	Hours
SST k-Omega	0.293	2.02	147550.00	40.99
Spalart-Allmaras	0.467	56.19	212485.70	59.02
Experiment	0.299			

The next analysis seeks to evaluate the flow prediction around the Ahmed body through two main post processing approaches. The first one encompasses a quantitative method through numerical data from the simulation, namely plots, forces and functions reports, in order to identify patterns and tendencies to compare analytical and experimental results. The second approach handles the qualitative method, primarily used for a complex overall phenomenon visualization through velocity contours, flux vectors and pathlines. Both methods are complementary, hence quantitative data display numbers essential to the validation of the results and qualitative data enable the comprehension of the complexity of details obtained from the simulations.

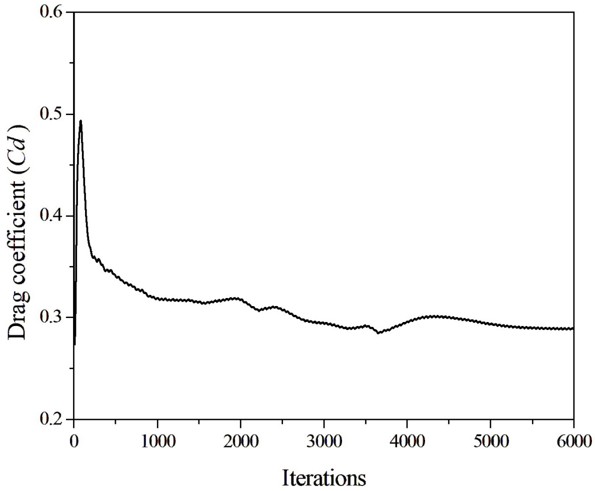
Therefore, the present paper explores both post processing approaches in order to evaluate OpenFOAM predictions. The analysis region concentrates on the rear part of the Ahmed body. This is because in that region the flow becomes more complex due to the associated phenomena such as detachment, recirculation, and vortex formation. Furthermore, as mentioned in (Ahmed *et al.*, 1984), the pressure drag related to the flow on the rear part is responsible for up to 85% of the total drag of the vehicle.

A first analysis evaluates the velocity profiles on the back of the body, assessed as a quantitative evaluation. The velocity profile indicates the magnitude of the velocity as a function of position. Thus, three positions were selected to enable the comparison between the two selected softwares and experimental results. The first position to be evaluated is located on the inclined surface and the other two locations are on the vehicle's wake, as shown in Figure 5.

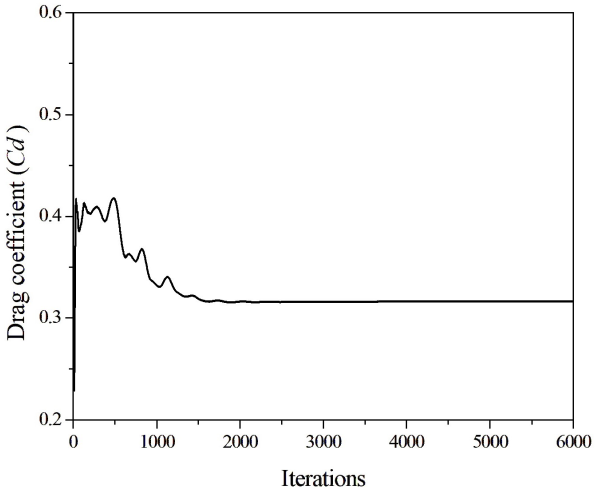
Moreover, Figure 6 presents a matrix configuration of plots containing the velocity profiles generated for the specific positions mentioned, with two different turbulence models. Each graph contains three curves in which two are with respect to the ANSYS Fluent simulation, OpenFOAM and the third one taken from experimental data from a wind tunnel. The columns in the matrix form refer to the analyzed position and the lines refer to the turbulence model. Therefore, the first line represents the SST k-Omega turbulence model and the second one represents Spalart-Allmaras. Thus, it is possible to evaluate the turbulence model influence on the prediction of the flow around Ahmed body and assess the differences between ANSYS Fluent and OpenFOAM curves simultaneously.



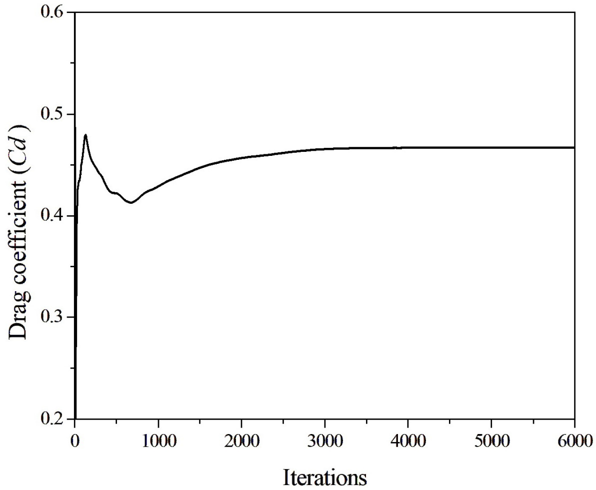
(a) ANSYS Fluent using SST k-Omega



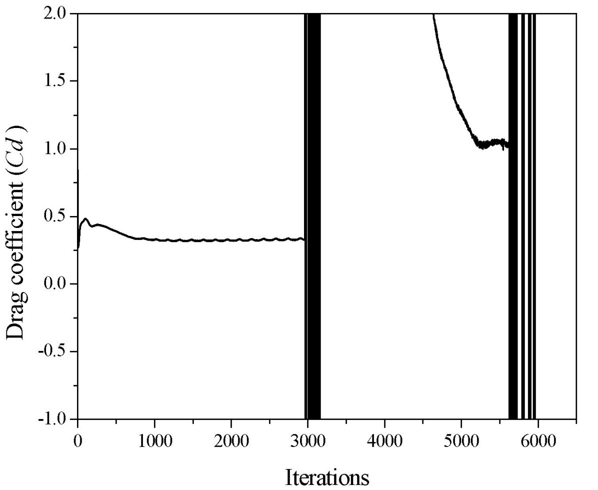
(b) OpenFOAM using SST k-Omega



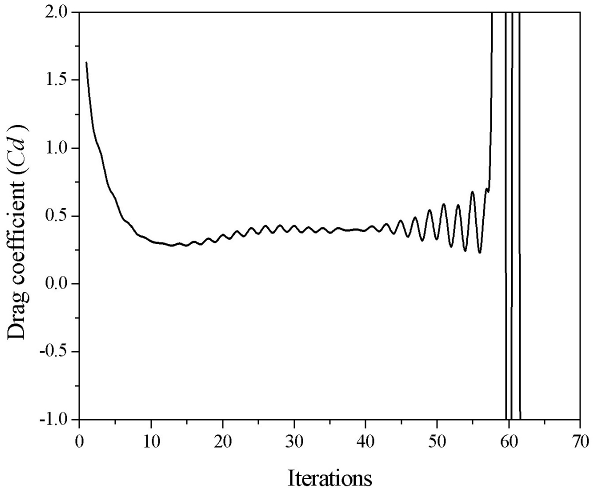
(c) ANSYS Fluent using Spalart-Allmaras



(d) OpenFOAM using Spalart-Allmaras



(e) ANSYS Fluent using Realizable k-Epsilon



(f) OpenFOAM using Realizable k-Epsilon

Figure 4. Drag coefficient response as a function of the number of iterations using different turbulence models in ANSYS Fluent and OpenFOAM.

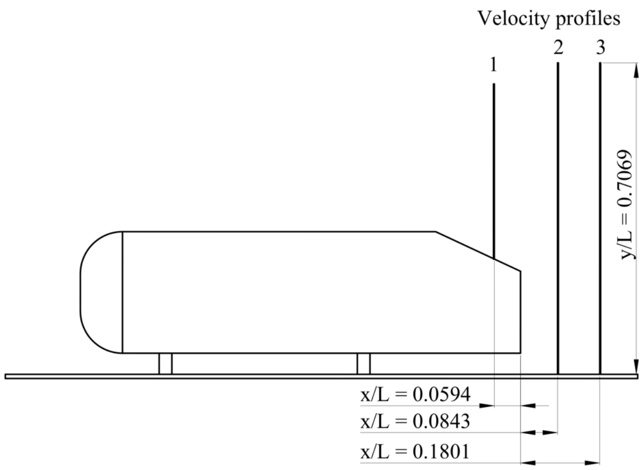


Figure 5. Velocity profile location analysis on the back of the Ahmed body.

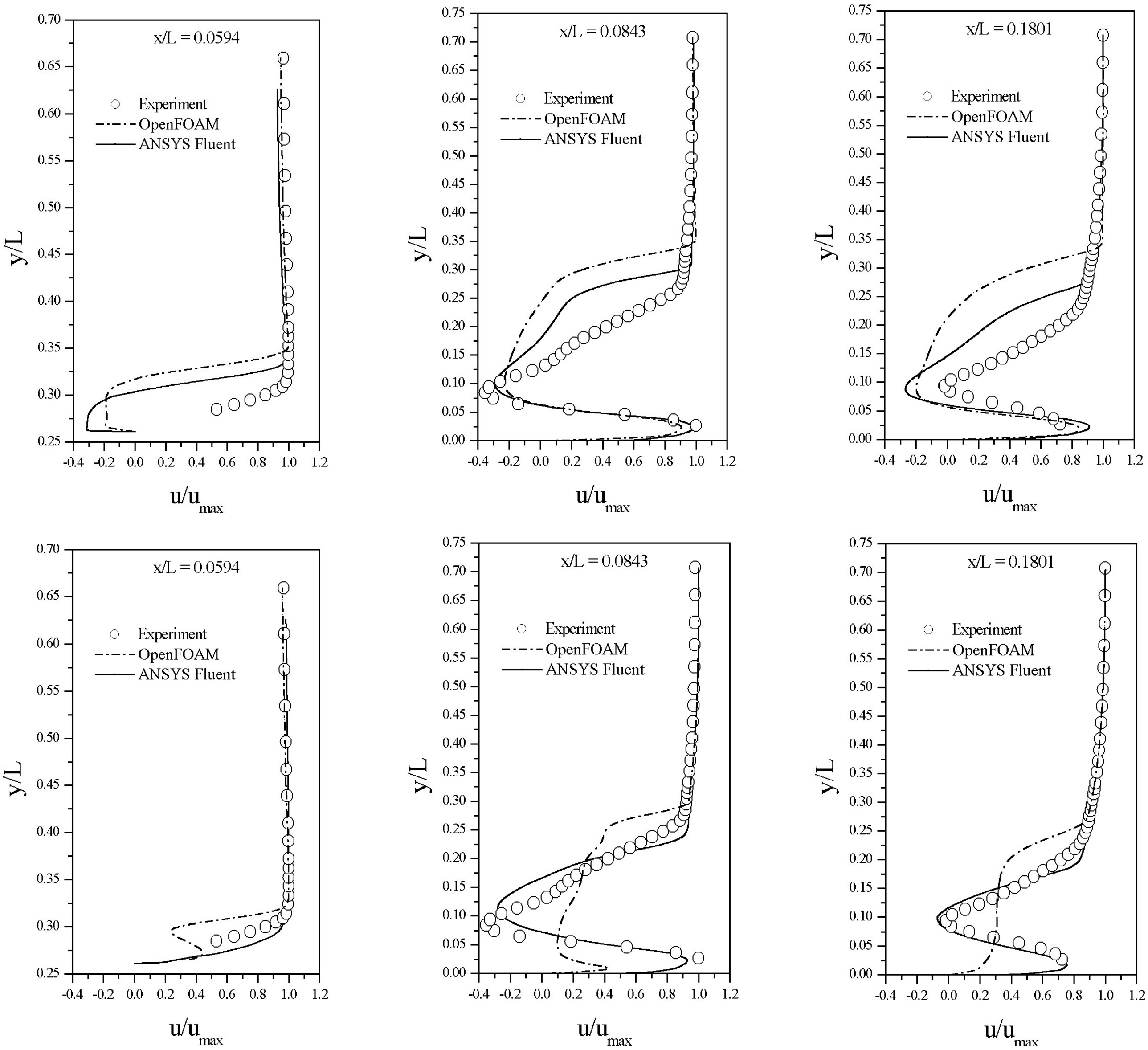
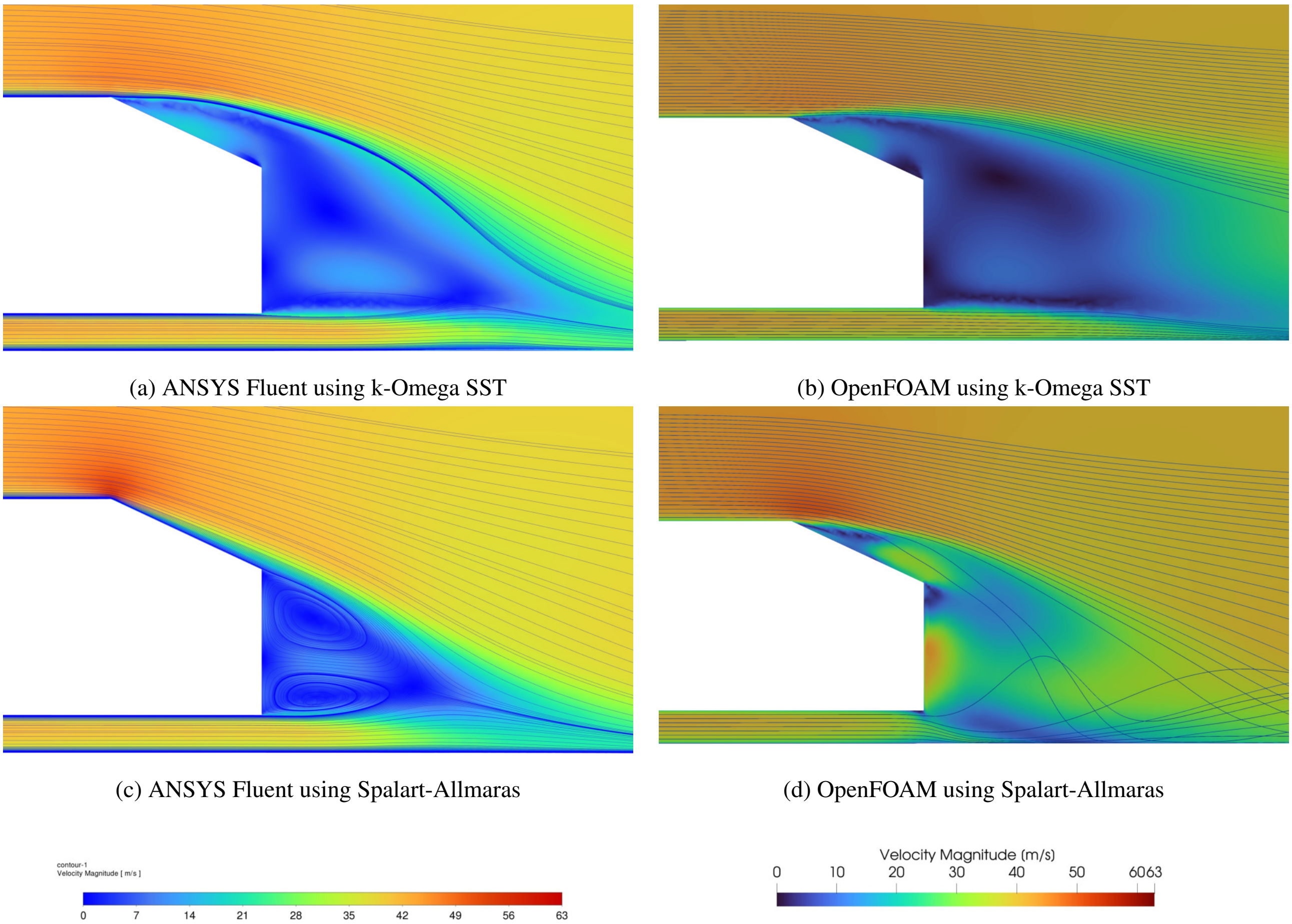


Figure 6. Comparison of velocity profiles predicted by different RANS turbulence models for the rear part of Ahmed body. The columns in the matrix form refer to the analyzed position and the lines refer to the turbulence model, the first line represents the SST k-Omega turbulence model and the second one represents Spalart-Allmaras.

The qualitative approach of the flow is demonstrated on Figure 7, which represents the velocity magnitude contour overlaid on the pathlines. The Figure 7(a) and Figure 7(b) display the results from the simulations using ANSYS Fluent and Open FOAM with the SST k-Omega turbulence model. It is possible to observe that both the recirculation regions and the boundary layer detachment occurred similarly in the simulations from both softwares. This is solidified since the drag coefficient (Cd) predicted values were similar. However, Figure 7(c) and Figure 7(d) exhibited dissimilarities since the simulation on OpenFOAM was not able to feature the flow behavior when compared to ANSYS Fluent. This is consistent with the discrepancies observed on the values for the drag coefficient.



(a) ANSYS Fluent using k-Omega SST

(b) OpenFOAM using k-Omega SST

(c) ANSYS Fluent using Spalart-Allmaras

(d) OpenFOAM using Spalart-Allmaras

Figure 7. Color map for velocity and streamlines showing flow circulation at the back of Ahmed body using different turbulence models.

4. CONCLUDING AND REMARKS

The present paper aimed for the validation of an opensource CFD software, designated OpenFOAM, through comparison between ANSYS Fluent simulations and experimental data. Initially, grid independence tests were taken in order to guarantee the quality of the results and enable the characterization according to the turbulence models. Therefore, the analysis was assessed quantitatively and qualitatively, seeking to compare the drag coefficient values, velocity profiles and the computational processing time for the flow around the Ahmed body using both softwares.

From the presented data, it is confirmed that the CFD analysis using OpenFOAM showed the same tendency as the simulations performed using ANSYS Fluent. This trend is confirmed from the observation of three aspects. The first one is related to the velocity profiles, which showed the same trend despite differences between curves. Further on, the second factor is exposed due to the occurrence of divergence in both softwares when using the Realizable k Epsilon turbulence model. Finally, when looking at the mesh evaluation in both softwares, it is possible to identify a break in the pattern of the drag coefficient when using the mesh of approximately 3.5 million elements.

Furthermore, it is worth noting that the use of OpenFOAM showed a good accuracy of the results when using the k-Omega SST turbulence model. The employment of experimental data as a reference demonstrates that OpenFOAM presented an error of 2.02% in contrast to the 3.35% observed by ANSYS Fluent, when adopting the same turbulence model.

Nonetheless, the computational time from the simulation performed in OpenFOAM was in average four times superior then the one performed in ANSYS Fluent. This is detrimental to the scalability and versatility of the software considering that it would require greater computational power to achieve ANSYS Fluent processing time. In addition, the error presented by OpenFOAM when using Spalart-Allmaras turbulence model is unfeasible, since it was greater than 50% .

Further on, new study possibilities arise from the investigation of the influence of significant parameters. The first one originates in the low orthogonal quality of the mesh, which reached a value of 0.027, lower than the one considered acceptable, which would be above 0.15. This may have occurred due to the complexity of the Ahmed body geometry and the three-dimensional mesh. The second factor may be due to the number of elements used in the discretization of the model, considering that a grid with a larger number of elements could better characterize the flow phenomena around the body. Furthermore, a significant evaluation would be the simulation in both softwares using the same operating system. Finally, further investigations are relevant to corroborate with the correlation of these aspects and validate the results attained in this study.

5. ACKNOWLEDGEMENTS

The authors express their sincere gratitude for the financial support granted by the Federal University of Uberlândia (UFU) and by the National Council for Scientific and Technological Development (CNPq), through the CNPq/UFU Scientific Initiation scholarship. It is with great recognition that we highlight the fundamental contribution of these institutions to the successful development of this work. As required by the scientific initiation scholarship notice, the same text will be presented in Portuguese. "Os autores expressam sua sincera gratidão pelo apoio financeiro concedido pela Universidade Federal de Uberlândia (UFU) e pelo Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq), por meio da concessão da bolsa de Iniciação Científica CNPq/UFU. É com grande reconhecimento que destacamos o contributo fundamental destas instituições para o desenvolvimento bem sucedida deste trabalho."

6. REFERENCES

Ahmed, S.R., Ramm, G. and Faltin, G., 1984. "Some salient features of the time-averaged ground vehicle wake". *SAE transactions*, pp. 473–503.

Lienhart, H., Stoots, C. and Becker, S., 2002. "Flow and turbulence structures in the wake of a simplified car model (ahmed modell)". pp. 323–330.

Meile, W., Brenn, G., Reppenhagen, A., Lechner, B. and Fuchs, A., 2011. "Experiments and numerical simulations on the aerodynamics of the ahmed body". *CFD letters*, Vol. 3, No. 1, pp. 32–39.

Robertson, E., Choudhury, V., Bhushan, S. and Walters, D.K., 2015. "Validation of openfoam numerical methods and turbulence models for incompressible bluff body flows". *Computers & Fluids*, Vol. 123, pp. 122–145.

Versteeg, H. and Malalasekera, W., 2007. *An introduction to computational fluid dynamics*. Prentice Hall, Philadelphia, PA, 2nd edition.

7. RESPONSIBILITY NOTICE