

**REVISTA CIENTÍFICA ACERTTE**
**ISSN 2763-8928**

# *TURBULENCE MODELS APPLIED TO CLASSICAL FLUID MECHANICS AND HEAT TRANSFER PROBLEMS: THE PERFORMANCE EVALUATION OF THE OPEN SOURCE CFD PACKAGE OPENFOAM*

# MODELOS DE TURBULÊNCIA APLICADOS A PROBLEMAS DE MECÂNICA DOS FLUIDOS E TRANSFERÊNCIA DE CALOR: AVALIAÇÃO DO DESEMPENHO DO PACOTE DE DFC DE CÓDIGO ABERTO OPENFOAM

Paulo Alexandre Costa Rocha, Ph.D[1], Felipe Pinto Marinho, MSc[2], Victor Oliveira Santos[1], Stéphano Praxedes Mendonça, MSc[3], Maria Eugênia Vieira da Silva, Ph.D[1]

**Submetido em: 05/10/2021** e1539
**Aprovado em: 15/11/2021** https://doi.org/10.47820/acertte.v1i5.39

**ABSTRACT**
Topics related to the modeling of turbulent flow feature significant relevance in several areas, especially in engineering, since the vast majority of flows present in the design of devices and systems are characterized to be turbulent. A vastly applied tool for the analysis of such flows is the use of numerical simulations based on turbulence models. Thus, this work aims to evaluate the performance of several turbulence models when applied to classic problems of fluid mechanics and heat transfer, already extensively validated by empirical procedures. The OpenFOAM opensource software was used, being highly suitable for obtaining numerical solutions to problems of fluid mechanics involving complex geometries. The problems for the evaluation of turbulence models selected were: two-dimensional cavity, Pitz-Daily, air flow over an airfoil, air flow over the Ahmed blunt body and the problem of natural convection between parallel plates. The solution to such problems was achieved by utilizing several Reynolds Averaged Equations (RANS) turbulence models, namely: k-ε, k-ω, Lam-Bremhorst k-ε, k-ω SST, Lien-Leschziner k-ε, Spalart-Allmaras, Launder-Sharma k-ε, renormalization group (RNG) k-ε. The results obtained were compared to those found in the literature which were empirically obtained, thus allowing the assessment of the strengths and weaknesses of the turbulence modeling applied in each problem.

**KEYWORDS:** Turbulence modeling. RANS. Computational fluid dynamics (CFD). Heat transfer. Finite volume method. OpenFOAM

**RESUMO**
Tópicos relacionados à modelagem de escoamentos turbulentos apresentam relevância significativa em diversas áreas, principalmente nas engenharias, uma vez que a grande maioria dos escoamentos presentes em projetos de dispositivos e sistemas são caracterizados como turbulentos. Uma ferramenta amplamente aplicada para a análise de tais fluxos é o uso de simulações numéricas baseadas em modelos de turbulência. Assim, este trabalho tem como objetivo avaliar o desempenho de diversos modelos de turbulência quando aplicados a problemas clássicos de mecânica dos fluidos e transferência de calor, já amplamente validados por procedimentos empíricos. Foi utilizado o software open source OpenFOAM, altamente adequado para a obtenção de soluções numéricas para problemas de mecânica dos fluidos envolvendo geometrias complexas. Os problemas de avaliação dos modelos de turbulência selecionados foram: cavidade bidimensional, Pitz-Daily, fluxo de ar sobre um aerofólio, fluxo de ar sobre o corpo rombudo de Ahmed e o problema da convecção natural entre placas paralelas. A solução para tais problemas foi alcançada utilizando vários modelos de turbulência de Equações Médias de Reynolds (RANS), a saber: k-ε, k-ω, Lam-Bremhorst k-ε, k-ω SST, Lien-Leschziner k-ε, Spalart- Allmaras, Launder-Sharma k-ε, grupo de renormalização (RNG) k-ε. Os resultados obtidos foram comparados com os encontrados na literatura os quais foram obtidos

[1] Departamento de Engenharia Mecânica da Universidade Federal do Ceará
[2] Departamento de Engenharia de Teleinformática da Universidade Federal do Ceará
[3] Mestre em Engenharia Mecânica pela Universidade Federal do Ceará



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



empiricamente, permitindo avaliar os pontos fortes e fracos da modelagem de turbulência aplicada em cada problema.

**PALAVRAS-CHAVE:** Modelagem de turbulência. RANS. Dinâmica de fluidos computacional (DFC). Transferência de calor. Método dos volumes finitos. OpenFOAM

## 1. INTRODUCTION

The use of numerical simulations as an integral part of some engineering systems projects, such as in the power generation, automotive, aerospace etc. is already a reality that brings several benefits, such as reducing the cost and time for product development. The increasing use of computational procedures is due to the advancement of the processors' performance as well as the numerical simulation software and the decreasing costs of hardware equipment (ROCHA; SILVEIRA, 2012; WEHMANN, et al., 2018). Given that most of the flows, which are observed in the practical context of designing devices and systems, are characterized by being turbulent (VERSTEEG; MALALASEKERA, 2007), the study on the modeling of such problems is of utmost importance in the training of professionals that will work with this subject, which occurs mainly in technological and exact sciences. In many cases, the evaluation of turbulent flows is made by the application of turbulence models, which are solved by the use of appropriate numerical methods. In this sense, the evaluation of the performance of turbulence models in the solution of problems commonly encountered in the engineering context becomes very important.

Numerical techniques for modeling and simulation are an efficient tool to analyze turbulent flows. Direct Numerical Simulation (DNS) of Navier-Stokes equations can solve the calculation of incompressible turbulent flows, being limited to low and moderate Reynolds numbers, because it requires a large computational effort (KUROKAWA; CORRÊA; DE QUEIROZ, 2018). Unsteady Reynolds Average Navier-Stokes (URANS) equations with high Reynolds k–ε turbulence model may be a useful alternative to Direct Numerical Simulation (DNS) in computational simulations (OZMEN-CAGATAY; KOCAMAN, 2010; QUECEDO, et al., 2005), however its modeling performance needs an efficient numerical method to result in an oscillation-free solution without introducing excessive artificial dissipation (KUROKAWA; CORRÊA; DE QUEIROZ, 2018).

The turbulence model chosen to run the simulation usually has a direct influence in the results. Each turbulent model can be appropriate to specific computational problems, reaching better results to specific cases. The most know turbulence models are the k–ε and k–ω, being successfully applied for industrial applications. The k–ε model is based in two equations which are used in the evaluation of the turbulent viscosity. This viscosity is mathematically modeled as a function of k, the turbulent kinetic energy, and ε, the viscous dissipation rate of turbulent kinetic energy. This model is recommended for fully turbulent flows. The k–ω model is also a two equations model, and it is most



**REVISTA CIENTÍFICA ACERTTE**
**ISSN 2763-8928**



recommended for near-wall regions, achieving better results in this kind of problem. Beyond the boundary layer region, the ω equation behaves largely sensitive, so an upgrade model called Shear Stress Transport (SST) k–ω was developed. This model is a combination/transition between the k–ω model in near-wall regions and the k–ε model in regions far from the boundary layer. This approach is known to be successful in cases with boundary layer separation (JONES; LAUNDER, 1972; MAZARBHUIYA; BISWAS; SHARMA, 2020; MENTER; KUNTZ; LANGTRY, 2003).

Over the years, turbulence models have been applied by many researchers to solve complex engineering problems that simulate real-world applications in CFD softwares. Khavaran (1999) consolidated the use of an acoustic analogy with a compressible Reynols-Avaraged Navier-Stokes (RANS) solution and k–ε turbulence model to predict jet noise. Menter, Kuntz and Langtry (2003) described the SST turbulence model, as well as this model enhancement, modifying near wall treatment of the equations. Quecedo et al., (2005) described and compared two mathematical methods for solving a dam-break problem using the finite element method, one by solving Navier-Stokes equations and other by solving Shallow Water equations. Considering the computational efforts required, the results showed that the Shallow Water approach should be used for computations involving large domains, and Navier-Stokes approach for fine calculations in which knowledge of the 3D structure of the flow is required. Similarly, Ozmen-Cagatay and Kocaman (2010) also got results to dam-break flows, comparing experimental and numerical results using a commercially available CFD program, solving RANS equations with k–ε turbulence model involving the Shallow Water equations.

Sacomano Filho, Fukumasu and Krieger (2013) applied numerical simulations based on RANS k–ε turbulence model within the stochastic separated flow formulation to analyze the physics of ethanol turbulent spray flames. Dos Santos *et al.* (2014) compared the use of Large Eddy Simulation (LES) and RANS with k–ε turbulent model in combined convective and radiative heat transfer for non-reactive turbulent channel flows. Yahiaoui et al. (2016) developed a numerical study by the commercial CFD code, ANSYS Fluent, using a finite volume method to solve the steady-state RANS equations, and the turbulence models Spalart-Allmaras, realizable k–ε and k–ω SST were used to produce a closed system of solvable equations. Rosa et al. (2017) compared four models to predict jet mixing noise, two derived from the Lighthill Acoustic Analogy (LAA) and other two from the Linearized Euler Equations (LEE) with added source terms. RANS equations, DNS and LES were used to the computational solution of the mean flow, and empirical functions were used to model turbulent correlations.

Kurokawa, Corrêa and de Queiroz (2018) analyzed the effectiveness of a numerical technique based on the finite-difference GENSMAC methodology coupled with high Reynolds k–ε turbulence model and the ADBQUICKEST upwind scheme to deal with the advective terms for a dam-break flow and a turbulent jet impinging orthogonally onto a flat surface. Coimbra and da Silva (2020) used RANS equations to solve the turbulent closure problem. The impact of mesh refinement levels and boundary conditions on the swirl number and overall flow structure was investigated. The realizable

k−ε has been determined to be the most predictive for high swirl numbers, showing best agreement with experimental results.

Computational Fluid Dynamics (CFD) softwares are also commonly applied in aerodynamic performance analysis of horizontal and vertical axis wind turbines. A NACA 0018 blade profile was used by Abdalrahman, Melek and Lien (2017) in ANSYS Fluent CFD software to determine the optimum pitch angles of a H-type Vertical-Axis Wind Turbine (VAWT) for several tip speed ratios. Mazarbhuiya, Biswas and Sharma (2020) investigated the effect of blade pitch angle on aerodynamic performance of a NACA 63-415 in a H-type VAWT. A finite volume method was used for the turbine performance simulations, and an SST transition model was applied on the validation of the experimental results.

Time and cost as well as flexibility of the simulation setup are the main advantages of using a CFD software, although commercial CFD tools are commonly expensive. As an alternative to these softwares, the opensource package OpenFOAM has been used mainly by researches in computational fluid analysis. This software brings a new alternative to the CFD community in the development of efficient and robust models, allowing the industrial sectors to be updated with all new models without any delay compared to the commercial CFD codes (KASSEM et al., 2011). OpenFOAM is especially robust on fluid dynamics studies, being freely distributed as an opensource code under the conditions and terms given by the GNU license. OpenFOAM is a large opensource C++ library which contains all kinds of tools for performing field operation and manipulation within the continuum mechanics paradigm (GJESING; HATTEL; FRITSCHING, 2009). This software also takes advantage of its ability to simulate two phase flows using advanced sub-models (YOUSEFIFARD; GHADIMI; MIRSALIM, 2015).

Gjesing, Hattel and Fritsching (2009) obtained the mean properties of a turbulent flow with OpenFOAM. Some RANS model modifications were tested, but the k–ε model was chosen because it yielded reasonable results with acceptable calculation times. Kassem *et al.* (2011) compared the implementation of eddy dissipation models between OpenFOAM and ANSYS Fluent. The code was validated in modeling of confined non-premixed methane jet flame. The results showed agreement for both qualitative and quantitative aspects. Fluent showed overpredictions in temperature values and underprediction in flame lengths. Yousefifard, Ghadimi and Mirsalim (2015) applied OpenFOAM to study the behavior of biodiesel fuel spray. An Eulerian-Lagrangian scheme was implemented to simulate the multiphase nature of the problem, modeling in-cylinder flow by the RANS paradigm and tracking the fuel droplet by the lagrangian scheme.

In front of the presented, our work aims to apply and test the turbulence models that use the Reynolds Averaged Equations (RANS), namely: k-ε, k-ω, Lam-Bremhorst k-ε, k-ω SST, Lien-Leschziner k-ε, Spalart-Allmaras, Launder-Sharma k-ε and renormalization group (RNG) k-ε; with the subsequent comparison of the numerical results obtained against those empirically known for the classic problems under study: two-dimensional cavity, Pitz-Daily, air flow over an aerodynamic profile,



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



air flow over the Ahmed body and the natural convection problem between parallel plates. The OpenFOAM opensource package was used in obtaining the numerical solutions, being highly efficient when working on problems involving complex geometries. The proposed approach allows the evaluation of the strengths and weaknesses of the turbulence models performance for each problem. As stated before, the turbulence models used in this work are classic and well documented (LAUNDER; SPALDING, 1974; WILCOX, 1988; MENTER, 1994; LAM; BREMHORST, 1981; LIEN; LESCHZINER, 1993; SPALART; ALLMARAS, 1992; YAKHOT et al., 1992; LAUNDER; SHARMA, 1974).

## 2 METHODOLOGY

To perform the numerical simulations, OpenFOAM opensource software was used, which applies the finite volume method for computational resolution of fluid dynamics and heat transfer problems. The finite volume method (FVM) is characterized by the discretization of the system of conservation equations (mass, linear moment, energy and turbulence) in conjunction with the equations of state to obtain a system of algebraic linear equations, that are solved by means of iterative methods, obtaining a numerical result that characterizes the behavior of the flow and the heat transfer fields of a given problem. FVM is based on the integration of properties and their flows into a control volume small enough to ensure volumetric uniformity and interpolation of the properties' values between faces (ROCHA; SILVEIRA, 2012; WEHMANN et al., 2018; PATANKAR, 1980).

In this work, the approach used to assess the problems was as follows: presentation of each problem and the information needed to solve them (still in the methodology section), comparison of the results obtained by applying each turbulence model for each problem against those found in the selected literature (in the results section).

### 2.1 Two-dimensional cavity

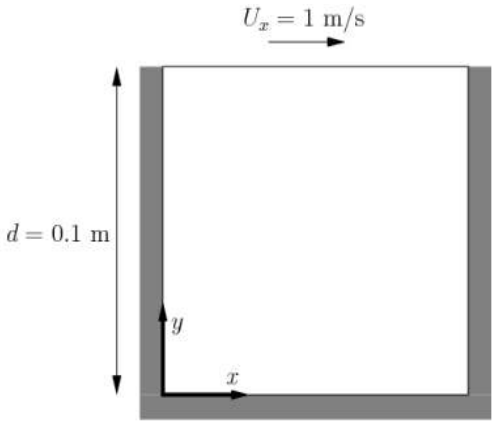
The first problem approached is that of the two-dimensional cavity consisting of an isothermal incompressible flow of a newtonian fluid, in a two-dimensional domain with the shape of a square, whose upper face presents a movement to the right, as illustrated in Figure 1.



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure 1: Schematic representation of the two-dimensional cavity (GREENSHIELDS, 2018)




The outline and initial conditions applied are as follows:

- It is assumed that the initial pressure field is uniform with an intensity of 0 m²/s²;

- The velocity field is assumed to be uniform with magnitude 0 m/s;

- The configuration of a movable upper contour (solid wall with constant velocity) and the remaining boundaries being stationary (solid walls) is applied;

- In the stationary boundaries, zero gradient conditions are used for pressure, and non-slip with null velocity;

- The upper wall received the non-slip boundary condition, with a fixed velocity equal to 1 m/s on the x-direction.

### 2.1.1 Transport properties

The only transport property required for this problem is the kinematic viscosity of the fluid, v, where it was used the hypothesis that it is equivalent to 10-5 m²/s, so it is possible to determine the value of the Reynolds number for the flow, Re, by Equation (1).

$$\mathrm{Re} = \frac{d \cdot |u|}{v} \tag{1}$$

Obtaining then a Re = 10000, which characterizes a fully turbulent regime for internal flows.

### 2.1.2 Mesh generation

Initially, a uniform mesh of 10 x 10 elements (10 mm x 10 mm) was used. Then the mesh went through a gradual refinement until the variation of the results was small enough for the numerical

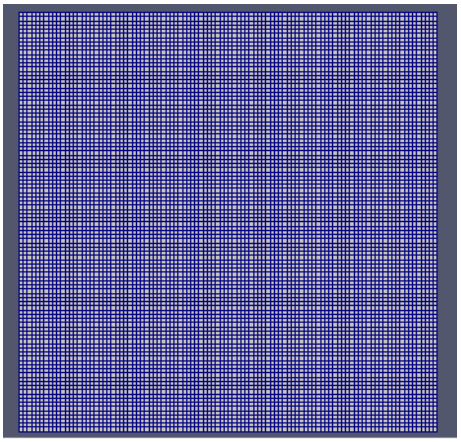


REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



solution to stabilize. The mesh generator tool provided by OpenFOAM, blockMesh, was used for the generation of the uniform mesh. The mesh used to obtain the results was 100 x 100 elements (1 mm x 1 mm), being the most refined, as well as the one that provided the most consistent results with the literature. The mesh used is illustrated in Figure 2.

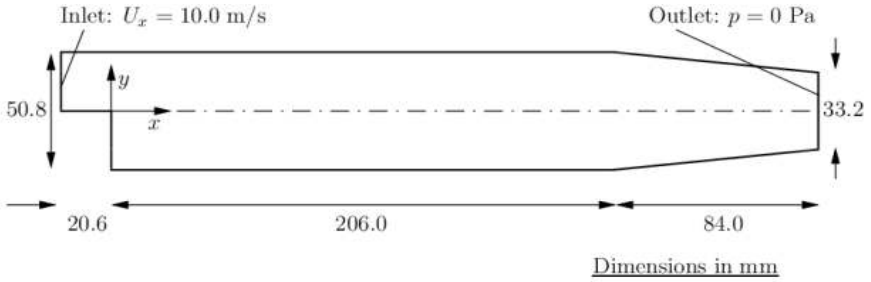
Figure 2: Schematic representation of the mesh used for the cavity problem



**2.2 Pitz-Daily**

The Pitz-Daily problem is based on the experiment carried out in 1981 by the researchers Pitz and Daily (1981). The study conducted by them aimed to analyze the development of the boundary layer and the stabilization of the combustion flame inside a test combustor, after the descending step present in the geometry shown in Figure 3. The measurements were performed by lasers and high-performance photographs for high speeds.

Figure 3: Illustration of the combustion geometry that was used in the Pitz-Daily experiment




The velocity profile along the geometry proposed by the authors was also analyzed, as well as the analysis of other properties (turbulence intensity, for example). Experiments were carried out for cases with and without combustion. For both cases, the effects were evaluated for different Reynolds numbers (1.5, 2.2 and 3.7 x $10^4$), depending on the average velocity in the inlet region, the air



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



viscosity and the size of the step present in the geometry. The simulations performed for this problem had as the main focus the evaluation of the recirculation zone formed by the flow in the posterior region of the step, as well as the location of the reattachment point of the flow. Given that, in this work the hypotheses applied to the numerical solution of the problem were incompressible, turbulent, two-dimensional, viscous and stationary (steady-state) flow.

In a more detailed way, the following boundary and initial conditions were adjusted as:

- The non-slip condition on the walls was considered;
- A uniform velocity field was defined at the inlet. As carried out in the Pitz-Daily experiment, the cases for which the entry velocity were 9.1 m/s were considered for the case where Re = 1.5 x $10^4$, 13.3 m/s and 22.2 m/s for Reynolds values of 2.2 x $10^4$ and 3.7 x $10^4$ respectively;
- A null pressure value (manometric) was considered at the outlet;
- For the initial value of the turbulent kinetic energy, k, a value of 5% of the velocity was considered. Thus, for the case in which U = 13.3 m/s, for example, k = 0.663 J by Equation (2) and by the hypothesis that the components of the random fluctuations of velocity $U'_x$, $U'_y$, $U'_z$, are equivalent:

$$U'_x = U'_y = U'_z = 0,05 \times 13,3 = 0,665 \text{ m/s}$$

$$k = \frac{U'^2_x + U'^2_y + U'^2_z}{2} = 0,663 \text{ J} \quad (1)$$

### 2.2.1 Adjusting the values of the turbulence models properties

Table 1 illustrates the initial values of the turbulent flow properties used in the models applied to the problem. The method of calculating these is well detailed by Versteeg and Malalasekera (2007).

Table 1: Initial values for turbulent properties

| Speed (m/s) | k (J) | ω ($s^{-1}$) | ε ($m^2/s^3$) |
|---|---|---|---|
| 9.1 | 0.311 | 407.253 | 11.399 |
| 13.3 | 0.663 | 594.637 | 35.482 |
| 22.2 | 1.665 | 940.641 | 140.955 |

### 2.2.2 Mesh generation

The check of the mesh quality was performed using different degrees of refinement and evaluating the change in the results for the velocity fields along the sections. The results considered in this comparison were obtained after a sufficient period for the flow simulation to be steady state. Figure 4 shows the curves of the horizontal component of the velocity, Ux, for an intermediate section (x = 0.1347 m) inside the three meshes considered. The results were obtained using the k–ε model for the case of Re = 2.2 x $10^4$. It can be perceived that the mesh with intermediate degree of refining is

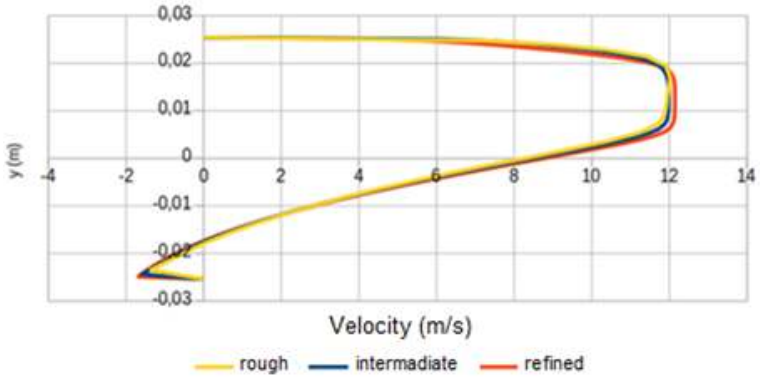


REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



able to produce results with a considerable degree of precision, making the use of a more refined mesh not necessary for the purposes of this work.

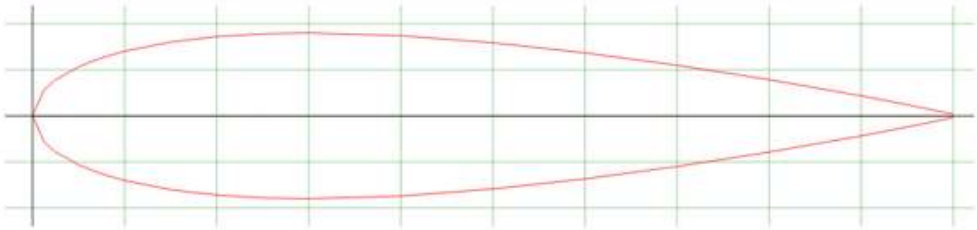
Figure 4: Velocity field over a plane located at x = 0.1347 m for different levels of mesh refinement




**2.3 Air flow over an aerodynamic profile**

For this problem, the profile studied was the NACA 0018, which belongs to the 4-digit NACA series. The value 00 means that the profile is not chambered (symmetric profile), while the final digits represent the percentage of the length corresponding to the maximum height of the profile related to the chord, in this case 18%. For the simulations, the model used has dimensions of 35 m x 6.3 m, as represented in Figure 5.

**Fig. 5** NACA 0018 profile



It is desirable that aerodynamic profiles or airfoils generate maximum lift while producing a minimum of drag. These forces depend on the density of the fluid, ρ, the free flow velocity, V, and the size, shape and orientation of the body. In this sense, a very relevant criterion, generally decisive to the choice of the profile for a given application, are the airfoil drag and lift coefficients, $C_d$ and $C_l$, which indicate in a nondimensional way, respectively, the components of the resulting force parallel



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



and perpendicular to the flow stream. A more detailed study on aerodynamic profiles is presented by Abbott and Von Doenhoff (1959) and Anderson Jr (2017).

With that in mind, the problem consisted in determining the $C_l$ and $C_d$ curves for the chosen profile, analyzing the influence of the Reynolds number and the angle of attack. The hypotheses applied to the flow are that it was incompressible, turbulent, two-dimensional, viscous and stationary.

The applied boundary and initial conditions in the simulation are:

- Uniform pressure field with zero gradient on the walls;
- For the flow velocity, a variable field was defined according to the direction of the considered angle of attack and the speed module defined for each Reynolds number. In addition, the non-slip condition was defined on the walls;
- The Reynolds number values considered are $2 \times 10^5$, $5 \times 10^5$ and $1 \times 10^6$;
- The fluid properties were defined as air at atmospheric pressure and at 0 ºC temperature. Furthermore, a density of 1.2928 kg/m³ and kinematic viscosity of $1.3304 \times 10^{-5}$ m²/s were used.

Table 2 illustrates the values of the free flow velocity for different Reynolds numbers, calculated by Equation 3.

$$\mathrm{Re} = \frac{\rho \cdot V \cdot l}{\mu} \tag{2}$$

Being μ the dynamic viscosity of the fluid and *l* the characteristic length of the airfoil, in this case the value of the chord length.

Table 2: Free stream flow speed for each Reynolds number studied in the airfoil problem

| Parameter | Re = $2 \times 10^5$ | Re = $5 \times 10^5$ | Re = $1 \times 10^6$ |
|---|---|---|---|
| Speed (m/s) | 0.058 | 0.1470 | 0.2940 |

**2.4 Air flow over the Ahmed body**

The Ahmed body is a classic and extremely important problem for the automobile industry. Because of its geometry, proposed by Ahmed, Ramm and Faltin (1984), it generates the essential characteristics of the flow field of a real vehicle, with the exception of more specific effects, as seen on the lower part of the vehicle and surface protrusions such as rear-view mirrors. Figure 6 illustrates the dimensions of the geometry used in the simulation. The dimensions are in mm and the angle adopted

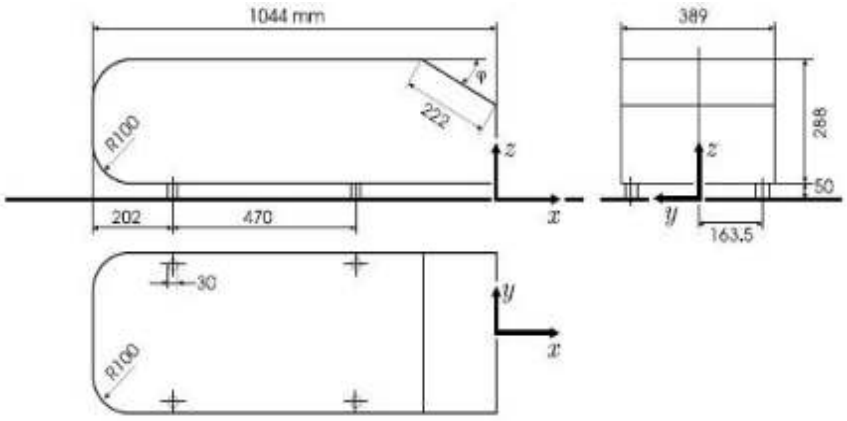
was φ = 30 °, which, according to Ahmed, Ramm and Faltin (1984), is the angle for which the drag is maximum.

Therefore, the problem of air flow over the Ahmed body consisted in determining its drag coefficients. The hypotheses used are of incompressible, turbulent, three-dimensional, viscous and stationary flow.

For the simulations, the boundary and initial conditions are:

- The non-slip condition over the walls is considered;
- A uniform velocity field was defined at the inlet with an intensity of 30 m/s;
- A null pressure value was considered at the outlet and null pressure gradient at the inlet and over the contact regions with solid surfaces, such as the ground floor and the body;
- The reference area for calculating the nondimensional coefficients was assumed to be equivalent to the cross-sectional area and the reference length as the length of the body, in the case: $A_{ref}$ = 0.112032 m², $l_{ref}$ = 1.044 m.

Figure 6: Dimensions of the Ahmed body used in the simulations




### 2.4.1 Mesh generation

To choose the level of mesh refinement, six different meshes were analyzed, whose results obtained are represented in Table 3.

Table 3: Variation of the Cd against the mesh refinement level

| Mesh | Number of elements | Cd | Percentage Variation (%) |
|---|---|---|---|
| 10x3x3 | 90 | 0.488503 | - |
| 15x4x4 | 240 | 0.419104 | 14.2 |
| 20x5x5 | 500 | 0.3880229 | 7.42 |
| 40x10x10 | 4000 | 0.3585465 | 7.59 |
| 60x15x15 | 13500 | 0.3357221 | 6.36 |
| 80x20x20 | 32000 | 0.322286 | 4.0 |



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



According to Table 3, it is perceived that the influence on the results due to the mesh refinement becomes small even for a significant increase in the number of elements, starting from the mesh with 13500 elements. Because of that, this mesh was chosen for the resolution and analysis of the problem.

### 2.5 Natural convection between parallel plates

This problem aims to study natural convection in a closed space. The geometry of interest consists of a cavity with a face of higher temperature, and another of lower temperature, separated by a small gap filled by air that works as insulating material, Figure 7. Temperatures are uniform along the area of the faces and steady in time. The hot wall is at a temperature of 307.75 K (34.6 ºC) and the cold wall is at 288.15 K (15 ºC). The dimensions of the cavity are W = 76 mm, D = 520 mm and H = 2180 mm. The air properties necessary for the solution of such problem are represented in Table 4. In addition, the hypotheses applied to the flow are that it is three-dimensional, steady-state, turbulent, compressible and viscous regime.

Table 4: Air properties required for convection problem analysis

| Property | Value |
|---|---|
| Molar weight | 28.96 g/mol |
| Pressure coefficient ($C_p$) | 1004.4 J/kg.K |
| Kinematic viscosity (v) | $1.831 \times 10^{-5}$ m²/s |
| Prandtl number (Pr) | 0.705 |

The boundary and initial conditions used in the simulation are:

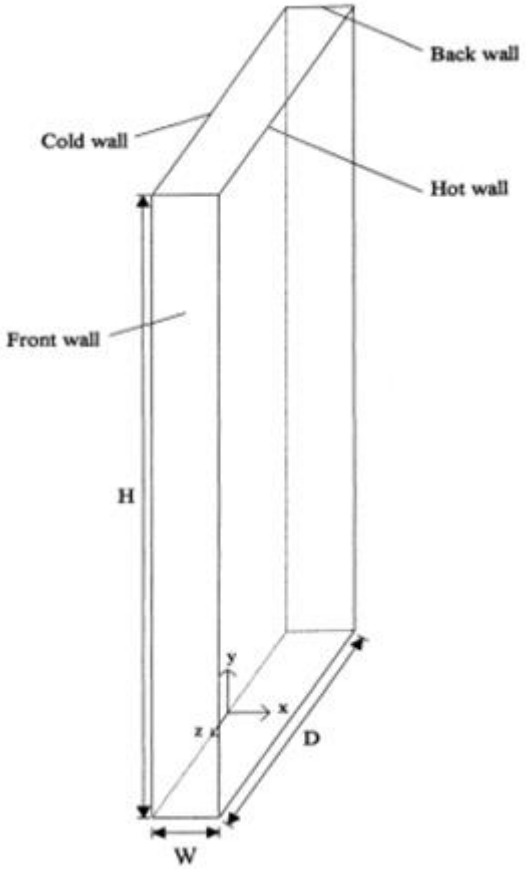
- The velocity field was initially considered null and the non-slip condition was applied over the 6 faces.
- The pressure was defined as being uniform of value $1 \times 10^5$ Pa.



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure 7: Representation of the geometry of interest to the air convection problem




### 2.5.1 Mesh generation

The k – ω SST method was used to select the mesh and the effect of the refinement on the temperature values at an intermediate point (y = 0.5 m) was evaluated, and this effect can be seen in Figure 8. The different refinement levels applied are illustrated in Table 5.

Table 5: Characteristics of the tested mesh refinement levels

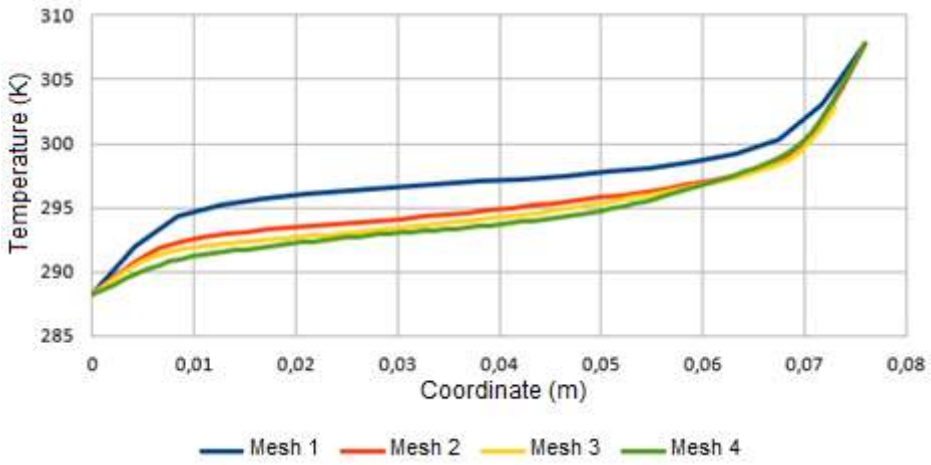
| Meshes | Mesh discretization | Number of elements |
|---|---|---|
| Mesh 1 | 18x75x8 | 10800 |
| Mesh 2 | 35x150x15 | 36750 |
| Mesh 3 | 50x200x20 | 200000 |
| Mesh 4 | 70x300x30 | 630000 |



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure 8: Effect of the mesh refinement on the temperature profile




Based on Figure 8, it is clear that from mesh 2 on, the effect on the temperature is no longer significant. Thus, this level of refinement was chosen for the other simulations of this problem.

## 3 Results

### 3.1 Two-dimensional cavity

The results for the problem of the two-dimensional cavity are represented in Figures 9 and 10, where the vertical component of the velocity is evaluated along the horizontal center axis of the cavity, while the horizontal velocity component is evaluated along the vertical center axis.

Figure 9: $U_y$ along the horizontal center axis

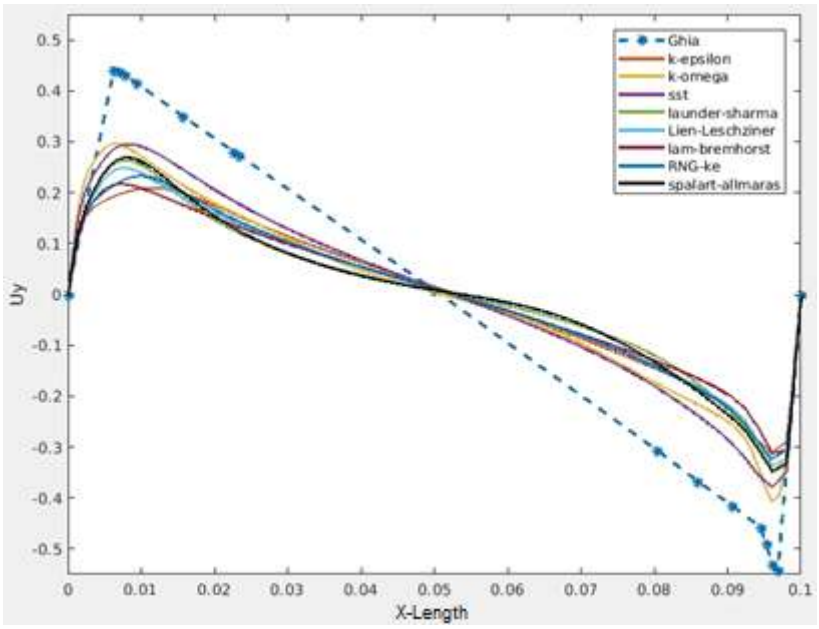
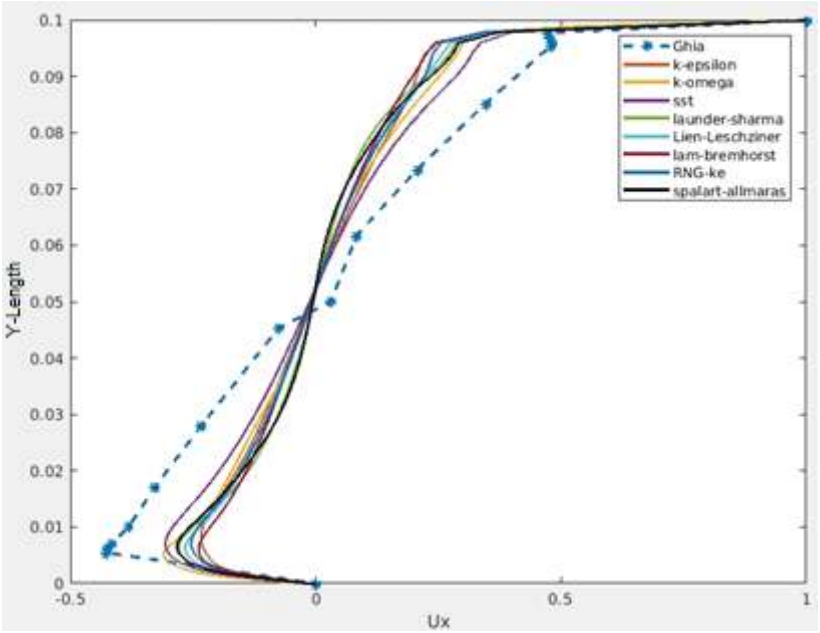


Figure 10: Ux along the vertical center axis




The numerical results obtained show a behavior very close to those found experimentally in the work of Ghia, Ghia and Shin (1982), as can be observed by Figures 9 and 10. However, this similarity occurred only when the most refined mesh was used, 100 x 100. Thus, it can be affirmed that all turbulence models presented a good performance in the numerical prediction of the flow field for this problem. This fact is expected since the problem of the two-dimensional cavity is considered a standard case to test the accuracy and efficiency of numerical algorithms. In this case, since all models used are considered classic and have already been extensively validated, it is expected that they presented a good behavior for this problem. For this computational domain, the greatest deviations occur at the corners of the cavity where there is a tendency to the formation of recirculating zones, mainly affecting the performance of the models k – ε, Lam-Bremhorst k – ε, Launder-Sharma k – ε, Lien-Leschziner k – ε. The best fit is achieved by k – ω and k – ω SST models. In the first, the great advantage is that its integration to the solid boundaries does not require wall functions, since it is a low Reynolds number model. Nevertheless, there is a problem related to the specification of the boundary conditions for the turbulent frequency, ω, in the free stream, because the same tends to zero causing the dynamic viscosity turbulent, $\mu_t$, tend to infinity. Because of this, small non-null values of ω must be specified in these regions. The k–ω SST model can adjust its behavior according to the region to be analyzed, since near the walls it presents a similar behavior to the k-ω model, while in the free stream it resembles the k – ε model, presenting the main advantages of each model. A better description of these characteristics is presented by Versteeg and Malalasekera (2007), Wilcox (1988) and Menter (1994).



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



### 3.2 Pitz-Daily

As already presented, one of the main goals of this problem is the determination of the reattachment point of the stream after the step. Thus, Table 6 shows the percentage errors, differences between the numerical results obtained and those found by Pitz and Daily where they are divided by the empirical results. These errors were found considering all the velocities addressed in the problem, 9.1, 13.3 and 22.2 m/s. It is noteworthy that when the Spalart-Allmaras model was applied, no numerical convergence could be achieved.

Table 6: Percentage errors for the location of the reattachment point

| Models | Percentage errors (Minimum-maximum) |
|---|---|
| k – ε | 2.074% - 3.771% |
| k – ω | 1.729% - 9.458% |
| k – ω SST | 4.206% - 5.574% |
| Lien-Leschziner k – ε | 1.433% - 7.942% |
| Lam-Bremhorst k – ε | 0.612% - 3.233% |
| Launder-Sharma k – ε | 5.544% - 8.800% |
| Renormalization Group k – ε | 4.892% - 9.806% |

The analysis of Table 6 shows that, in general, all turbulence models show good performance, highlighting the k–ε and Lam-Bremhorst k–ε models. The former is widely used in industrial applications, because it presents good performance in areas of free stream with small pressure gradients and for being extensively well validated. However, its performance becomes poor in regions close to walls where there is a decrease in the Reynolds number and in situations in which there are high gradients of adverse pressure, as well as in some unconfined flows. The Lam-Bremhorst k–ε (LB), Launder-Sharma k–ε (LS) and Lien-Leschziner k–ε (LL) models are modified versions of the k–ε model for cases where the flow presents regions of low Reynolds numbers. The difference between the first two is in the boundary conditions of the rate of dissipation of turbulent energy, ε. This difference is well explained by Garg (1998). The latter was developed to replicate the behavior of the one equation Wolfshtein, Norris and Reynolds model, as well as providing the correct asymptotic response in the vicinity of the wall, as illustrated by Leschziner (2015). The Spalart-Allmaras model was designed for applications of external aerodynamic flows, in addition to presenting a good performance in boundary layers with adverse pressure gradients, which are important for the prediction of the stall phenomenon (VERSTEEG; MALALASEKERA, 2007). In general, the model is inappropriate for many internal flows, which may justify its numerical non-convergence.

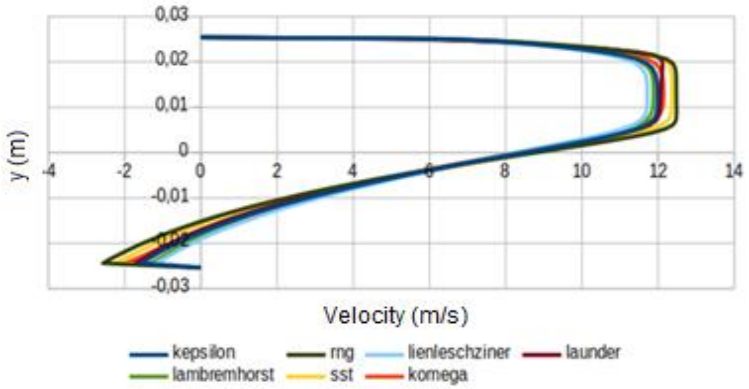


REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure 11 shows the velocity fields plotted for a plane located at x = 0.1347 m for the case where Re = 2.2 x $10^4$, thus illustrating, by the similarity of the velocity profiles in the x direction, that the model applied does not directly impact the behavior of the field in this direction.

Figure 11: Velocity field plotted for a plane located at x = 0.1347 m for the case Re = 2.2 x $10^4$




### 3.3 Air flow over an aerodynamic profile

The values used as reference were provided by the database of the Research Group on Applied Aerodynamics of the University of Ilinois (UIUC AIRFOIL DATA SITE, 2019), and are presented in Figures 12 and 13. The green curve corresponds to Re = 2 x $10^5$, purple to Re = 5 x $10^5$ and orange to Re = 1 x $10^6$. As it can be seen, $C_l$ lies between 1 and 1.3 and $C_d$ ranges from 0.012 to 0.022. It is worth to note that the model which presented the best performance was Spalart-Allmaras, and the results obtained by it are shown in Figure 14. This good concordance of the prediction provided by the Spalart-Allmaras model was already expected, since, as already discussed in this work, this model, which is characterized by being of a transport equation over one auxiliary variable that is related to the turbulent viscosity, is designed for applications involving external aerodynamic flows. For a better detailing on the mathematical description, one can recommend the references Versteeg and Malalasekera (2007) and Spalart and Allmaras (1992).



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure. 12: Experimental results for $C_l$ versus the angle of attack (NACA 0018, 2019)

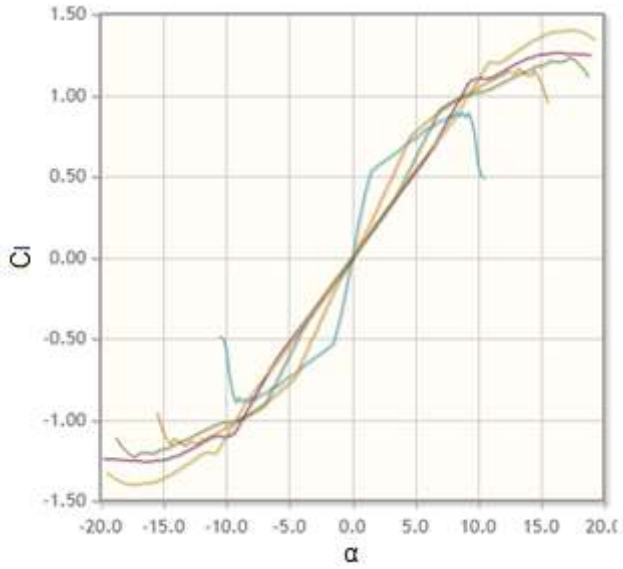
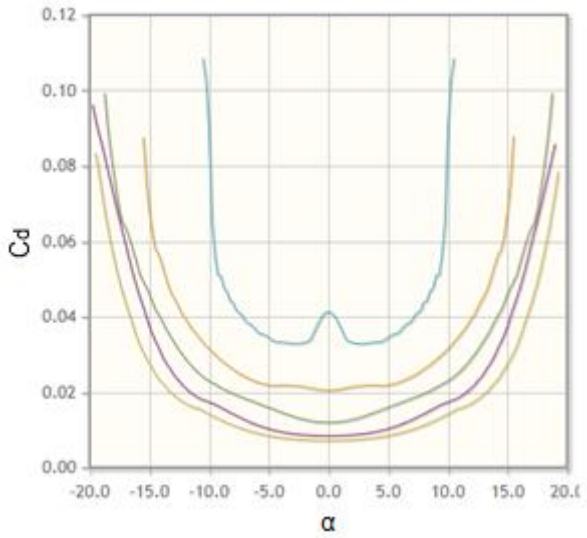



Figure. 13: Experimental results for $C_d$ versus the angle of attack (NACA 0018, 2019)

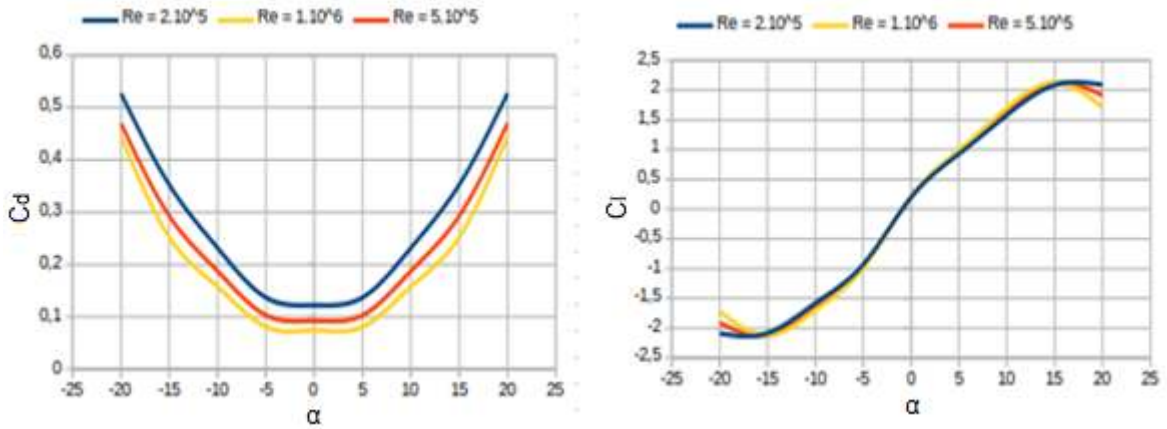




REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure 14: Results provided by the Spalart-Allmaras model




### 3.4 Air flow around the Ahmed body

The values of the drag coefficient for all the models applied and their empirical values are shown in Table 7.

Table 7: Values of the drag coefficient obtained by the tested models and the experiment

| Method | Drag coefficient |
|---|---|
| Experimental | 0.375 |
| k – ε | 0.626 |
| k – ω | 0.340 |
| k – ω SST | 0.322 |
| Lien-Leschziner k – ε | 0.616 |
| Lam-Bremhorst k – ε | 0.461 |
| Launder-Sharma k – ε | 0.616 |
| Group of renormalization k – ε | 0.424 |
| Spalart-Allmaras | 0.370 |

It is clearly perceived the improved performance of the Spalart-Allmaras model, proving that in fact this model is highly suitable for aerodynamic applications, where external flow occurs. The greatest discrepancy was observed in the result provided by the k – ε model, which, as already discussed, may present poor performance for some unconfined flows. Amongst the k – ε models, the renormalization group (RNG) k – ε was the one that provided the best result. This was developed by



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928

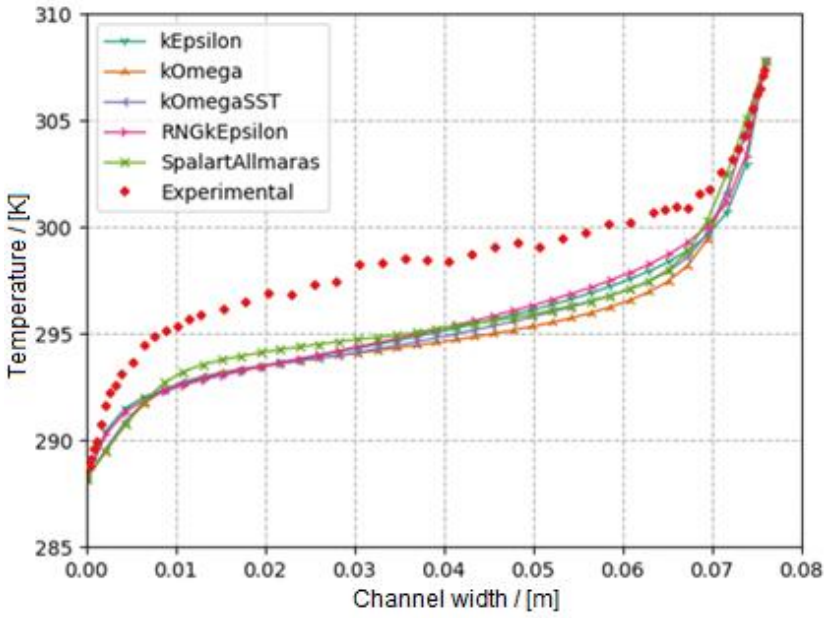


the application of the renormalization group theory, mathematical formalism from the statistical mechanics to the standard k – ε model. The main advantage is related to the presence of a deformation-dependent correction term, since one of the main limitations of the k – ε model is in the transport equation for the rate of dissipation of turbulent energy for flows that experience high deformation rates. For more details, it is recommended the reference Yakhot et al., (1992). The k – ω model also showed good performance, mainly due to its good characteristics in the evaluation of the boundary layer near the boundaries of the profile.

### 3.5 Natural convection between parallel plates

The results for the temperature distribution and the velocity field in a position that is half the height of the gap between plates are represented in Figures 15 and 16, where the percentage of 50% serves only to indicate that the position corresponds to half of the distance between the plates. The experimental results were obtained by Betts and Bokhari (2000).

Figure 15: Temperature distribution on a plane on half of the distance between the plates

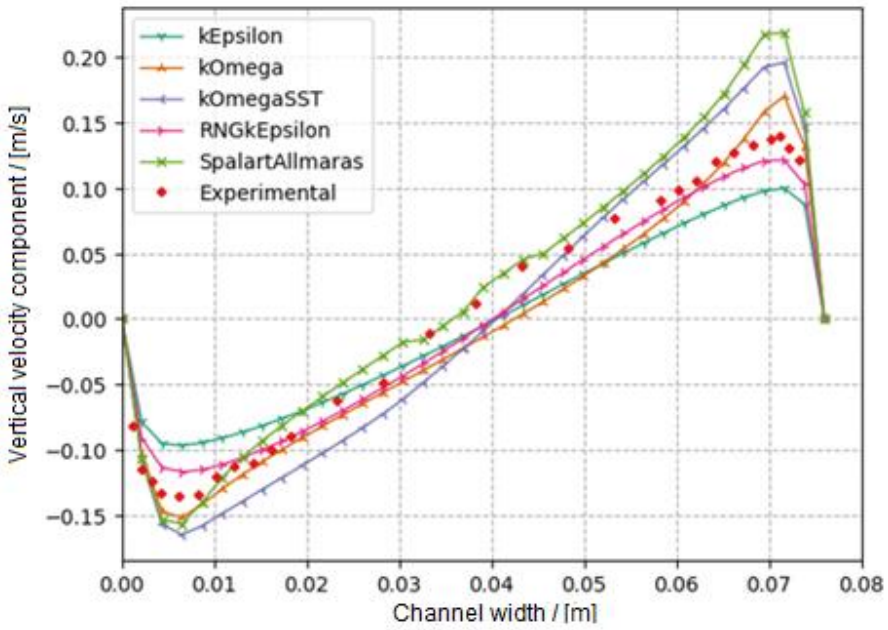




REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



Figure 16: Velocity field on a plane on half of the distance between the plates




It is perceived that in the temperature prediction all models provided estimates that had significant deviations in relation to the experimental values. In determining the velocity distribution, it can be considered that no model accompanied the experimental behavior observed for the vertical component of velocity. The discordant results obtained for the temperature distributions and vertical component of velocity, compared to experimental results, can be justified by the fact that, in natural convection, the circulation of the fluid due to density variations, caused by the temperature gradient, and the action of buoyancy forces usually take into consideration the density as a function of temperature for the Boussinesq approximation. In the conservation of momentum equations, it becomes strongly coupled with the energy equation, which makes obtaining adverse numerical results consistent. For a more detailed analytical description on natural convection one can recommend the reference Incropera et al., (2008).

## FINAL CONSIDERATIONS

The present work has applied several turbulence models using Reynolds Averaged Equations (RANS), namely: k–ε, k–ω, Lam-Bremhorst k–ε, k–ω SST, Lien-Leschziner k–ε, Spalart Allmaras, Launder-Sharma k–ε, renormalization group (RNG) k–ε, for the numerical simulation of five classic problems of fluid mechanics and heat transfer, which have already been extensively validated by empirical procedures. Numerical results were compared with those found in the literature, illustrating what the potential and shortcomings encountered for each turbulence model used. With this method, it



REVISTA CIENTÍFICA ACERTTE
ISSN 2763-8928



was possible to observe that the realization of computational experiments, carried out in the open source CFD package OpenFOAM, were completely adequate to the assessment of the performance of turbulence models employed for each problem considered, which may be of great assistance to the modeling of problems commonly encountered in engineering.

## REFERENCES

ABBOTT, I. H.; VON DOENHOFF, A. E. **Theory of wing sections:** including a summary of airfoil data. New York: Dover Publications, 1959, 705 p.

ABDALRAHMAN, G.; MELEK, W.; LIEN, F.S. Pitch angle control for a small-scale Darrieus vertical axis wind turbine with straight blades (H-Type VAWT). **Renewable energy**, v. 114, p. 1353-1362, 2017.

AHMED, S. R.; RAMM, G.; FALTIN, G. Some salient features of the time-averaged ground vehicle wake. **SAE Transactions**, p. 473-503, 1984.

ANDERSON JR., J. D. **Fundamentals of aerodynamics**. 6. ed. New York: Tata McGraw-Hill Education, 2017, 1154 p.

BETTS, P. L.; BOKHARI, I. H. Experiments on turbulent natural convection in an enclosed tall cavity. **International Journal of Heat and Fluid Flow**, v. 21, n. 6, p. 675-683, 2000.

COIMBRA, A. P. N.; DA SILVA, L. F. F. Modelling of a turbulent lean premixed combustor using a Reynolds-averaged Navier–Stokes approach. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 42, n. 5, p. 1-16, 2020

DOS SANTOS, E. D. *et al.* A numerical study of combined convective and radiative heat transfer in non-reactive turbulent channel flows with several optical thicknesses: a comparison between LES and RANS. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 36, n. 1, p. 207-219, 2014.

GARG, V. K. **Applied computational fluid dynamics**. New York: CRC Press, 1998, 438 p.

GHIA, U. K. N. G.; GHIA, K. N.; SHIN, C. T. High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. **Journal of computational physics**, v. 48, n. 3, p. 387-411, 1982.

GJESING, R.; HATTEL, J.; FRITSCHING, U. Coupled atomization and spray modelling in the spray forming process using open foam. **Engineering Applications of Computational Fluid Mechanics**, v. 3, n. 4, p. 471-486, 2009.

GREENSHIELDS, C. OpenFOAM v6 User Guide: 2.1 Lid-driven cavity flow. **CFD Direct: The Architects of OpenFOAM**. Avaliable at: https://cfd.direct/openfoam/user-guide/v6-cavity/. Accessed 9 february 2019.

INCROPERA, F. P. *et al.* **Fundamentos de transferência de calor e de massa**. 6. ed. Rio de Janeiro: Ltc, 2008. 664 p.



**REVISTA CIENTÍFICA ACERTTE**
**ISSN 2763-8928**



JONES, W. P.; LAUNDER, B. E. The prediction of laminarization with a two-equation model of turbulence. **International journal of heat and mass transfer**, v. 15, n. 2, p. 301-314, 1972.

KASSEM, H. I. *et al.* Implementation of the eddy dissipation model of turbulent non-premixed combustion in OpenFOAM. **International Communications in Heat and Mass Transfer**, v. 38, n. 3, p. 363-367, 2011.

KHAVARAN, A. Role of anisotropy in turbulent mixing noise. **AIAA journal**, v. 37, n. 7, p. 832-841, 1999.

KUROKAWA, F. A.; CORRÊA, L.; DE QUEIROZ, R. A. B. Numerical simulation of 3D unsteady turbulent free surface flows using κ−ε model and ADBQUICKEST scheme. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 40, n. 4, p. 1-16, 2018.

LAM, C. K. G.; BREMHORST, K. A modified form of the k-ε model for predicting wall turbulence. **Journal of Fluids Engineering**, v. 103, n. 3, p. 456-460, 1981.

LAUNDER, B. E.; SHARMA, B. I. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. **Letters in heat and mass transfer**, v. 1, n. 2, p. 131-137, 1974.

LAUNDER, B. E.; SPALDING, D. B. The numerical computation of turbulent flows. **Computer Methods In Applied Mechanics And Engineering**, v. 3, n. 2, p. 269-289, 1974.

LESCHZINER, M. **Statistical turbulence modelling for fluid dynamics-demystified:** an introductory text for graduate engineering students. London: ICP, 2015, 424 p.

LIEN, F. S.; LESCHZINER, M. A. A pressure-velocity solution strategy for compressible flow and its application to shock/boundary-layer interaction using second-moment turbulence closure. **Journal of Fluids Engineering**. v. 115, n. 4, p. 717-725, 1993.

MAZARBHUIYA, H. M. S. M.; BISWAS, A.; SHARMA, K. K. Low wind speed aerodynamics of asymmetric blade H-Darrieus wind turbine-its desired blade pitch for performance improvement in the built environment. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 42, p. 1-16, 2020.

MENTER, F. R. Two-equation eddy-viscosity turbulence models for engineering applications. **AIAA journal**, v. 32, n. 8, p. 1598-1605, 1994.

MENTER, F. R.; KUNTZ, M.; LANGTRY, R. Ten years of industrial experience with the SST turbulence model. **Turbulence, heat and mass transfer**, v. 4, n. 1, p. 625-632, 2003.

NACA0018 (naca0018-il). **NACA0018 - NACA0018 airfoil**. Avaliable at: http://www.airfoiltools.com/airfoil/details?airfoil=naca0018-il. Accessed 9 february 2019.

OZMEN-CAGATAY, H.; KOCAMAN, S. Dam-break flows during initial stage using SWE and RANS approaches. **Journal of Hydraulic Research**, v. 48, n. 5, p. 603-611, 2010.

PATANKAR, S. V. **Numerical heat transfer and fluid flow**. Washington, DC: Hemisphere Publishing Corp., 1980, 210 p.

PITZ, R. W.; DAILY, J. W. Experimental study of combustion in a turbulent free shear layer formed at a rearward facing step. *In.:* **19th Aerospace Sciences Meeting**. 1981. p. 106.



**REVISTA CIENTÍFICA ACERTTE**
**ISSN 2763-8928**



QUECEDO, M. *et al.* Comparison of two mathematical models for solving the dam break problem using the FEM method. **Computer Methods in Applied Mechanics and Engineering**, v. 194, n. 36-38, p. 3984-4005, 2005.

ROCHA, P. A. C.; DA SILVEIRA, J. V. P. Estudo e aplicação de simulação computacional em problemas simples de mecânica dos fluidos e transferência de calor. **Revista Brasileira de Ensino de Física**, v. 34, n. 4, p. 1-8, dez. 2012.

ROSA, V. *et al.* Comparison of RANS-based jet noise models and assessment of a ray tracing method. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 39, n. 6, p. 1859-1872, 2017.

SACOMANO FILHO, F. L.; FUKUMASU, N. K.; KRIEGER, G. C. Numerical simulation of an ethanol turbulent spray flame with RANS and diffusion combustion model. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 35, n. 3, p. 189-198, 2013.

SPALART, P.; ALLMARAS, S. A one-equation turbulence model for aerodynamic flows. *In.:* **30th aerospace sciences meeting and exhibit**. 1992. p. 439.

UIUC AIRFOIL DATA SITE. **UIUC Applied Aerodynamics Group: Departament of Aerospace Engineering**. Avaliable at: https://m-selig.ae.illinois.edu/ads.html. Accessed 13 february 2019.

VERSTEEG, H. K.; MALALASEKERA, W. **An introduction to computational fluid dynamics:** the finite volume method. Harlow: Prentice Hall, 2007, 517 p.

WEHMANN, C. F. *et al.* Estudo e aplicação de simulação computacional em problemas simples de mecânica dos fluidos e transferência de calor – Parte II: Problemas clássicos de transmissão de calor. **Revista Brasileira de Ensino de Física**, v. 40, n. 2, 2018.

WILCOX, D. C. Reassessment of the scale-determining equation for advanced turbulence models. **AIAA journal**, v. 26, n. 11, p. 1299-1310, 1988.

YAHIAOUI, T. *et al.* Experimental and CFD investigations of turbulent cross-flow in staggered tube bundle equipped with grooved cylinders. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 38, n. 1, p. 163-175, 2016.

YAKHOT, V. S. A. S. T. B. C. G. *et al.* Development of turbulence models for shear flows by a double expansion technique. **Physics of Fluids A: Fluid Dynamics**, v. 4, n. 7, p. 1510-1520, 1992.

YOUSEFIFARD, M.; GHADIMI, P.; MIRSALIM, M. Numerical simulation of biodiesel spray under ultra-high injection pressure using OpenFOAM. **Journal of the Brazilian Society of Mechanical Sciences and Engineering**, v. 37, n. 2, p. 737-746, 2015.