Simulation of Turbulent Flow in an Asymmetric Air Diffuser

Abubaker E.M Elbalsohi, Junling Hu, Ruoxu Jia
Department of Mechanical Engineering
University of Bridgeport, Bridgeport, CT, USA
a.elbalos@my.bridgeport.edu, jihu@bridgeport.edu, ruoxujia@bridgeport.edu

Abstract—This research paper compared three $k-\varepsilon$ family turbulence models, Standard $k-\varepsilon$, RNG $k-\varepsilon$ and Realizable $k-\varepsilon$, and two $k-\omega$ family models Standard $k-\omega$ and Stress-Strain Transport, SST $k-\omega$. The turbulent flow characteristics were predicted in a two-dimensional of $10^6$ half-angle diffuser using the five turbulence models with the ANSYS FLUENT 13.0 code. Numerical results were validated by comparing them to experimental fluid dynamics (EFD) results. Velocity profiles, turbulent kinetic energy profiles and skin friction coefficients were presented to validate the numerical results. Contours velocity-streams functions were shown as well. One of the most interesting observations of comparing numerical solutions to EFD data was apparently that $k-\varepsilon$ family models have a valid prediction of flow characteristics that are far away from wall effects, however, $k-\omega$ models have a significant prediction of flow behavior nearby the wall boundaries. In addition, the changes in the quality of meshing elements and its number have noticeable influences on computational fluid dynamics (CFD) results. Personally, the present CFD investigation obviously has given a deep insight of the most important fluid dynamics concepts that were studied in the computational fluid dynamics course.

Keywords—Turbulence models; Computational Fluid Dynamics, CFD; Experimental Fluid Dynamics, EFD; Air Diffuser.

I. INTRODUCTION

In the present time, fast computers performance allows engineers to simulate such complex computational geometries. Solving numerically a set of coupled partial differential equations, PDE’s, for continuity, momentum, and energy equations by using one of modern Computational Fluid Dynamics, CFD codes has become common and more powerful tool in the most modern industrial fields in satisfied accuracy. CFD codes have provided a great deal discretizing such complicated PDE’s into a set of algebraic equations and then getting numerical solutions by using one of numerical methods. Many predicted functions and design of different interesting engineering problems, either in industry fields or even in medical fields, are numerically predicted and developed. Any developed technique, equipment, or an industrial tool will nowadays be marketing until its performance or function has been simulated by using one of CFD codes.

Complicated coupled partial differential equations, like continuity, Navier-Stokes, Energy equation or any constitutive equations are extremely unsolved analytically because of its difficult getting a solution. Nowadays, with high power of computer processors and a wide range storage capacity of temporary or permanent computer memories, engineers have had more capability to solve such those PDE’s numerically and then predicting their performance under certain operational conditions.

Generally, CFD has become the core of comprehension of the basic concepts of fluid flow processes, like Heat-Transfer, Mass-Transport, Fluid-Flow…etc. and of analyzing the numerical solutions results.

Surely, CFD’s users will not be able to use CFD codes until they understand the fluid dynamics aspects and comprehend the necessary processes of implementations of CFD. A successful CFD engineer is not only who can run successfully CFD code, but the one who can interpret and analyze the physical flow phenomena and makes the right decision based on those numerical observations.

In the fluid dynamics, the flow is classified into three categories; laminar flow, transient flow, and turbulent flow. The turbulent flow is the most common flow in most practical engineering systems. Every single flow pattern is dominating by unique flow characteristics. One of the most interesting flow properties is Reynolds number (Re). The Reynolds number is the key to distinguish between those flows patterns. Reynolds number is defined as the ratio of the inertia force to viscous force of a fluid flow.

In turbulent flow, many of fluid dynamics phenomena occur. Disturbances in the fluid motion and the fluctuation of the fluid velocity make the flow rapidly transient into turbulent flow. Because of the chaotic and unstable state of the turbulent motion of flow particles, eddies and vortexes will be created. Large eddies and small eddies with different turbulent scales, length scale and time scale, will be transported through in the flow direction under vortex stretching process, thus the turbulence flow will continue.

The investigation of a starting point of turbulence regime depends on the ratio of inertia force to friction force. At high
Reynolds number, the inertia effects are enough large to magnify the disturbances and rapidly transferring a flow into turbulent flow by affecting on velocities components and the other flow characteristics to vary in unstable and random way.

Flow characteristics in laminar simple cases are totally calculated by the continuity and momentum equations and can be solved analytically. However, there are no analytical solutions for turbulent flows for most turbulent problems.

Turbulent flow can be treated numerically with CFD turbulence-modeling approaches. There are many turbulence models develop for various kinds of flow. It is very important to understand these turbulence models in order to appropriately use them to model flow phenomena. Icacciarino [1] used a benchmark problem to compare two turbulence models in three commercial CFD codes. This paper compared five turbulence models using a bench mark problem – flow in an asymmetric diffuser.

II. MATHEMATICAL MODELS

In general, the turbulence equations which are more popularly known as the Reynolds-averaged Navier-Stokes equations are similar to the laminar flow equations except of some additional turbulent terms. The most additional interesting quantities are Reynolds stresses terms, eddy viscosity, \( \mu_r \), turbulent kinetic energy, \( k \), turbulent diffusivity, \( \Gamma_t \), turbulent Prandtl number, \( P_{RT} \), and the rate of dissipation of turbulent energy, \( \varepsilon \).

Because of the additional turbulent terms, there are more additional calculated unknowns and then solving such complex sets of partial differential equations will require more effort and won’t be easy if not impossible.

\[
\frac{\partial \sigma}{\partial t} + \frac{\partial \nabla u}{\partial x} + \frac{\partial \nabla v}{\partial y} = \frac{1}{\rho} \frac{\partial}{\partial x} \left( \nabla u \frac{\partial u}{\partial x} + \nabla v \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial y} \left( \nabla v \frac{\partial u}{\partial x} + \nabla v \frac{\partial v}{\partial y} \right) - \frac{\partial}{\partial x} \left( \nabla u \frac{\partial u}{\partial x} + \nabla v \frac{\partial v}{\partial y} \right)
\]

(2)

\[
\frac{\partial \sigma}{\partial t} + \frac{\partial (\sigma \mu)}{\partial x} + \frac{\partial (\sigma \mu)}{\partial y} = \frac{1}{\rho} \frac{\partial}{\partial x} \left( \nabla u \frac{\partial u}{\partial x} + \nabla v \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial y} \left( \nabla v \frac{\partial u}{\partial x} + \nabla v \frac{\partial v}{\partial y} \right) - \frac{\partial}{\partial x} \left( \nabla u \frac{\partial u}{\partial x} + \nabla v \frac{\partial v}{\partial y} \right)
\]

(3)

\[
\frac{\partial \sigma}{\partial t} + \frac{\partial (\sigma u)}{\partial x} + \frac{\partial (\sigma u)}{\partial y} = \frac{k}{\rho C_p} \frac{\partial u}{\partial x} + \frac{k}{\rho C_p} \frac{\partial u}{\partial y} - \frac{\partial}{\partial x} \left( \nabla u \frac{\partial u}{\partial x} + \nabla v \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial y} \left( \nabla v \frac{\partial u}{\partial x} + \nabla v \frac{\partial v}{\partial y} \right)
\]

(4)

Generally, \( \nabla \) is a mean quantity of flow characteristics and \( \nabla u' v' \) is a component of the Reynolds stresses components. Many turbulent modeling approaches have tried to describe the turbulence phenomena by modified turbulence considerations and reduce the above transport equations. Launder and Spalding (1974) successfully invited a new turbulent modeling approach by simplified and adding two-equation to the Reynolds-averaged Navier-Stokes equations to their transport calculations by accommodating and identifying two turbulent quantities, the turbulent kinetic energy, \( k \), and the rate dissipation of turbulent energy, \( \varepsilon \), as;

\[
\frac{\partial k}{\partial t} + u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} = \frac{\partial}{\partial x} \left( \nu \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y} \left( \nu \frac{\partial k}{\partial y} \right) + P - D
\]

(5)

\[
\frac{\partial \varepsilon}{\partial t} + u \frac{\partial \varepsilon}{\partial x} + v \frac{\partial \varepsilon}{\partial y} = \frac{\partial}{\partial x} \left( \nu \frac{\partial \varepsilon}{\partial x} \right) + \frac{\partial}{\partial y} \left( \nu \frac{\partial \varepsilon}{\partial y} \right) + \frac{\varepsilon}{k} (C_{e1} P - C_{e2} D)
\]

(6)

where \( D \) is the destruction term and \( P \) is the production term.

\[
P = 2\nu T \left( \frac{\partial u}{\partial x} \right)^2 + 2\nu T \left( \frac{\partial v}{\partial y} \right)^2
\]

(7)

\[
k = \frac{1}{2} u_i u_i
\]

(8)

\[
i, j = 1, 2, 3
\]

(9)

And,

\[
\varepsilon = \nu T \left( \frac{\partial u_i}{\partial x} \right) \left( \frac{\partial u_j}{\partial y} \right)
\]

(10)

\( \alpha \) is the thermal diffusivity

\[
\alpha = \frac{k}{\rho C_p}
\]

(11)

\( \nu \) is the kinetic turbulent/eddy viscosity

\[
\nu_T = \frac{\mu_T}{\rho}
\]

(12)

\[
\mu_T = c_{ ud} k^2
\]

(13)

From \( k-e \) equation, we found out that \( k-e \) is a consequence of the eddy viscosity concept. The \( k-e \) model is well established and widely validated, in addition, it gives sensible numerical solutions to most engineering flows.

In flows with large and rapid extra strains, especially for complex geometries with highly curved velocity/thermal boundary layers and sharp diverging streamlines, \( k-e \) turbulence modeling will fail to completely predict the sensible effects of the streamlines curvature on turbulence.

Because \( k-e \) equation model considers the turbulent stresses (Reynolds stresses components) have a linear relationship with the rate of the strain (velocity gradients) by a scalar turbulent viscosity, \( k-e \) model has some deficiencies and invalid predictions of the normal stresses calculations in addition, a back flow zone that appears in curvature areas won’t be predicted. Briefly, \( k-e \) model could be only used to predict unconfined flows.

In general, the major disadvantage of \( k-e \) model is its lack to describe shear layers far-wake and handle well within a mixing-layer of separated flows. Another important weaknesses of using \( k-e \) model is its oblivion to predict body forces which are resulting from rotation of a plane frame of reference. To overcome some of \( k-e \) model deficiencies, some turbulent constants should be accordingly adjusted.

Standard \( k-e \) model is one member of \( k-e \) family models. This kind of \( k-e \) model is used as a starting point of any type of turbulent flows to give a general prediction for turbulence analysis. Clearly, standard \( k-e \) model has weakness points in its numerical solutions and these numerical errors might be minimized by adjust turbulent constant that are existing in \( k-e \) equation.
Consequently, there are two adjusted members of k-ε family models, Renormalization Group, RNG k-ε model and Realizable k-ε model. These two models were proposed by Yakhov et al. (1992) and Shih at al. (1995) respectively as alternative turbulence models of standard k-ε model to improve numerical prediction where standard k-ε model failed.

At the diffuser wall boundary layers, the turbulent fluctuations are restrained nearby the wall and thus the viscous influences become dominated in this area which is well known as the "viscous sub-layer". Therefore, the k-ε family models could not be valid in use in such a case. Other turbulence models are required to satisfy this kind of flow condition at the near-wall region.

Based on the Low Reynolds-number concept, LRN, Wilcox (1998) invented a new turbulence model, k-ω model, where $\omega$ is a frequency of the large turbulent eddies. The main goal of k-ω model was to be valid in use especially close to walls in boundary-layer flows. Similarly, of solving $\omega$-equation in k-ε equation model, $\omega$-equation is solving in the same way to calculate its local distribution within the fluid flow as well. Combining k-ω two- equation model solution to a modeled transport equation to solve quantity and momentum equations creates k-ω turbulence model.

III. SIMULATION PROCESSES

For this research paper, the CFD commercial code ANSYS 13 -FLUENT, was used to predict the turbulent flow under steady-state condition by providing numerical solutions of solving governing transport equations of continuity and momentum equations in two dimensions.

A. Problem statement

The computational domain of a 2-D asymmetric diffuser is shown in Fig. 1. It consists of three sections – a small channel with a length of H1 and height of V1, a 10° half-angle expansion section, and a big channel with a length of H2 and height of V2. The dimensions of the diffuser are taken \(^{[2]}\) as H1= 60 m, V1=2 m, H2=70 m, and V2= 9.4 m. The expansion section has a length of 42 m. Air at 20ºC enters the small channel with a free stream velocity of 1.25 m/s and exits the big channel at a fully developed condition. Non-slip boundary conditions are taken for the top and bottom walls. The material properties of air and the inlet and outlet boundary conditions are listed in Tables 1-3, respectively. The Reynolds number based on the free-stream velocity, $U$ of 1.25 m/s and inlet channel height, $V1$, is approximately $1.7 \times 10^4$.

<table>
<thead>
<tr>
<th>Property</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>(\rho)</td>
<td>kg/m(^3)</td>
<td>1</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>(\mu)</td>
<td>kg/m/s</td>
<td>0.000147</td>
</tr>
</tbody>
</table>

TABLE II, INLET BOUNDARY CONDITIONS

<table>
<thead>
<tr>
<th>Variable</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>x-velocity</td>
<td>U</td>
<td>m/s</td>
<td>1.25</td>
</tr>
<tr>
<td>y-velocity</td>
<td>V</td>
<td>m/s</td>
<td>0</td>
</tr>
<tr>
<td>Inlet Pressure</td>
<td>P</td>
<td>pa</td>
<td>--</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>K</td>
<td>m(^2)/s</td>
<td>0.0018</td>
</tr>
<tr>
<td>Turbulent Dissipation Rate</td>
<td>E</td>
<td>m(^2)/s</td>
<td>9.63\times10(^{-4})</td>
</tr>
</tbody>
</table>

TABLE III, OUTLET BOUNDARY CONDITIONS

<table>
<thead>
<tr>
<th>Variable</th>
<th>Symbol</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>y-velocity</td>
<td>V</td>
<td>m/s</td>
<td>--</td>
</tr>
<tr>
<td>Outlet Gauge Pressure</td>
<td>P</td>
<td>pa</td>
<td>0</td>
</tr>
<tr>
<td>Backflow Turbulent Intensity</td>
<td>--</td>
<td>%</td>
<td>3.25</td>
</tr>
<tr>
<td>Backflow Turbulent length</td>
<td>--</td>
<td>m</td>
<td>0.0035</td>
</tr>
</tbody>
</table>

B. Meshing Processes

A non-uniform structured mesh was generated in a two-dimensional-10\(^8\) half-angle diffuser domain. A non-uniform structured mesh of 59 × 59 cells are generated for each of the three sections of the computational domain. As shown in Fig. 2, finer meshes are concentrated near the top and bottom walls boundaries and mesh size in y direction gradually increases as it moves away from the walls. In the similar way, finer meshes in the x direction are set in the expansion region and where the expansion section connects with the channels. The stretched structured provide a computationally efficient solution to resolve the turbulence viscous layers near the top and bottom walls and the large gradient regions in and near the expansion section.

The stretched mesh is generated through the Bias Factor Option ANSYS. Fig. 3 shows an example of setting up 59 non-uniform meshes in x direction for the expansion section. Selecting the top and bottom walls in the expansion section as the two edges, 59 cells are specified in these two edges to have non-uniform mesh sizes. The bias type generates finer mesh sizes near the two ends of these two edges and coarser mesh as it moves inward. The ratio of the largest cell size to the smallest cell size on these two edges was set by the bias factor, which is 1.8593 in this example.

![Fig. 1. A schematic of the 2-D computational domain and boundary conditions of a 10° half-angle Diffuser (not to scale)](image1)

![Fig. 2. A non-uniform rectangular mesh 100 half-angle air diffuser.](image2)
C. Turbulence Models

The turbulence flow was simulated with five turbulence models in ANSYS 13 Fluent package. They include three turbulence models in the k-ε family – Standard k-ε, RNG k-ε and Realizable k-ε; and two turbulence models in the k-ω family – Standard k-ω and Stress-Strain Transport (SST) k-ω. Enhanced wall treatments are chosen for the turbulence boundary layers. This type of wall treatment method combines a two layer model with enhanced wall functions. This near wall treatment model can work with fine mesh to resolve the near wall boundary layer and also with course mesh using wall functions. The simulations conducted with the five turbulence models are based on the same computational domain and mesh, as shown in the Workbench project management window in Fig. 4.

The simulations conducted with computational domain and mesh, as shown in the Workbench project management window in Fig. 4.

![Fig. 3. Bias Factor options window for top & bottom wall boundaries.](image)

![Fig. 4. Final 2-D 10^5 Half-angle Diffuser Workbench Window](image)

IV. RESULTS

When air enters the expansion section of the diffuser, the flow is separated due to the adverse pressure gradient and a large recirculation bubble is generated. The CFD simulation results are examined in the expansion section to predict the size of the recirculation bubble and its effect on velocity profiles and skin friction coefficient.

The CFD results are compared with the experimental fluid dynamics (EFD) data that were measured at seven specified locations. Boundary layer profiles and other results generated with the five turbulence models were examined in detail and compared with EFD data at the seven locations along the bottom wall. These lines were defined in FLUENT by Line/Rake option. The coordinates of those lines are showing in the Table 4. The first five locations are in the expansion section and the last two are in the big channel.

| Position-2 | 82  | -4.23 | 82  | 2  |
| Position-3 | 86  | -4.9371 | 86  | 2  |
| Position-4 | 98  | -7.053 | 98  | 2  |
| Position-5 | 102 | -7.4  | 102 | 2  |
| Position-6 | 110 | -7.4  | 110 | 2  |
| Position-7 | 118.5 | -7.4 | 118.5 | 2 |

**TABLE 4. EXPERIMENTAL DATA POSITIONS**

<table>
<thead>
<tr>
<th>Surface Name</th>
<th>X0 (m)</th>
<th>Y0 (m)</th>
<th>X1 (m)</th>
<th>Y1 (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Position-1</td>
<td>78</td>
<td>-3.52</td>
<td>78</td>
<td>2</td>
</tr>
</tbody>
</table>

The streamwise velocity and turbulence kinetic energy are modified to be clearly shown in one figure at the seven specific locations. Table 5 list the definitions of the two modified functions and skin friction coefficient, where \( r_w \) is shear stress at wall.

The modified computed velocity profiles at specified different positions within the expansion zone geometry were predicted through each k-ε family models and k-ω family models which are illustrated in the following comparing figures. In addition, to validate the numerical predictions, the numerical results are compared versus EFD data.

In the same above manner, the modified computed turbulent kinetic energy, TKE, and skin-friction coefficients against the bottom wall of 10^5 half-angle diffuser are also presented in comparison figures for each turbulence-modeling approaches and comparing with experimental data.

**A. Comparison of k-ε models**

The modified streamwise velocity profiles obtained with the k-ε family models are compared with the experimental data in Figs. 5-7. It can be seen in Fig. 5 that the three k-ε family models give a very close results, with standard k-ε model gives a slightly different results. Comparing with the experimental results, the realizable k-ε model gave the best result among the three k-ε models. However, all the three k-ε models predicted a very small recirculation bubble. Remarkable deviations from the experimental data are observed in the recirculation region. The velocity streamlines in Fig. 8 show that a very small recirculation region predicted by standard k-ε model.

![Fig. 5. Modified velocity vs Standard k-ε vs Realizable k-ε vs RNG k-ε](image)
Figures 9 and 10 report the modified profiles of the turbulent kinetic energy, $k$, predicted by using $k$-$\varepsilon$ family models. The results of the three $k$-$\varepsilon$ models are still close to each other and the Realizable $k$-$\varepsilon$ model gives a result closest to the EFD data. The TKE profiles agree with the EFD data at the beginning of the expansion section, i.e. at $x = 78\text{,} 82\text{,}$ and $86$ m. As the flow move downstream, we noticed that the modified TKE profiles have been deviated from the experimental data, predicting a higher TKE near the bottom wall and a much lower TKE in the flow compared to EFD. These reduce flow recirculation and prohibit back flow at the bottom wall.

Figure 11 reports the skin friction at the bottom wall. The negative value of skin friction coefficient corresponds to the back flow region at the bottom wall. The skin friction coefficients predicted by the $k$-$\varepsilon$ models agree well with the EFD data after the beginning expansion and before the recirculation region and the discrepancy increases downstream of the recirculation region. It shows that $k$-$\varepsilon$ models predicted a much smaller recirculation region than the EFD data.

B. Comparison of $k$-$\omega$ models

The modified velocity profiles, KTE profiles, and skin friction coefficient at the bottom wall obtained by $k$-$\omega$ models
are compared with the EFD data, in Figs. 12-14, respectively. It can be seen that SST k-ω model gives slightly better results than standard k-ω model does. The k-ω models predicted a very large recirculation bubble as shown by the streamline in Fig. 15. They predicted much higher KTE in the flow streams and higher skin friction coefficient magnitude than the EFD data.

C. Comparison of k-ε models and k-ω models

Figures 16-18 compared the results of standard k-ε model and standard k-ω model with EFD data. It can be clearly seen that k-ε models failed to capture the back flow bubble and under-predicted KTE, k-ω models could capture the recirculation region, but over-predicted the recirculation region and KTE. k-ε models predict closer to EFD flow profiles than k-ω models, especially in the region without significant adverse pressure gradient. k-ε models was better at predicting skin friction coefficient at the bottom wall than k-ω models. Table 6 compared the pressure drop in the diffuser and friction force on the top wall obtained with k-ε and k-ω models. It is found that k-ε predicts a 16.6% higher pressure drop and 9.78% higher friction force at the top wall.
Finally, under the same flow condition, the next figures 17, 18, and 19 illustrate that the modified skin-friction coefficients against the bottom wall can be predicted by $k-\varepsilon$ family models better than $k-\omega$ family models.

Contour Velocity Stream Function:
Another investigation of comparing between two turbulence models is the investigation of recirculation flow zone of back flow. Clearly, from the following contour velocity-stream functions figures, the standard $k-\omega$ has the advantage of prediction such a flow behavior, i.e. the back flow at the 10° half-angle diffuser geometry, more than standard $k-\varepsilon$.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Total pressure difference between inlet and outlet (Pa)</th>
<th>Total friction force on the top wall (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SST</td>
<td>0.3</td>
<td>1.7</td>
</tr>
<tr>
<td>$k-\varepsilon$</td>
<td>0.35</td>
<td>1.8</td>
</tr>
<tr>
<td>Relative difference</td>
<td>-16.6%</td>
<td>-9.78%</td>
</tr>
</tbody>
</table>

V. CONCLUSION
This paper simulated the turbulent flow of air in a 10° diffuser with five turbulence models. The flow characteristics, such as like velocity profiles, turbulent kinetic energy, and the skin-friction coefficients were compared and validated against EFD data.

It was found that the results generated within each turbulence model family are close to each other. The $k-\varepsilon$ family models, Standard $k-\varepsilon$, RNG $k-\varepsilon$, and realizable $k-\varepsilon$, give very close results, with realizable $k-\varepsilon$ gives the best result for the diffuser simulation. The standard $k-\omega$ model and SST $k-\omega$ model give very close results.

$k-\varepsilon$ models predicted reasonable the flow characteristics, such as velocity profiles, turbulent kinetic energy and the skin-friction coefficients, but they failed to capture the flow separation at the wall and under-predicted turbulent kinetic energy and thus recirculation region.

$k-\omega$ models predicted well the flow characteristics, such as velocity profiles, and turbulent kinetic energy near wall boundary-layers, but they over-predicted turbulent kinetic energy and thus recirculation region.

$k-\varepsilon$ models and $k-\omega$ models have been widely used in industries to predict flow characteristics. It can be seen that each type of model has its own characteristics. $k-\varepsilon$ models are good for fully turbulent flow away from boundary layers, but not good at capturing complex flows involving severe pressure gradient and separation. $k-\omega$ models have a better near wall treatment, and can predict the complex boundary layer flows such as flow in a diffuser, but they typically have an excessive and early prediction of flow separation.

The differences of any numerical solutions and analytical or experimental results are referring to a fact that the numerical solutions have numerical errors and those errors come from many different sources, including from turbulence models. Now turbulence-modeling approach is expected to be commonly used for many industrial flows. Therefore, the investigation, like what this research paper has done, can be the good guidance to figure out which CFD turbulent model can be used for an engineering problem.

As it is saying before, the turbulence phenomena could be described similarly to “The invention of the Devil on the Seventh day of creation, when the God lord wasn’t looking” [1]

VI. REFERENCES

