

Assessment of Turbulence RANS Models for Conical Diffuser with Tailpipe

Mohammad Faizan

Abstract — In this paper, four common turbulence models were selected to assess the predictions of the velocity profiles and static pressure coefficient in an experiment-studied conical diffuser. The four models chosen were the standard $k-\epsilon$ model, the standard $k-\omega$ model, the shear-stress transport $k-\omega$ SST, and the Reynolds stress model. The steady RANS equations for turbulent incompressible fluid flow and turbulence closure were solved using the commercial code of ANSYS Fluent 14.0. It was found that the standard $k-\omega$ model and the shear-stress transport $k-\omega$ SST model failed to predict accurate velocity profiles and the static pressure recovery in the tailpipe. The model results were compared with the published experimental data. The standard $k-\epsilon$ model presented the same capability of as Reynolds stress model to capture flow pattern in the diffuser and tailpipe. Numerical results also revealed that the standard $k-\omega$ model succeeded to predict an accurate static pressure recovery in the diffuser but failed to predict accurate velocity profiles.

Index Terms — Conical diffuser, diffuser performance, pressure recovery, RANS, turbulence models.

I. INTRODUCTION

A substantial amount of research effort is devoted to the development of numerical methods of Computational Fluid Dynamics. The methods can be grouped into the three categories; Turbulence models for Reynolds-averaged Navier-Stokes (RANS), Large Eddy Simulation (LES) and Direct numerical simulation (DNS). The RANS method has been proven to be suitable for industrial flow computations due to low cost in terms of computing resources [15].

The predictive performance of linear and nonlinear eddy-viscosity models and differential stress-transport closures for separated flow in a nominally two-dimensional, asymmetric diffuser was investigated [1]. It was demonstrated that standard linear eddy-viscosity models failed to represent adequately the flow characterized by a strong adverse pressure gradient that provokes separation while the SST $k-\omega$ model showed some of the predictive advantages.

The predictions of turbulent separated flow in an asymmetric two-dimensional diffuser using commercial CFD are investigated [9]. The codes of CFX, Fluent, and Star-CD were employed and showed very similar characteristics in terms of convergence and accuracy while appreciable differences were obtained when the $k-\epsilon$ model was employed. Reference [6] compared the three dimensional calculations of RANS, LES and LES-fine grid models for asymmetric diffusers and the results showed that RANS simulation

strongly over-predicted the strength of separation while the LES computations showed a much better agreement with the experimental data, especially for the fine grid.

A comparative study of turbulence models performance for separating flow in a planar asymmetric diffuser using six turbulence models of commercial software code, FLUENT 6.3.26 [7]. The results showed that the standard $k-\omega$, SST $k-\omega$ and v^2-f models have better agreement with the experiments than other models and the RSM model failed to predict the location of separation and attachment points.

Flow characteristics and performance of conical diffusers of high half-cone angle with different inlet velocity distortions by inserting center bodies at inlet was studied by Faizan et al. [8]. Three types of center bodies were employed viz. simple (streamlined), triangular grooved and square grooved at diffuser inlet. The velocity profiles, flow separation and static pressure recovery using CFD commercial code, Fluent 14 was calculated and compared with their own experimental results. It was concluded that SST $k-\omega$ turbulent model showed a good agreement with the experimental results.

Most Computational Fluid Dynamics analysis in high adverse pressure gradient are employed to simulate flow separation and approximately predict the point at which flow reattaches to the wall and predicting the boundary layer development downstream of such point [3]. Such flow separation is affected by diffuser geometry and inlet flow conditions. Ariff et al. [2] demonstrated that the accuracy of the results in all the numerical analyses, particularly those using RANS formulation, are dependent on the turbulence models used, near-wall treatments applied and discretization schemes employed. The five most common turbulence models to predict the attached separation flow in a planar diffuser were studied by Chen et al. [5] with the commercial software Fluent. It was demonstrated that the $k-\epsilon$ model failed to predict the attached separation while $k-\omega$ model presented the same capability with SST $k-\omega$ model in capturing the separation bubble.

In the computational fluid dynamics study of flow in diffusers, the computational domain extends a sufficient length downstream of the diffuser in order to apply proper flow condition of either exit static pressure or fully developed flow condition. However, the experimental results showed that when a diffuser flow discharges through a tailpipe, the overall pressure rise in the diffuser and the tailpipe combination is higher than the pressure rise in a diffuser with free discharge [13] and [14]. Therefore, the validation of turbulence models for the diffusers with extended tailpipe to minimize the effect of flow prediction is needed. The experimental results of the conical diffuser performance with a tailpipe published by Nakamura et al. [14] are considered to assess the validity of various turbulence models in the present study.

Manuscript published on 30 October 2014.

* Correspondence Author (s)

Mohammad Faizan, Department of Mechanical Engineering, College of Engineering, Taif University, Taif, KSA, Saudi Arabia.

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an [open access](#) article under the CC-BY-NC-ND license <http://creativecommons.org/licenses/by-nc-nd/4.0/>

The aim of the present study is to examine the validity of four RANS turbulence models on the flow structure in conical diffuser equipped with tailpipe and uniform velocity profile at inlet. The examined turbulence models are the standard $k-\varepsilon$ model (SKE), the standard $k-\omega$ model (SKW), the shear-stress transport $k-\omega$ (SST) and the Reynolds stress model (RSM) via ANSYS FLUENT flow simulation program.

II. COMPUTATIONAL MODEL AND BOUNDARY CONDITIONS

The governing steady flow field equations are the continuity and Reynolds averaged Navier-Stokes equations, as follows:

$$\frac{\partial u_i}{\partial x_j} = 0 \quad (1)$$

$$u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(u \frac{\partial u_i}{\partial u_j} - \bar{u}_i \bar{u}_j \right) \quad (2)$$

In the present study, four different turbulence models SKE, SKW, SST and RSM and their empirical constants were used as they are implemented in the Fluent software.

A. The standard $k-\varepsilon$ model (SKE)

This is a semi-empirical model and most commonly used of all the turbulence models. The $k-\varepsilon$ model is by far the most widely used and tested two-equation model, with many improvements incorporated over the years [17]. It is classified as a two equation model and the transport equation is solved for two turbulent quantities, turbulence kinetic energy, k and its dissipation rate, ε . The model is a high Reynolds model [10] and requires a wall function and fine grid in solving the near-wall problem.

B. The standard $k-\omega$ model (SKW)

Another ‘successful’ model and also widely used is the $k-\omega$ model. The initial form of the model was proposed by Kolmogorov in 1942 [17]. This model includes two equations of the turbulence kinetic energy, k and the specific dissipation rate, ω which can also be thought of as the ratio of ε to k . The standard $k-\omega$ model in Fluent is based on Wilcox $k-\omega$ [16]. The model incorporates a damping coefficient for the turbulent viscosity causing a Low-Reynolds number correction.

C. The shear-stress transport $k-\omega$ (SST)

This model was developed by Menter [12] which combines two models, the $k-\omega$ and the $k-\varepsilon$. The formulation of the $k-\omega$ model is used in the near-wall region all the way down to the wall through the viscous sub-layer, and be used as a low-Reynolds turbulence model without any extra damping function. The use of a $k-\varepsilon$ formulation in the free-stream can avoid the $k-\varepsilon$ problem that is too sensitive to the inlet turbulence properties.

D. The Reynolds stress model (RSM)

The RSM model closes the Reynolds-averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for the dissipation rate [11]. The exact Reynolds stress transport equation accounts for the directional effects of the Reynolds stress fields.

The schematic of the computational domain is shown in Fig. 1. The basic geometric parameters are the same as implemented by Nakamura et al. [14]. The entrance pipe had a length of 280 mm and 79.8 mm diameter. The conical diffuser had an axial length of 457 mm and a cone angle of 10 degrees with the length of tail pipe discharge as 2092 mm. The dimensionless lengths of inlet section, diffuser length and the tail pipe were -7.0, 11.5 and 52.5 respectively as shown in Fig. 1.

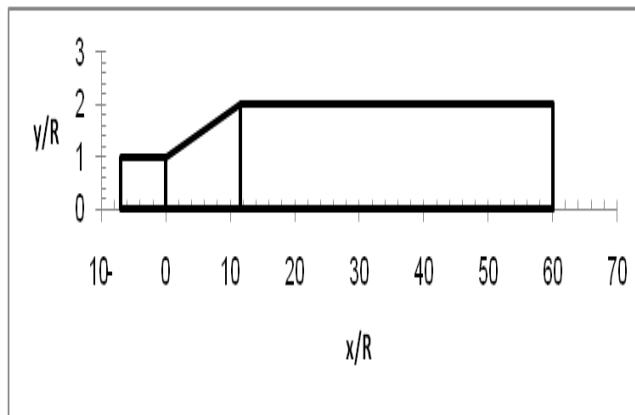


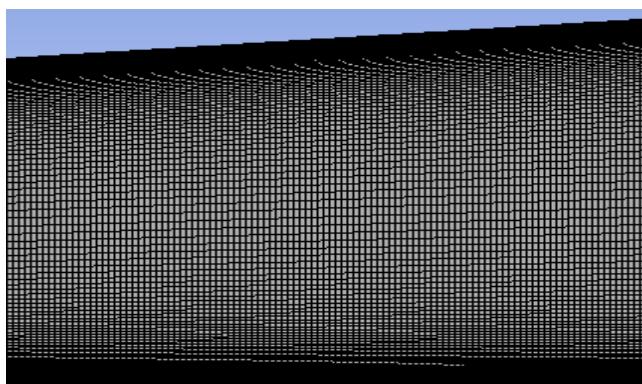
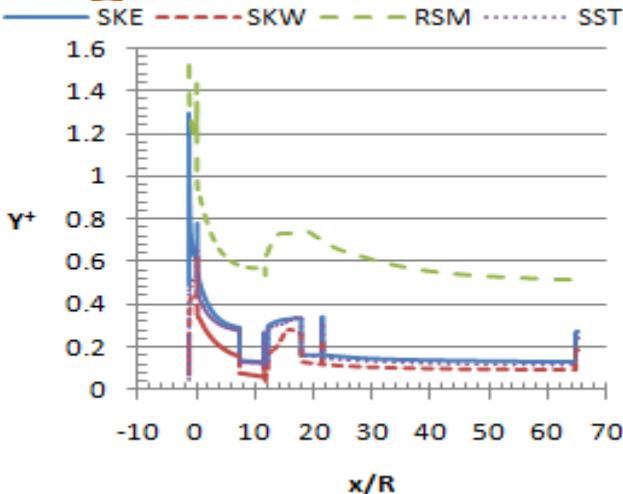
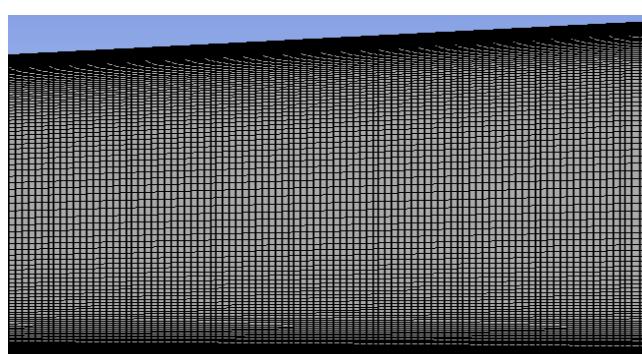
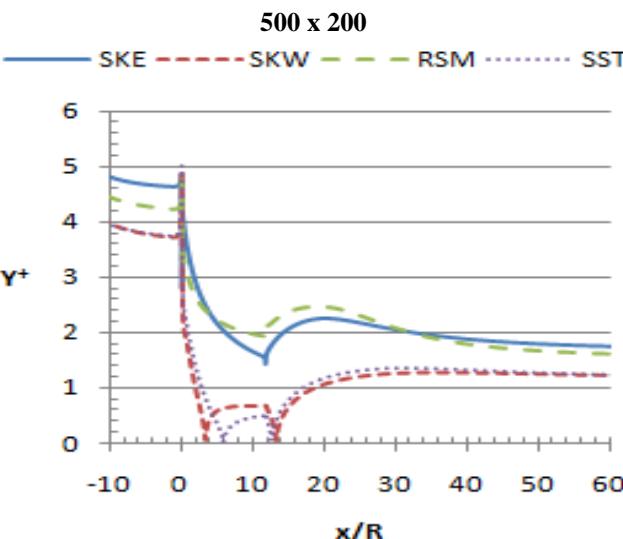
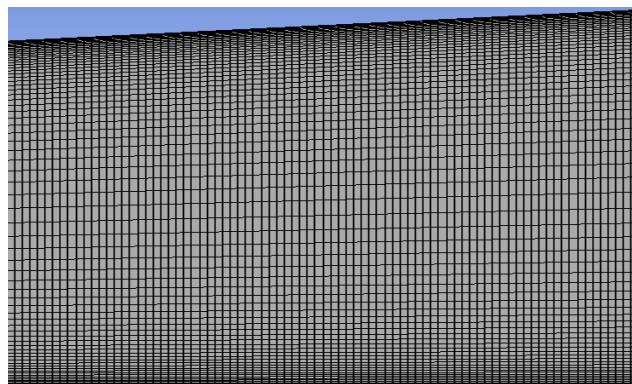
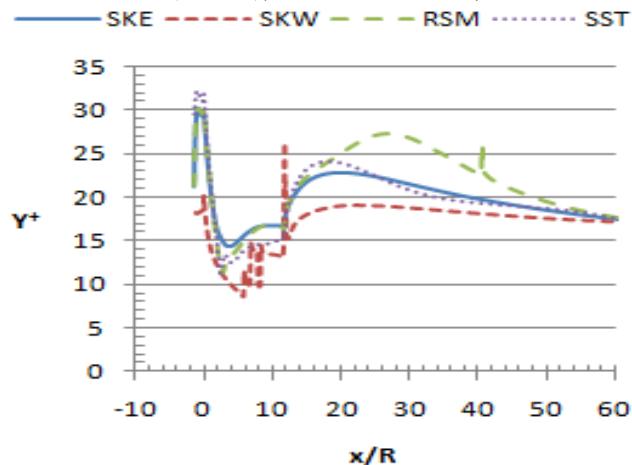
Fig. 1: Computational domain

ANSYS Workbench CFD was used to generate the geometry and associated mesh and ANSYS FLUENT 14 was used to solve the problem numerically and analyze the results. For validation of numerical predictions, the experimental results by Nakamura et al. [14] were considered. Four different RANS turbulence models provided in the FLUENT were investigated for validation. Enhanced wall treatment was used to model the near-wall region. The enhanced wall treatment combines the two-layer model with enhanced wall functions.

All turbulence models implemented a SIMPLE scheme to couple the pressure and velocity and the spatial discretization was employed by a second-order accurate upwind scheme for the momentum and a FLUENT standard scheme for the pressure. In all cases, the corresponding calculation residuals were monitored to convergence at 1×10^{-6} .

The appropriate boundary conditions were selected to solve the system of governing differential equations. In general four types of boundaries were specified viz. velocity inlet, outlet, symmetry axis and solid wall. Inlet velocity was prescribed as fully developed at the inlet with a centerline velocity of 20 m/s and the exit domain was specified as pressure outlet. On the symmetry axis, the radial derivative for variables were equal to zero while no-slip conditions were applied at the wall boundaries as usual.

Three different mesh sizes of fine, medium course and coarse grids were applied in the grid test as shown in Fig. 2, 3 and 4 and four turbulence models were used for the test analysis. The fine and medium course grids showed very close results and hence the medium coarse grid was selected as a good compromise of computational cost and agreed with the FLUENT recommendations for $Y+$ (less than 5).

Fig. 2: Variations of Y^+ with fine grid ($Y^+ \approx 1.0$)Fig. 3: Variations of Y^+ with medium coarse grid ($Y^+ \approx 5$)
500 x 125Fig. 4: Variations of Y^+ with coarse grid ($Y^+ \approx 30$)

500 x 75

III. RESULTS AND DISCUSSION

The comparison between the experimental and computational streamwise velocity profiles at several stations with $x/R = -1.4, 2.8, 8.5, 12.8, 33.2$ and 35.2 are shown in Fig. 5. The comparison showed that reasonable predictions were established at $x/R = -1.4$ especially with SKW and SST turbulence models. At initial stage of high adverse pressure gradient the RSM model at $x/R = 2.8$ presented the best results when compared with the experimental data followed by SKE model and these observations were also clear at $x/R = 8.5$. At the proximity of diffuser exit, $x/R = 12.8$, the velocity profile of the experimental results revealed a good resemblance of an axisymmetric distribution without flow separation and the SKE and RSM models again predicted the velocity profiles in good agreement with the experimental data while SKW and SST models failed to capture the velocity profile accurately due to smaller predicted boundary layer thickness.

The velocity profiles in the tailpipe at $x/R = 23.2$ and 35.2 indicated slight deviation of flow pattern from axisymmetric behavior very close to the walls. The turbulence model RSM turned out to be in good agreement with the experimental results. Poor predictions were shown with SKW and SST turbulence models. In brief, it was clear that the RSM turbulence model followed by SKE model turned out to be the best choice overall to capture the velocity distributions across the flow from the wall to the diffuser center in comparison with other models.

Assessment of Turbulence RANS Models for Conical Diffuser with Tailpipe

□ Experimental — SKE — RSM — SKW — SST

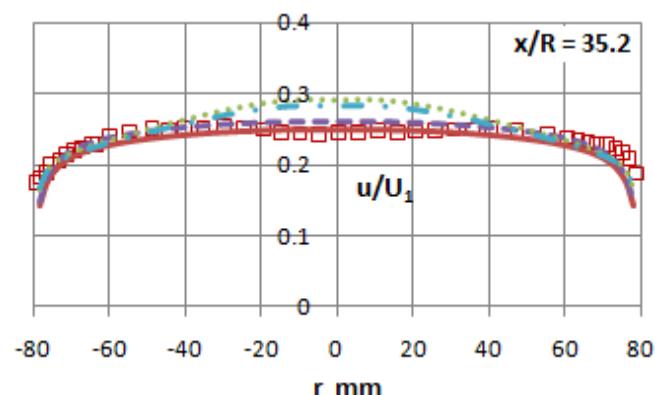
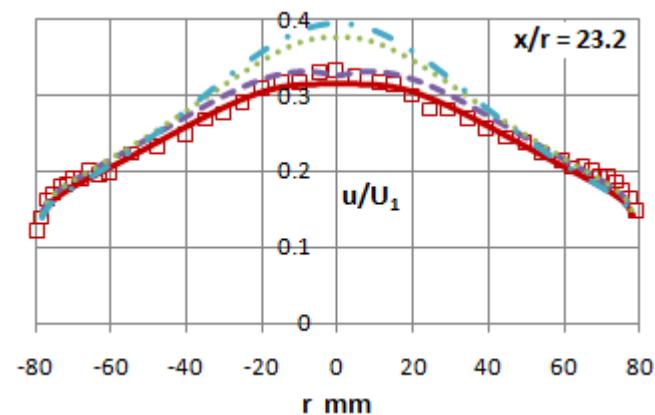
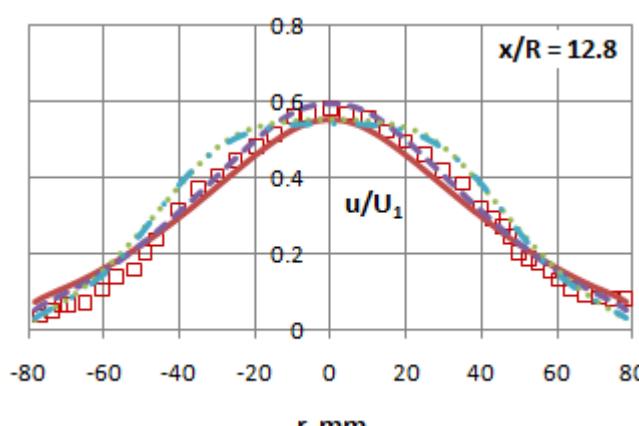
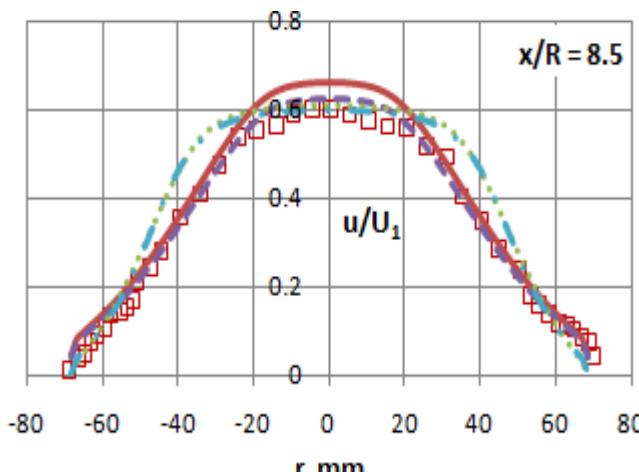
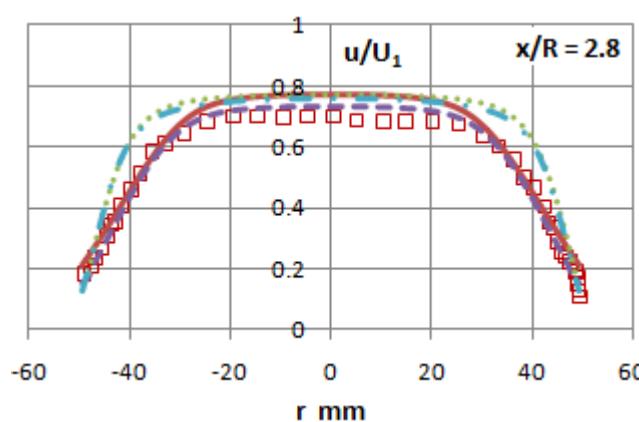
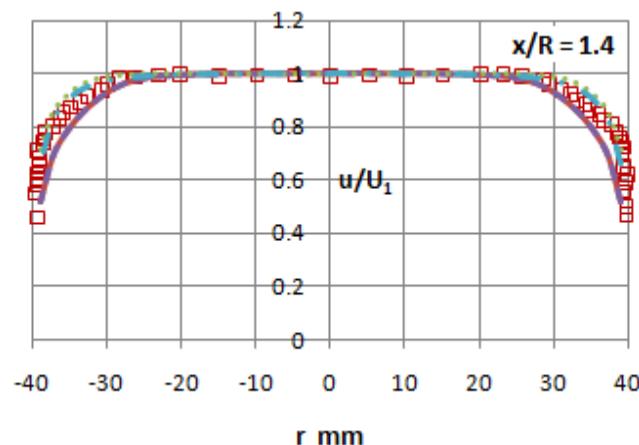


Fig. 5: Streamwise velocity profiles at various sections in diffuser and tailpipe

The non-uniformity of velocity profiles was expressed along the streamwise direction of the diffuser to aid a quantitative comparison of velocity between the predicted values from turbulence models and experimental data. The non-uniformity " σ " is defined as the ratio between the maximum velocity and the mean velocity over the cross-sectional area and is given as:

$$\sigma = \frac{u_{\max}}{\bar{u}} \quad (3)$$

□ Expt. — SKE — RSM — SKW — SST

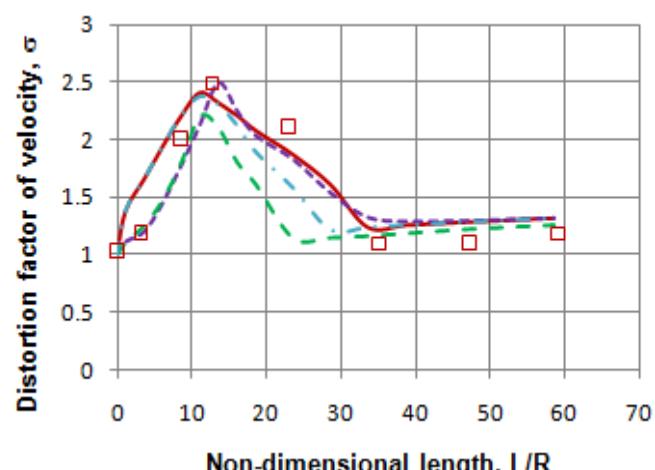


Fig. 6: The non-uniformity of velocity profiles

The comparison between the predicted and experimental results of streamwise variation of non-uniformity of velocity profiles are illustrated in

Fig. 6. The value of σ increased in the stream-wise direction of conical diffuser and attained a maximum value near the diffuser exit. Such non-uniformities decreased rapidly towards the tailpipe end. Again RSM model showed a good agreement with experimental results up to $x/R = 30$. Overestimated values of σ are shown for all turbulence models at $x/R > 30.0$ due to the deviation of flow from axisymmetric pattern in the tailpipe.

A comparison between the experimental results and model predictions for streamwise variations of the normalized kinetic energy flux "K" is shown in Fig. 7. The kinetic energy flux is defined as the turbulent kinetic energy flux at any cross section relative to the inlet section of diffuser as under:

$$K = \frac{\int_A \frac{1}{2} \rho u^3 dA}{\int_{A_1} \rho u_1^3 dA} \quad (4)$$

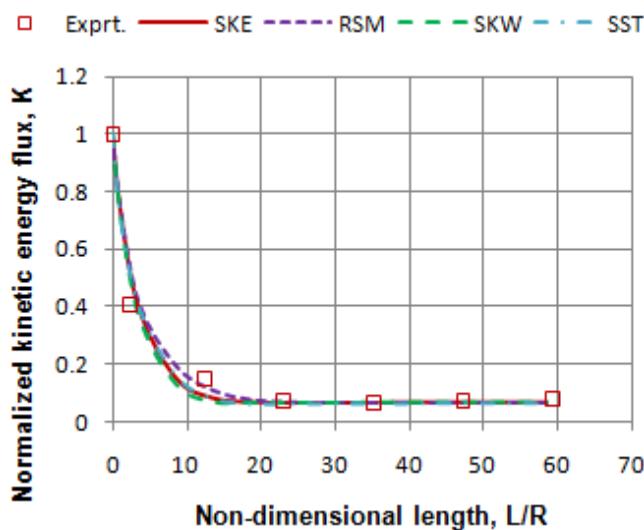


Fig. 7: Streamwise variations of kinetic energy flux

It was evident from the results that the dissipation rate of normalized kinetic energy flux decreased in the streamwise direction. The RSM turbulence model showed the best overall prediction for kinetic energy flux, while other models presented good results in the inlet and tailpipe section.

The comparison between the predicted static pressure coefficient and the experimental results is shown in Fig. 8. To represent the pressure distribution in dimensionless form, a static pressure coefficient was defined as under

$$C_p = \frac{p - p_1}{1/2 \rho u_1^2} \quad (5)$$

Where p_1 and u_1 are pressure and average velocity at inlet reference section. The comparison with experimental data showed a reasonable agreement in the diffuser section with all turbulence models however, in the tailpipe section the SKE and RSM turbulence models predicted the pressure coefficient better than SKW and SST models. The percent errors between the predicted and measured static pressure coefficients at the diffuser and tailpipe exits are shown in Table I. It was clear that the error due to adopting SKW model is very small at the diffuser exit and such model presents better static pressure distribution in the diffuser. The SKE and RSM models are also acceptable to present the static pressure recovery for the whole domain of calculation. The SKW and SST turbulence models under-predicted the static pressure recovery in the tailpipe section.

Table I: Errors of C_p at diffuser and tailpipe exit

	SKE	SKW	RSM	SST
Error at diffuser exit (%)	5.36	-0.28	5.62	-6.36
Error at tailpipe exit (%)	2.07	-9.73	2.27	-7.85

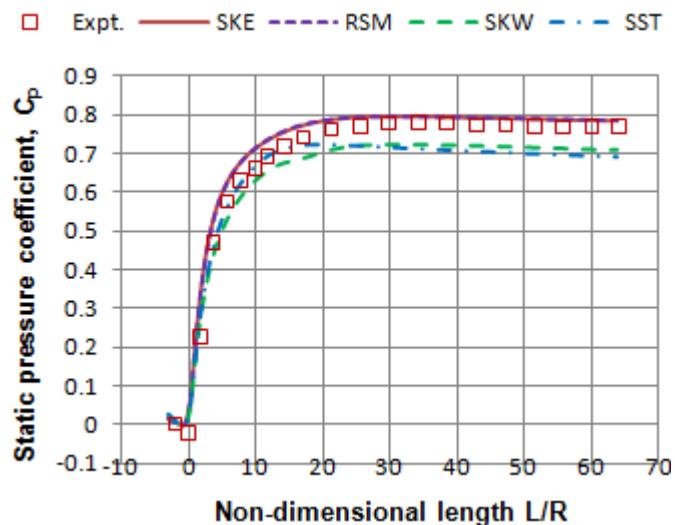


Fig. 8: Streamwise variation of static pressure coefficient

IV. CONCLUSIONS

In the present study four Reynolds-averaged Navier-Stokes (RANS) turbulence models have been employed to validate the flow characteristics in conical diffuser with tailpipe. Based on the numerical experiments, it was concluded that the standard $k-\epsilon$ (SKE) and the Reynolds stress (RSM) models were capable to capture the velocity profiles as well as the static pressure recovery within acceptable error limit. The estimations of non-uniformity of velocity profiles and the streamwise variations of kinetic energy flux also consolidated the $k-\epsilon$ (SKE) and the Reynolds stress (RSM) schemes as the choice of the best turbulence models. The standard $k-\omega$ (SKW) and shear stress model (SST) models showed very good agreement for the static pressure recovery distribution in the diffuser section but failed to predict the same accurately in the tailpipe section.

REFERENCES

- [1] D. D. Apsley, and M. A. Leschziner, "Advanced turbulence modelling of separated flow in a diffuser", *Flow, Turbulence and Combustion*, Vol. 63 no. 1-4, pp. 81-112 (2000).
- [2] M. Ariff, S. M. Salim and S. C. Cheah, "Wall $y+$ approach for dealing with turbulent flow over a surface mounted cube: part 1-low Reynolds number", *Proceedings of Seventh International Conference on CFD in the Minerals and Process Industries*, Melbourne, Australia, (Dec. 2009).
- [3] H. Bassily, M. A., EL-Kersh, A. M. Bassiouny and M. R. Gomaa, "Air flow characteristics in an asymmetric plane diffuser under different inlet conditions", *Bulletin of the Faculty of Engineering*, Minia University, vol. 27, no. 2 (2008).
- [4] R. A. Berdanier, "Turbulent flow through an asymmetric plane diffuser", Masters Thesis, *Purdue University* (2011).

- [5] L. Chen, A. Yang, R. Dai, and K. Chen, "Comparative study of turbulence models in separated-attached diffuser flow", *The 4th International Symposium on Fluid Machinery and Fluid Engineering*, Beijing, China No. 4ISFMFE – Ch 04, pp. 215-220, (Nov. 2008)..
- [6] E. M. Cherry and Glaccarino, "Separated flow in a three-dimensional diffuser: preliminary validation", *Center for Turbulence Research*, Annual Research Briefs (2006).
- [7] S. M. El-Behery and M. H. Hamed, "A comparative study of turbulence models performance for separating flow in a planar asymmetric diffuser", *Computers and Fluids*, vol. 44, no. 1, pp. 248-257 (2011).
- [8] M. Faizan, A. Alosaimy, and A. M. El- Kersh, "Performance of conical diffuser equipped with center bodies at inlet", *Minia Journal of Engineering and Technology (MJET)*, vol. 32, no. 1, pp. 174-187 (2013).
- [9] G. Iaccarino, "Predictions of a turbulent separated flow using commercial CFD codes", *Journal of Fluids Engineering*, vol. 123, no. 4, pp. 819-828 (2001).
- [10] B. E. Launder and D. B. Spalding, "Lectures in Mathematical Models of Turbulence", Academic Press, London, England (1972).
- [11] B. E. Launder, G. J. Reece and W. Rodi, "Progress in the development of a Reynolds-stress turbulence closure", *Journal of Fluid Mechanics*, vol. 68, no. 3, pp. 537-566 (1975).
- [12] F. R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", *AIAA Journal*, vol. 32, no. 8, pp. 1598-1605 (1994).
- [13] D. S. Miller, "Internal flow systems design and performance prediction", 2nd Edition, *Gulf Publishing Company*, Houston, Texas USA (1990).
- [14] I. Nakamura, K. Ishikawa, and Y. Furuya, "Experiments on the Conical Diffuser Performance with Asymmetric Uniform Shear Inlet Flow: Effect of tailpipe discharge", *Bulletin of JSME*, vol. 24, no. 196, pp. 1729-1738 (1981).
- [15] H. K. Versteeg and W. Malalasekera, "An introduction to computational fluid dynamics: the finite volume method", Pearson Education (2007).
- [16] D. C. Wilcox, "Turbulence modeling for CFD", DCW Industries, Inc., La Canada, California (1998).
- [17] C.D. Argyropoulos and N.C. Markatos, "Recent advances on the numerical modeling of turbulent flows: Review Article", *Applied Mathematical Modeling*, In Press, Available online 14 July 2014.

Dr. Mohammad Faizan is a Faculty of Mechanical Engineering at Taif University Taif, KSA since 2009. He obtained a Bachelors degree in Mechanical Engineering from Aligarh Muslim University, Aligarh India. His Masters degree in Engineering from Youngstown State University, Youngstown Ohio USA and Ph.D. degree from University of Akron, Akron Ohio USA. He has numerous publications in international journals and conference proceedings. His areas of research are heat and mass transfer in material processing, lead-free solders, comfort air-conditioning and computational fluid dynamics. He is a member of TMS, IMECE, IST and SAE.