Simulation of Turbulent Flow in an Asymmetric Air Diffuser

Abubaker Elbaloshi and Junling Hu
Department of Mechanical Engineering, University of Bridgeport

Abstract

This research paper compared three k-ε family turbulence models; Standard k-ε, RNG k-ε and Realizable k-ε, and two k-ω family models; Standard k-ω and Stress-Strain Transport, SST k-ω. The turbulent flow characteristics were predicted in a two-dimensional of 10° 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 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-ε family models have a valid prediction of flow characteristics that are far away from wall effects, however, k-ω 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.

Introduction

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 continuity, momentum, and energy partial differential equations, PDE's, numerically and then predicting their performance under certain operational conditions in satisfied accuracy. Any developed technique, equipment, or an industrial tool will not be marketing until its performance or function has been simulated by using one of CFD codes. 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 as well.

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.

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 developed for various kinds of flow. It is very important to understand these turbulence models in order to appropriately use them to model flow phenomena. This paper compared five turbulence models using a bench mark problem – flow in a asymmetric diffuser.

Simulation Processes

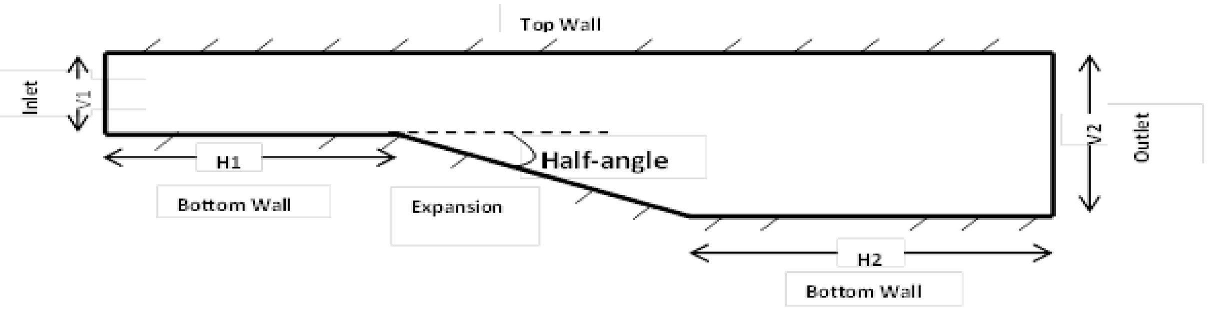

Fig. 1. A schematic of the 2-D computational domain and boundary conditions of a 10o half-angel Diffuser (not to scale).

The computational domain 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, where, H1= 60 m, V1=2 m, H2=70 m, and V2= 9.4 m.

Table 1. Boundary Conditions

Variable	Symbol	Unit	Value
x-velocity	U	m/s	1.25
y-velocity	V	m/s	0
Inlet Pressure	P	pa	--
Turbulent Kinetic Energy	K	m^2/s^2	0.0018
Turbulent Dissipation Rate	E	m^2/s^3	9.63×10^{-5}

Variable	Symbol	Unit	Value
y-velocity	V	m/s	--
Outlet Gauge Pressure	P	pa	0
Backflow Turbulent Intensity	--	%	3.25
Backflow Turbulent length	--	m	0.0035

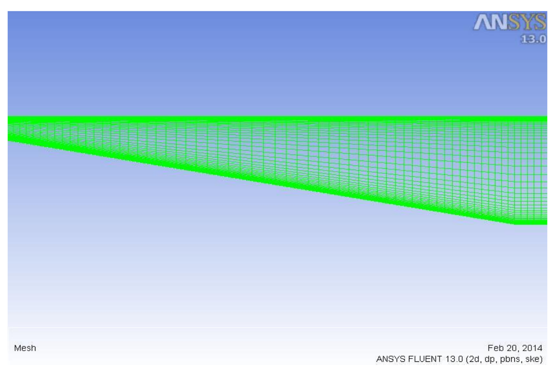

Fig. 2. A non-uniform rectangular mesh 10° half-angle air diffuser.

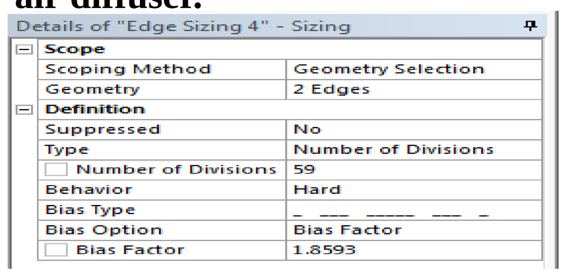

Fig. 3. Bias Factor options window for top & bottom wall boundaries .

Meshing Processes

A non-uniform structured mesh was generated in a two-dimensional-10o 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.

Computational Results

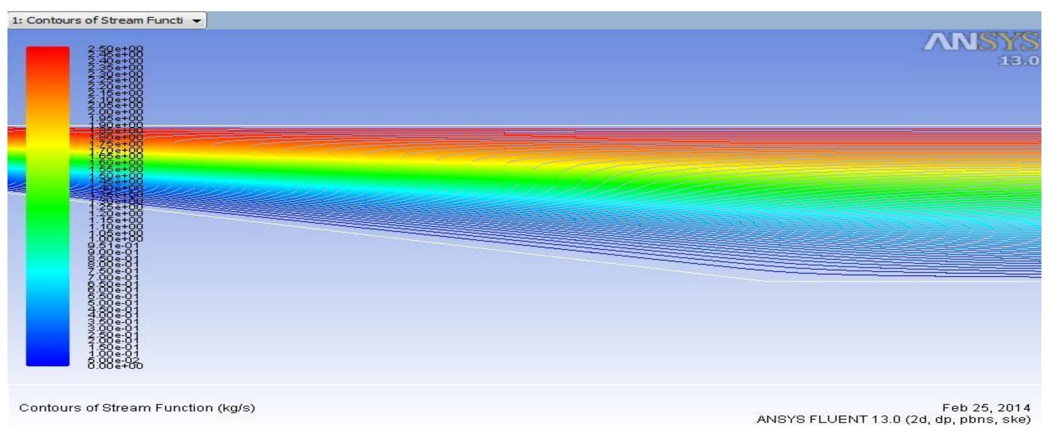

Fig. 4. The Velocity Stream of k-ε

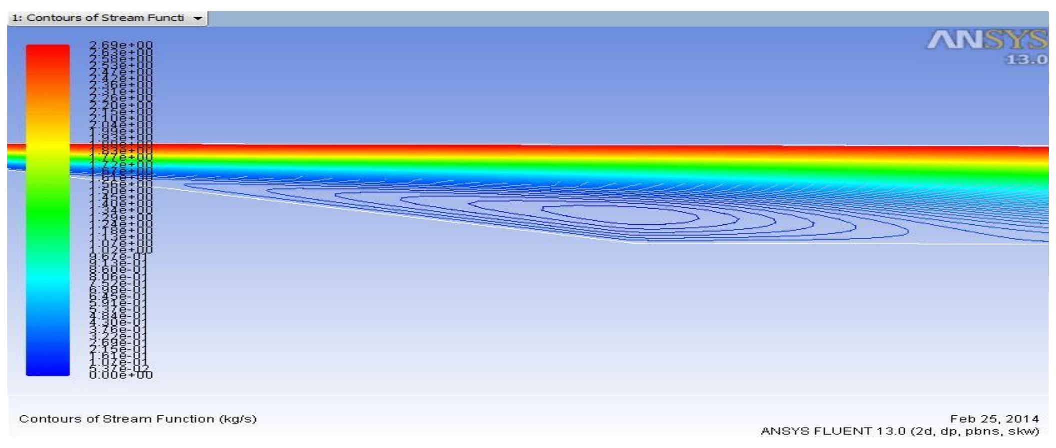

Fig. 5. The Velocity Stream of k-ω

Comparison the Results of the Five Models with Experimental Results

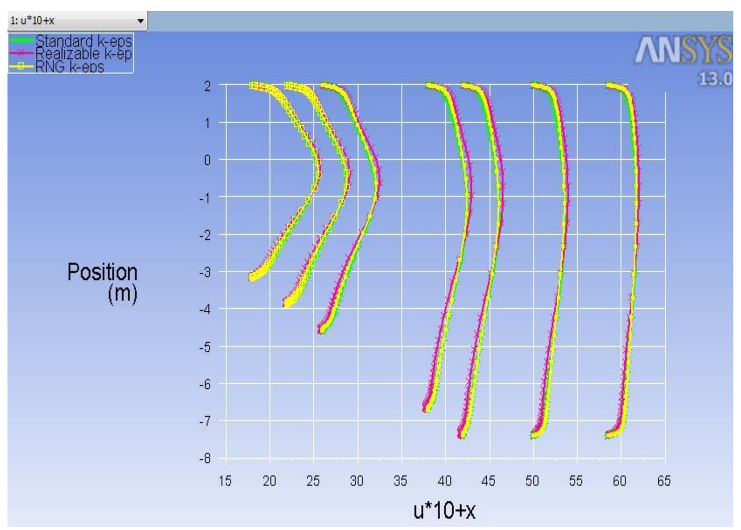

Fig. 6. Modified velocity vs Standard k-ε vs Realizable k-ε vs RNG k-ε .

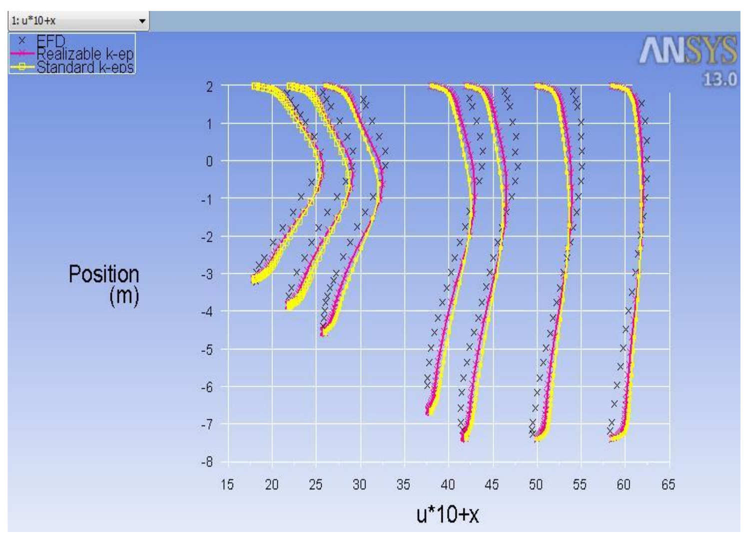

Fig. 7. Modified velocity vs Standard k-ε vs Realizable k-ε vs EFD.

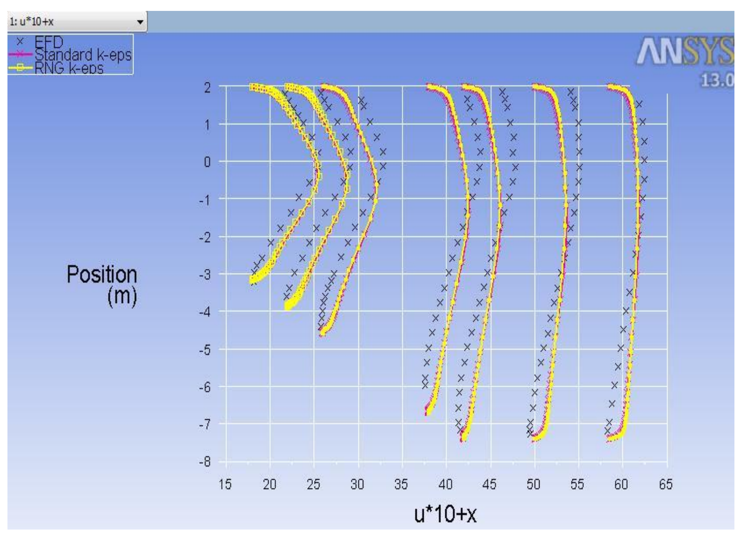

Fig. 8. Modified Velocity vs Standard k-ε vs RNG k-ε vs EFD.

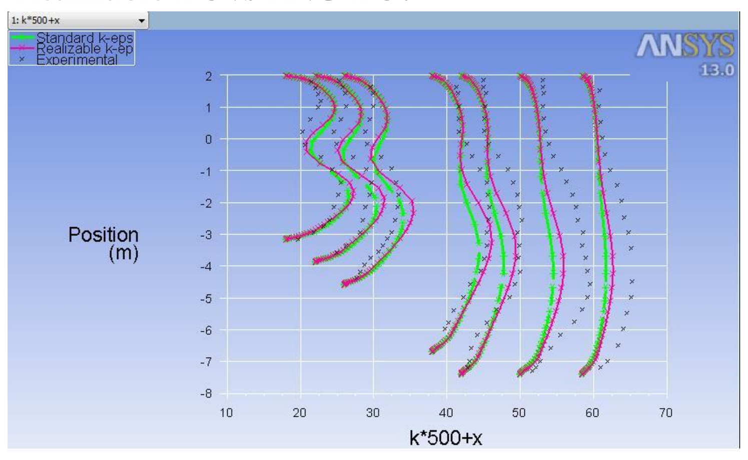

Fig. 9. Modified TKE of Standard k-ε vs Realizable k-ε vs EFD.

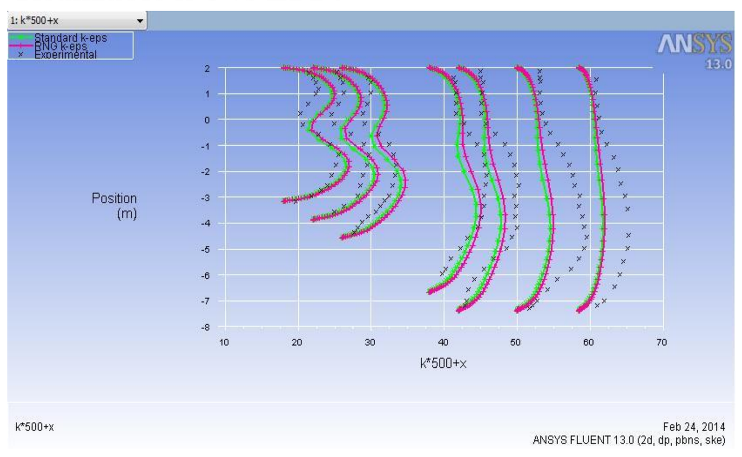

Fig. 10. Modified TKE of Standard k-ε vs RNG k-ε vs EFD.

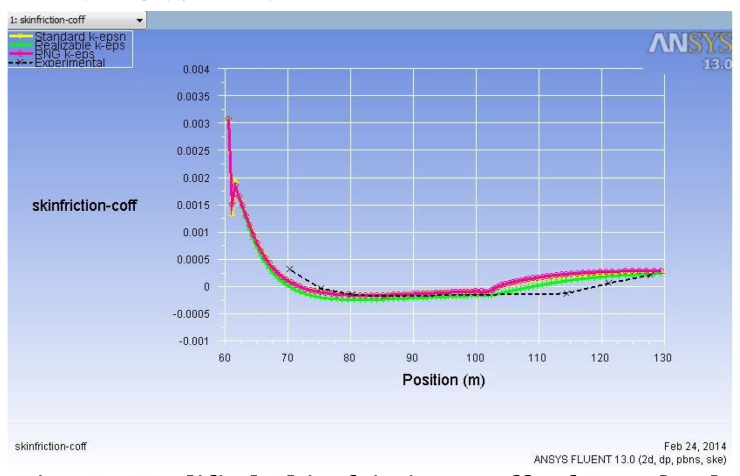

Fig. 11. Modified Skin-friction coeff. of Standard k-ε vs Realizable k-ε vs RNG k-ε vs EFD.

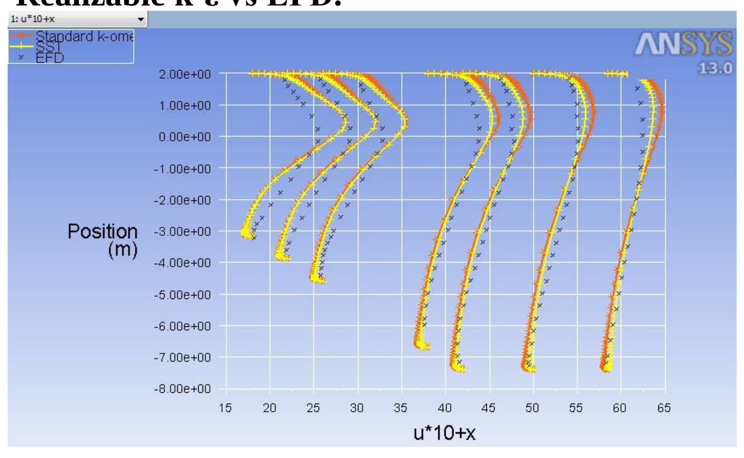

Fig. 12. Modified velocity of Standard k-ω, SST and EFD.

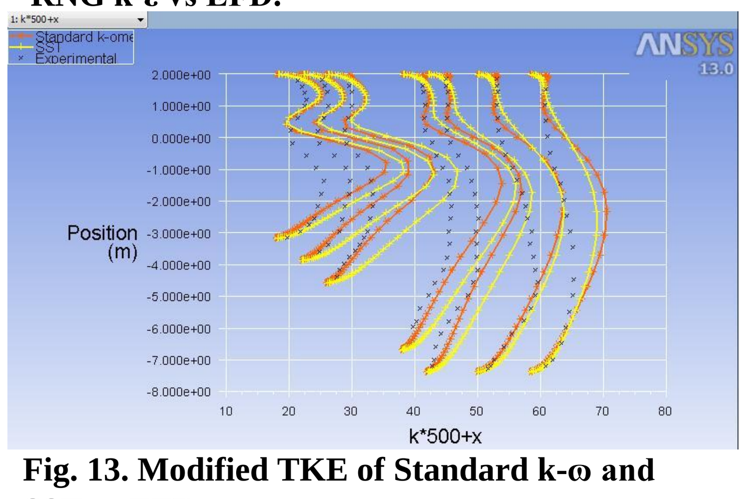

Fig. 13. Modified TKE of Standard k-ω and SST vs EFD.

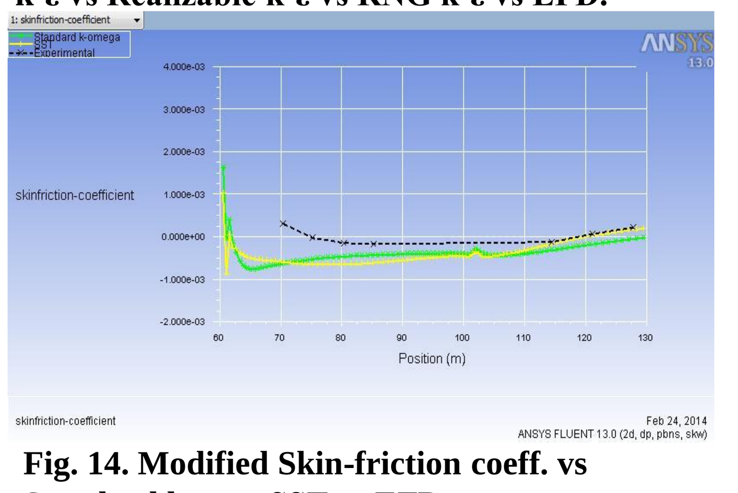

Fig. 14. Modified Skin-friction coeff. vs Standard k-ω vs SST vs EFD.

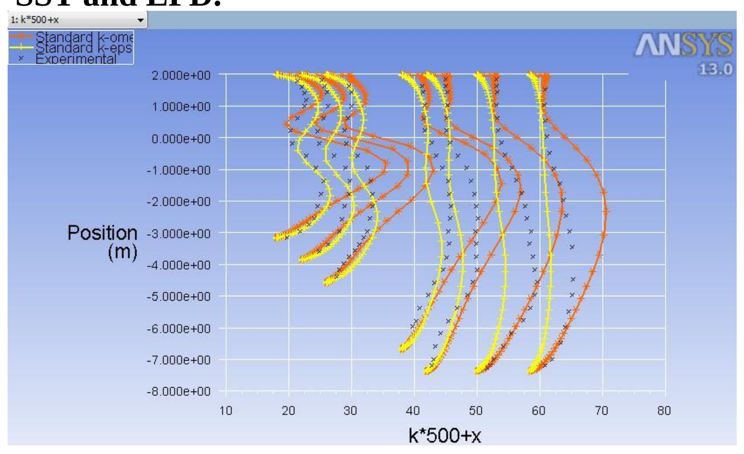

Fig. 15. Modified velocity of Standard k-ω, Standard k-ε, and EFD.

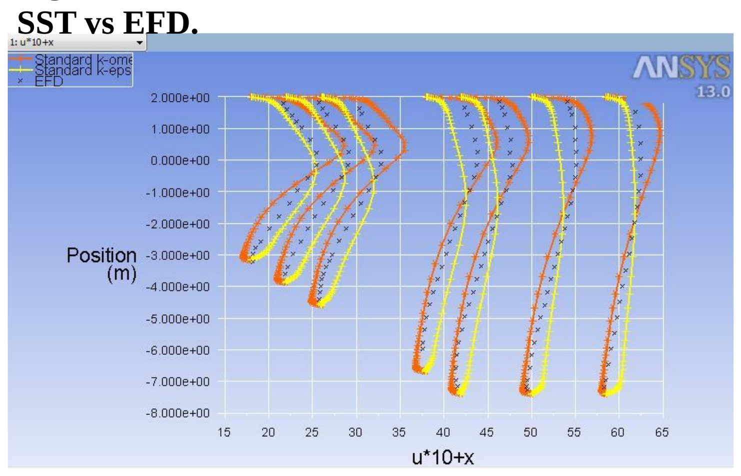

Fig. 16. Modified velocity of Standard k-ω, Standard k-ε and EFD.

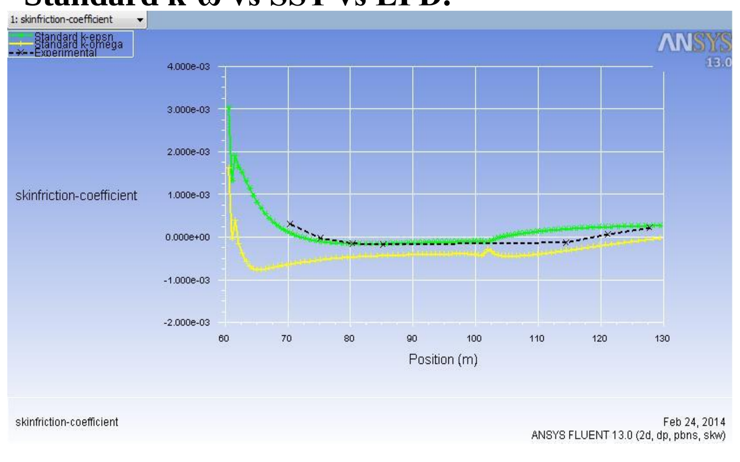

Fig. 17. Modified TKE of Standard k- ε, and Standard k-ω.

Conclusion

This paper simulated the turbulent flow of air in a 10o 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-ε family models, Standard k-ε, RNG k-ε, and realizable k-ε, give very close results, with realizable k-ε gives the best result for the diffuser simulation. The standard k-ω model and SST k-ω model give very close results. k-ε 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-ε models and k-ω 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-ε 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- ω 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.