The Asian Review of Civil Engineering
ISSN: 2249-6203 (P) Vol.2 No.2, 2013, pp.30-34
© The Research Publication, www.trp.org.in
DOI: https://doi.org/10.51983/tarce-2013.2.2.2192

Numerical Studies of a Two Dimensional Symmetric Diffuser in a Turbulent Flow Using CFD

Chougala Kumar[1] and Abhinandan S. Kabbur[1]
[1&2]*Assistant Professor, Department of Mechanical Engineering, KLECET, Chikodi - 591201, Karnataka, India*
E-mail: kumar.chougala@gmail.com, askabbur@gmail.com
(Received on 18 June 2013 and accepted on 30 July 2013)

Abstract - **The numerical studies of a two dimensional asymmetric diffuser in turbulent flow with maximum pressure at the exit is obtained using computational fluid dynamics. The three Reynolds number based on the bulk mean velocity and the channel height at the diffuser entrance is 18000, 40,000 and 60,000. The maximum pressure is obtained to be zero skin friction along the diffuser wall. The turbulent flow inside the diffuser is predicted using three Reynolds average navier stokes based turbulence models are considered for the study i.e., K-ε standard, K-ε RNG and K-ω SST. The designed diffuser geometry has analyzed with fluent for its design intent. The variation of static pressure, velocity, skin friction coefficient and pressure coefficient are studied for three different Reynolds numbers. The designed diffuser geometry has been used in gas turbine engine exhaust. The function of diffuser is the kinetic energy of incoming exhaust gases from the power turbines is converted into static pressure through the diffuser. It also helps in discharging of exhaust gases from high speed to low speed, and also to turn the flow of gases from axial to radial direction.**

Keywords: **CFD, Diffuser, K-ε RNG Model, K-ω SST Model, K-ε Standard Model, Pressure coefficient, Reynolds numbers, Skin friction coefficient, Turbulent flow**

I. Introduction

The diffuser is a device that converts the kinetic energy of the flow into the potential energy by broadening its height to decelerate the velocity and recover the pressure. However, when the diffuser height increases too much in the stream-wise direction, flow separation occurs inside the diffuser, resulting in a serious pressure loss. Therefore, flow separation is an important factor that should be considered to enhance the performance of the diffuser.

Researches on modifying the diffuser wall shape have been conducted to improve the diffuser performance. In his pioneering work, startford theoretically obtained a pressure distribution that maintains zero skin friction throughout the region of pressure rise inside a diffuser and experimentally constructed a diffuser having a theoretical pressure distribution by controlling each segment of the diffuser wall. Maintaining zero skin friction along the diffuser wall suggests that the diffuser should be widened as much as possible but preventing flow separation.

In the first decade of this country, a debate regard in the learned societies as to the practical utility of placing an exhaust diffuser downstream of a hydroelectric turbine. Experts argued back and forth that the exhaust diffuser would or would not, improve the performance of the turbine. The opponents essentially maintained that the fluid had already left the turbine and little could be done. The expansion ratio across the turbine rotor, by the reduction in rotor back pressure with the use of a well designed diffuser. Today's arguments and concerns over the role of the diffuser are significantly more advanced. Nonetheless, the details of diffuser design and performance are, in some instances, just as vague as the early debate on diffuser application for hydro turbines.

Fluid machinery is conveniently broken into Positive displacement and turbo machinery categories. The distribution follows directly along the lines of Newton's second law of motion as applied either in a Cartesian co-ordinate system or a cylindrical co-ordinate system. Newton's second law takes the form of the torque being proportional to the change in angular momentum, this principle leads directly to the Eulers turbo machinery equation, which expresses the energy transfer through turbo machinery as the change in

UCθ. Thus, energy transfer in turbo machinery involves the exchange of significant levels of Kinetic energy in order to accomplish the intended purpose. As a consequence, very large levels of residual kinetic energy frequently accompany the work input and work extraction processes, sometimes as much as 50% of the total energy transferred.

Therefore diffusers are absolutely essential for good turbo machinery performance. With kinetic energies of this magnitude, it is not hard to appreciate that the performance of the diffuser directly and often strongly influences the overall efficiency of the turbo machine, a change of 0.01 in pressure recovery can be equivalent to a few tenths of a point of stage efficiency or can be as much as two to four points of stage efficiency.

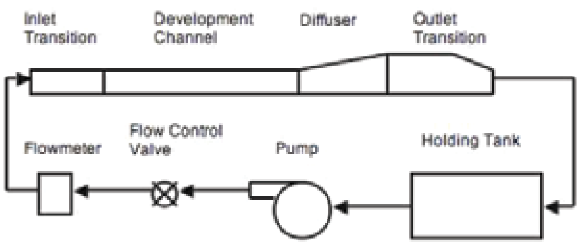


Fig.1 Application of diffuser in Pump

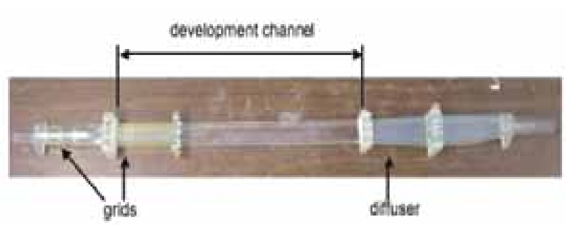


Fig.2 The development channel, the diffuser, and the outlet transition duct as fabricated using stereo lithography manufacturing

In gas turbine engines, the final stage of air compression occurs within the annular diffuser just upstream of the combustor. This component must satisfy connecting goals of recovering kinetic energy exiting the compressor while supplying reasonably uniform flow and consistent mass splits into the various sections of the combustor. The key challenge in designing the diffuser is to make it as short as possible while avoiding any possibility of massive flow separation. Pressure losses due to the separated flow reduce engine performance while unsteadiness and recirculating flow associated with separation can cause catastrophic engine failure. An optimal design probably operates very near separation for some part of the engine's operating envelope. Accurate design analysis tools are needed to find the optimum and to avoid unexpected failures during prototype testing.

II. Diffuser Performance Parameters

A. Geometric Parameters

1. Channel Diffusers

The geometric specification of a channel diffuser is, at first appearance, comparatively straightforward.

2. Aerodynamic Parameters of Performance

In the early periods these parameters were usually ignored but, with time, it was found that a number of aerodynamic parameters became crucial in evaluating the performance of the diffuser, depending on the type of diffuser involved.

3. Aerodynamic Blockage

Viscosity is an important parameter in any fluid dynamic process and normally appears in the form of a Reynolds number typically, diffusers are characterized by a Reynolds number based on an inlet hydraulic diameter.

4. Inlet Velocity Profiles

No convention has been developed to specify the inlet velocity profile to a diffuser. However, various research programs have shown the effect to be significant. Both simple skewed inlet profiles and highly distorted inlet profiles have been considered and reported in the open literature.

III. Introduction to Fluent

CFD is primarily used as a design aid for predicting the performance characteristics of equipment involving fluid flow and heat transfer. The ability to simulate heat transfer and fluid flow problems numerically even before a prototype being built, reduces the cost and most essentially time of development to a greater extent. Obviously, CFD results have to be continuously checked experimentation ensuring the numerical predictions are reliable. Thus a cycle is formed involving theoretical predictions, CFD and experimentation. FLUENT is one of the most widely used computational fluid dynamics (CFD) software package to simulate fluid flow related problems. It uses the finite - volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc.

Once a grid has been read into FLUENT, all remaining operations are performed within the solver like setting boundary conditions, defining fluid properties, iterating the solution and post processing the results.

A. Discretization of Governing Equations

Discreation is the process whereby the continous governing differential equations are replaced by their counterparts. The differential equations are transformed to algebraic equations, which should correctly approximate the transport properties of the physical process. Discreatization identifies the node locations used to model the physical problem configuration. In order to solve these algebraic equations it is necessary to calculate the transport property at the cell faces, which the upwinding differencing scheme does. The important steps involved in the discretization procedure are listed below:-

1. Division of the computational domain into discrete control volumes using a general curvilinear grid.
2. Integration of the governing equations on the individual control volumes to construct the algebraic equation for discrete unknowns.

IV. Design Methodology

A. Meshing

The two dimensional mesh of the diffuser was achieved easily with good equiangle skewness using a Quad/pave meshing scheme. Two separate meshes of the diffuser were made using an interval size of 0.5, 0.8. Figure 3 shows grid of interval size of 0.8.

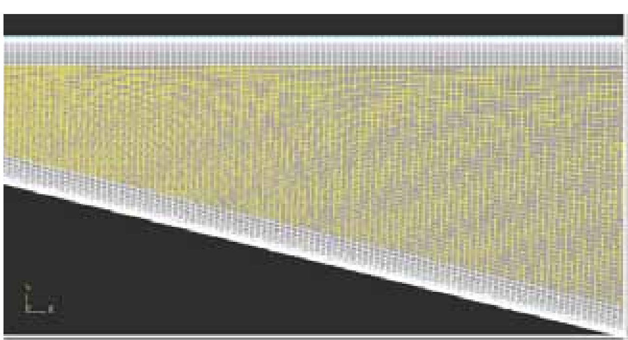


Fig.3 Face Mesh Creation

Boundary conditions and control parameters prescribed flow conditions were applied at the inlet and outlet of the grid. Water was chosen from the fluid database defining material and chosen as an incompressible fluid. The operating pressure was also defined to 101325 pascal. For the boundary conditions, one velocity inlet was selected for the diffuser inlet, with a velocity of 4.14 m/s. the diffuser upper portion was defined as a top wall and lower portion was defined as a bottom wall. One pressure outlet was prescribed for the diffuser outlet of 0 pascal (gauge). For a more accurate solution, the second order upwind differencing scheme was chosen for discretization alone applicable and the residual convergence criteria lowered to 0.001 for all solution parameters.

Fig. 4 Boundary conditions

V. Results and Discussion

The analysis of each solution began with a check of the diffuser total pressure. The results are shown below in Table I.

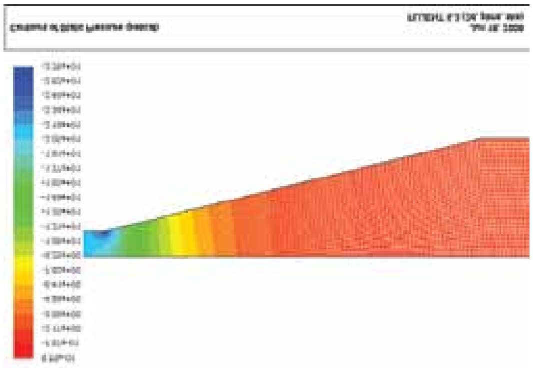
Table I Results of Total Pressure

S. N.	Turbu-lence Model	Reynolds No	Inlet Static Pressure	Outlet Dynamic Pressure	Total Pressu re
1	K-ε Standard	18000	-0.9504	0.1008	0.1902
		40000	-5.97	0.534	0.8886
		60000	-14.50	1.119	1.295
2	K-ε RNG	18000	-1.07	0.1013	0.4795
		40000	-6.476	0.4978	0.6696
		60000	-14.905	1.116	1.5755
3	K-ω SST	18000	-0.813	0.1086	0.2421
		40000	-5.47	0.5241	1.075
		60000	-13.197	1.168	2.36

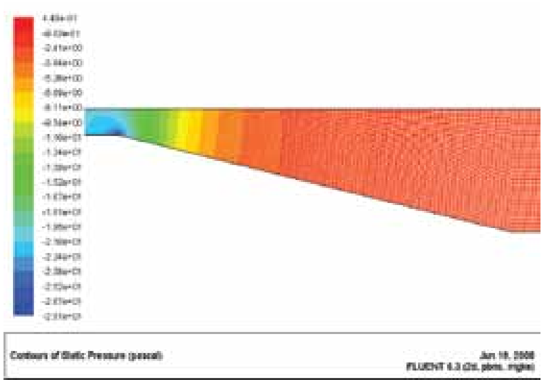
The three Reynolds numbers were compared with three turbulence models are K-ε standard, K-ε RNG and K-ω SST. Out of these three turbulence models K-ω SST gives maximum pressure at the outlet of the diffuser. Hence for design intent K-ω SST model is better.

The higher Reynolds number 60000 gives maximum pressure at the outlet of the diffuser.

K-Ɛ Standard Model Reynolds Number = 60000



K-Ɛ RNG Model Reynolds Number = 60000



K-ꞷ SST Model Reynolds Number = 60000

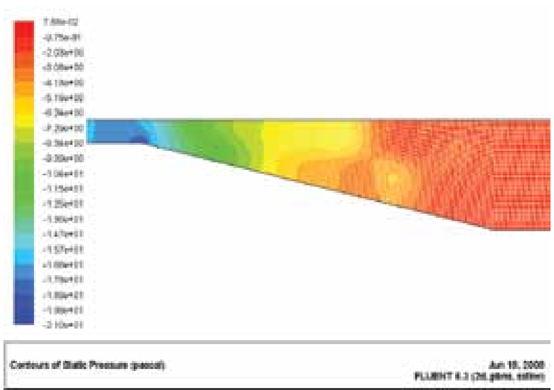
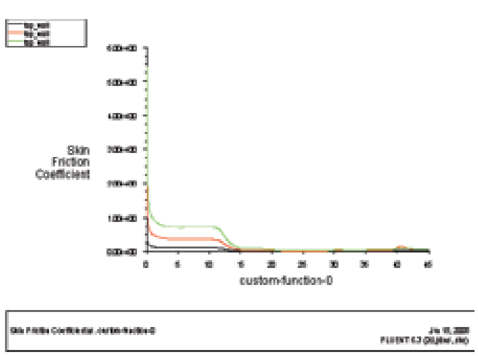
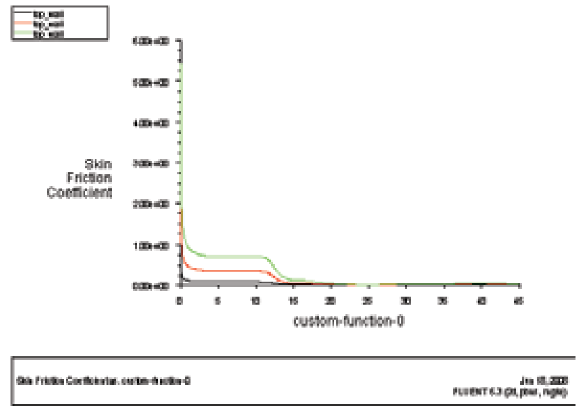


Fig.5 Effect of Reynolds Number on Contours of Static Pressure

K-Ɛ Standard Model



K-Ɛ RNG Model



K-ꞷ SST Model

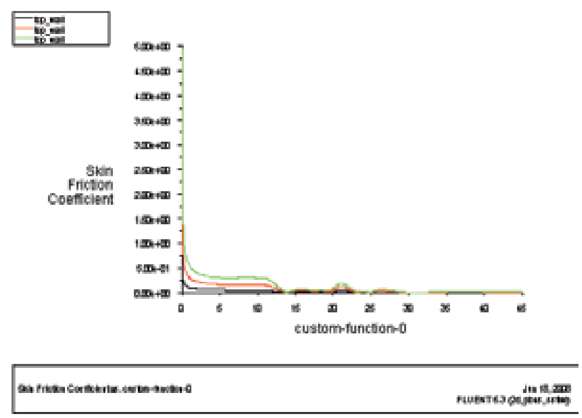
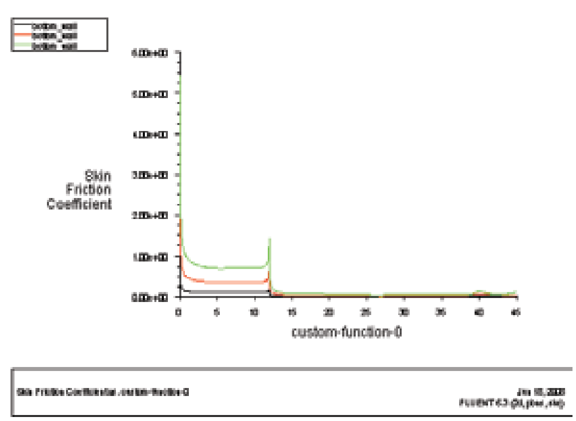


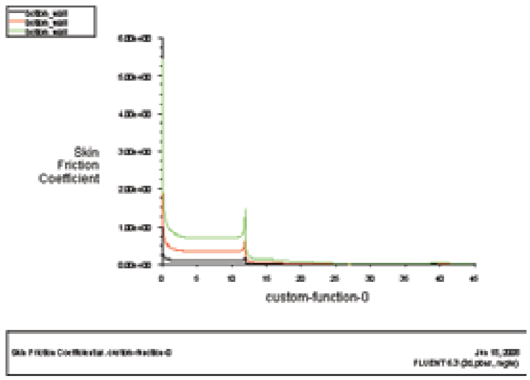
Fig.6 Effect of Reynolds Number on Top Wall Skin Fiction Coefficient

Figure shows the distribution of the skin friction co-efficient along the top walls of the diffuser for three different Reynolds number. The skin friction coefficient nearly zero in the diffuser region except near the diffuser and throat. The flow separation is not removed in the diffuser region as velocity increases the skin friction coefficient decreases. But at the inlet of the diffuser the skin friction coefficient is not zero.

K-Ɛ Standard Model



K-ε RNG Model



K-ω SST Model

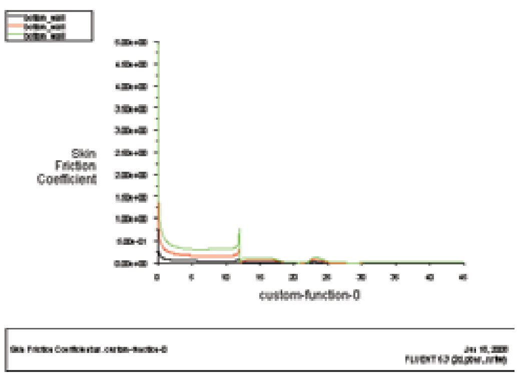


Fig. 7 Effect of Reynolds Number on Bottom Wall Skin Fiction Coefficient

Figure shows the distribution of the skin friction co-efficient along the bottom walls of the diffuser for three different Reynolds number. The skin friction coefficient nearly zero in the diffuser region except near the diffuser. The flow separation is not removed in the diffuser region as velocity increases the skin friction coefficient decreases.

Comparison of Skin friction coefficient for experimental and fluent results

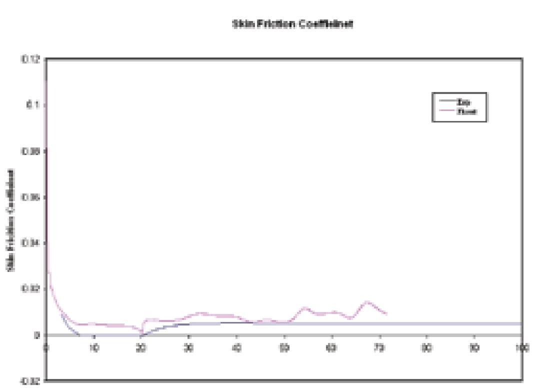


Fig.8 Shows the comparison of experimental and fluent results

Figure shows the distribution of skin friction co-efficient on top wall of the diffuser for 18000 Reynolds number. The experimental results nearly match with the fluent results.

VI. Conclusion

The results of various turbulence models were compared with available literature. The results from the studies with commercial code fluent are in good agreement with published literature. Higher Reynolds number is performing better compared to lower Reynolds number. The higher Reynolds number gives maximum pressure at the outlet of the diffuser. K-w SST turbulence model gives maximum pressure at the outlet of the diffuser.

References

[1] E.G.Reid,"Performance Characteristics of Plane Wall, Two Dimensional Diffusers", NACA TN 28888, 1953.

[2] L.R.Reneau, J.P.Johnston, and S.J. Kline,"Performance and Design of Straight Two Dimensional Diffusers", Report PD-8, Thermosciences Division, Mechanical Engineering Department, Stanford University, 1964.

[3] David Japikse, Nicholas C.Baines, "Diffuser Design Technology", USA Edwards brothers incorporated, 1998.

[4] D.Japiske, "Turbomachinery diffuser Design Document", 1984.

[5] Versteeg and Malalasekara, "Introduction to Computational Fluid Dynamics", 1998.

[6] G.Biswas,V.Esvaran, "Turbulent Flows – Fundamentals, Experiments and Modeling", Delhi, Narosa Publishing house, 2000.

[7] Ching-Jen Chen, Shen-Yuh Jaw, "Fundamentals of Turbulence Modelling", USA, Taylor and Francis, 1998.

[8] P.A. Aswatha Narayan, K.N.Seetharamu, "Engineering Fluid Mechanics", Delhi, Narosa Publishing House, 2000.

[9] J.Bardina, "A Prediction method for Planar Diffuser flows", Trans ASME Journal Fluids Engineering, Vol. 103, pp. 315-321, 1981.