

Validation Study and CFD Analysis of K- ϵ Model by Using Discrete Jet Swirler

Prasanna Gouda¹, Sagar H N², Doreswamy G V³, B M Krishne Gowda⁴

Assistant Professor, Department of Mechanical Engineering, S.K.I.T Bangalore, Karnataka, India^{1,2,3,4}

ABSTRACT: The purpose of this study is to quantify as to how well standard k- ϵ turbulence model simulate a turbulent, three-dimensional swirling flow field with discrete jets in a confined model combustor. Analyses are carried out using the commercially available ANSYS Fluent and the results obtained were compared with the experimental measurement obtained using Laser Doppler Velocimetry (LDV) and published in literature [5].

KEYWORDS: Discrete jet swirler, k- ϵ model, LDV

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is the numerical analysis of a system involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. This technique is very powerful and spans a wide range of industrial and non-industrial areas of application. CFD constitutes a new "third approach" in the philosophical study and development of whole discipline of fluid dynamics. In this present study, k-epsilon turbulent model has been used.

II. GEOMETRICAL DETAILS OF DISCRETE JET SWIRLER AND COMBUSTOR CHAMBER

The swirler studied in this study is a discrete jet swirler and its photocopy is shown in Fig.1 CAD model of swirler assembly in the test chamber is shown in Fig.2

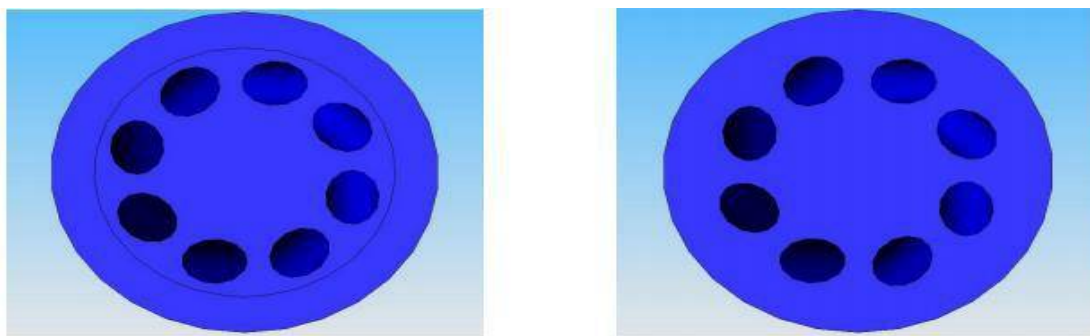


Fig.1 Solid Model of Discrete Swirler [5]

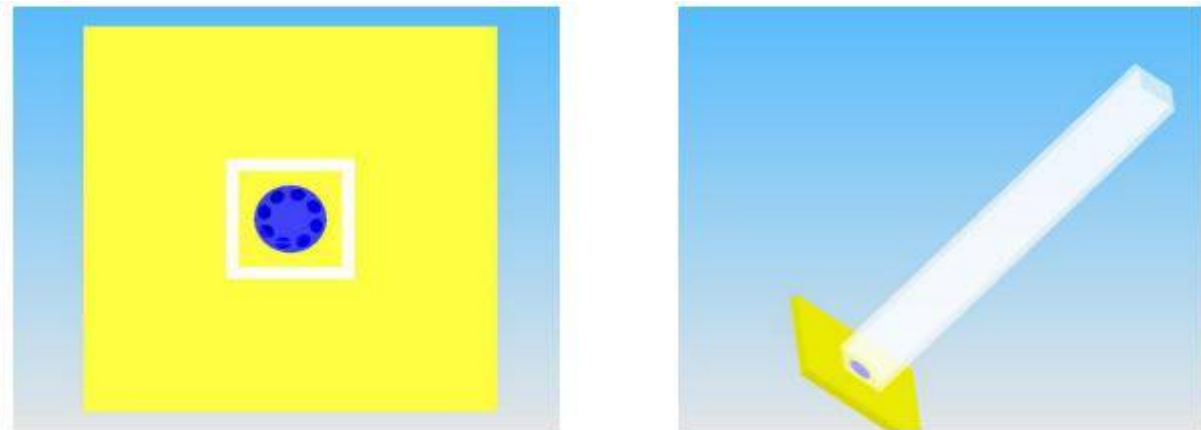


Fig.2 CAD Model of Swirler Assembled in a Test Chamber [5]

The swirler has 22.86 mm inch outlet diameter at the exit and 25.4 mm inner diameter. At the edge of swirler, a step was designed to help mounting the swirler on the swirler holder. Eight 3.175 mm diameter holes were drilled around a 12.7 mm diameter circle on the swirler and are tangential to the circle. Also, each hole has a 35-degree tilt angle relative to longitudinal axis of the swirler.

The combustor chamber was designed to simulate a section of the multi-swirler array. Experiments were conducted in a rectangular combustor chamber of dimension 25.4×25.4×304.5 mm³. Fig.3 shows enlarged view of CAD model showing swirler surface and its surrounding. A full view of this CAD model is shown in the Figure 4

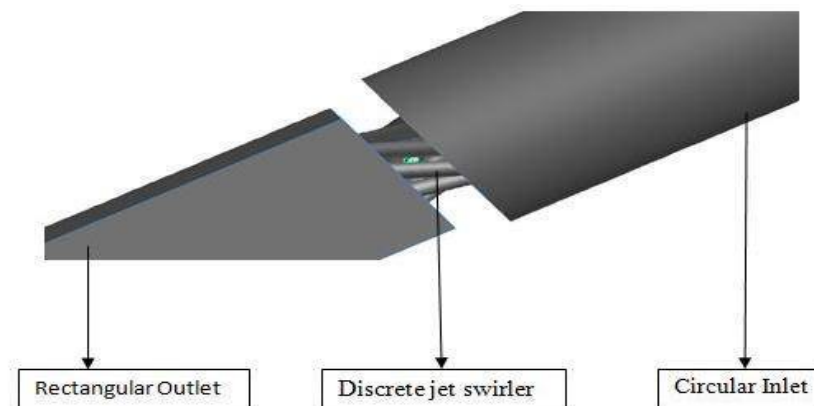


Fig.3 3D Model created using Gambit

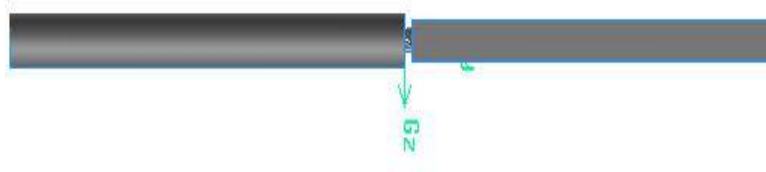


Fig.4 Full View of the Model

IV. GRID GENERATION

The Computation grid has been generated in Gambit 2.4 using hexahedral elements. The grid generated for a full sector model is totally hexahedral with 40 lakh cells. Fig.5 shows the grid generated. The full view of grid generated is shown in Fig.6.

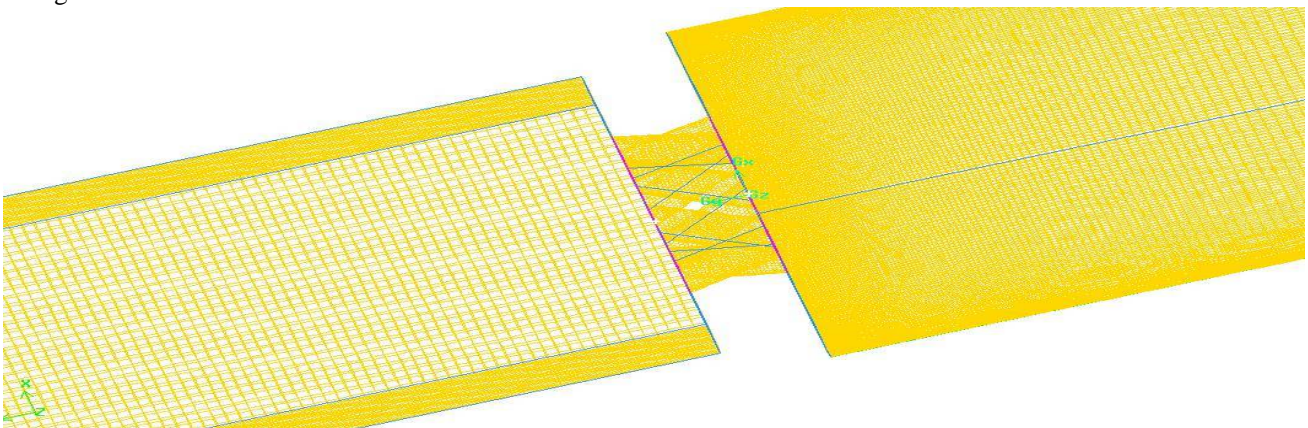


Fig.5 Test Model Mesh created using Gambit

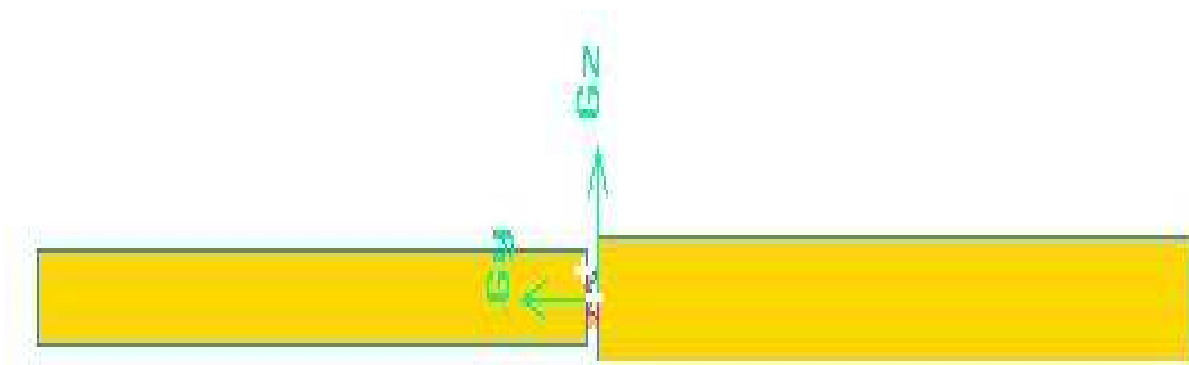


Fig.6 Test Model Geometry created using Gambit

V. BOUNDARY DETAILS

Inlet: Total pressure is specified at the inlet along with turbulent intensity and hydraulic diameter.

Wall: Adiabatic no-slip boundary condition is applied.

Outlet: Static pressure has been applied at the exit of the domain.

VI. SOLUTION AND RESULTS

Steady flow analysis in a discrete jet swirler has been carried out using the standard k-ε model. The results obtained from CFD analysis have been compared with experimental results in terms of velocity contours profiles.

Contour Plots of Axial Velocity

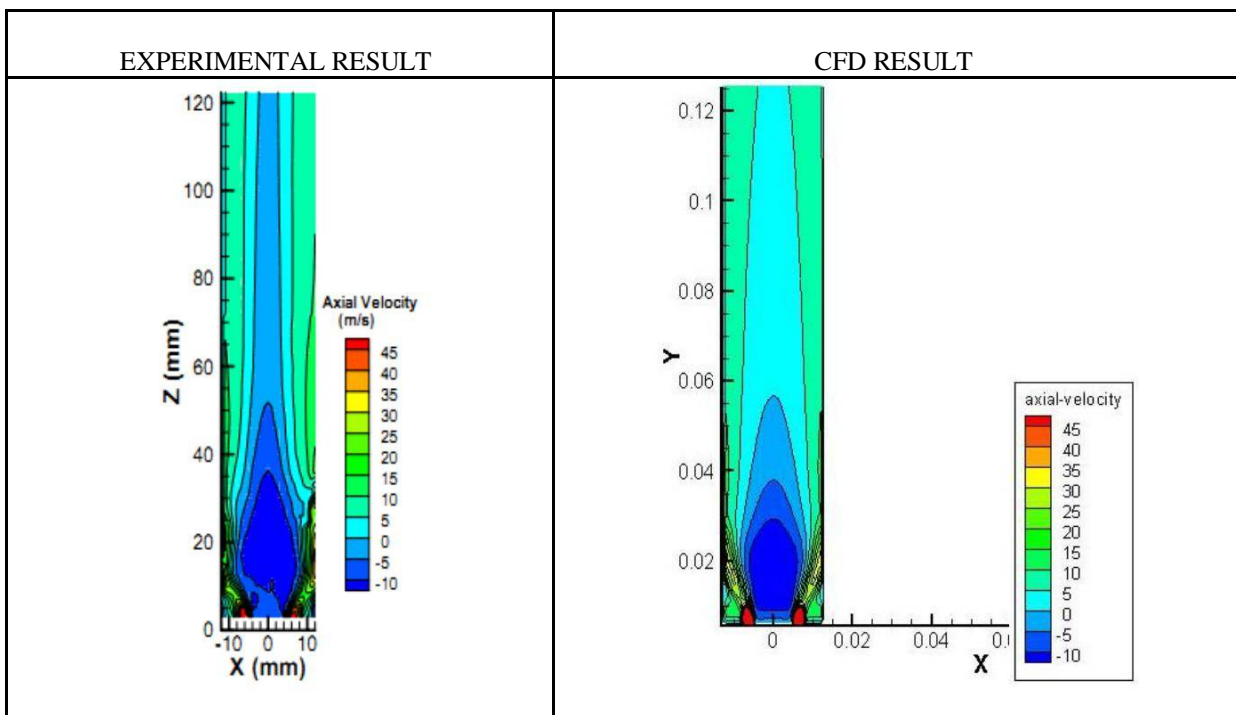


Fig.7 Axial Velocity Contours

Fig.7 shows the comparison of axial mean velocity at z=0 plane. In the figure, the CFD data displayed starts from z=0 Swirler Exit Plane. While the first data measured in LDV measurement is 3 mm downstream of swirler. From Fig.7, comparing experimental results with CFD results standard k-ε model did a reasonably good job in predicting the flow structures of swirling flow. It has predicted the strong central recirculation zone induced by swirling flow. It also predicted the strong outgoing swirling jet at the exit of swirler and strong wall jet after the swirling jet attaches to the wall. In both the results, the recirculation zone can be divided into three different regions: expansion region, constant width region and shrinking region. Comparing the results of CFD and experiments it can be observed that the fluid velocity trends predicted in the recirculation zone are quite accurate, which implies that standard k-ε model is good for predicting these kind of flow features.

Contour Plots of Tangential Velocity

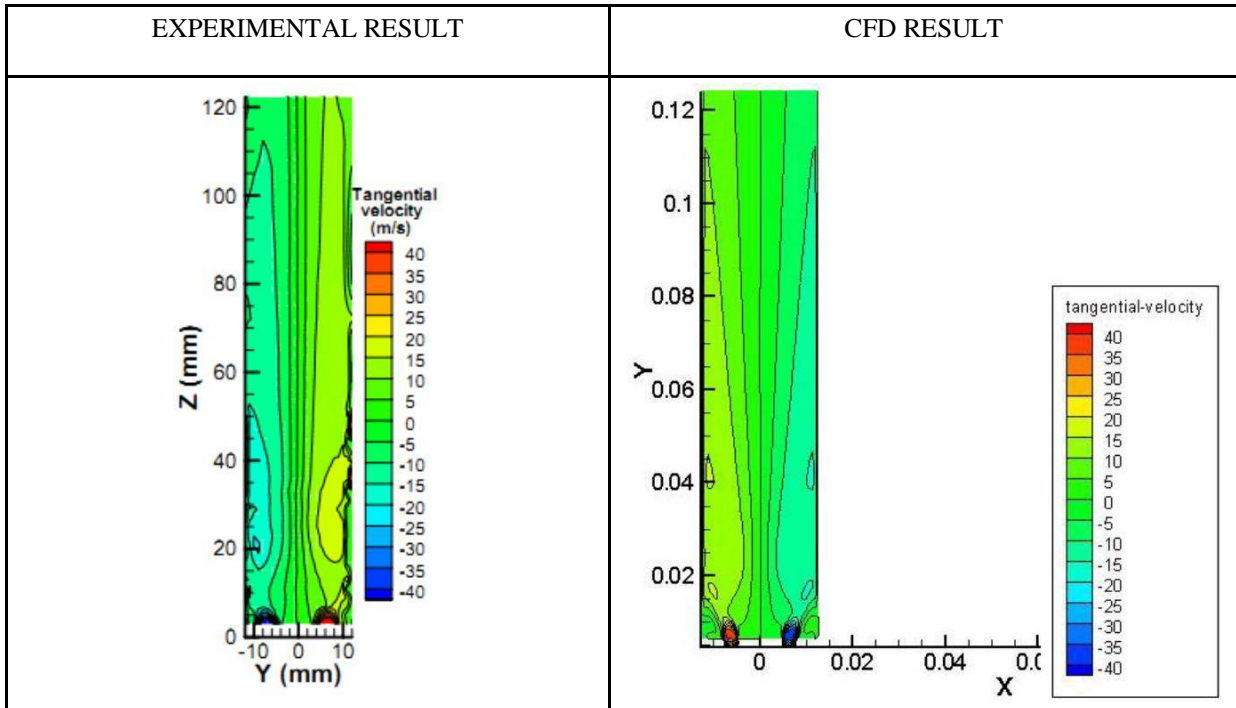


Fig.8 Tangential Velocity Contours

Fig.8 shows the comparison of tangential mean velocity at the cutting plane of $z=0$. Contours of the experimental data show a solid-free combined vortex structure in the location between 20mm to 50 mm downstream, where the swirling jet attaches to wall and the wall jet starts developing. In this region the solid vortex structure dominates the flow field. In respect to this, the standard k- ϵ model also shows a similar trend.

Contour Plots of Turbulent Kinetic Energy

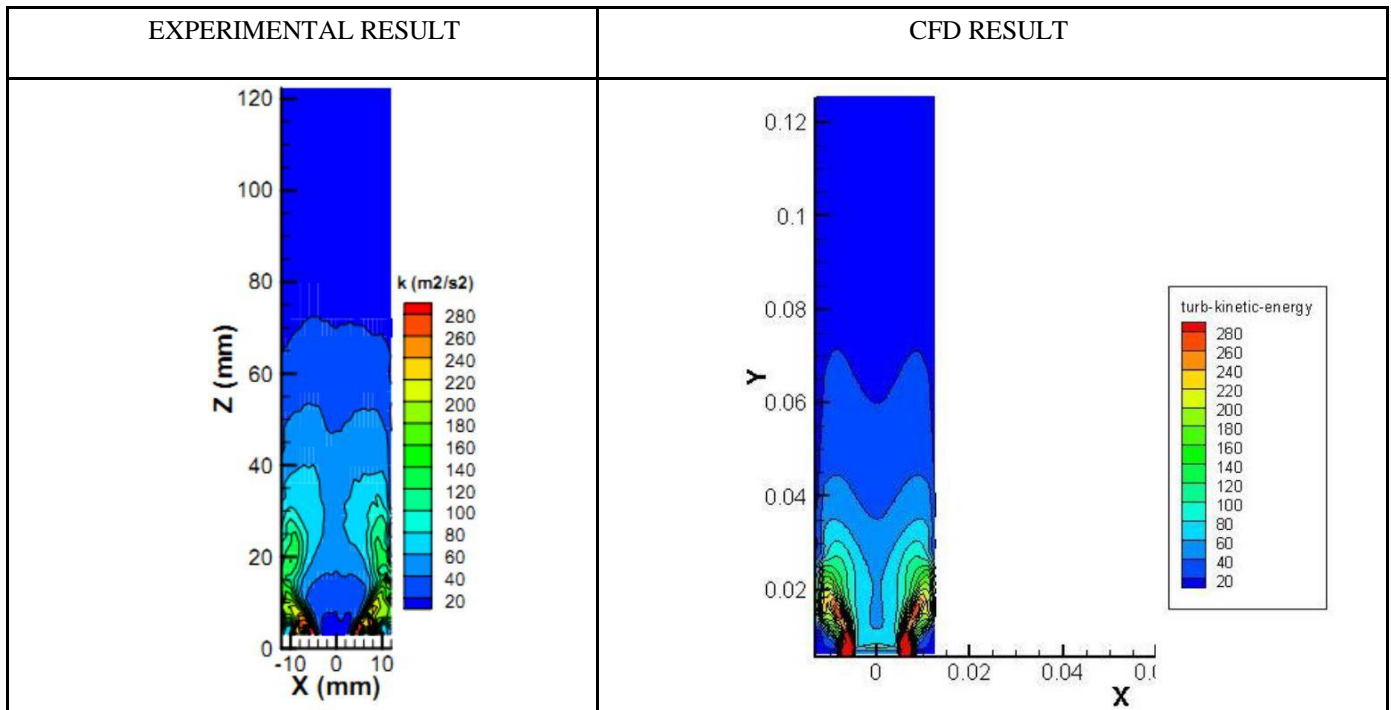


Fig 9 Turbulent Kinetic Energy Contours

Fig.9 shows comparison of turbulent kinetic energy between CFD and experiment on xz plane at z=0. Experimental results show that the turbulent kinetic energy peaks are located at the edge of exiting swirling jets. Subsequent to that, the swirling jets attach to the side wall and a strong turbulent kinetic energy area forms near the side wall due to high velocities and velocity gradients in this area. The standard k-ε model describes this kind of characteristic correctly. It also predicts the turbulent kinetic energy more accurately in the central recirculation zone.

VII. CONCLUSION

In this present work, CFD analysis of a three-dimensional swirling flow field with discrete jets in a confined model combustor has been carried out and the results obtained have been compared with the available experimental values.

REFERENCES

1. Lefebvre, A. H., 1998, "Gas Turbine Combustion", Second Edition, Taylor and Francis.
2. Muthukumar, P. S. and Balakrishna, S. R., 2003, "CFD Analysis of Recirculating Flows Induced by Axial Swirler", International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol.2, Issue 6.
3. Arun Raj, E., Sujith, R. I. and Chakravarthy, S. R., 2009, "Investigation of the Effect of Degree of Swirl on Swirl Flow Characteristic using PIV", 8th International Symposium on Particle Image Velocimetry - PV09, Melbourne, Australia.
4. Ilker and Yilmaz, 2013, "Effect of Swirl Number on Combustor Characteristic in Natural Gas Diffusion Flame", Journal of Energy Resources Technology, Vol. 135/042204-1.
5. Jun Cia, 2008, "Aerodynamics of Lean Direct Injection Combustor with Multi Swirler Arrays", Ph.D Thesis, University of Cincinnati.
6. Khodadadi, J. M. and Vlachos, N. S., 1987, "Effect on Turbulence Model Constants on Computation of Confined Swirling Flows", Vol. 28, Technical Notes, AIAA Journal.
7. Yehia A. Eldrainy, Hossams. Aly and Khalid M., 2009, "A Multiple Inlet Swirler for Gas Turbine Combustor", Word Academy Science, Engineering and Technology, 53, 2009, publication / 242598240.
8. Lester C. Lichty, Combustion Engine Processes, 1967, McGraw-Hill, Inc., Library of Congress 67-10876.
9. Mattingly, Jack D. (2006). Elements of Propulsion: Gas Turbines and Rockets. AIAA Education Series. Reston, VA: American Institute of Aeronautics and Astronautics. ISBN 1-56347-779-3.