

Assessment of Turbulent Boundary Layer Modeling Methods by Using Computational Fluid Dynamics for Gas Turbine Engine Afterburner Diffuser

Bheemaraddi.S.B. Dr.S.Kumarappa

PG Student, Department of Mechanical Engineering, BIET Davanagere, Karnataka, India

Professor and PG Coordinator, Department of Mechanical Engineering, BIET Davanagere, Karnataka, India

Abstract: In the present study we focus on simulating the high temperature subsonic compressible flow in the diffuser duct of an afterburner unit of an air breathing jet engine. The diffuser duct was designed as per the guidelines explained in the standard engine design data hand book and corresponding experimentally measured results are referred for validation purpose. Aerofoil struts were incorporated in the design and the pressure loss due to the struts is assessed by CFD analysis. The CFD analysis is carried out with SST K-Omega Turbulence model and direct resolution of shear stress approach employed in Mentors Shear Stress Transport Turbulence model. The pressure loss computed for diffuser with various struts configurations and compared without strut

Keywords: CFD, Jet Engine, Pressure loss, SST K-Omega, Turbulent Boundary layer.

I. INTRODUCTION

A gas turbine, also called a combustion turbine. The basic operation of the gas turbine is similar to that of the steam power plant except that air is used instead of water. Fresh atmospheric air flows through a compressor that brings it to higher pressure. Jet engine thrust is governed by the general principle of mass flow rate. Thrust depends on two things: the velocity of the exhaust gas and the mass of that gas. A jet engine can produce more thrust by either accelerating the gas to a higher velocity or by having a greater mass of gas exit the engine. To generate the increased power with a more compact engine for short periods of time, an engine requires an afterburner. The afterburner increases thrust primarily by accelerating the exhaust gas to a higher velocity. The turbojet (and the turbofan) can also be fitted with an afterburner. An afterburner is simply a long tube placed in between the turbine and the nozzle in which additional fuel is added and burned to provide a significant boost in thrust. An afterburner or "reheat jet pipe" is a device added to the rear of the jet engine. It provides a means of spraying fuel directly into the hot exhaust, where it ignites and boosts available thrust significantly; a drawback is its very high fuel consumption rate.

An afterburner (or a reheat) is an additional component present on some jet engines, mostly military supersonic aircraft. Its purpose is to provide an increase in thrust, usually for supersonic flight, takeoff and for combat situations. Afterburning is achieved by injecting additional fuel into the jet pipe downstream of (i.e. after) the turbine. The advantage of afterburning is significantly increased thrust; the disadvantage is its very high fuel consumption and inefficiency.

Resolution of Turbulent boundary layer physics for compressible flows in the gas turbine engine combustion systems using Computational Fluid Dynamics(CFD)is a critical task due the presence of flow obstruction devices like exhaust cone struts, fuel spray bars, V-gutter, chutes etc. These obstruction devices will result in strong vortex fluid zones of high turbulence, which enhances the better mixing of air and fuel there by resulting in higher combustion efficiency, however these obstruction devices will also result in higher pressure losses. This low static pressure enhances the power of the turbine and the thermal efficiency whether the gas turbine is used in simple cycle or combined cycle application.CFD simulations are common practice to predict the flow complexity of exhaust diffusers. High-fidelity CFD models can require millions of meshes for which conservation of mass, momentum and energy are calculated in every one of these cells. However, the computational challenge and the mesh load of numerical modelling can be an uneconomical use of resources.

II. MATHEMATICAL MODELS

The commercial FLUENT software package, FLUENT 6.3.26, was used for solving the set of governing equations. The numerical method employed is based on the finite volume approach. Fluent provides flexibility in choosing discretization schemes for each governing equation.

A. Governing Equations

The Governing Navier-Stokes equations for the flow physics considered in this work are written in vector form as,

$$\frac{\partial U}{\partial t} + \frac{\partial G_1}{\partial x} + \frac{\partial G_2}{\partial y} + \frac{\partial G_3}{\partial z} = \frac{\partial G_{1v}}{\partial x} + \frac{\partial G_{2v}}{\partial y} + \frac{\partial G_{3v}}{\partial z}$$

Where,

U is a vector of conserved variables, given by

$$U = \begin{bmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho E \end{bmatrix}$$

G_1, G_2 and G_3 are the Inviscid Flux vectors,

$$G_1 = \begin{bmatrix} \rho u_1 \\ P + \rho u_1^2 \\ \rho u_1 u_2 \\ \rho u_1 u_3 \\ P u_1 + \rho u_1 E \end{bmatrix} \quad G_2 = \begin{bmatrix} \rho u_2 \\ P + \rho u_2^2 \\ \rho u_2 u_1 \\ \rho u_2 u_3 \\ P u_2 + \rho u_2 E \end{bmatrix} \quad G_3 = \begin{bmatrix} \rho u_3 \\ \rho u_3 u_1 \\ \rho u_3 u_2 \\ P + \rho u_3^2 \\ P u_3 + \rho u_3 E \end{bmatrix}$$

G_{1v}, G_{2v} and G_{3v} are the viscous flux vectors given by

$$G_{1v} = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ u_1 \tau_{xx} + u_2 \tau_{xy} + u_3 \tau_{xz} - q_1 \end{bmatrix} \quad G_{2v} = \begin{bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \\ u_1 \tau_{yx} + u_2 \tau_{yy} + u_3 \tau_{yz} - q_2 \end{bmatrix} \quad G_{3v} = \begin{bmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \\ u_1 \tau_{zx} + u_2 \tau_{zy} + u_3 \tau_{zz} - q_3 \end{bmatrix}$$

τ_{xx}, τ_{yy} and τ_{zz} are the normal stresses, given by

$$\tau_{xx} = 2\mu \frac{\partial u_x}{\partial x} + \frac{2}{3}\mu \left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} \right) \quad \tau_{yy} = 2\mu \frac{\partial u_y}{\partial y} + \frac{2}{3}\mu \left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} \right)$$

$$\tau_{zz} = 2\mu \frac{\partial u_z}{\partial z} + \frac{2}{3}\mu \left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} \right)$$

Where μ is the dynamic viscosity coefficient

$$\tau_{xy} = \tau_{yx} = \mu \left(\frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \right) \quad \tau_{yz} = \tau_{zy} = \mu \left(\frac{\partial u_z}{\partial y} + \frac{\partial u_y}{\partial z} \right) \quad \tau_{zx} = \tau_{xz} = \mu \left(\frac{\partial u_x}{\partial z} + \frac{\partial u_z}{\partial x} \right)$$

B. Turbulence models

1. The transport equations for turbulent kinetic energy k and dissipation rate ϵ are written in tensor form as,

$$\left(\frac{\delta}{\delta x_i} \right) (\rho u_i) = \left(\frac{\delta}{\delta x_i} \right) \left(\frac{\mu_{eff}}{\sigma} \right) \left(\frac{\delta}{\delta x_i} \right) + Pk - \rho \quad \left(\frac{\delta}{\delta x_i} \right) (\rho u_i) = \left(\frac{\delta}{\delta x_i} \right) \left(\frac{\mu_{eff}}{\sigma} \right) \left(\frac{\delta}{\delta x_i} \right) + c_1 \left(\frac{-}{k} \right) (P - c_2 \rho)$$

The turbulence generation term P_k can be expressed in tensor notation as,

$$P_k = \mu_t (\delta u_i / \delta x_i) [(\delta u_i / \delta x_j) + (\delta u_j / \delta x_i)]$$

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 3, Issue 1, January 2014

2. Direct Resolution of Boundary Layer Region Approaches (Mentors SST model)

The SST (Shear Stress Transport) model of Mentor (1994) is an eddy-viscosity model which includes two main novelties:

1. It is combination of a k- ω model (in the inner boundary layer) and k- ϵ model (in the outer region of and outside of the boundary layer).
2. A limitation of the shear stress in adverse pressure gradient regions is introduced.

The k- ϵ model has two main weaknesses: it over-predicts the shear stress in adverse pressure gradient flows because of too large length scale (due to too low dissipation) and it requires near-wall modification damping functions/terms.

This model is almost identical to the Mentor baseline model. Only one constant and the expression for turbulent eddy viscosity are different. The two-equation model (written in conservation form) is given by the following equation

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right], \quad \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma}{\nu_t} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho \sigma_\omega \omega}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

C. Grid independence study

Table 1. The grid independence studies are carried out to finalize the mesh which captures the results accurately

| Sl No | Grid size, Number of elements | Y-plus | Total pressure (absolute) at section 6.1, Pascals | | Remarks |
|---|-------------------------------|--------|---|--------|---|
| | | | Experimental | CFD | |
| Afterburner diffuser duct without struts | | | | | |
| 1 | 80031 | 78 | 525249 | 503689 | Y-plus value is high |
| 2 | 135268 | 38 | | 521963 | Y-plus value is acceptable |
| 3 | 157575 | 32 | | 525003 | Mesh finalized for CFD analysis |
| Afterburner diffuser duct with NACA 4412 aerofoil struts(Standard wall function) | | | | | |
| 4 | 219713 | 74 | Not available | 510697 | Y-plus value is high |
| 5 | 259871 | 35 | | 521321 | Y-plus value is acceptable |
| 6 | 286961 | 34 | | 522632 | Mesh finalized for CFD analysis |
| Afterburner diffuser duct with NACA 4412 aerofoil struts(SST K-Omega model) | | | | | |
| 7 | 296520 | 32 | Not available | 509382 | Y-plus value is high |
| 8 | 315230 | 6 | | 520289 | Y-plus value is high |
| 9 | 328010 | 3 | | 517239 | Y-plus value is acceptable, Mesh finalized for CFD analysis |

D. Boundary Conditions

The following boundary conditions have been imposed for CFD analysis,

- A. Inlet: Absolute Total pressure of 529924 Pa and Total temperature of 686.5 Degree Kelvin is imposed at the inlet boundary.
- B. Outlet: Mass flow value of 12.96 Kg/sec corresponding to 45 degree sector is imposed at the outlet.
- C. Wall: The walls of the diffuser duct, aerofoil strut surfaces are imposed with no-slip boundary condition.
- D. Periodicity: Rotational periodicity is imposed on both the periodic surface.

E. Geometrical Modelling and Grid Generation

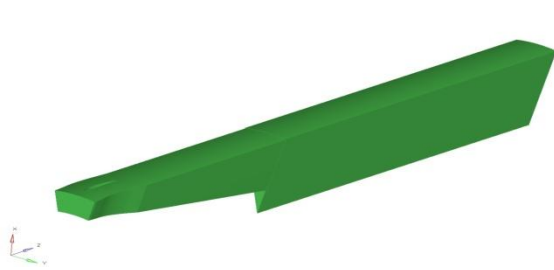


Fig 1. Computational domain of the after burner diffuser duct with struts (45 degree)

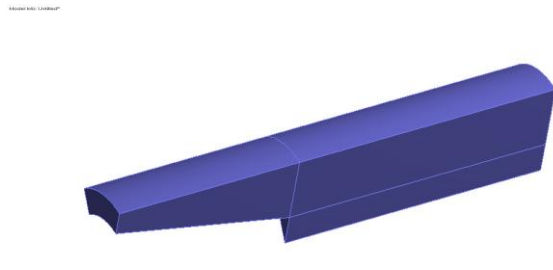


Fig 2. Computational domain of the after burner diffuser duct without struts (45 degree)

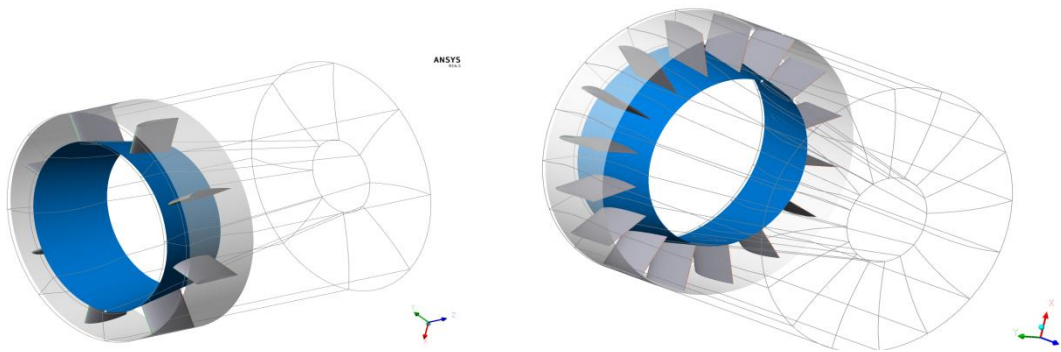


Fig 3. Computational domain of the after burner diffuser duct with struts

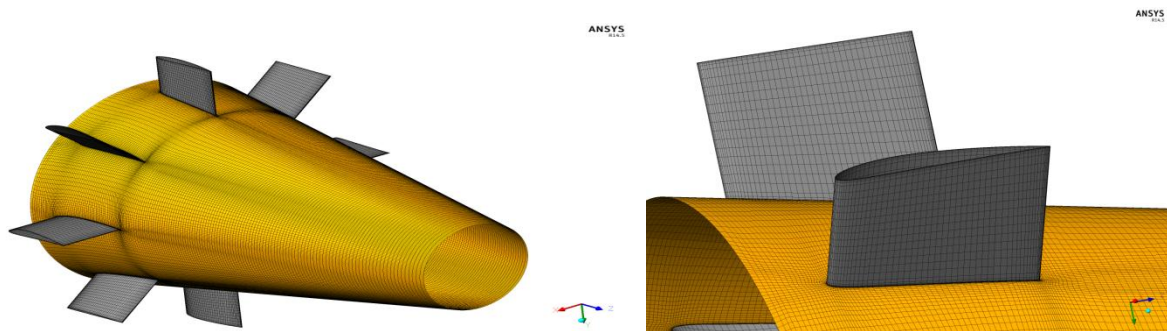


Fig 4. Body fitted grid generated around the Aerofoil Struts

III. RESULTS AND DISCUSSIONS

CFD Analysis is carried out for afterburner diffuser duct without struts and with different number struts for each case. For example here 8, 10,12,14,16 numbers of struts are used on the diffuser duct for each case and pressure loss is computed for every case. Pressure counters are shown below,

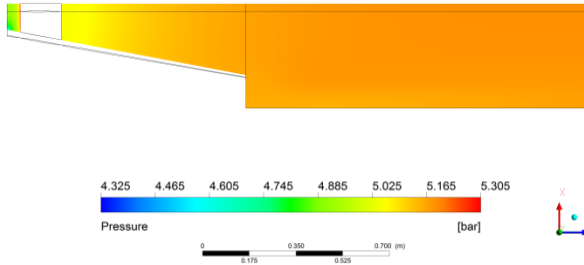


Fig 5. Pressure counters for 8 numbers of struts

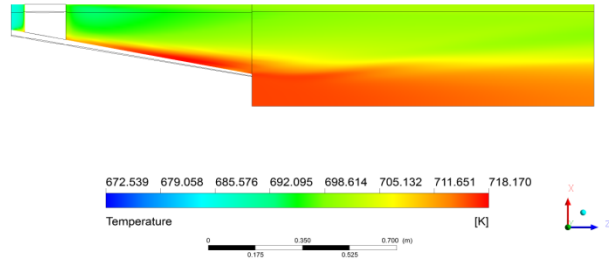


Fig 6. Temperature counters for 8 numbers of struts

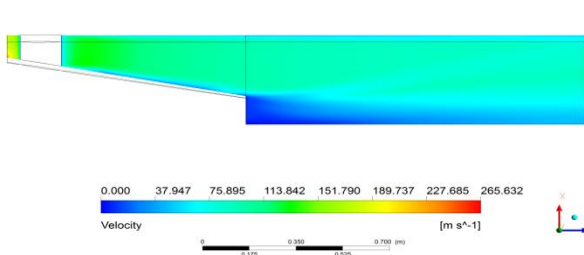


Fig 7. Velocity counters for 8 numbers of struts

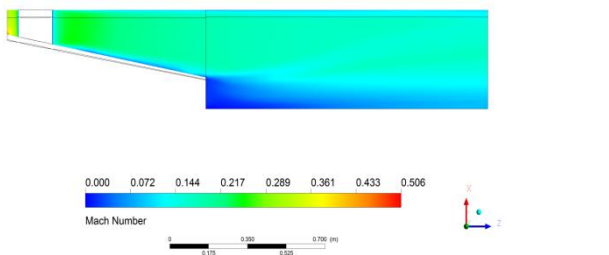


Fig 8. Mach number for 8 numbers of struts

The same procedure is carried out for 10, 12, 14, 16 numbers of struts. Finally pressure loss is found out for each case.

Table 2. Pressure loss comparison

| Experimental Pressure Drop(pa) | CFD Pressure Drop (without struts)(pa) |
|--------------------------------|--|
| 4675 | 4562 |

Table 3. Total pressure drop with Numbers of struts

| Numbers of Struts | Total Pressure drop (Pa) |
|-------------------|--------------------------|
| 0 | 4562 |
| 8 | 15629.04 |
| 10 | 21713.80 |
| 12 | 27735.72 |
| 14 | 29056.16 |
| 16 | 29020.96 |

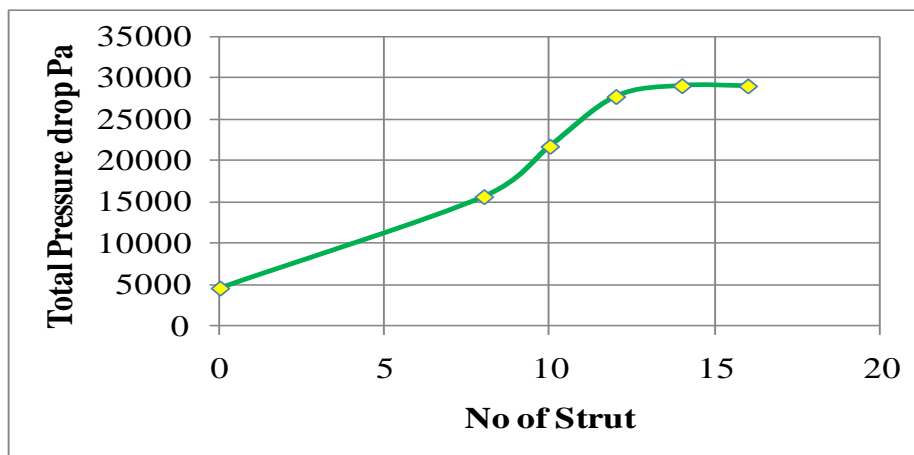


Fig 9. Total pressure drop with Numbers of struts

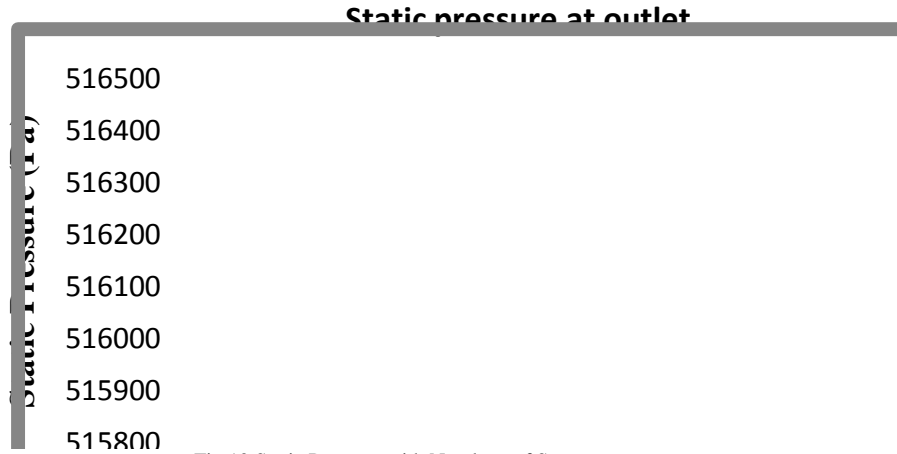


Fig 10. Static Pressure with Numbers of Struts

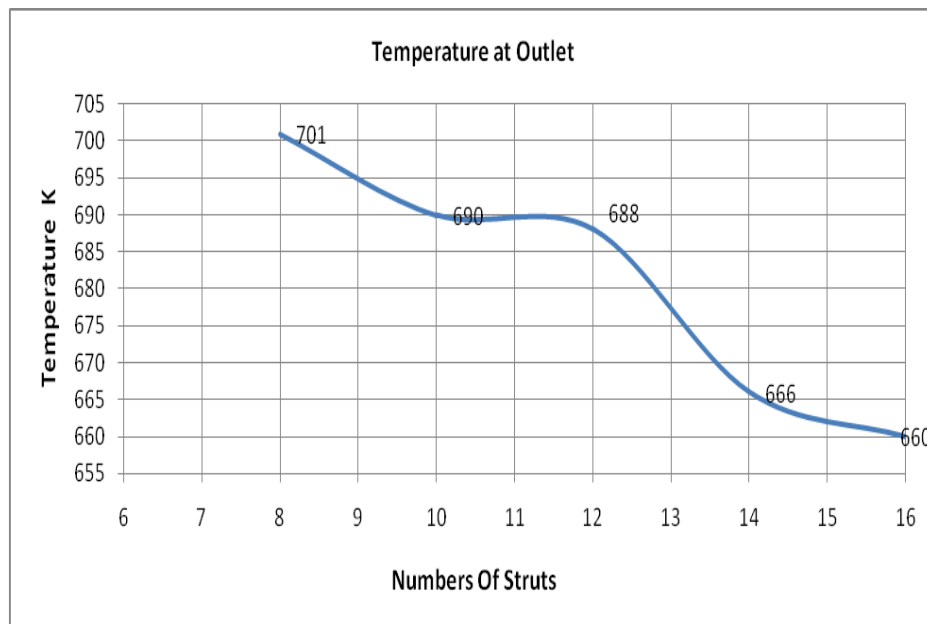


Fig 11. Temperature with Numbers of struts

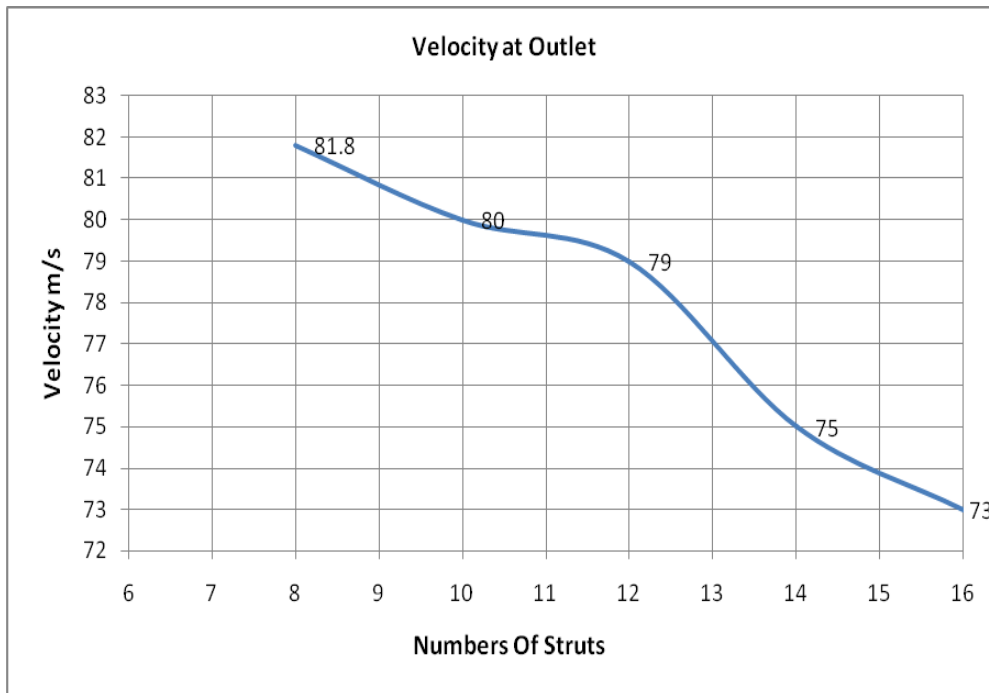


Fig 12. Velocity with Numbers of struts

The experimental pressure drop as per the aircraft design data hand book Jack Mattingly is 4692 Pascal, and pressure drop as per the CFD analysis is 4562 Pascal. This shows that CFD analysis for pressure drop gives nearly accurate values and the prediction of pressure drop by the CFD gives near values. Pressure drop starts attaining constant after numbers of struts 12. For Afterburner with numbers of struts 14 and struts 16 the pressure drop becomes almost equal. So it is clear that as the number of struts increases turbulent kinetic energy increases, it leads to increase in total pressure loss. Pressure loss becomes equal at some point this shown in the figure number 9 during number of struts 14 and 16. The above results and figure number 11 of temperature with numbers of struts shows that, temperature between inlet and outlet of the diffuser decreases as the numbers of the struts increases. For 8 numbers of struts temperature is 701 K, for 16 numbers of struts temperature is 660K. This shows that as the numbers of struts increases temperature decreases. Blockage at the inlet of the diffuser affects the temperature. Since the sharp ends are not favourable for the flow, stream lined centre body has been used. From the velocity plots we can observe the magnitude and direction of velocity in the flow field. The colour code and the corresponding velocity magnitude are given in the left side margin. In a plane along the struts, velocity vectors close to the conical wall are distorted because of the boundary layer development and the struts wakes. we can observe velocity vector variations in struts plane and Magnitude of velocity in the near wall regions is less because of the boundary layer development and in the core regions velocity magnitude is high. Struts and their wakes reduce the flow area in the region close to the struts. As the flow develops the velocity reduces due to diffusion. In the stream wise direction, the boundary layer thickness increases because of adverse pressure gradient and also the velocity gradient near the wall. The boundary layer characteristics on both the walls are different. Hence, the peak value of velocity may not occur at the centre line region. The magnitude of maximum velocity value shifts from the wall region due to boundary layer. From the inlet to the outlet of the diffuser mean velocity reduces and this reduction in velocity is converted into pressure rise.

IV. CONCLUSION

CFD analysis is carried out for the after burner diffuser duct with and without struts. CFD analysis results for the after burner diffuser duct without struts is compared with the Experimental data. The SST k-omega model is found to be a satisfactory physical model to give good predictions for various flow characteristics of an annular diffuser. Total Pressure loss is well predicted by the numerical model which has close agreement with experiment. The diffusion in the diffuser with struts is interrupted by the reduction of the flow area due to the struts and their wakes and this is constant after number of blades or struts are 12. The pressure loss is significantly increased by the presence of struts. Pressure

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 3, Issue 1, January 2014

recovery development in the diffuser without struts is higher than the pressure recovery in the diffuser with struts. The effectiveness of the diffuser with aerofoil shaped struts is more than the diffuser without struts even though the blockages are same. Thus struts play an influential role in the diffuser performance. The present study indicates that the prediction procedure used in the paper can be used with confidence in the development of diffusers for the gas turbine afterburners.

REFERENCES

- [1] Dr.N.Mohammed Sheriff et.al,” CFD analysis of flow in Afterburner”, Proceedings of the 6th WSEAS International Conference on HEAT and MASS TRANSFER (HMT'09)
- [2] Yogesh TVet.al “Effect of Exhaust Diffuser on Gas Turbine After Burner Diffuser performance “Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power December 16-18, 2010, IIT Madras, Chennai, India.
- [3] N.Maheshwara Reddy, E G Tulapurkara and V Ganesan”Optimization of the fuel manifold location inside a jet engine afterburner” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power December 16-18, 2010, IIT Madras, Chennai, India.
- [4] Samy M. El-Behery, and Mofreh H. Hamed,” A Comparative Study of Turbulence Models Performance for Turbulent Flow in a Planar Asymmetric Diffuser”, World Academy of Science, Engineering and Technology 53 2009
- [5] Vance Dippold, III,” Investigation of Wall Function and Turbulence Model Performance within the Wind Code”, 43rd AIAA Aerospace Sciences Meeting and Exhibit, 10 - 13 January 2005, Reno, Nevada
- [6] Jack.D.Mattingley et.al, “Aircraft Engine Design”AIAA Education series, 2001
- [7] C.VanErtbruggen, P.Corieri, R.Theunissen, M.L.Riethmuller,C.Darquenne.”Validation of CFD prediction of flow in a 3D alveolated bend”2000.
- [8] NasserAshgriz and JavadMostaghimi “Introduction to computational fluid dynamics” 1998.
- [9] Versteeg and Malalsekera “An Introduction to Computational Fluid Dynamics” 1995
- [10] KampaAshirvadam “Combustion instability screech in gas turbine afterburner” Department of aerospace engineering IISc 2007.
- [11] KouichiSHIZAKA,SusumuWAKAZONO,Masanori YURI and Ronald TAKAHASHI “ CFD studies of Industrial gas turbine exhaust diffusers” proceedings of gas turbine conference 1999.
- [12] W.A.Sirigano and F,Liu “Performance Increases for Gas-Turbine Engines Through Combustion Inside the Turbine”vol 15,January February 1999
- [13] M.J Zucrowvolume 1 “ Aircraft and missile propulsion” 2003
- [14] SM.Yahya,”Fundamentals of Compressible flow “New Age International Publishers, 2003