iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** VIII **Month of publication:** August 2017

DOI: http://doi.org/10.22214/ijraset.2017.8306

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

Internal Flow Analysis of a Dump Diffuser Using CFD

Jagadish Hungund[1], Anandkumar S Malipatil[2]
Dept of Thermal Power Engineering, VTU Regional Centre Kalaburagi, Karnataka, India.

Abstract: Internal flow analysis of a dump diffuser used in marine gas turbines and modern aircraft engines has been carried out. Analysis has been carried out using k-ε turbulence model in ANSYS FLUENT. Modelling has been done in UNIGRAPHICS with suitable dimensions used in the previous papers. Bluff body is placed in the diffuser to distract the fluid flow. Suitable governing equations are solved. With suitable format model prepared is imported to ANSYS. Meshing is done in ANSYS. Tetrahedral type of meshing is carried out. Fine mesh is preferred for more accurate results. Different diffuser angles are used for fluid flow analysis. Suitable boundary conditions are used for this analysis. This is a cold flow analysis. We concluded that there will be no effect of diffuser when the diffuser angle is increased. If the velocity is not decreased in the diffuser the high velocity fluid damages the inner walls of the gas turbine and hence reducing the efficiency of the turbine. Ideal diffuser angle has been found in which all the requirements suits good for the performance of the diffuser. Effects of flow recirculation and boundary layer separation have been discussed. Four different planes have been created for the representation of different contours. Pressure contours, velocity contours, turbulence kinetic energy contours, streamlines and velocity vectors are represented on the planes created on the diffuser. Velocity versus diffuser angle, Pressure versus diffuser angle and turbulence kinetic energy versus diffuser angle graphs has been plotted. Validation has been carried out for the present results and the previous results.

Keywords: Dump diffuser, Diffuser angle, Pressure contours, Velocity contours, Turbulence kinetic energy.

I. INTRODUCTION

A diffuser is a contraption for diminishing the speed and expanding the pressure gradient of a liquid going through a structure. Diffusers are utilized to moderate the liquid's speed while expanding its pressure gradient. The liquid's static weight ascends as it passes however a conduit is regularly alluded to as pressure recuperation. Interestingly, a spout is frequently expected to expand the release speed and lower pressure while coordinating the stream in one specific heading. Frictional impacts amid examination can now and again be vital, yet as a rule they are ignored. Conduits containing liquids streaming at low speed can more often than not be investigated utilizing Bernoulli's guideline. Investigating channels streaming at higher speeds with mach numbers in overabundance of 0.3 normally require compressible stream relations.

A Sheeba and V Ganesan[1] introduced numerical and test examination of a straight centre annular diffuser normally utilized as a part of marine gas turbine combustor. The stream in the annular diffuser has been recreated by comprehending the conditions of preservation of mass and energy. The standard K-ε turbulence show is utilized for examination. A five gap test and hot wire anemometer were utilized to quantify mean and fluctuating segments of speed. The examination of the trial and anticipated mean speed esteems in outspread course, at various hub areas of annular diffuser is introduced and talked about. Stream improvement in the annular diffuser with various strut geometries has been investigated. V R Sanal Kumar, Santosh kumar sahoo and S Raghunathan[2] dealt with geometrical enhancement of dump diffusers in present day flying machine motors. They completed the examinations utilizing SST K-w turbulence demonstrates. In this full verifiable limited volume plan of compressible, Reynolds arrived at the midpoint of, navier stokes condition are utilized. Numerical stream recreation is done in straight walled pre diffuser with two diverse divider geometries $\alpha1=10$ and $\alpha2=2$, later again they completed with various geometries $\alpha1=5$ and $\alpha2=4$, different measurements are kept same. In both cases static pressure recuperation is higher for $\alpha2=4$ contrasted with $\alpha2=2$. Leilei xu, Yue huang, Can ruan and Peiyong wangand Fri Ying[3] utilized a glorified physical model of dump diffuser with straight walled pre diffuser utilizing the base valuesof run of the mill diffuser. The stream speed was 0.25m/s. The test office utilized water as medium. The PIV framework comprises of quasic laser light source, optics focal point with reflect, one CMOS camera seeding particles, and PIV programming. V R Sanal kumar, A sameen, HD Kim, T setoguchi, S mastuo and S raghunathan[4] taken a shot at numerical re-enactment with the assistance of k-w turbulence show. Parametric investigations were finished utilizing dump diffuser with decrease walled fire tube. In this paper low delta speed v1=93m/s has been chosen. Dump hole proportion (d/h2) of the diffuser are fluctuated viz 0.43 and 1.3. These qualities are named as case 1 and case 2. On the off chance that 1 (1.3) speed profile is allegorical in nature. On the off chance that 2 speed profile is invert illustrative in nature. It was seen that at little estimations of dump hole proportions the fire tube head incites stream arch at the pre diffuser leave plane, hence avoiding the stream towards the dividers. This outcome in a much uniform outlet speed profile and restrains division. G Ananda Reddy and V Ganesan[5] depicted the endeavours made in the advancement of pre-diffuser geometry of an air gas turbine burning chamber for getting least aggregate weight misfortunes and for good static weight recuperation. Advancement of pre-diffuser has been attempted by changing the geometric parameters like length, disparity point and divider shape. A 20° area has been considered for the progressing study due to the periodicity as for the air impact atomizer. John M Smith and Albert J Juhasz [6] researched the execution of a short annular dump diffuser with enhanced stream extension by utilizing suction to give stable vortices. The examination was completed in test office stream framework. Encompassing temperature air is passed to the framework by remotely set compressor station. The weight of the air is 1 super pascal total. CR Fishenden and ST Stevens [7] introduced the consequences of an exploratory examination of the execution of an annular combustor dump diffuser. The geometry utilized was straight walled pre diffuser taken after by sudden extension in which stream partitions and goes to the two concentric annuli encompassing the stream tube. Tests were taken to discover the impact of pre diffuser geometry, separate between two concentric annuli. Prakash Ghose, Amitava Datta and Achintya Mukhopadhyay[8] considered axi symmetric dump diffuser with turbulent isothermal stream. This investigation was performed by utilizing k-€ show with and without whirl. Diverse arch shapes have been taken for the recreation, viz hemispherical, curved and vertical circular. Prakash Ghose, Amitava Datta and Achintya Mukhopadhyay[9] performed numerical investigation in an axi symmetric diffuser , to dissect the stream structure and weight recuperation in the model. Stream structures are numerically anticipated with various pre diffuser edges and channel whirl levels. Streamline conveyance and weight plots on the packaging and liner dividers are examined.

For better design parameters, in this paper, closely approximated computational fluid dynamic analysis has been carried out to demonstrate the effect of diffuser angle on the fluid flow. Pressure, velocity and turbulence kinetic energy contours have been represented.

II. COMPUTATIONAL DETAILS

A three dimensional model of a dump diffuser is modelled using UNIGRAPHICS with suitable geometries. Figure 1 shows the modelled diffuser with mesh. Meshing is carried out in ANSYS. Mesh generated is tetrahedral type mesh. It is found that number of nodes are 13264 and number of elements are 66313. Figure 2 shows the schematic cross sectional view of straight core diffuser. It contains a bluffed body at the center.

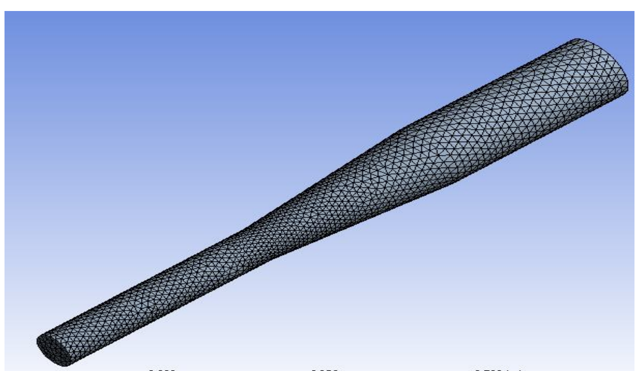
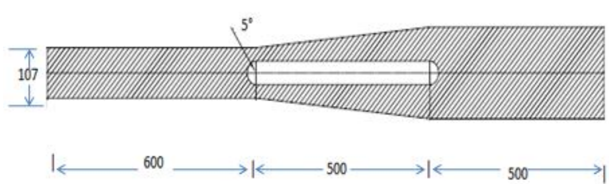


Fig.1 Modelled geometry of a diffuser with mesh



International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

Fig.2 Cross sectional view of diffuser with bluff body

A. Boundary Conditions

Boundary conditions are incorporated in ANSYS FLUENT. This study has three boundary conditions they are inlet, outlet and wall boundary.

At the inlet, velocity specification method is taken as magnitude and normal to boundary. Velocity magnitude is taken as 20 m/s. Turbulent intensity is taken as 5% and turbulent viscosity ratio is taken as 10.

At the outlet pressure and velocity specifications are not required. Backflow turbulent intensity is kept 5% and backflow turbulent viscosity ratio is kept 10.

At the wall boundary, the walls are assumed to be adiabatic with no slip condition. The wall is taken as stationary wall and shear condition as no slip. Wall roughness is taken as 0.5.

B. Governing Equations

It is somewhat difficult to choose which model is best for the analysis, nevertheless, with the help of previous works on the diffuser, in this analysis we have chosen standard k-ε turbulence model for the simulation. It is a two equation model. Flow is simulated by using equations of conservation of mass and momentum. Finite volume method is used to convert governing equations to algebraic equations. Transport equations for standard k-ε model are

For turbulent kinetic energy k

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

For dissipation ϵ

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(P_k + C_{3\epsilon}P_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

Turbulent viscosity

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

Production of k

$$P_k = -\rho \overline{u_i' u_j'} \frac{\partial u_j}{\partial x_i}$$

$$P_k = \mu_t S^2$$

Where S is the modulus of the mean rate of strain tensor, defined as:

$$S = \sqrt{2S_{ij}S_{ij}}$$

III. VALIDATION

For the validation velocity contour is taken into account. Inlet velocity of the air in the diffuser is taken as 20 m/s. Flow is considered as steady and incompressible flow. The experimental results are compared with present numerical results. Velocities at different lengths of the diffuser are plotted. Velocity versus length graph is plotted. Thus we can ensure that flow is turbulent as the Reynolds number is 1.465×10^5, at the inlet and develops under unfavourable pressure gradient i the diffuser. The total flow in the diffuser is incompressible.

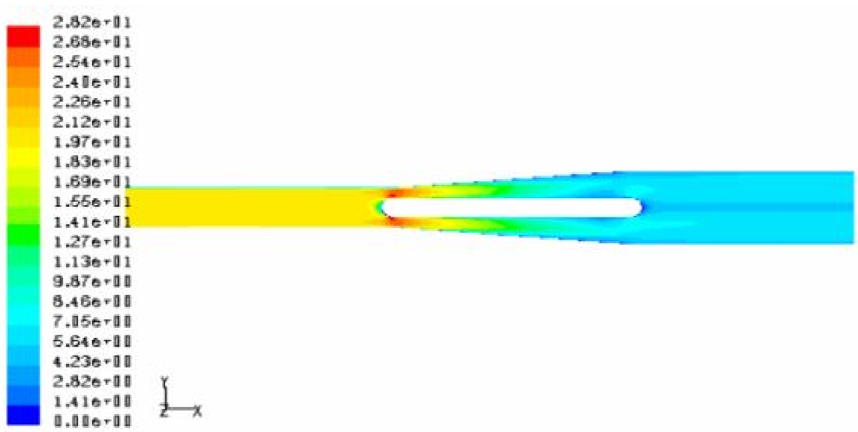


Fig.3 Axial velocity along mid plane at 20 m/s inlet velocity

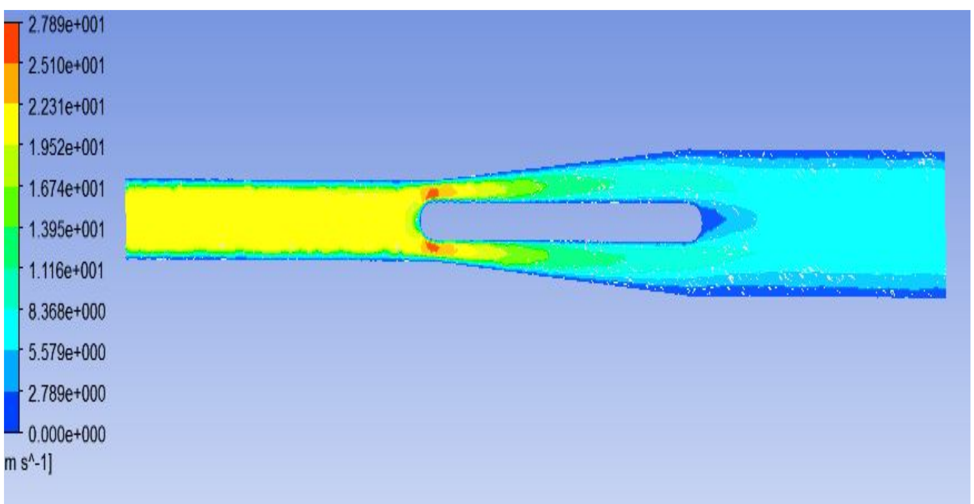


Fig.4 Axial velocity along mid plane at 20 m/s inlet velocity

TABLE I

EXPERIMENTAL AND CFD RESULTS

Experimental Results		CFD Results	
Velocity, m/s	Length, mm	Velocity, m/s	Length, mm
2.23	100	2.26	100
1.95	600	1.97	600
1.67	1100	1.63	1100
1.39	1600	1.41	1600

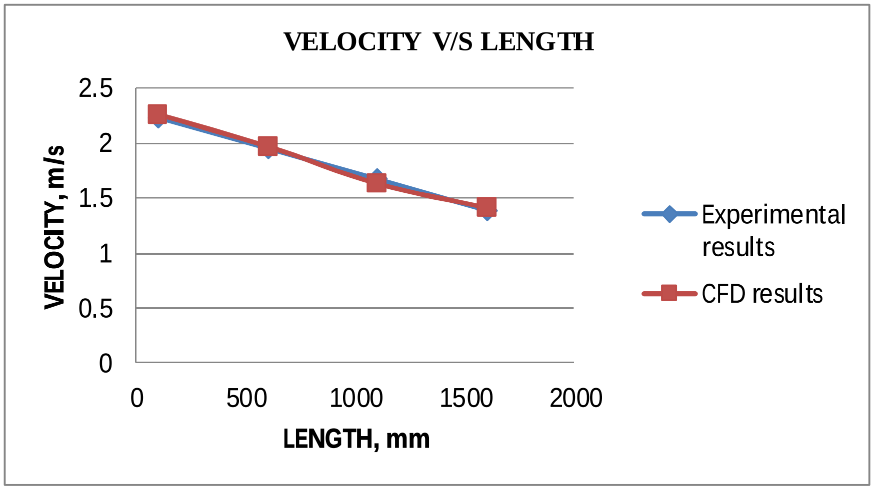


Fig.5 Axial velocity versus length graph

IV. RESULTS AND DISCUSSION

In this paper attention is focussed mainly on diffuser angle. The diffuser angle is changed and other geometries are kept constant. Different diffuser angles taken are 5°, 10°, 16°, 20° and 22°. The effect of diffuser that is decrease in velocity and increase in static pressure is taken into account and accordingly diffuser angles are picked. As the diffuser angles is below 5° and more than 22° the effect of diffuser will be absent. Pressure contours, velocity contours, turbulent kinetic energy contours, streamlines and velocity vectors are represented on the different planes created.

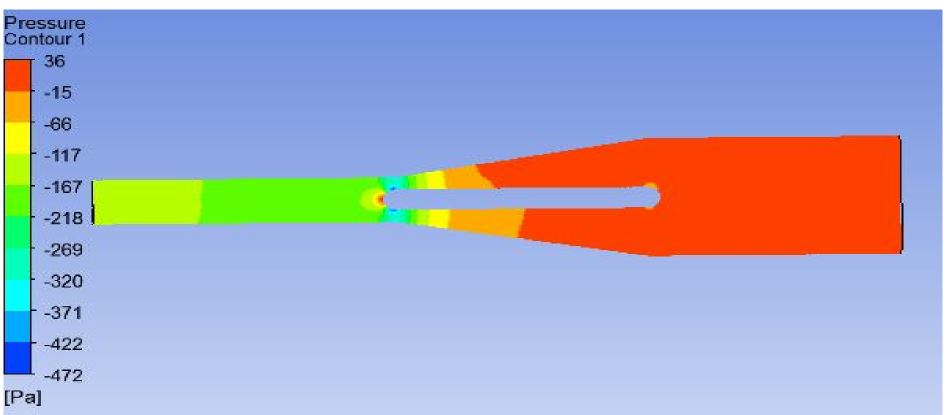


Fig.6 Pressure contour along plane at 10° diffuser angle

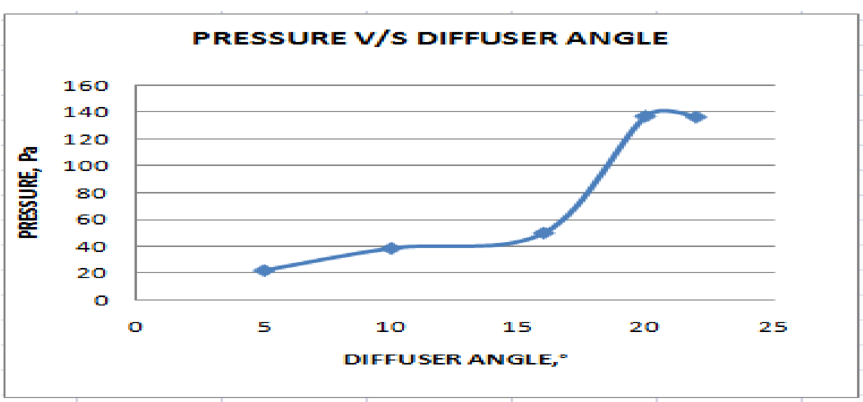


Fig.7 Pressure versus diffuser angle

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

Figure 7 shows the pressure versus diffuser angle plot. Pressures obtained during analysis for different diffuser angles are plotted. Pressure gradually increases for every diffuser angle change. From 5° diffuser angle to 22° diffuser angle pressure increases. This gradual increase can be seen in the graph plotted. Due to the flow recirculation and boundary layer separation the pressure gradient increases while the velocity of the fluid decreases. Boundary layer separation takes place when the flow near the wall or edge re circulates or reverses in the diffuser.

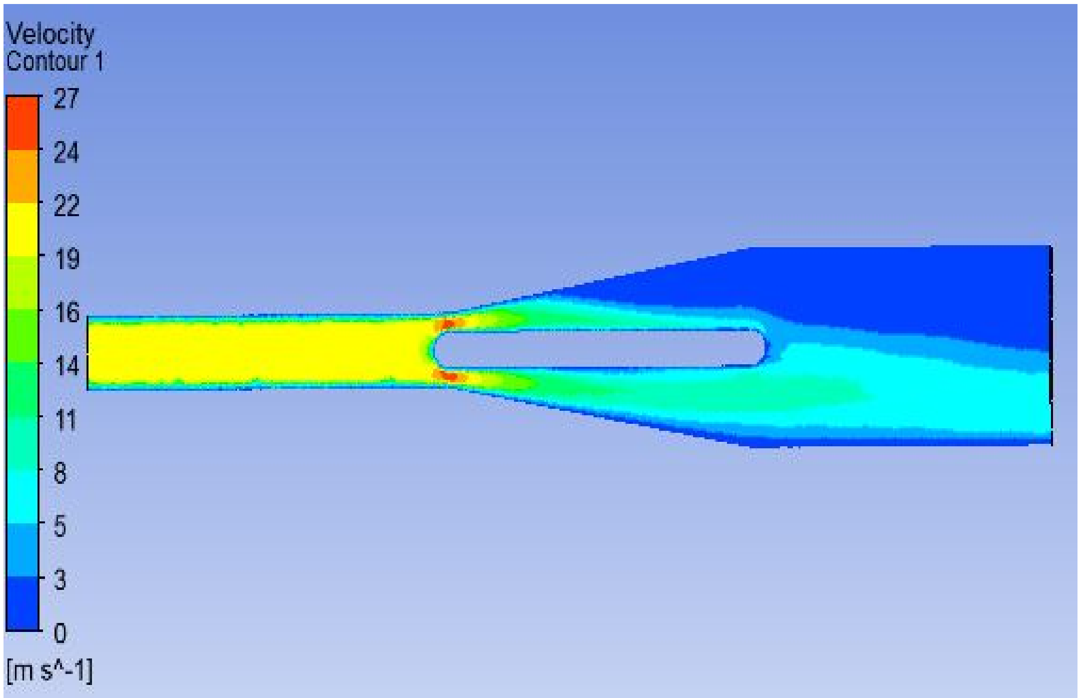


Fig.8 Velocity contour along plane at 10° diffuser angle

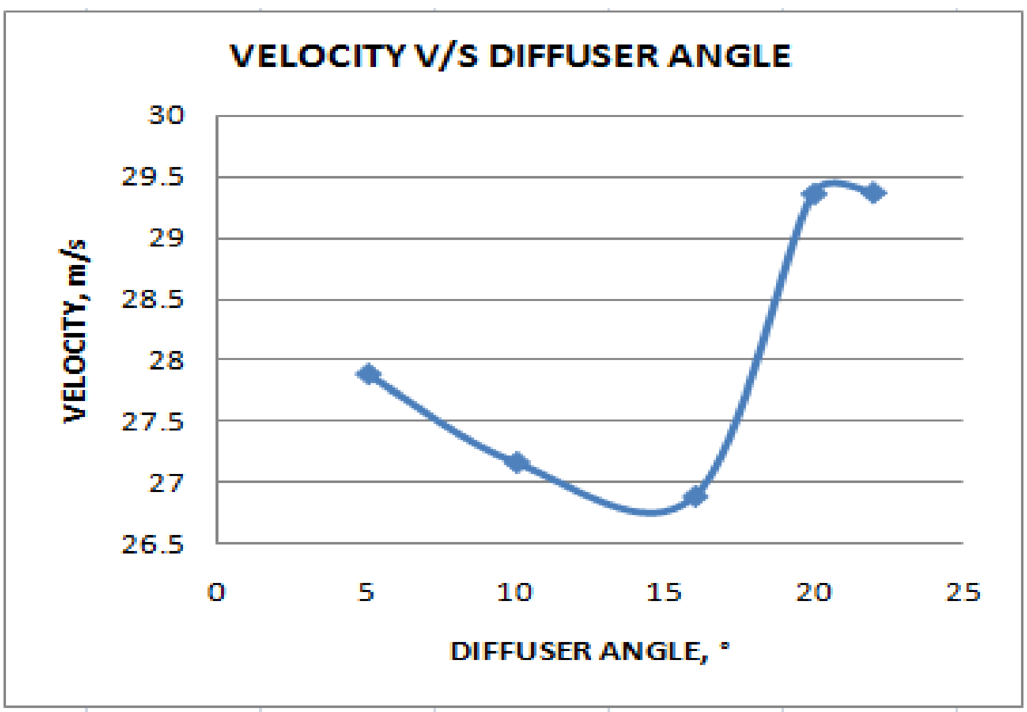


Fig.9 Velocity versus diffuser angle

Figure 9 shows the velocity versus diffuser angle plot. Velocities obtained during analysis for different diffuser angles are plotted. From diffuser angle 5° to 16° the velocities goes on decreasing. From 16° to 22° the velocity keeps on increasing. From this we can tell that diffuser angle more than 16° no more behaves like a diffuser. Effect of diffuser is no more after 16°. Due to the boundary separation the velocity gradient decreases or sometimes goes to negative velocity gradient. Boundary layer separation depends on the distribution of undesirable velocity gradient along the surface. Due to flow recirculation the velocity in the diffuser reduces.

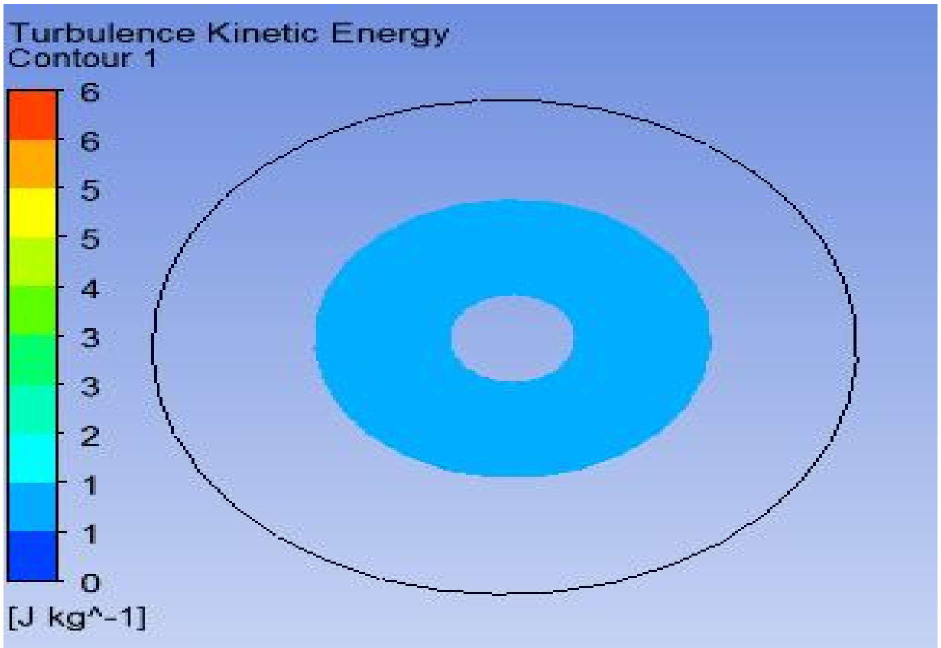


Fig.10 Turbulence kinetic energy along plane at 10° diffuser angle

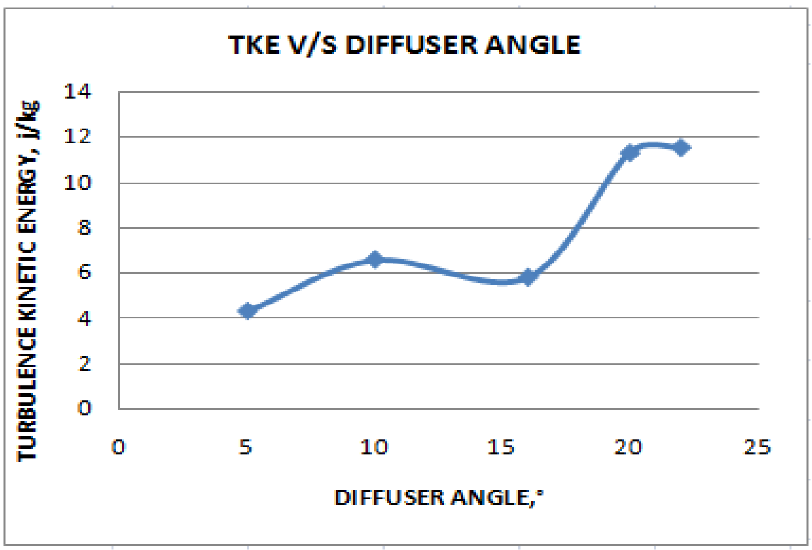


Fig.11 Turbulence kinetic energy versus diffuser angle

Figure 11 shows turbulence kinetic energy versus diffuser angle plot. Turbulence kinetic energy obtained for different diffuser angles are presented on the graph. Turbulence kinetic energy increases as the diffuser angle increases. From 10° diffuser angle to 16° diffuser angle the turbulence kinetic energy decreases slightly and for diffuser angle more than 16° turbulence kinetic energy increases. It is associated with eddies as the flow in the diffuser is turbulent. As the eddies produced near the walls and edges increases the turbulence kinetic energy also increases.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

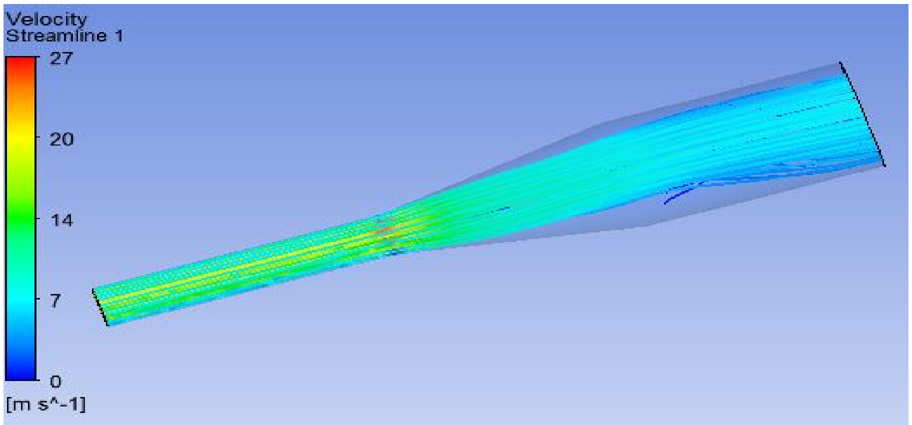


Fig.12 Streamlines along plane at 10° diffuser angle

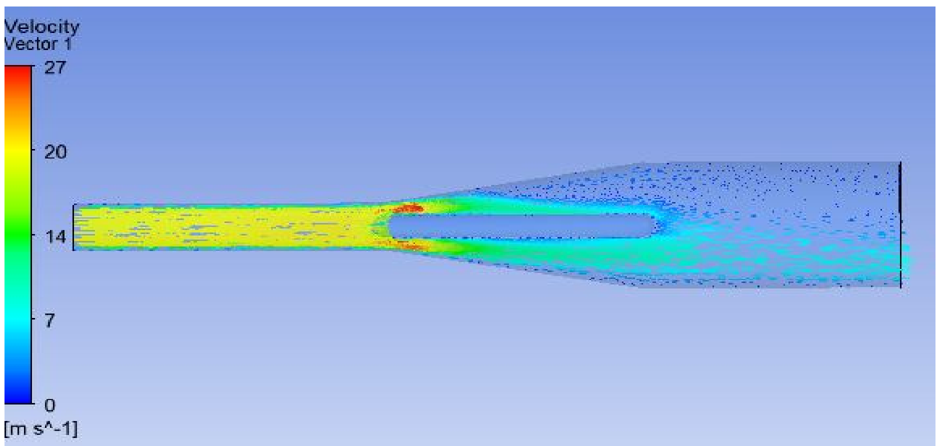


Fig.13 Velocity vectors along plane at 10° diffuser angle

From the above figures and analysis for the 10° diffuser angle the maximum velocity is 27.17 m/s, pressure is 38.75 Pa and turbulent kinetic energy is 6.55 J/kg. Similarly analysis has been carried out for different diffuser angles. Streamlines and vectors are also represented for all the diffuser angles. Due to the flow recirculation and boundary layer separation the velocity gradient of the fluid decreases and pressure gradient increases in the diffuser.

V. CONCLUSIONS

A. Computational fluid flow analysis is carried out for annular diffuser with angles ranging from 5,10,15,20 and 22 degrees.
B. The present analysis examines the detailed insight of flow patterns inside the diffuser.
C. Plots of Velocity, Pressure, Turbulent Kinetic energy and streamlines coloured by velocity is plotted to understand the effect of diffuser angle on the flow parameters for the fixed inlet velocity.
D. It is concluded from the analysis that pressure at the diffuser outlet raises as the diffuser angle increases from 5 to 20 degree and remains same for angle above 20 degrees.
E. It is found that higher the diffuser angle higher the Turbulence in the diffuser.
F. It is also found that velocity at the outlet reduces for the angle 5 to 15 degree and there after increases for 20 and 22 degrees.

VI. ACKNOWLEDGMENT

My special gratitude to my guide Mr. Anandkumar S Malipatil, Assistant professor, Dept. of PG Studies, VTU Regional Office, Kalaburagi, for his inspiration constant supervision, direction and discussions in successful completion of this project. I thank my Parents whose sacrifice in all respect from time to time with love and affection has made me to reach here. I also express my

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VIII, August 2017- Available at www.ijraset.com

immense thanks to Dr. Baswaraj Gadgay, Special Officer, VTU Regional Office, Kalaburagi for his help and encouragement during the tenure of the course.

REFERENCES

[1] A Sheeba and V Ganesan 'Modelling of annular pre diffuser for marine gas turbine combustor using CFD' journal of naval architecture and marine engineering (2005), 1813-8535.

[2] V R Sanal Kumar, Santosh Kumar Sahoo and S Raghunathan 'Internal flow simulation of dump diffusers for modern aircraft engines' AIAA 2009-4832.

[3] Leilei xu, Yui huang, Can ruan and Peiyong wangand fri ying 'Study of the dump diffuser optimization for gas turbine combustors' Elsevier , procedia engineering , volume 99, 2015, page 828-834.

[4] V R Sanal Kumar, A Sameen, HD Kim, Tsetoguchi, S mastuo and S Raghunathan' Influence of dump gap on the performance characteristics of dump diffusers' ISTP-16, 2005, Prague.

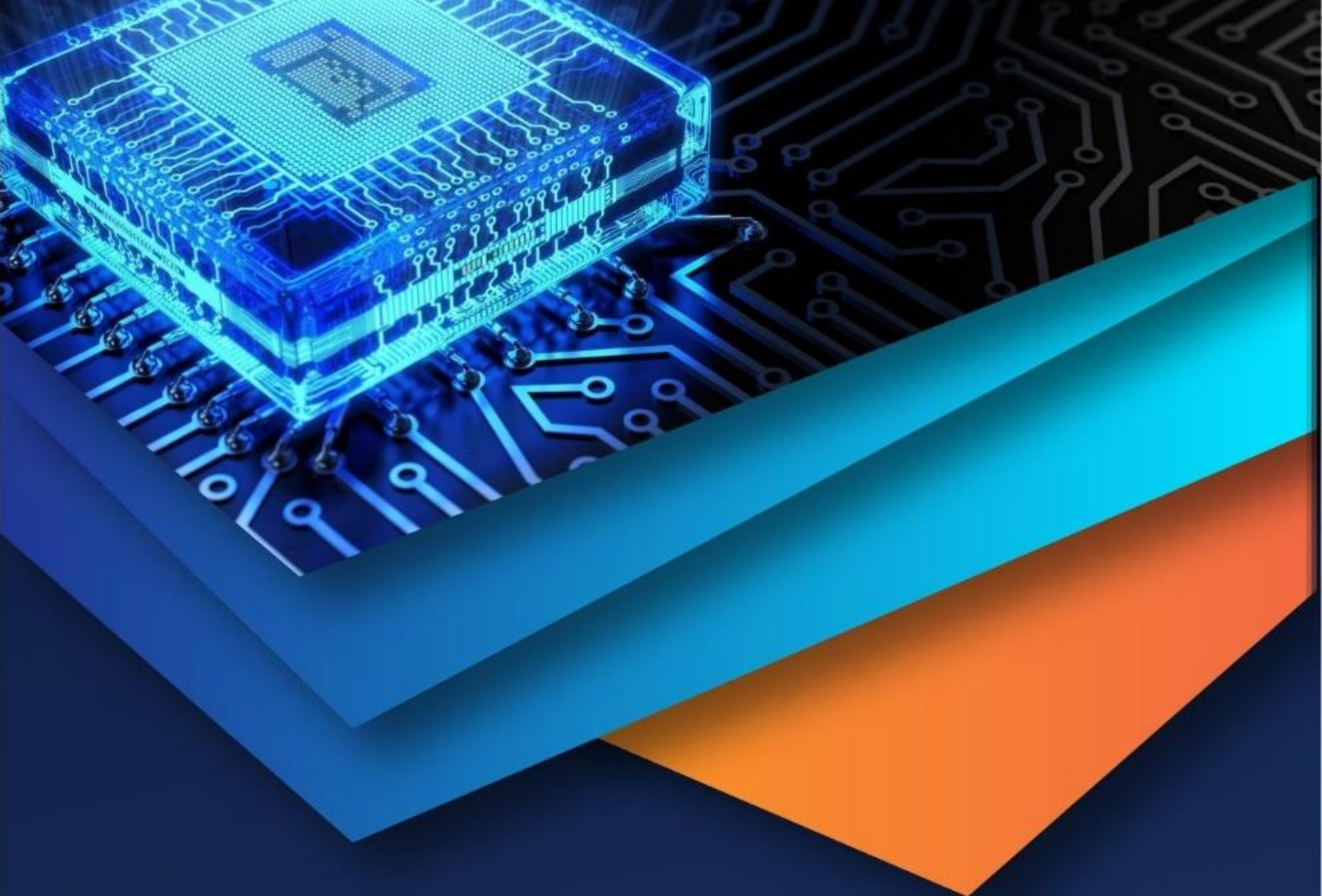
[5] G Ananda Reddy and V Ganesan, ' Study of pre diffuser optimization of an aero gas turbine combustion chamber', Indian journal of engineering and material sciences, Vol 12, august 2005, pp 281-291.

[6] John M Smith and Albert J Juhasz, 'Performance of a short annular dump diffuser using suction stabilised vortices at inlet mach numbers to 0.41' NASA Technical paper 1194.

[7] CR Fishenden and ST Stevens, 'Performance of annular combustor dump diffusers' AIAA/SAE, Volume 14, No 1, 1976.

[8] Prakash Ghose, Amitava Datta and Achintya Mukhopadhyay, 'Effect of dome shape on static pressure recovery in a dump diffuser at different inlet swirl' IJETAE, volume 3, 2013, pages 465-471,

[9] Prakash Ghose, Amitava Datta and Achintya Mukhopadhyay, 'Effect of pre diffuser angle on the static pressure recovery in flow through casing liner annulus of a gas turbine combustor at various swirl levels' TSEA-14-1133.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY