IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 16, Issue 3 Ser. III (May. - June. 2019), PP 07-12
www.iosrjournals.org

CFD Analysis, Theoretical Verification and Experimental Validation on the Effects of Reynolds Number and Cone Angle on Pressure Recovery Coefficient (PRC) of a Diffuser

Engr. Stephen Tambari[1], Engr. Ayodele Akinola Samuel[2], Engr(Mrs) Gloria I.F Dan-Orawari[3]
[1,2,3]Department Of Mechanical Engineering Ken Saro Wiwa Polytechnics, Bori Rivers State, Nigeria

Abstract: *The purpose of this study is to investigate the effects of Reynolds number (Re) and Cone angle (2θ) on the Pressure Recovery Coefficient (PRC) of the diffuser. It also borders on comparing the advantages and disadvantages of the theoretical, experimental and computational analysis of a diffuser with different cone angles. The above aims, were achieved by performing experimental , computational and theoretical analysis on a given diffuser for three different cone angles (In this case; 5º, 24º and 60º). This was done in order to verify and select the cone angle that has the best optimal Pressure Recovery Coefficient (PRC) and to know the relationship between the PRC for Theoretical, Experiment and CFD for a given diffuser. The basic reason behind this is to improve diffuser performance which is a function of Pressure Recovery Coefficient (PRC) and thus, increase efficiency and useful work of the entire system for which the diffuser is a component.*

Keywords: *CFD, Reynolds, cone angle, diffuser, PRC, Analysis*

Date of Submission: 20-05-2019 Date of acceptance: 05-06-2019

I. Introduction

The Fundamental principle behind the diffuser performance can best be describe when a diffuser is used at the exhaust of fluid machines like the gas turbine and also at the exhaust of a motor vehicle. The basic reason why diffusers are mounted at the exhaust of these machines is to reduce the velocity of the exhaust gases(Kinetic energy becomes zero near the wall and is gain towards the mid stream of the flow) and increase the pressure of the gases (It works on Bernoulli's principle) that is, converting dynamic pressure to static pressure. This increase in pressure (Kinetic Flow Energy) will cause an increase pressure ratio across the entire system and subsequently cause a pressure difference at the outlet of the diffuser. This difference in pressure created at the outlet of the diffuser will make it easy for the gases to be expelled out of the system since the pressure created will be greater than the atmospheric pressure. This increase in pressure difference created by the diffuser is very important, like in the case of the gas turbine. It reduces the work load on the turbine to dispense the exhaust gases and therefore increasing the useful work of the turbine and the efficiency of the gas turbine system. In the absence of a diffuser, the reverse will be the case as the atmospheric pressure will tend to push the exhaust gases back into the system and thus causing a back flow which will result in a drop in the system performance. Since the working fluid is incompressible (Air), the density remains constant(Principle of conservation of mass) and since the system is isolated, total energy is conserved since heat and work is negligible. Therefore, the following principles applies:

- Principle of conservation of mass
- The law of ideal gas
- principle of conservation of energy
- The principle of Bernoulli

II. Methodology

EXPERIMENTAL METHOD

An experiment was carried out on three diffusers having three (3) different cone angles 5º, 24º and 60º. The diffusers comprises of three basic sections: The inlet pipe, the diffuser (cone section) and the outlet pipe. The system works in conjunction with a fan connected in series and linked o a dial regulator. The inlet pipe and pipes are connected to two inclined manometers carrying liquids in millimeter water (mmH20) for measuring the pressure differences. The inlet section has a diameter of 89mm with length 267mm, the cone (diffuser) section has length of 584.34mm and the outlet has diameter of 140mm and length 700mm. The distance from the inlet of the diffuser to the point that leads to the entrance of both manometers was noted and marked as Li (Li=54mm) and the distance from the exit of the diffuser to the point that leads to the other ends of the manometers was also noted and marked as Lo (Lo=115mm) . The cone angle of the diffuser was also noted as 2θ in each of the diffusers.

EXPERIMENTAL PROCEDURE:

- The experiment was started with a diffuser of 60^0 cone angle. This angle was confirmed and recorded.
- The distance Li and Lo were noted and recorded
- The tubes were then adjusted to achieve a perfect level for each tube and the zero errors of ± 2mm and ± 3mm where noted and recorded accordingly.
- The was then turned on starting it first at a maximum position using the dial regulator connected to it.
- The system was allowed a few minutes to ensure that the flow stabilizes before taking the readings.
- Readings were taking and recorded from both manometers for pressures in mmH_20 with accuracy of ± 2mm and ± 3mm.
- The knob of the fan regulator was then turned to a medium position and minimum positions to decrease the inlet velocity of the air flow .
- The above steps were repeated for other cone angles and readings were taken accordingly.

NUMERICAL METHOD

The following assumptions were made in carrying out the numerical analysis:

- It was assumed to use a **2D axisymmetric flow** in order to avoid circular flow but turbulence and the following models applies: The **K-Epsilon model** and **the standard wall function**
- The flow was assumed to be a **steady flow** since there was no obstruction within the diffuser and fluid properties becomes constant making it to obey the mathematical law:

$$P_{a-}P_1 = \frac{1}{2}\rho V_1^2$$

- The flow is also assumed to be **incompressible** since it uses Air as its working fluid and air has a constant density. This assumption was backed with the following mathematical principle (principle of conservation of mass):

$$\rho V_1 A_1 \ = \rho V_2 A_2$$

- The fluid was also assumed to have an **isothermal flow** since heat transfer is neglected and the velocity equals zero towards the wall.
- The flow was assumed to use a **high Reynolds** number which leads to a high pressure difference thus making the flow turbulent especially the boundary region.

THE CFD LAB PROCEDURE:

- A two dimensional (2D) axisymmetric Model of the diffuser (used during the experiment) with half angle of 5º was created using the given dimension in the design modeler of ANSYS 16 workbench.
- The surface was created and the solution domain was partitioned and both were generated accordingly.
- The boundaries of the diffuser (where the viscous forces and the inertia force are of same order as the inertia forces and the velocity tends to zero):The Inlet, Wallin, Wallout, Outlet and Axis.
- The system was then meshed with a bias factor of 3 and the mesh quality was checked using the aspect ratio by ensuring that all elements have aspect ratio less than 5 especially the elements near the wall to control separation due to viscous effects.
- The system was then Simulated in 2D axysimmetric solver mask in FLUENT to avoid circular flow in the radial direction.
- The standard wall function was used to Model the near wall flow since it borders on turbulence and grid size. And the K-epsilon (2eqn) was used to simulate the viscous laminar flow.
- The material property was set using air as the working fluid and the boundary conditions were set for; Turbulence intensity (%) and hydraulic diameter(m) for outflow and maximum velocity at the inlet.
- Standard Pressure condition was selected for the solution methods with Second Order Upwind set from Momentum, Turbulent Kinetic Energy, and Turbulent Dissipation rate.
- The monitors were set in the residual for absolute criteria and the fluid variables (pressure, velocity) and surface monitors were created.
- The solution was initialized from inlet and calculated with 200 to 400 number of iterations to ensure convergence.

- Mesh independent and Convergence criterion study was done after ensuring that the solution for the boundary layer thickness (Y^+) lied between 30 and 100 and also to have same values for the fluid variables at different convergence criteria.

III. The Results

THE NUMERICAL RESULTS

The conversion of the solutions obtained into plots and contours was done using FLUENT Post Processing interface.

FACTORS AFFECTING SOLUTION ACCURACY

The plots for the Pressure is shown in Figure 1.0. This plot clearly indicates that the static pressure is constant along the whole length of the inlet pipe of the diffuser. It also showed that the pressure increased along the length of the diverging side of the diffuser and a steady or constant state was observed along the length of the outlet pipe.

From the Velocity plot shown in Figure 2.0. The history shows that, the velocity decreased from the inlet down to the outlet since kinetic energy was converted to pressure energy. It also showed that, the centre line velocity is higher than the boundary line velocity which is a clear case of frictional losses due to unfavorable pressure difference.

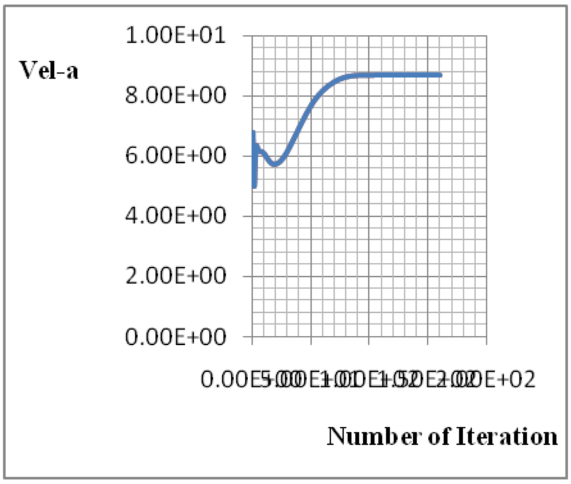

Figure 2.0 Velocity plot

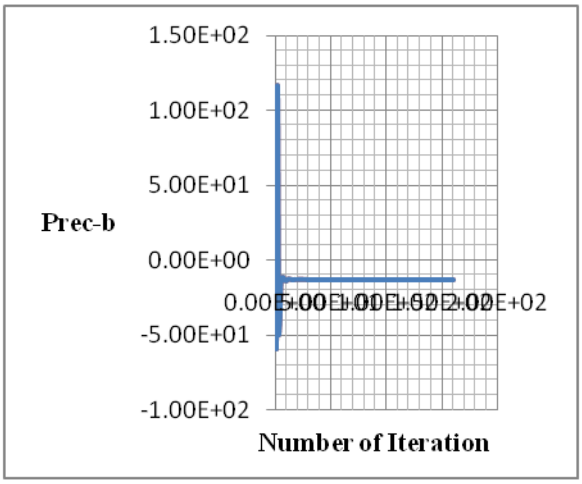

Figure 1.0 Pressure plot

The Residual contours shown in Figure 3.0. Shows contours of turbulence (that is, turbulence intensity, turbulence kinetic energy) And it depicts that, there is a fast change at the divergence of the diffuser. It also indicates that the residual history for initial mesh conditions before convergence, contained Residual RMS errors values greater than 1E-04 and the imbalances were more than 1% with non steady monitor points (Since It is assumed to be a steady state Simulation). When the mesh was refined, number of Iteration increased and the solution converged; the residual errors were now less than 1E-04, the monitors were also in a steady state and the imbalances also fall below 1%.

The Convergence Criteria is plotted in Figure 4.0. The monitored quantity used was the PRC against the Convergence criterion. The PRC was chosen for the convergence criteria study because the performance of the diffuser depends greatly on its capability to recover dynamic flow energy which appears in the form of static pressure increase. The plot shows the same value for the PRC for all the convergence criterion confirming that the flow has converged and steady towards the exit of the diffuser due to steady pressure gradient. It also confirmed that the solution can be used freely with any of the convergence criterion for the medium and minimum velocity. the PRC value was calculated using the equation below;

$$\mathrm{PRC} = \frac{\mathrm{C} - \mathrm{b}}{0.5 * \rho \mathrm{V}^2}$$

where, ρ is the density, V_1 is the inlet velocity and c and b are pressure values

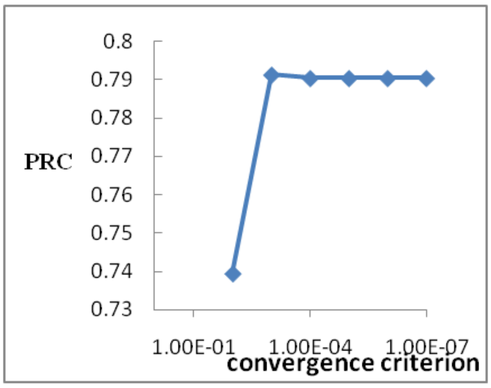

Figure 4.0 The convergence criterion study

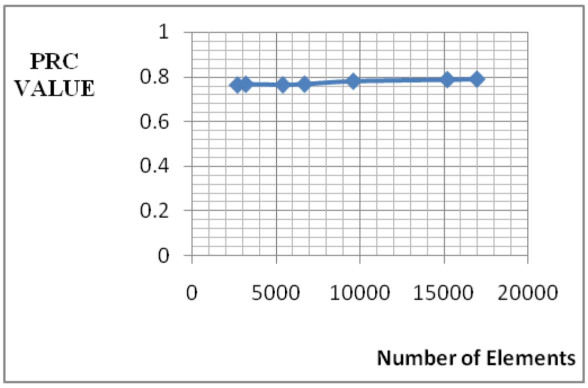

Figure 5.0 The mesh independence study

The Mesh Independence Study is shown in Figure 5.0. As shown, the PRC value was also chosen because it showed the best convergence rates. After the flow was Simulated for initial mesh and solution converged, the boundary layer thickness contour was checked and confirmed to be within the acceptable range of; $30 < Y^{+} < 100$. The mesh density was increased and the monitor quantities(PRC, Vel-a, Pres-b, Pre-c) were obtained(with the changes gradually becoming smaller). The mesh density was increased further, the mesh quality became finer and the difference between the monitor quantities was very small to less than 5% confirming that the solution has converged and now mesh independent. The trend plotted means that, the elements at which it converged can be used directly to solve a similar problem without guessing which element will be suitable and to solve the problem of delay time allotted to meshing.

The simulated diffuser with half angle of 5^0 was carefully meshed knowing too well that mesh is strongly affected by Aspect ratio, skewness and size variations which can possible affect the quality of the mesh and hence the convergence and accuracy. The meshed sized was controlled using the aspect ratio by ensuring that the aspect ratio at each point was less than 5. This was by increasing the number of divisions for each sections of the diffuser. The density of the elements near the wall was increased so high (High density gives good quality results) because of the swirl and viscous effects and using the hexahedral mesh and making the grid lines align with the flow. The solution accuracy was maintained by avoiding the following during meshing; large coarse mesh, large skewness, high aspect ratios, improper boundary layer mesh.

The Figure 7.0 Shows the plot of the Wall Y-Plus (Y^{+}). The plot indicates that the Wall Y-Plus value was controlled within the 30 and 100 which is a prove of convergence. The boundary layer thickness Y^{+} is the distance from the wall where there is high turbulence due to high Reynolds number with serious viscous effects down to the mid stream of the flow. The Y^{+} as can be seen from the plot in Figure 7.0 tells when the boundary layer is too thick (that is, when $Y^{+} > 100$) which affects the inlet to the diffuser. The mesh to the inlet was reduced to bring down below 100. The Y^{+} also tells when the boundary layer is too thin (that is, $Y^{+} < 30$) which affects the outlet of the diffuser and the mesh density was increased to bring is up above 30. The Y^{+} was also controlled by reducing the bias factor down to 2 or 3.

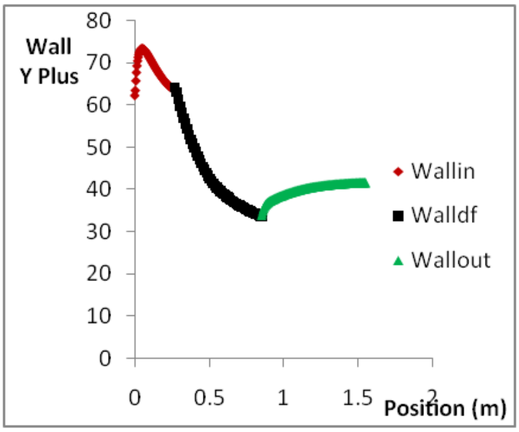

Figure 7.0 Wall Y-Plus plot

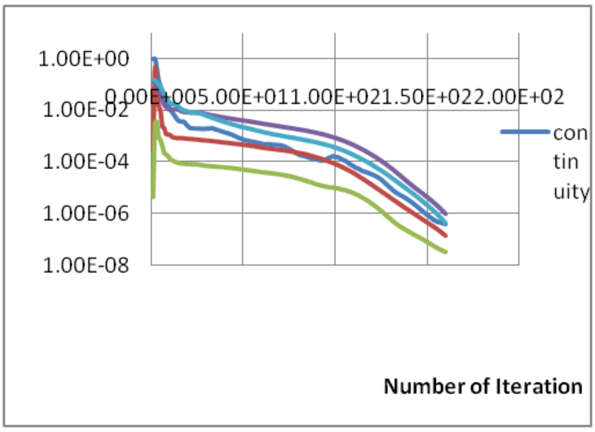

Figure 3.0 The Residual plot

EXPERIMENTAL RESULTS

The readings obtained from the inclined manometers were converted to pressure differences using the hydrostatic equation; $\Delta P = \rho\, g\, \Delta h$ where ΔP is the pressure difference in Pascal, g is the acceleration due to gravity and ρ is the density. The Experimental PRC was computed using the equation;

$$PRC = \frac{P_2 - P_1}{0.5 * \rho V^2}$$

where $P_2 - P_1$ is the pressure difference

The Reynolds number at the inlet was also computed using the equation below:

$$R_{ei} = \frac{V_1\ d}{v}$$

where v is the kinematic viscosity, d is the inlet diameter and V_1 is the inlet velocity.

The Ideal PRC was also obtained using the equation below;

$$\text{Ideal PRC} = 1 - \frac{d^4}{D^4}$$

The experimental PRC, IDEAL PRC and the CFD PRC were then plotted to compare their relationships. Figure 8.0 and Figure 9.0 shows this correlation.

IV. Result Description

(EXP PRC , IDEAL PRC, CFD PRC VS Reynolds and Cone angles)

From the Figures 8.0 The PRC value for CFD tends to be very close to the theoretical value and far higher than the experimental values. This is because, in the CFD many assumptions: (such as: No losses were taken into account, flow was assumed to be isothermal, the flow was assumed to be steady state, viscous effects and eddies were not considered, flow is incompressible, Reynolds number is high) were made. But in the experiment, the losses were taken into account, such losses are; losses due to frictional effects, losses due to separation, losses caused by viscous effects, losses caused by eddies as there is high turbulence near the wall, with zero velocity and shear stress and a high pressure gradient due to high Reynolds number.

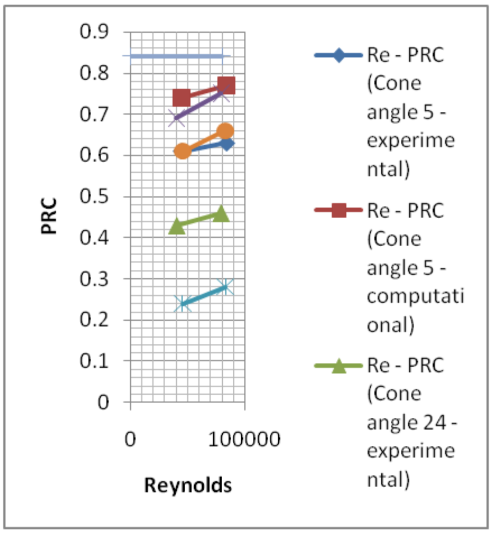

Figure 8.0 PRC VS REYNOLD NUMBER

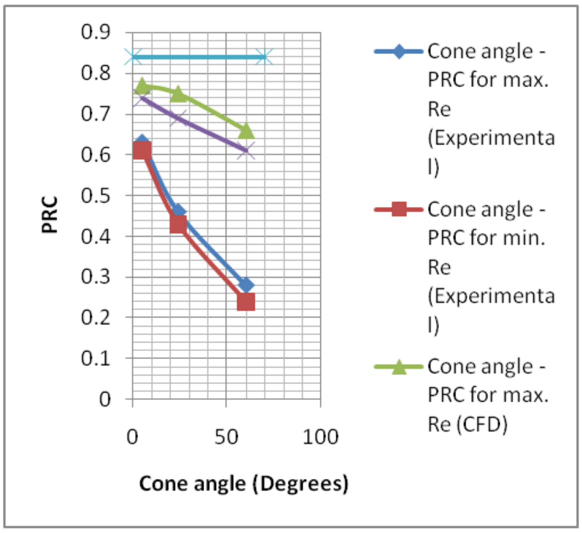

Figure 9.0 PRC VS CONE ANGLES

PRC tends to be constant in the case of PRC VS Reynolds number, which is a strong indication that PRC does not depend on inlet values since the Reynolds number is a function of inlet value like Velocity.

In the case of PRC VS Cone angle, the PRC for the larger cone angle tends to be very low showing separation of the boundary layer since Large angles imposing a very large pressure difference that the boundary layer cannot sustain. This is a strong indication that, PRC depends strongly on geometry.

The Ideal PRC is the highest PRC since it is a perfect state. And thus, any value above the ideal value was ignored and not selected for the PRC. Secondly, some data had the same PRC for both minimum and maximum Reynolds number, such values were also filtered as the PRC cannot be the same.

V. Discussion And Conclusion

- Pressure Recovery Coefficient (PRC) tends to be constant when plotted against Reynolds number showing that PRC is not affected by inlet values (Velocity) since Reynolds number is a function of velocity.
- Pressure Recovery Coefficient (PRC) is greatly affected by the cone angle showing it depends on geometry. This effect is due to separation of the boundary layer and losses due to viscosity and eddies.
- For a very small cone angle, the positive pressure gradient is very small and thus no separation takes place
- For a large cone angle, a large positive pressure gradient is imposed on the boundary layer. This pressure gradients becomes too great for the boundary to sustain and thus separates from the wall.
- From the results obtained, the wall Y-plus was found to be between 30 and 100 showing that the solution converged.
- The mesh independent study and the convergence criterion were done successfully
- A high mesh density was maintained near the wall and aspect ratio was kept at less than 5.
- From the results and plots, it can clearly be seen that the CFD values of the PRC is higher than that of the experimental values which means that CFD produces good results than the experimental method.
- From the correlation graphs it can clearly be seen that the diffuser with the smallest angle produces an optimal value of PRC and those of larger angles gave low PRC owing to separation.
- The CFD method is less expensive, easy to use, valid and gives good results but it goes with so many assumptions and it requires proofs.
- The Experimental method is very good to use when handling real life problems but it is very costly, it requires going over the procedure several times and the data are not usually very accurate.
- The Theoretical method is fast method, less costly and it bends you to understand the basic principles but it is not suitable for solving very tough problems and it also requires estimation when using the values.

References

[1]. John D. Anderson, JR. Department of Aerospace Engineering, University of Maryland" Computational Fluid Dynamics the basics with applications" International edition,1995

[2]. Bernard Massey and John Ward-Smith "Mechanics of Fluids" 8th edition, 2006

[3]. R.Prakash et al, "CFD Analysis of Flow through a conical exhaust diffuser" InternationalJournal of Research in Engineering and Technology, 2014

[4]. E. Karunnakaran and V Ganesan " Mean Flow Field Measurements in an AxisymmetricConical diffuser with and without inlet Flow distortion" Indian Journal of Engineering and Materials Science

[5]. Dr. Alberto Marzo, Department of Mechanical Engineering, University of Sheffield. Lecture notes " MEC6016 Fluids: CFD in a nutshell 1 & 2"

Engr. Stephen Tambari. "CFD Analysis, Theoretical Verification and Experimental Validation on the Effects of Reynolds Number and Cone Angle on Pressure Recovery Coefficient (PRC) of a Diffuser." IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , vol. 16, no. 3, 2019, pp. 07-12.