International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-5, March 2020

Effect due to Variation in Bend Angles of Intake Manifold on Turbulent Kinetic Energy for Diesel Engine

Shahim Haider Abidi, M. M Hasan

***Abstract:** One of the positive results for enhancing turbulence is to improve swirl, which is an important factor of air motion in a diesel engine. Other than enhancing mixing and improvement in combustion processes it also influences heat transfer, combustion quality, and engine raw emissions. To improve swirl intensities in-cylinder parameters like velocity, pressure, temperature and turbulence intensity are to be considered. There are two ways to create a swirl, modification in the intake system and valve design. So this work done contains modifications in the design of manifold to enhance turbulence during the intake stroke. Designs of manifold having different bend angle of 15º, 30º, 45º, 60º and 75º were used, all parts of numerical analysis were carried out on Ansys Fluent. The 200mm long intake model having a 20 mm diameter, with a bend on 160mm along length was used to find out the best bend angle configuration from the above orientations. K-epsilon model was used to simulate flow dynamics; variations turbulent kinetic energy was studied. After analyzing these results it was concluded that best-optimized design (in terms of turbulent kinetic energy) to get better swirl was for 75º. This work gives the understanding to find new techniques for further improvement in mixing by increasing turbulent kinetic energy. This work emphasizes on the techniques to enhance turbulent kinetic energy of any flow, and can also be applied to different fields related to mixing of fluids other than diesel engine.*

Keywords: CFD, Intake Manifold, Turbulent kinetic energy

I. INTRODUCTION

As pressure arises on automobile makers due to compliance with BSVI standards in India, these standards' main focused area is emissions. So the scope of finding new techniques to reduce the percentage of unburned fuel is the need of the hour. In this sphere the main problem is to get a homogenous mixture of air and fuel, as air motion influences the atomization and injected fuel distribution in a combustion chamber, due to this modification in the design and orientation of intake manifold is always required [1]. Various researchers have developed methods for increasing the turbulence in the intake manifold by using guide vanes, ring-type generator with four curvilinear blades, variation in plenum length [2], and using different types of the internal thread, helical and spiral shape [3]. By using these techniques researchers were able to enhance the turbulence of inlet air which in turn improves the mixing of fuel and air, resulting in more efficient combustion of fuel.

Engines are one of the greatest Mechanical Engineering applications developed in the 90s. Compression ignition engines are very robust, durable and efficient. The volumetric efficiency of the CI engine is higher because of the absence of throttle losses in comparison to the SI engine. The flow of air through the intake manifold considerably affects the power and volumetric efficiency of CI engines. In the present investigation, the existing intake manifold of the CI engine is modified and manufactured by additive manufacturing techniques. The optimized design was finalized by undergoing computational fluid dynamic analysis. The new intake manifold is fitted to the engine intake and performance tests were performed. For all load conditions, the volumetric efficiency, brake power, and brake thermal efficiency were considerably improved. The brake specific fuel consumption was also reduced.

In terms of durability and efficiency, CI engines have leverage on SI engines, but with these advantages, it has a problem in terms of emissions. Various techniques related to in-cylinder and also after combustion treatment are being developed to minimize the effects of harmful output. In the IC engine mainly there are three types of efficiency thermal, volumetric and mechanical. As far as mechanical efficiency is concerned a significant amount of optimization has been achieved during past research work, but still, there is a scope of work in the other two (thermal & volumetric). In the field of designing intake manifold, various geometries were used by researchers to increase the mixing of fuel and air, as it plays an important role in combustion. The intake manifold design is a major deciding factor for an efficient engine in terms of power and emissions. In this work, an effort is done to discuss the different arrangement of the intake manifold to serve the purpose of enhanced mixing and the possible ways to further increase the turbulence of inlet air. The main objective is to enhance the turbulent kinetic energy of intake air.

II. METHODOLOGY

A. Standard k-ε turbulence model

This model is applicable to a large number of turbulent processes [4].

Equation for turbulent kinetic energy k

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon \quad (1)$$

For dissipation ε

Revised Manuscript Received on March 30, 2020.
* Correspondence Author

Abidi Shahim Haider*, Research scholar, Department of Mechanical Engineering, Jamia Millia Islamia (a central university) in New Delhi, INDIA.

Hasan M.M, Senior Professor, Department of Mechanical Engineering, Jamia Millia Islamia (a central university) in New Delhi, INDIA.

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (2)$$

Where u_i represents velocity component in corresponding direction, E_{ij} represents component of rate of deformation, μ_t represents eddy viscosity

$$\mu_t = C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

Table I: Value of adjustable constants

C_μ	σ_k	σ_ε	$C_{1\varepsilon}$	$C_{2\varepsilon}$
0.09	1.00	1.30	1.44	1.92

B. Steps in fluent analysis

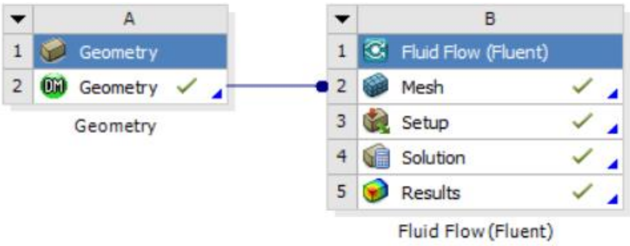

Fig.1. steps in fluent module

III. WORKBENCH MODEL

The first step is to create an intake manifold model on design modeler Ansys. As shown in the fig. 2 below, the length of the intake manifold is 200mm having a radius 10mm and bend is at around 160 mm.

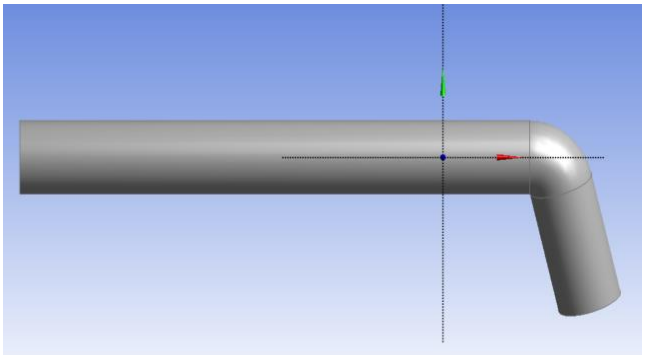

Fig. 2. Intake manifold design in design modelor

The fluid inside the intake manifold is air and material is aluminum.

Table II: material properties

Aluminum Properties	
Density	2710 kg/m3
Poisson's ratio	0.346
Coefficient of thermal expansion	2.36e-5 /0K
Yield strength	9.5e7 N-m2
Young's' modulus	7e+010N m2

Table III: fluid properties

Air properties	
Density	1.225 kg/m^3
Viscosity	1.7894e-05 kg/m-s

Different cases having variations in bend angle (clockwise with respect to x-axis) has to be investigated and shown below.

Table IV: Bend angles for different cases

Case A	Case B	Case C	Case D	Case E
15°	30°	45°	60°	75°

All other dimensions and conditions are same for all cases

IV. MESHING

Mesh quality is responsible for accurate results. So it is to be done in a very careful manner. There are various mesh control methods like sizing, refinement face meshing, pinch, match control, and inflation. In sizing curvature advanced size function is used, the relevance center and smoothing are taken as medium and active assembly opts for initial size seed with the slow transition. All other settings are taken as defaults value. The very important control in this problem as to inflation layer meshing is required for near boundary regions of wall-bounded turbulent flow. In below table inflation settings are mentioned. It this problem's first layer thickness option of inflation taken with a maximum of five layers. During meshing simulation, the physical domain is divided into a discrete grid as the accuracy of the simulation result is dependent on the mesh geometry and the choice of the grid dimension is not trivial

Table V: Inflation parameters

Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Layer Thickness
First Layer Height	0.11 mm
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No

All default values are taken for other controls, name selection is also done during selection like inlet, and outlet and the wall are assigned in this problem..

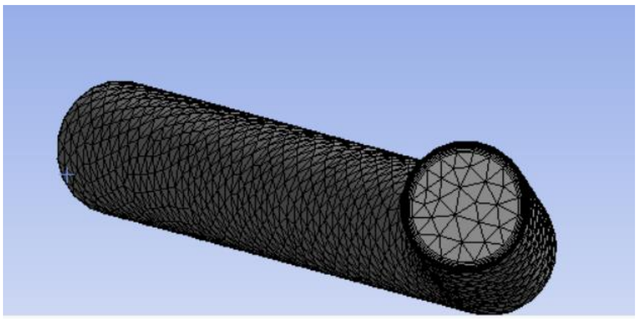

Fig.3. Meshing

V. BOUNDARY CONDITIONS

In this transient problem, a pressure-based solver with absolute velocity formulation is taken. Standard k-ε turbulence model has opted; inlet velocity of air given as 5 m/s and gauge pressure value is given in the table. For turbulence, intensity and viscosity ratio method is selected.

Retrieval Number: E2973039520 /2020©BEIESP
DOI: 10.35940/ijitee.E2973.039520
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-5, March 2020

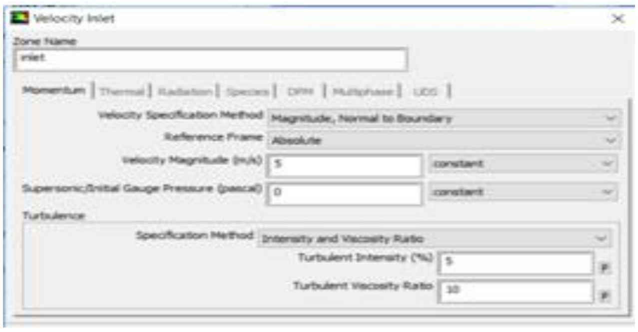

Fig. 4. velocity inlet conditions

Reference The reference value for temperature is 288.16k, a SIMPLE scheme is selected, a least-square cell-based method is for gradient, second-order upwind is taken for momentum, turbulent kinetic energy, turbulent dissipation rate, and pressure is the second order for this first-order implicit transient formulation. Default values for relaxation factor were being chosen; residual monitor for continuity, x-velocity, y-velocity, z-velocity, k, and epsilon was observed and converged. For run calculation, the time step size is taken as 0.01 and around 4000 iterations were computed in a fixed time-stepping method.

VI. RESULTS AND DISCUSSION

CFD-Post was used for extracting contours and charts for the variation of turbulent kinetic energy at outlet of the intake manifold.

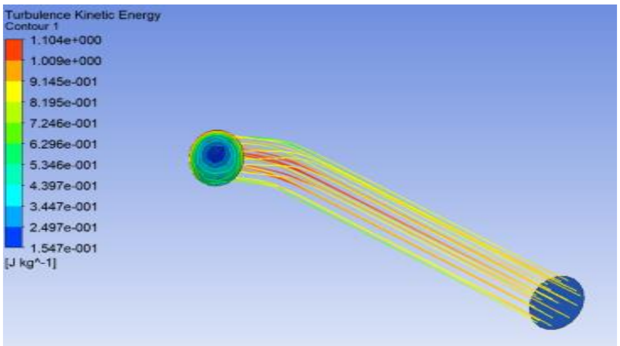

Fig.5. Contours for 15°

In above fig.5. variation of turbulent kinetic energy can be seen through intake manifold for bend angle 15 degree. The average turbulent kinetic energy is 0.659493

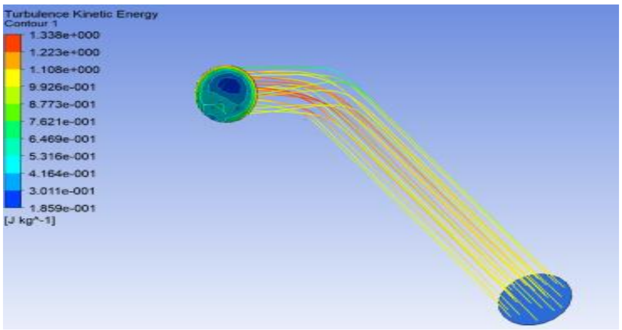

Fig. 6. Contours for 30°

The above fig.6 shows the variation for a 30-degree bend angle. In this case the maximum turbulent kinetic energy and minimum turbulent kinetic energy are 1.338 JKg^{-1} and 0.1859 JKg^{-1}. By comparing it by 15-degree case it can be observed that in this case, turbulent kinetic energy is more than the previous one. The average turbulent kinetic energy in this case is 0.727683

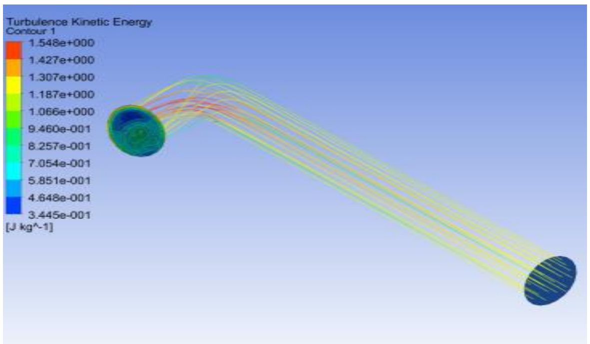

Fig. 7. Contours for 45°

For 45 degree case, the value of minimum turbulent kinetic energy is 0.3445 JKg^{-1}. and maximum turbulent kinetic energy is1.548 JKg^{-1}.. In this case, the average value is more than both of the previous cases. The average turbulent kinetic energy value is 0.8782124

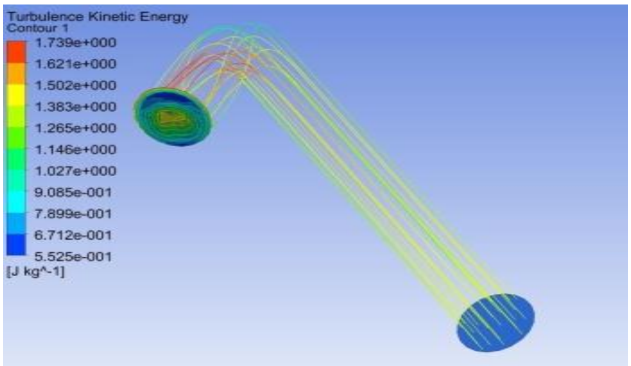

Fig. 8. Contours for 60°

In case 60-degree bend angle in the intake manifold. The maximum and minimum turbulent kinetic energy value is 1.739 JKg^{-1} and 0.5525 JKg^{-1}.

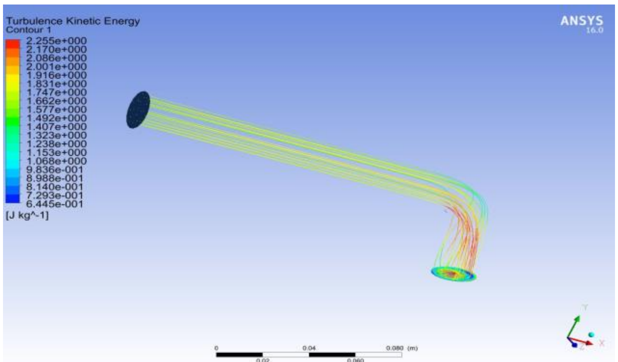

Fig. 9. Contours for 75°

This case had a 75-degree bend angle that gives maximum turbulent kinetic energy among all cases. In this case, the maximum and minimum values are 2.25516 JKg^{-1} and 0.644486 JKg^{-1}. The average value is 1.272136 JKg^{-1}.

We can see that with increasing bend angle in intake manifold the turbulent kinetic energy increases.

Table: Average turbulent kinetic energy at outlet (JKg^{-1})

Case A	Case B	Case C	Case D	Case E
0.659493	0.727683	0.8782124	1.006673	1.272136

They studied five different angles of inclinations which were 15°, 30°, 45°,60° and 90° (on a horizontal plane) and reported that the highest angle of inclination(bend angle along the axis) produced the highest turbulent kinetic energy (TKE)

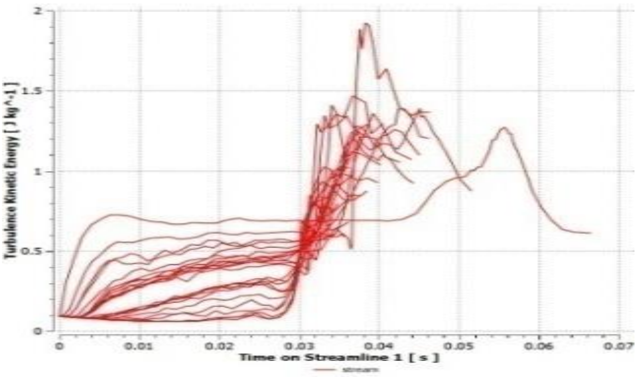

Figure 10

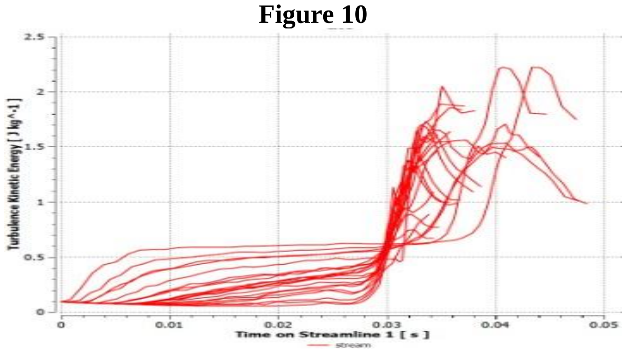

Figure 11

In the above two figures variation of turbulent kinetic energy concerning time on streamline is seen for case D and case E. Maximum peaks can be noticed for these two cases.

VII. CONCLUSIONS

Based on this research work, it was observed that the bend angle or inclination angle plays an important role in the mixing of air and fuel in the combustion chamber of the diesel engine. By variation in the design of intake manifold, it was seen that turbulent kinetic energy changes with bend angles. It can be concluded that as bend angle increases, turbulent kinetic energy increases and it will lead in improvement of mixing and performance. Still, after doing this analysis we can further increase mixing by incorporating dimples in the best case among these all cases of the intake manifold, which is for the 75-degree case. This work focuses on geometrical techniques to enhance turbulent kinetic energy of any fluid flow and can be applied to other fields like gas turbines.

ACKNOWLEDGMENT

I am thankful to Almighty that I had worked successfully in the computational dynamics field. Firstly the credits go to my Supervisor, who had shown faith in my work. I want to deliver special thanks to Department of Mechanical engineering, AMU, Aligarh, INDIA for allowing me to work in CFD labs at their campus

REFERENCES

1. Heywood J. B., Internal combustion engine fundamentals, McGraw-Hill, Inc., USA, 1988.
2. Amit Kumar Gupta and Abhishek Mishra, "Design and Development of Inlet Manifold for Six Cylinder Engine for Truck Application" Volume: 3 | Issue: 7 | July 2014 ISSN - 2250-1991 Paripex - Indian Journal Of Research.].
3. Paul, V. Ganesan, "Flow field development in a direct injection diesel engine with different manifolds" International Journal of Engineering, Science and Technology Vol. 2, No. 1, 2010, pp. 80-91
4. Rahiman Abdul, Abdul Razak R. K., Mohammad Samee A. D., Ramis M. K. CFD Analysis of Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds. American Journal of Fluid Dynamics 2014, 4(3): 102-113 DOI: 10.5923/j.ajfd.20140403.03
5. Idris Saad and Bari S. CFD Investigation Of In-Cylinder Air Flow To Optimize Number Of Guide Vanes To Improve CI Engine Performance Using Higher Viscous Fuel. International Journal of Automotive and Mechanical Engineering (IJAME) ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online); Volume 8, pp. 1096-1107, July-December 2013©Universiti Malaysia Pahang.
6. Saad Idris and Bari Saiful. Optimizations of Vane Height of Guide Vane Swirl and Tumble Device to Improve in Cylinder Airflow Characteristics of a Diesel Engine Running with Vegetable Oil. Adv Automobile Eng 3:106. doi:10.4172/2167-7670.1000106
7. Shenghua, L., Ou, C., and Won, H., Development of New Swirl System and Its Effect on DI Diesel Engine Economy, SAE Technical Paper 1999-01- 2889, https://doi.org/10.4271/1999-01-2889
8. S. A. Sulaiman, S. H. M. Murad, I. Ibrahim and Z. A. Abdul Karim. Study Of Flow In Air-Intake System For A Single-Cylinder Go-Kart Engine. International Journal of Automotive and Mechanical Engineering (IJAME) ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online); Volume 1, pp. 91-104, January-June 2010 ©Universiti Malaysia Pahang DOI: http://dx.doi.org/10.15282/ijame.1.2010.8.0008

AUTHORS PROFILE

Abidi Shahim Haider, is a research scholar at the Department of Mechanical Engineering, Jamia Millia Islamia (a central university) in New Delhi, INDIA. His research areas are the application of computational fluid dynamics and molecular fluid dynamics in a different field.

Hasan M.M, is a senior Professor at the Department of Mechanical Engineering, Jamia Millia Islamia (a central university) in New Delhi, INDIA. He has 31 Years of Teaching and Research experience, his research areas are I.C.Engines & Combustion, Air Pollution, Energy Studies, and Heat Transfer. He has 70 plus publications in the field of mechanical engineering