International Journal of Advanced Culture Technology Vol.3 No.1 1-12 (2015)
http://dx.doi.org/10.17703/IJACT.2015.3.1.1

IJACT 15-1-1

Influence of intake runner cross section design on the engine performance parameters of a four stroke, naturally aspirated carbureted SI engine

Somendra Pratap Singh, Vasu Kumar, Dhruv Gupta, Naveen Kumar

Delhi Technological University, Delhi, India
somendra.dtu@gmail.com

Abstract

The current scenario of the transportation sector reflects the urgent need to address issues such as depletion of traditional fuel reserves and ever growing pollution levels. Researchers around the world are focussing on alternatives as well as optimisation of currently employed devices to reduce the pollution levels generated by the commonly used fuels. One such optimisation involves the study of air flow within the intake manifolds of SI engines. It is a well-known fact that alterations in the air manifolds of engines have a significant impact on the engine performance parameters, fuel consumption and emission levels. Previous works have demonstrated the impacts of runner lengths, diameter, plenum volume, taper angle of distribution manifolds and other factors on in-cylinder fluid motion and engine performance. However, a static setup provides an optimal configuration only at a specific engine speed. This paper aims to investigate the variations in the same parameters on a four stroke, naturally aspirated single cylinder SI engine through varying the cross section design over the intake runner with the aid of Computational Fluid Dynamics. The system consists of segments that form the intake runner with projections on the inside that allow various permutations of the intake runner segments. The various configurations provide the optimised fluid flow characteristics within the intake manifold at specific engine speed intervals. The variations such as turbulence, air fuel mixing are analysed using the three dimensional CFD software FLUENT. The results can be used further for developing an automated or manually adjustable intake manifold.

Keywords: *Segmented Intake Runner, Computational Fluid Dynamics, runner length*

Nomenclature

CFD	Computational Fluid Dynamics
rpm	Rotations per minute
cc	Cubic centimetres
Tl	Turbulence Intensity

Manuscript Received: Feb. 10, 2015 / Revised: Mar. 13, 2015 / Accepted: Apr. 20, 2015
Corresponding Author: somendra.dtu@gmail.com
Tel: +91-11-27871018, Fax: +91-11-27871023
Delhi Technological University

1. INTRODUCTION

With the problem of depleting fossil fuels reserves available, it has become a widespread research topic around the world to search for alternatives and to optimise the currently available technology in terms of engine thermal efficiency and fuel economy. Developing nations pose another problem that they cannot invest in new and improved technologies due to economic constraints. For example, technologies such as fuel injection and direct injection are known to improve fuel efficiency and at the same time reducing exhaust emissions by a considerable factor. But still, in developing nations such as India, these technologies cannot be incorporated in small scale engines as the new technologies push up product costs. Therefore, the need is to design systems that are cost effective and tackle developing nations' economic constraints. These products would have a significant impact as these small capacity engines are used on a large scale to meet the transportation requirements. Despite the tremendous development in the field of fluid flow in any engineering sector, manifold designing still remains a subject with a large potential for improvement and optimisation.

Important design criteria include minimal flow restriction, equal flow distributions between the engine cylinders, sufficient heating to ensure more efficient vaporisation of fuel and ram air intake effects if applicable[9]. However, the flow feature required in an engine manifold is a characteristic feature of the type of engine and operating conditions of the same. This is also the reason why manifold designs such as geometry and inner surface finish, vary from one manufacturer to another depending on the type of engine[8]. The flow features of fuel air charge that this system aims to target is the manifold turbulence which can enhance the air fuel mixing at various engine speeds in carburetted engines. At high speeds, turbulence may lead to increased back pressure in the manifold prior to induction and thus reducing the volumetric efficiency. Thus, a manifold with variable turbulence drives the idea behind the project. The atomisation and vaporisation of fuel is a function of the intake temperature and the flow of air within the manifold runners prior to and after the carburetion. In terms of processes, the homogenisation of air fuel is determined by thermal diffusion, diffusion property of the species and forced distribution through interaction between motions of two species. With the temperature being treated as constant, the air fuel mixing is determined by the flow turbulence within the manifold as the effect of forced diffusion is much larger than natural diffusion in case of high turbulence. Also, the flow feature within the intake manifold or runners determines the flow within the engine cylinder geometry.

A significant study of the intake and exhaust manifold designs was done by Ugur Kesgin, where various geometric parameters were varied by the author and the effect on engine performance was observed [1]. The parameters that were given importance were the diameter, length, the L/D ratio and valve timing. The link between these parameters and the working of devices such as turbochargers was also established. A highlighted fact was that the diameter of the exhaust manifold should be at least equal to the cylinder bore for maximum turbocharger performance. The increase in the engine efficiency was also shown to be a function of the valve timing and valve lift profiles in addition to optimisation of intake geometry to eliminate excessive backflow. A important aspect for engine intake geometry was highlighted by S.A. Sulaiman et al on the experimental research on intake geometry of a Go-kart engine [6]. According to their results, the variation in the intake manifold did not produce a significant impact on the engine performance at low valve lifts. Appreciable changes in engine performance were obtained only when the intake valve lift was increased beyond 3mm. An even dedicated investigation of the geometrical factors in designing involved the variation of other parameters such as taper angle, type of taper, angle of taper etc. between two manifold pipes with runners to distribute the flow. Jimmy C.K. Tong et al conducted extensive study by varying various factors to obtain identical flow rates in the manifold runners[4]. It was observed that the increase in cross sectional area helped obtain better uniformity in flow rates in the runners. The study involved two steps. Firstly, the linear taper angle in the distribution pipe was observed and it was found that increase in taper angle reduces the non uniformity as compared to the non tapered arrangement. The second part involved the type of taper and out of the three arrangements, namely concave up, concave down and a combination of both, the concave down arrangement showed the best flow rate uniformity whereas the concave up arrangement was most ineffective. A significant observation made was when the both the distribution and collection manifolds were given linear tapers. It was observed that the advantage due to one taper was nullified by the taper of the other manifold and the flow rate remained nearly same as in the stock configuration. The problem of obtaining identical flow rates in runners from a primary distribution manifold was also addressed by A.S. Gren through a different approach. A common duct was chosen as a distribution manifold with runners of varying dimensions [3]. Modifications in designs offered better flow rates, however it was a permanent feature with all configurations that the runner closest to the inlet received the maximum flow rate. For multi cylinder engines, the variation in plenum volume also has a significant impact on all the aspects of engine performance, especially at low speeds. This effect was presented by M.A. Ceviz et. al. in an experiment where the only parameter of the manifold varied was the plenum length, which in turn varied the plenum volume[7]. The test setup involved a nearly 1800cc engine with electronically controlled fuel injection. Even though the fuel consumption was expected to show a marginal increase due to use of electronic fuel injection, a larger plenum length was highly effective in increasing the fuel efficiency at low engine speeds. The effects on the parameters such as brake power, brake torque, and thermal efficiency was highly favourable for urban and sub-urban operations. Among these, the highest variation showed was that in case of thermal efficiency, which increased from 0.27 to 0.30 with an increased plenum length. However, there was no direct relation to determine the most optimum plenum volume. Though increase in volume increased performance, the increase beyond a certain plenum length had negative effects on the same parameters. A similar approach was presented in an experiment whereby an optimal design was compared to a stock design of the intake manifold using two approaches, CFD and experimental. The pressure, velocity and turbulence fields were shown for both the configurations at two crank angles during the intake stroke[2]. A significantly higher degree of in cylinder turbulence, swirl and tumble were observed in the case of the optimised manifold. Through the CFD software, it was possible to study the variation in pressure, velocity etc. For the turbulence field, the two factors considered were the turbulent kinetic energy and the turbulence dissipation rate. A higher degree of turbulence was observed in the optimised configuration and since the turbulence level is proportional to the air fuel homogenisation, the experimental results for the optimised manifold showed increased performance at a large band of engine speeds. Increase in brake power by upto 16% and torque by 59.9Nm at 1500 rpm were the most noteworthy observations. In addition, thermal efficiency also increased from 26% to 27.5%.

This paper aims at studying an intake configuration on a single cylinder, four stroke, and carburetted SI engine. The parameter to be varied is the cross section of the runner that feeds the cylinder. A distinct feature is that the intake runner is segmented and can result in a set of combinations depending on the relative angular separation. The configurations are tested for turbulence intensity, mass fraction distribution of species and the flow velocities. Normal aspirated engines have a configuration which is optimised for a specific band of engine speeds. A segmented configuration allows for a large set of permutations that is expected to broaden the band of engine speeds where the intake manifold is optimised. The task can be implemented by using a segmented assembly of smaller runner elements. In each element, the internal cross section is modified as shown in figure 2. When the relative position of the internal cross section of each segment is changed, there is obstruction to the flow fluid and causes haphazard motion of fluid particles at the junction of two runner elements. The relative position can either be manually adjusted or guided by rotation through a motor. The haphazard motion raises the turbulent intensity in the manifold system and is dependent on the relative angular separation between two runner elements.

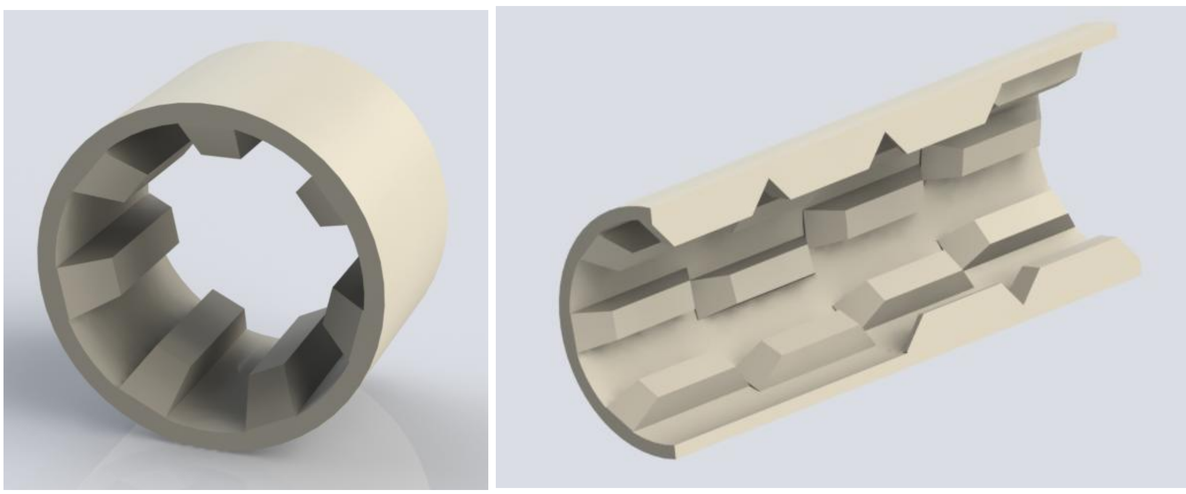

Figure 1. Design of a segment of the intake runner and arrangement of intake runner segments

The segmented design strategy decided for the experiment necessitated the use of circular runners. Since a relative angular separation is imparted to one segment relative to the adjacent segment as shown in figure 3, the axis and hence the flow field will remain uninterrupted only in case of circular runners. For the calculations, the setup decided is that of a single cylinder naturally aspirated engine without any ram air intake effect. This also eliminated the complexities of flow distributions and plenum volume since only a single runner feeds air to the engine. The only parameter that is varied is the relative angular separation between two adjacent segments with the other boundary conditions remaining the same.

The analysis was done through the 3 dimensional CFD package FLUENT on ANSYS 14. Firstly the basic part was designed, i.e. the segment of the intake runner. After designing the component, the theoretical permutations that should optimise the flow characteristics in the manifold were assembled. The fluid flow column was modelled for the final analysis.

2. EXPERIMENT:

2.1 Design:

The figure below shows the basic design of one segment of the intake runner. A pipe segment with an outer diameter of 5.2 cm and inner diameter of 5 cm was initially designed. Within the inner surface, five projections along the length were created covering the entire segment length. The edges of the projections were chamfered at 45 degrees to avoid abrupt backflow at the junctions. The length of one segment was restricted to 5 cm. The overall length of the intake runner was set to 40cm.

The overall runner length is longer than that observed in normal small displacement engines. This was done so that the gradients of velocity, pressure, turbulence could be observed for a larger area and a cleared distinction could be made.

The extended diameter of the runner was decided because of the loss in fluid flow volume in the segment due to the projections. Three configurations were decided as follows:

1. A relative angular separation of 0 degrees between 2 adjacent segments.
2. A relative angular separation of 30 degrees between 2 adjacent segments.
3. A relative angular separation of 15 degrees between 2 adjacent segments.

It is expected that the reduction in the angular separation of the adjacent segments would be an optimum setup for increasing engine speeds due to low flow restriction to the high air-fuel flow rates. For the analysis purpose, 2 engines speeds were decided upon on the basis of the band of speeds that are covered by single cylinder engines during operation. The 1500 rpm corresponds to the lower engines speed and the 6000 rpm engine speed corresponds to the average higher speeds used by single cylinder engines for operation.

Parameters such as mass fraction distribution, turbulence and velocity in the cross section were observed for the four configurations at the two engine speeds. Based on the parameters, the combination of runners that favours an engine speed was decided.

2.2 Calculations for flow rates:

For the purpose of modelling the air fuel mixing inside the flow volume, the inlet area was decided into 2 areas. The central circular area with a radius of 0.5cm was used as a fuel mass flow inlet area and the remaining area from the 2.5cm radius section of the inlet was used for air mass flow inlet. The considerations made for the purpose of calculations were:

Table 1. Details of the engine for the purpose of calculations

Engine Displacement	150cc
Lower engine speed for analysis	1500rpm
Higher engine speed for analysis	6000rpm
Volumetric efficiency(average)	85%
Compression ratio	9.5:1
Stoichiometric A/F ratio	14.7

The general calculation procedure was as follows:

1. Let the engine make x revolutions per minute.
2. For x revolutions per minute, the number of intake strokes were x/4 per minute.
3. Since the displacement of the engine is 150cc and the compression ratio is 9.5, the clearance volume can be calculated.
4. Since the clearance volume is never devoid of air or air fuel mixture, the mass of air ingested corresponds to the volume of the displacement of the engine.
5. Considering the volumetric efficiency of 85%, at x/4 intake strokes per minute, the volume flow rate is (150*0.85*x)/1000*4 Litres/minute.
6. Since at 35 degree C, the density of air is 1.1455 kg/cubic metres, the mass flow rate can be calculated.
7. Taking the value of 14.7 as the AF ratio, the mass flow rate of fuel is calculated.

Table 2. Calculated values for analysis

Engine speed	Air mass flow rate (kg/s)	Fuel mass flow rate(kg/s)
1500 rpm	$1.82574*10^{-3}$	$1.241425*10^{-4}$
6000 rpm	$7.302563*10^{-3}$	$4.9657*10^{-4}$

3. SETUP AND BOUNDARY CONDITIONS

The fluid flow area for all the four configurations were designed on ANSYS Design Modeller. The Fluent analysis module was used to carry out the further operations. After meshing the model was transferred to Fluent and the models used were Energy equation, Realizable K-epsilon model, and species transport model. The Species transport model was enabled to calculate the mass fraction distribution within the runner. Since the process is not for injection model, the DPM model was switched off.

The temperature for the entire walls and boundaries was set at 35 degree C since it can be taken as an average for the conditions surrounding the intake runner. For the mass fraction composition of air, the mass fraction values were kept identical to that of the atmospheric composition. For the fuel inlet, since the major component that undergoes combustion is isooctane, the mass fraction of isooctane at the inlet was kept 1.

4. RESULTS AND DISCUSSIONS

Calculations were run for the two engine speeds and the four configurations. For the purpose of analysis, the results are expressed as a comparison between 2 configurations for the two engine speeds. The factors considered are velocity, turbulent intensity and mass fraction distribution of isooctane. Configuration 1 and configuration 2: These two correspond to the zero angular separation and maximum angular separation between the adjacent segments respectively. At high rpm, it was observed that the end velocity in case of configuration 2 was 42.05% higher as compared to that of configuration 1. However, if we compare the velocity gradient throughout the length and from the wall to the centre, the former showed a more uniform and smooth gradient in both the directions. On comparing the turbulent intensity, higher values were present in case 1 only at the start of the first projection and at the end of the last projection due to the formation of eddies. Case 2 had higher turbulent intensity of 48% that spanned across major areas near the wall. Even at the centre, TI values of 20.3% were obtained which shows the effect that the end projections can make at the centre. The maximum value of 58.3% occupied regions near the end and start of projections as expected. The reason being that the end and start of the projections change the direction and magnitude of the velocity by a great factor which raises the turbulent intensity. At the end face or the outflow, the TI dropped to 5.9% in case 1 whereas it stayed high at 10.1% to 29.1% in case 2. The final factor considered was the mass fraction distribution of isooctane. The decrease in the mass fraction values was more visible in case 2. However, is the decrease in mass fraction along the length was considered, then case 1 showed early mixing of isooctane particles. Still, the mass fraction values were lower and more uniform in case 2 as compared to case 1. At the outflow, in case 1, the mass fraction ranged between 0.04 to 0.22 as compared to the range of 0.02 to 0.12 in case 2.

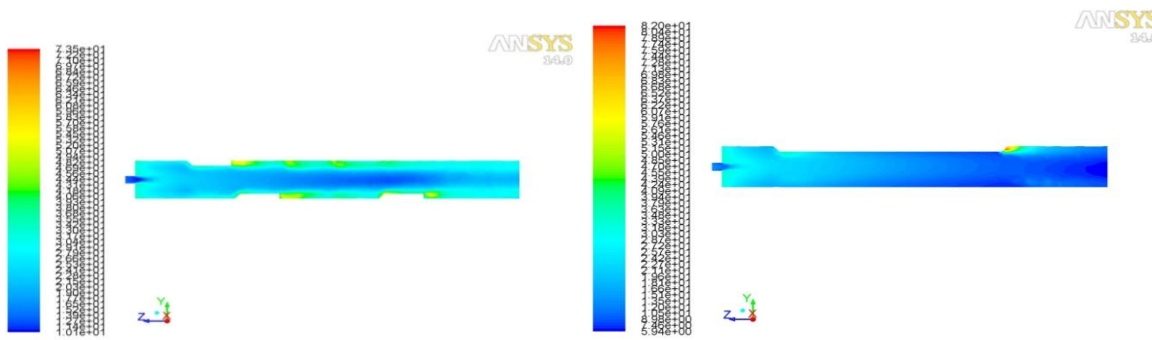

Figure 2. Turbulent intensity in case 2 at high rpm and case 1 at high rpm

At lower engine speed of 1500rpm, case 2 showed areas of zero velocity near the boundary wall that were not present in case 1. As the previous analysis, case 1 had a more uniform velocity gradient as compared to case 2 and also the peak value of velocity was higher in case 2 compared to case 1. As expected, the values of TI were higher for case 2 with values ranging from 11.3% at the walls to 6.68% near the centre. Surprisingly, the mass fraction distribution in case of part 1 was found to be lower and has a constant value near the outflow region along the entire area. Compared to this, case 2 had higher stratification and lower level of air fuel mixing. This is in opposition to the expected result, whereby the configuration 2 was expected to cause proper air fuel mixing at lower rpm. In case 1, the mass fraction values lied between 0.02 and 0.06 and was the best mass fraction homogenisation obtained for any arrangement.

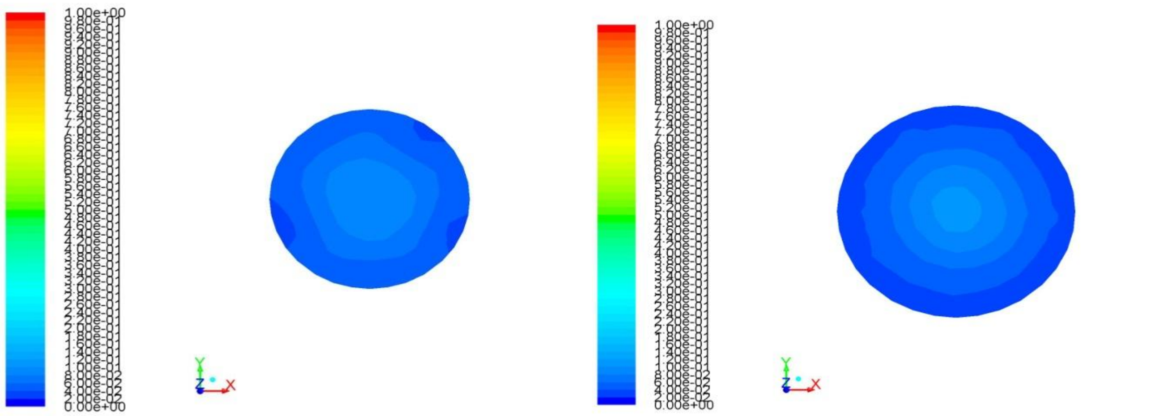

Figure 3. Mass fraction distribution in case 1 at low rpm and in case 2 at low rpm

Configuration 2 and Configuration 3: Due to relatively increase area of obstructions in case 2, the velocities obtained at the centre and at the ends has a higher magnitude in case 2 by about 10.13%. Both the cases 2 and 3 had a much higher TI value as compared to case 1. If turbulence at the centre has to be considered, case 2 had a higher TI but the difference was much lower as compared to the difference between case 2 and case 1. For the same regions, case 2 had higher TI values for almost all regions as compared to case 3. In this comparison, mass fraction distribution showed that case 2 was more efficient with fuel air mixing as compared to case 3 at higher rpm. Major area was occupied by regions of 0.02 to 0.16 mass fraction in case 2 as compared to area of 0.04 to 0.20 mass fraction in case 3.

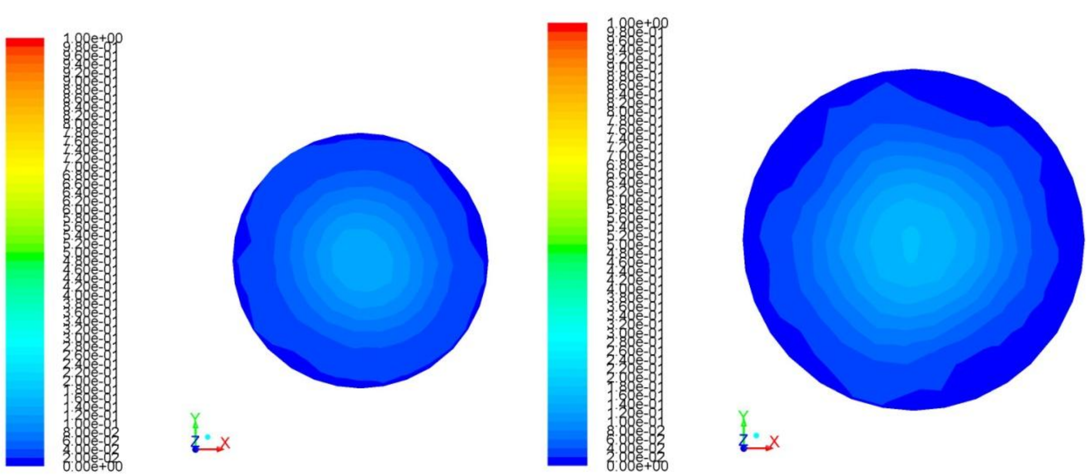

Figure 4. High rpm mass fraction distribution in case 2 and case 3

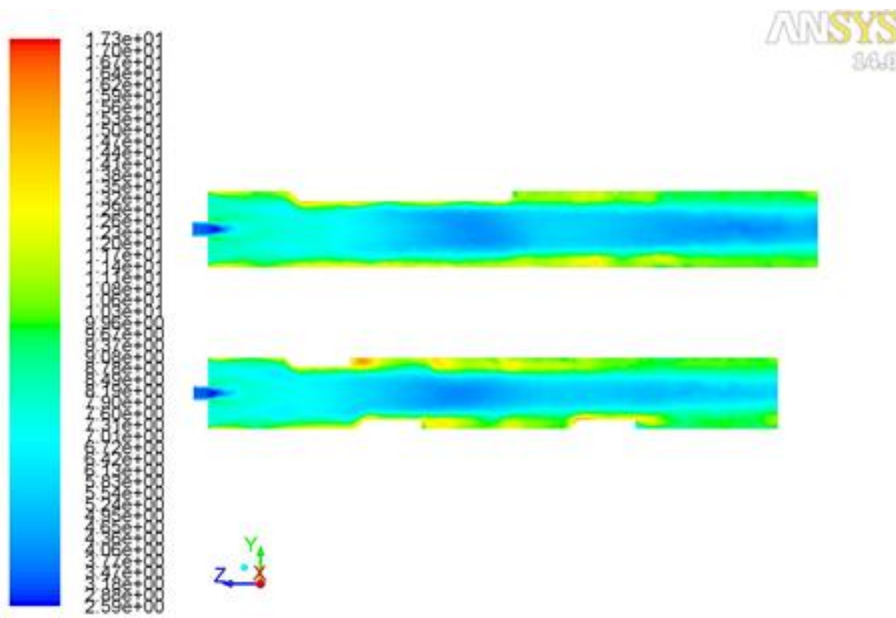

Figure 5. Low rpm turbulent intensity in case 3(upper) and case 2(lower)

At low engine speeds, the velocity distributions were fairly identical with the terminal velocity being higher in case 2 by only 0.11m/s. TI distribution was also very similar but the variation and values were higher in case 2 as expected. However, the variations lied only near the centre and the values near the wall had minor differences. Even the mass fraction distributions were similar but the regions of higher mass fractions existed over larger outlet area in case 3. It was concluded that the configuration with more angular separation was more favourable in terms of velocity profiles, the turbulent intensity and mass fraction distribution.

Configuration 1 and Configuration 3: Comparison between the 0 degree separation and 15 degree separation revealed the increase in velocity regions in the sections with projections. However, in configuration 1, the velocity profile showed rapid decrease afterwards. In case 3, the reduction in magnitude was of much less as compared to case 1. The maximum velocity at the centre of configuration 3 was higher by 16.41%. Turbulent intensity had a maximum value of 82% in case 1 but it was irrelevant since the reason for this was the flow reversal in a small space at the end of the projection. In comparison, case 3 had a maximum value of 59.5% even in regions where flow reversal did not take place. At the centres, the values for case 1 and case 3 were 16.4% and 8.95% respectively. Mass fraction distribution showed very similar pattern in the two configurations at high engine speed. The values ranged from 0.06 to 0.24 in both the cases but the homogenisation was better in case 1 as compared to case 3. At the low engine speed of 1500 rpm, the mass fraction distribution improved in case 3 and was more evenly spread as compared to the high rpm. However, the best mass fraction homogenisation was obtained in case 1 for low engine speed operation.

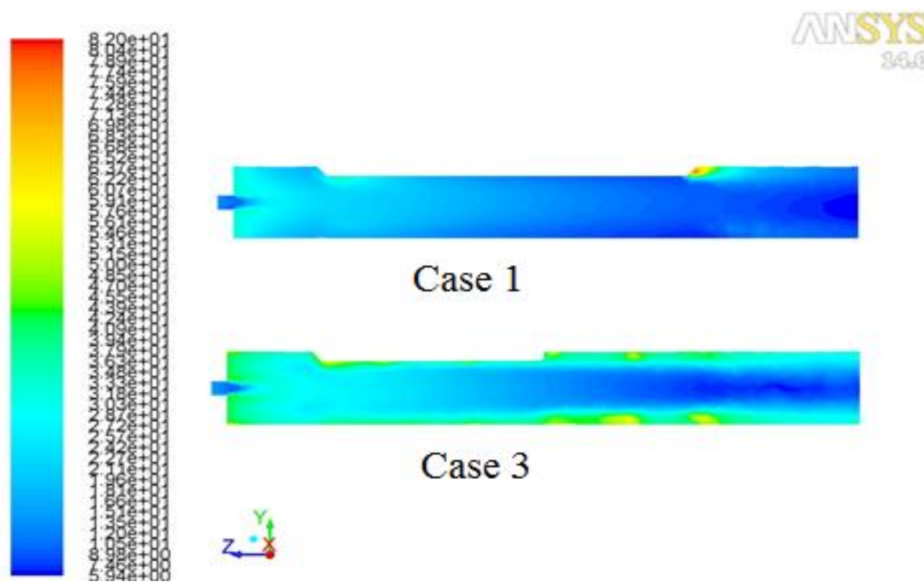

Figure 6. High rpm TI

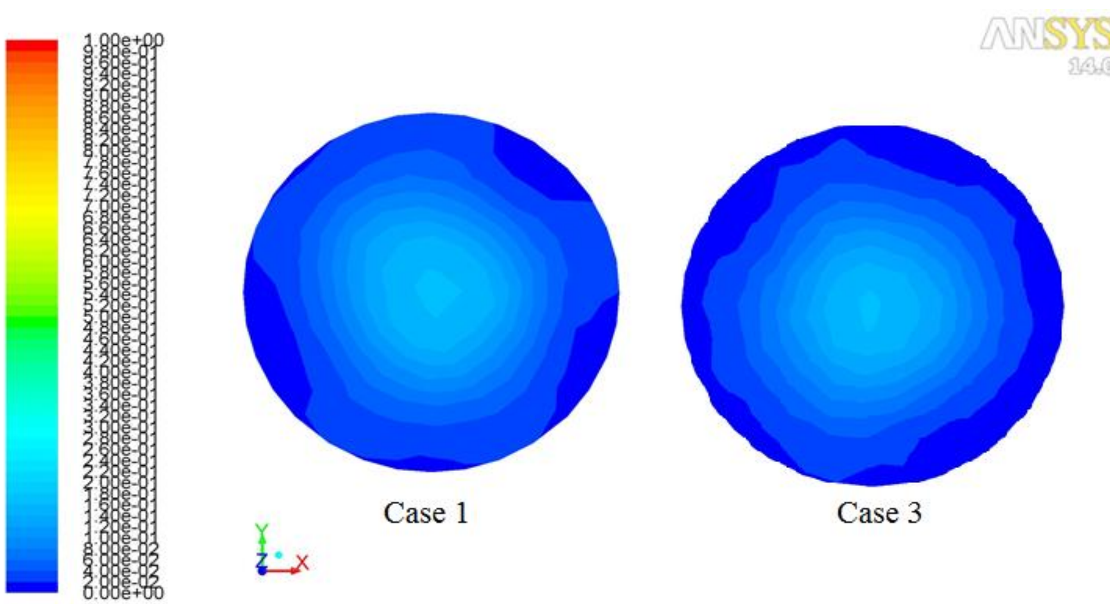

Figure 7. High rpm mass fraction

Backflow in the flow space: The backflow property for the configurations was studied at high engine speed since the effect would be amplified in case of higher engine speeds. To observe flow reversal, the fact was considered that the fluid is flowing in the negative z direction based on the designing process. The contour was observed for a positive velocity in the z axis and it was observed that flow reversal took place at the end of each projection for all configurations. The degree of backflow can be measured through comparing the magnitudes of positive z velocities. It was observer that as the angular separation increase, flow reversal took place at the end of the projections where the fluid. Also, as the angular separation increased, the magnitude of backflow increased. The chamfered edges reduced the backflow to a certain extent but reverse flow magnitudes of upto 1.1 m/s were still observed in configuration 2.

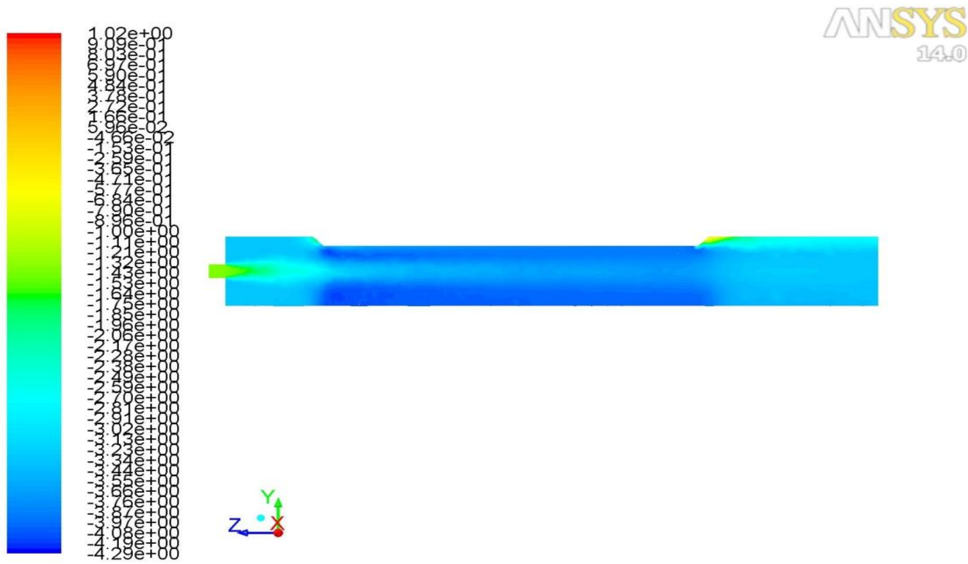

Figure 8. Backflow in case 1

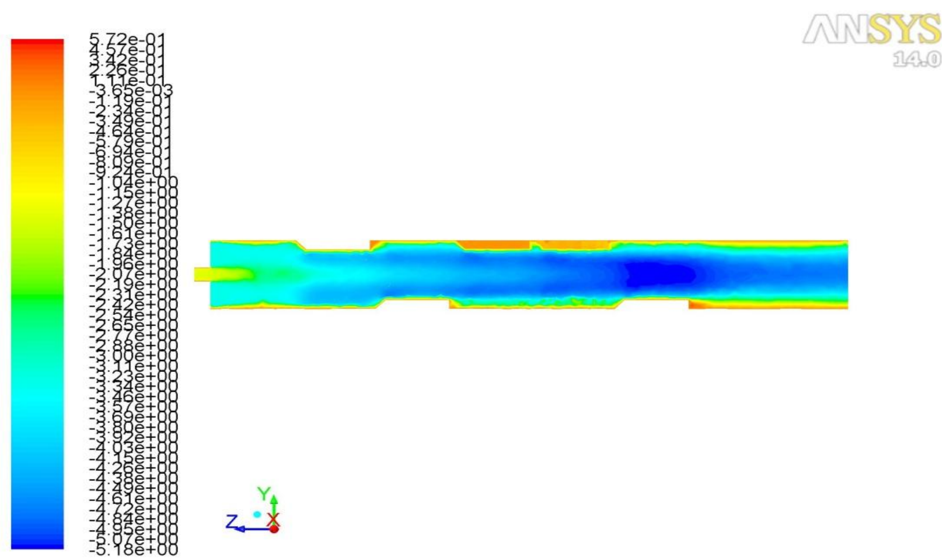

Figure 9. Backflow in case 2

The increase in the turbulent intensity in the flow field was expected to cause better homogenisation of the air fuel mixture at the outflow surface. The comparison between maximum and zero angular separation was in agreement of the expected result at high engine speed. However, the mass fraction distribution in case of minimal turbulent intensity showed the best result among the results obtained for various configurations. The relation between the turbulence and mass fraction distribution was again held true in the comparison between the low and high angular separation cases at both engine speeds. But the comparison between low angular separation and zero separation again showed contradictory to the relation between the turbulence and air fuel mixing that was observed in the second comparison. It can be concluded that there exists a complex relation between the properties of fluid flow and cannot be easily generalised. The expected increase in turbulent intensity and velocity profiles were in agreement to the CFD results but the fuel mixing characteristics of the system showed varying results.

5. CONCLUSIONS

The results showed that an exact prediction of fluid flow through a direct relation cannot be made and proper analysis through CFD is necessary. Also, the characteristics analysed are a function of engine speed and intake geometry. For the given setup, the CFD results reveal that

1. Though turbulent intensity can be raised though the projections on a segmented intake, the mass fraction distribution cannot be predicted solely on the basis of turbulence.
2. The best mass fraction distribution was obtained at a configuration with the lowest turbulent intensity at the low engine speed.
3. For other configurations, the increase in turbulence caused better homogenisation of air fuel at both the engine speeds selected.

It is thus clear that the performance of the manifold, if implemented through this approach, bears a complex relation with the angular separation. Actual manifold take into consideration thermal diffusion effects, though their major contribution is at the heated valves under continuous operation. In devising an automated or a manually adjustable intake runner assembly, it is important to test the configuration initially to obtain data of the performance parameters involved. Based on the data obtained, an optimum function can be evolved to increase the performance across a larger power band. From the results in the current study, it can be realised that a compromise between the turbulence, backflow within the manifold and mass fraction distribution has to be struck which can be most accurately determined through experimental data for a specific engine. With the data of the experimental setup, a completely automated intake system can be devised to benefit the large fraction of small capacity engines that still rely on carburetion in developing nations.

6. REFERENCES

[1] Ugur Kesgin, “Study on the design of inlet and exhaust system of a stationary internal combustion engine,” Energy Conversion and Management, Vol. 46, Issues 13–14, pp. 2258-2287, August 2005, , ISSN 0196-8904, http://dx.doi.org/10.1016/j.enconman.2004.10.015.
http://www.sciencedirect.com/science/article/pii/S0196890404002626
Keywords: Valve timing; Inlet pipe; Exhaust pipe; Junction; Natural gas engine; Efficiency

[2] Mohamed Ali Jemni, Gueorgui Kantchev, Mohamed Salah Abid, “Influence of intake manifold design on in-cylinder flow and engine performances in a bus diesel engine converted to LPG gas fuelled, using CFD analyses and experimental investigations,” *Energy*, Vol. 36, Issue 5, pp. 2701-2715, May 2011, ISSN 0360-5442, http://dx.doi.org/10.1016/j.energy.2011.02.011.
http://www.sciencedirect.com/science/article/pii/S0360544211000892
Keywords: Alternative fuels; CFD; Intake manifold; In-flow; Model of turbulence; Experiment

[3] A.S Green, T Moumtzis, “Case study: use of inlet manifold design techniques for combustion applications,” *Applied Thermal Engineering*, Vol. 22, Issue 13, pp. 1519-1527, September 2002, ISSN 1359-4311, http://dx.doi.org/10.1016/S1359-4311(02)00070-4.
http://www.sciencedirect.com/science/article/pii/S1359431102000704
Keywords: Burner intake manifold; Flow distribution; CFD modelling

[4] Jimmy C.K. Tong, Ephraim M. Sparrow, John P. Abraham, “Geometric strategies for attainment of identical outflows through all of the exit ports of a distribution manifold in a manifold system,” *Applied Thermal Engineering*, Vol. 29, Issues 17–18, pp. 3552-3560, December 2009, ISSN 1359-4311, http://dx.doi.org/10.1016/j.applthermaleng.2009.06.010.
http://www.sciencedirect.com/science/article/pii/S1359431109001884
Keywords: Flow uniformity; Distribution manifold; Collection manifold; Computational fluid dynamics

[5] Junye Wang, “Theory of flow distribution in manifolds,” *Chemical Engineering Journal*, Vol. 168, Issue 3, pp. 1331-1345, April 15, 2011, ISSN 1385-8947, http://dx.doi.org/10.1016/j.cej.2011.02.050.
http://www.sciencedirect.com/science/article/pii/S1385894711002300
Keywords: Flow distribution; Distributor; Parallel channels; Manifold; Spargers; Maldistribution

[6] 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

[7] M.A. Ceviz, M. Akın, “Design of a new SI engine intake manifold with variable length plenum,” *Energy Conversion and Management*, Vol. 51, Issue 11, pp. 2239-2244, November 2010, ISSN 0196-8904, http://dx.doi.org/10.1016/j.enconman.2010.03.018.
http://www.sciencedirect.com/science/article/pii/S0196890410001329
Keywords: Intake manifold; Intake plenum; Engine performance

[8] Pulkrabek, Engineering Fundamentals of the Internal Combustion Engine

[9] Heywood, Internal Combustion Engine Fundamentals