TRANSACTIONS B: Applications Vol. 29, No. 2, (February 2016) 255-263

International Journal of Engineering

Journal Homepage: www.ije.ir

Intake Manifold Flow Assessment on a 3-cylinder Natural Aspirated Downsized Engine Using CFD and GT-SUITE

M. H. Shojaeefard, A. Khalkhali, A. Firouzgan*

Automotive Simulation and Optimal Design Research Laboratory, School of Automotive Engineering, Iran University of Science and Technology, Tehran, Iran

PAPER INFO

Paper history:
Received 02 December 2015
Received in revised form 27 December 2015
Accepted 26 January 2016

Keywords:
Intake Manifold
Computational Fluid Daynamics
Intake Manifold Model
Engine Simulation Code
Volumetric Efficiency

ABSTRACT

In this paper, the intake manifold as the most effective part on engine's volumetric efficiency was investigated in detail with emphesizing on the flow behavior and characterestics. Eight different designs were prepared and imported to the CFD software. Five objective functions with different weights were applied and investigated by TOPSIS method. The weights were supposed so that, one of the objectives was dominant compared to others. Mass flow rate, mass flow rate deviation between runners, flow coefficient, turbulence intensity and velocity at the runner end were considered as objective functions for flow detail assesment. Then, all geometrical dimentions were applied to a verified ESC. The engine simulation model was developed for I3 engine considering turbulence combustion model named " SI-TURB". The ESC was validated by experimental combustion profile data. The experimental and numerical results had good compatibality, so that the ESC could simulate and predict the engine performance by 6.15% accuracy. This paper shows that the hybridization of 3D analysis with ESC in the closed loop way is mandatory for developing of IM in order to have the best compromising design. Finally, combination of CFD and ESC revealed, considering the same weights for objective functions in TOPSIS, good performance in point of engine power out put.

***doi**: 10.5829/idosi.ije.2016.29.02b.15*

NOMENCLATURE

BSFC	Brake Mean Specific Consumption	TOPSIS	Technique for Order of Preference by Similarity to Ideal Solution
ESC	Engine Simulation Code	RPM	Revolution Per Minutes
MCDM	Multi Criteria Decision Making	IM	Intake Manifold
I4	Inline Four Cylinder engine	I3	Inline Three Cylinder engine
CF	Coefficient of Flow	Dp	Plenum diameter
Di	Air Inlet Diameter	3D	Three Dimentional
CFD	Computational Fluid Daynamics	ICE	Internal Combustion Engine
R1	First Runner	R2	Second Runner
R3	Third Runner	R	Redious of runner profile
Dr1	Runner first part lenght	Dr2	Runner second part lenght
FKGM	Flame Kernel Growth Multiplier	TFSM	Turbulence Flame Speed Multiplier
BCD	Best Compromising Design	LPS	Liter Per Second

1. INTRODUCTION

Many car producers attempt to improve the performance of internal combustion engines. In this regard, the intake system must be designed properly in order to supply sufficient air in all speeds and loads of engine. This means the researchers' ambitions are to maximize the volumetric efficiency. The task of IM is conducting and dividing of fluid to the cylinders. Desirable performance of the IM is to distribute uniform and homogeneous air and mixture between the cylinders in order to minimize

*Corresponding Author's Email: *A_Firouzgan@auto.iust.ac.ir* (A. Firouzgan)

Please cite this article as: M. H. Shojaeefard, A. Khalkhali, A. Firouzgan, Intake Manifold Flow Assessment on A 3-cylinder Natural Aspirated Downsized Engine Using CFD and GT-SUITE, International Journal of Engineering (IJE), TRANSACTIONS B: Applications Vol. 28, No. ٥, (May 2015) 255-263

engine's vibration. Thus, the cylinders can have the same and uniform performance.

In recent years because of the increasing power of computers and development of the computational fluid dynamics (CFD) codes, design of manifold can be accomplished more scientifically and accurately.

Although 3D software are used favorably to investigate the flow behavior in engine parts design, but the output cannot be trustable while being applied on engine. So, using ESC for assessment of the engine performance will be mandatory to have the best compromising design. In the other words, software such as GT-POWER and WAVE have a great ability to design IM, but these software do not have sufficient capability for considering distribution of flow characteristics in the engine air paths. They cannot model local effects of flow in engine parts in detail like 3D software. According to consideration of three-dimensional details in these software, more accurate and realistic values are obtained for calculating the volumetric efficiency.

An accurate design of the intake manifolds enables the engineers to manipulate the flow characteristics to the desired level for obtaining the best engine charging. Kesgin presented good results by investigation of the air inlet systems for a stationary natural gas engine [1]. Ceviz investigated the effects of intake plenum volume variation on the engine performance and emissions to constitute a base study for variable intake plenum [2]. The results of his study showed that the variation in the plenum caused an improvement in the engine performance and the pollutant emissions. The effects of intake plenum length on the performance for a spark-ignited engine with electronically controlled fuel injectors was also studied. The results revealed that longer runners could have more power in low engine revolutions [3]. The effects of intake runner length on the performance of single-cylinder SI engine were also investigated by Potul et al. [4].

Jafarmadar and Barzegar simulated the combustion processes and emission formation in pre and main chamber of a Lister 8.1 IDI diesel engine with CFD code [5]. Teylor et al. investigated the variable length intake manifold, which was applied to a 1.4-litre turbocharged GDI gasoline engine and made smooth torque of engine [6].

Discharge coefficient, mass flow rate, mass flow rate deviation between runners, velocity and turbulence at the runner's end are known as the most important design variables in the optimization process. But some of these goals are in conflict with each other. Therefore, a multi-criteria decision making (MCDM) approach can be adopted as a useful method to handle such challenge. There are numerous articles in this field that show that MCDM has been widely used.

The two decision making methods, namely AHP and VIKOR, were integrated in order to make the best use of information available by Tavakkoli-Moghaddam and Mousavi [7]. Zegordi et al. [8] proposed a model which allows risks to be ranked based on the management priorities using a combined fuzzy-ANP and fuzzy-TOPSIS method. Velasquez and Hester [9] performed common MCDM methods and examined the advantages and disadvantages of the identified methods. Then explained how their common applications relate to relative strengths and weaknesses. In order to enhance the uncertainty effect, Yahyaei et al. [10] applied the clustering analysis to find similar regions by consideration of different characteristics. Vahdani et al. [11] proposed a compromise model, based on a new method, to solve the multi-objective large-scale linear programming problems with block angular structure involving fuzzy parameters. To resolve the multiplication drawback, Wang [12] utilized a relative preference relation that is from fuzzy preference relation in fuzzy generalized simple additive weighting method. Aghajani Mir et al. [13] introduced an improved version of TOPSIS which applied to obtain the best municipal solid waste management method by comparing and ranking the scenarios [13]. Lourenzutti and Krohling introduced a generalization of the TOPSIS method, called Group Modular Random TOPSIS (GMo-RTOPSIS), which provides freedom for the decision makers to express their individuality and opinions. The method is based on dealing with an imperfect setting where each decision maker can define independently from the criteria set, the weight factor and the underlying factors which may affect the alternatives' ratings and the type of information they want to use in each criterion [14].

The duty of IM is to feed air uniformly among the runners to achieve consistent and equal performance for cylinders and meanwhile applying minimal resistance to flow in order to obtain the highest volumetric efficiency. The maximum output power, which could be achieved at any condition of engine, depends on the amount of air trapped in cylinders. The volumetric efficiency is a criterion that represents the interaction of input and output systems which compares the standard and real engine filling with air.

The volumetric efficiency is defined as the ratio of the trapped air mass in real condition to the air mass which could be feed in the cylinder displacement at standard condition. This paper, with using the experience and results of the previous research of authors [15], emphasis on 3D analysis effects for IM' geometry in order to obtain more volumetric efficiency. The previous work of the authors proved that the second runner length (i.e, L2) has dominant role in the engine performance and BSFC. Then, the plenum could have more effective role.

TABLE 1. Engine specification

Parameter	Engine
Type	I3
	3cylider linear - 4 stroke
Displacement	1.3 L
No. valve	4
Compression ratio	11
S/B ratio	86/78.6=1.08
Fuel system	Gasoline MPFI
Power (kW)	69.96*@6500
BSFC(g/kWh)	327.46*@6500
*After final analysis	

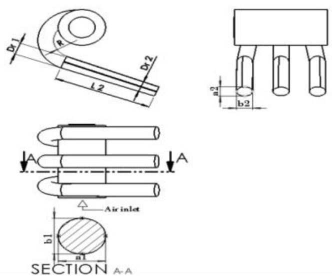

Figure 1. Schematic of I3 IM

As the third priority, the first length part of the runner (i.e, L1) have more sensitivity on the objective functions. Since the first part runner length had negligible effect on the engine performance, the authors decided to consider it as a constant value at the present research. So, based on the previous studies the total length (i.e, L) and Vp have been kept as design variables and eight different designs have been prepared and analyzed in 3D CFD software.

In this study, the flow behavior is observed with the help of 3D simulation at steady state condition. Eight different designs of IM are prepared in CATIAV5R21. Then, all configurations are imported to ANSYS FLUENT in order to investigate the flow characteristics in detail. The flow parameters which are investigated in the 3D analysis include flow coefficient, mass flow rate, mass flow rate dissipation from mean value, turbulence intensity and the velocity of flow at the end of runner length. In the second part of this research all designs are investigated in verified ESC for power and BSFC. Finally, all the results from 3D and ESC are discussed and analyzed.

2. DESIGN OF INTAKE MANIFOLD

In present research a 3-cylinder SI natural aspirated engine with specification listed in Table 1 has been selected. This engine has been carried over from the base engine named I4 version which has 1.7l and 370 mm plenum and runners length, respectively. According to the existing design, the 3D model for each design has been prepared with CATIA software for I3 engine.

The detailed geometrical dimensions of I3 IM have been shown in Table 2. Also the schematic of I3 IM has been illustrated in Figure 1.

All different IM' designs are generated by changing geometrical design variables, then are analyzed and compared with each other from the flow behavior point of view. These designs which are selected from benchmarking and experience of authors are listed as follows:

- The first model, 18% reduction in volume of plenum chamber and 19% reduction in length of runners named ST0.
- The second model, 18% reduction in volume of plenum chamber and 62% increase in length of runners named ST1.
- The third model, 17% reduction in volume of plenum chamber and 54% increase in length of runners named ST2.

TABLE 2. The geometrical specification of each design for I3 IM

	STB	ST0	ST1	ST2	ST3	ST4	ST5	ST6
Dp	a1=105, b1=95	a1=105, b1=95	a1=105, b1=95	a1=105, b1=95	a1=105, b1=95	a1=105, b1=95	a1=105, b1=95	a1=105, b1=95
Di	60	60	60	60	60	60	60	60
Dr1	38	38	38	38	38	38	38	38
Dr2	a2=37, b2=27	a2=37, b2=27	a2=37, b2=27	a2=37, b2=27	a2=37, b2=27	a2=37, b2=27	a2=37, b2=27	a2=37, b2=27
L	370	300	600	570	540	510	480	370
R	80	80	80	80	80	80	80	80
Vp(l)	1.432	1.429	1.429	1.442	1.429	1.429	1.429	1.442
Air entrance is in side position of IM								

- The fourth model, 18% reduction in volume of plenum chamber and 46% increase in length of runners named ST3.
- The fifth model, 18% reduction in volume of plenum chamber and 38% increase in length of runners named ST4.
- The sixth model, 18% reduction in volume of plenum chamber and 29% increase in length of runners named ST5.
- The seventh model, 17% reduction in volume of plenum chamber and no change in length of runners named ST6.
- The eighth model has no reduction either in plenum or in runner length. It has been supposed as the base IM carried over from I4 version, named STB.

3. GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

The equations which have been applied in the numerical analysis are as follows:

A) Conservation of mass (continuity)

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho u_i u_j\right)=S_m \tag{1}$$

where S_m is the mass added, during the phase changing.

B) Conservation of momentum

$$\frac{\partial}{\partial t}\left(\rho u_i\right)+\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)=-\frac{\partial P}{\partial x_i}+\frac{\partial \tau_y}{\partial x_j}+\rho g_i+F_i \tag{2}$$

where P is the static pressure, ρg_i and F_i are body force of gravity and external forces, respectively. τ_y is stress tensor that is expressed by the following equation (Equation (3)).

$$\tau_y=\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]-\frac{2}{3}\mu\frac{\partial u_j}{\partial x_j} \tag{3}$$

where μ is the molecular viscosity.

C) The equation of energy

$$\frac{\partial}{\partial t}\left(\rho E\right)+\frac{\partial}{\partial x_i}\left(u_i\left(\rho E+p\right)\right)=\frac{\partial}{\partial x_i}\left[K_{eff}\frac{\partial T}{\partial x_i}-\sum h_j h_{j'}+u_j\left(\tau_{ij}\right)_{eff}\right]+S_h \tag{4}$$

where K_{eff} is effective conductivity coefficient, $J_{j'}$ are the diffusion fluxes of components, and S_k is heat-induced chemical reactions or any other heat sources per unit volume. The three terms of right-hand of the equation are energy transfer by conduction, particles diffusion and viscosity dissipation, respectively.

In this study, it is assumed that the flow is fully turbulent and thus the effects of molecular viscosity be ignored. Therefore, this model will be valid for fully turbulent flows. k and ε are obtained from two transfer Equations (5) and (6).

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma k}\right)\frac{\partial}{\partial x_i}\right]+Gk+Gb-\rho\varepsilon-Y_M \tag{5}$$

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\partial\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k+C_{3\varepsilon}G_b\right)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

where G_k represents the turbulent kinetic energy produced by the average velocity gradient, G_b is the turbulent kinetic energy produced by the buoyancy and Y_m belongs to fluctuations due to the expansion of the compressible turbulence. C_{1z}, C_{2z} and C_{3z} are constants. In this model, turbulent viscosity can be expressed as:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{7}$$

where C_μ is constant. Constants of k-ε model have been used in FLUENT software as shown in Table 3. In this paper, the turbulence of the flow has been considered and investigated. In order to obtain good convergence in turbulence equation for steady state condition, the k-ε model is used in the turbulence numerical analysis.

TABLE 3. Constant of k-ε model

Constant	Value
C_{1z}	1.44
C_{2z}	1.92
C_μ	0.09
σ_k	1
σ_z	1.3

4. THE STEADY STATE SIMULATION PROCESS AND VERIFICATION

In the three-dimensional calculation, changes of the variables in all directions (i.e, x, y and z) are investigated. This type of simulation because of the nature for three-dimensional calculations is more complicated than the one-dimensional simulation. So, in this paper, the numerical solution of conservative equations are considered in three-dimensional way for more accuracy. The numerical simulation has been done as the following sequences:

1. Three-dimensional part for each design has been created
2. Each model has been meshed

TABLE 4. Different designs with their corresponding flow characteristics

	Design variables		Objective functions				
Design no.	**L (mm)**	**Vp (L)**	**CF_mean**	**M_mean**	**V_mean**	**Tur_mean**	**Deviation_ mean**
StB	370	1.432	0.8998278977	0.05982984143	53.96154192	10.11458448	0.0035231487
St0	300	1.429	0.86033	0.0572039	56.72423333	6.10411667	0.002088306595
St1	600	1.429	0.89442	0.0594705	52.67899667	11.48424667	0.003846648
St2	570	1.442	0.898754	0.0597583	52.70782333	6.306144667	0.004028141
St3	540	1.429	0.909173384	0.0604510	53.98761667	6.965046333	0.004231821
St4	510	1.429	0.9063167	0.0602611	53.20160667	6.599212667	0.00429516
St5	480	1.429	0.90988	0.06049842	53.79159333	6.936732333	0.0031047655
St6	370	1.442	0.9199212	0.06116567	54.63892333	8.406592	0.005405179

TABLE 5. CFD verification with experimental flow rate

	R1	**R2**	**R3**
CFD	69.10	69.45	69.23
EXP.	66.2	66.1	65.4
Errors (%)	4.39	5.07	5.87

3. A proper model of turbulence (i.e, k-ε) and appropriate numerical methods for equations discretization have been chosen.
4. An independent survey grid has been applied.
5. Final program has been run.
6. The results of the simulations have been analyzed.
7. Three-dimensional manifold negative volume model has been prepared in CATIA V5 R21 software and the 3D model has been meshed using ANSYS FLUENT.
8. To investigate the effect of the mesh quantity on the simulation results in steady state flow, meshes with 290000, 330000, 350000 and 365000 cells produced by ANSYS FLUENT software automatically. The independency from the quantity of mesh has been investigated with adapting meshing method in order to achieve an accuracy of at least 1e-6. The type of mesh configuration is unstructural and Tet/Hybrid. The most meshing configuration has four faces, except in corners with six faces and 8 nodes which has been applied.
9. Finally, due to small differences in the results, the mesh with 350000 cells has been used as reference mesh.
10. In the steady state simulation, the pressure boundary conditions were 100 kPa and 95 kPa, respectively at the inlet and the outlet of runners. Numerical simulation has been done on the eight different geometries at a Full-load engine condition and 6500 RPM. The deviation of mass flow rate between the runners has been calculated by Equation (8). Also CFD analysis will be verified based on experimental data from flow bench equipment with specification listed in the literature [15].

$$\text{Sigma}=\sqrt[2]{\left(\frac{(m1-Mmean)^2+(m2-Mmean)^2+(m3-Mmean)^2}{3}\right)} \quad (8)$$

where;

$$\text{Mmean}=(\text{m1}+\text{m2}+\text{m3})/3 \quad (9)$$

Table 5 shows the CFD accuracy which is limited by 5.87%. This verification has been done on STB design configuration.

5. ENGINE SIMULATION AND VERIFICATION

In this section the reliable ESC with experimental combustion pressure has been built. Among the available functions for solving combustion including Wiebe function and turbulence model, the turbulence model has been selected. Turbulence flame speed has a great effect on slope and rate of burning fuel and heat release rate in the combustion chamber. Hence, calibration and verification of ESC model is performed by sweeping FKGM and TFSM parameters.

Thus, in this model, all of the parameters related to geometry of the combustion chamber including speed of flame, air to fuel ratio and characteristic of the flow like swirl and tumble in the cylinder are assumed to be constant. In this way, affecting parameters on the speed of turbulence flame including FKGM and TFSM are swept. In the calibration process, it has been tried to close the experimental and theoretical combustion pressure curves and overlapped to each other as much as possible. Based on Figure 8, the maximum value of error between theoretical and experimental profiles is restricted to 6.15% and this fact shows the quality of calibration.

So, this verified model is applied for engine' performance calculations like torque and BSFC.

6. RESULTS AND DISCUSSION

6. 1. CFD Analysis Table 4 shows all different designs and flow specification indexes such as; mean of flow coefficient, mean mass flow rate, mean runner end flow velocity, mean turbulence and deviation of mass flow rate in three runners. For better discussion and ranking of the CFD results, the TOPSIS as a well-known method in MCDM with different weights have been applied. Table 6 shows the description of the five CFD objective functions considered in TOPSIS method. The weights of these objective functions have been listed in Table 7.

The weights of important functions have been selected, so that one function could be see as dominant among others. Based on Table 7, considering the highest mass flow rate (case 3) the design ST1 with 1.429l and 600 mm for plenum and runner's length has been selected. But, in point of minimum flow deviation between runners it is concluded that the design ST0 will have better performance. Considering the same weights for all the objectives functions, the design ST3 will be chosen as the final result from the 3D analysis. Based on Figure 2, ST1 has the best turbulence intensity and the highest flow velocity at runner's end, but the deviation of flow between the runners is not so acceptable.

Figure 3 clearly shows that the ST6 has maximum mass flow rate and flow coefficient but it cannot be nominated as the final design because of high deviation of mass flow rate between three runners.

Figures 4 to 6 illustrate the performance of three runners for each design. In point of mass flow rate (Figure 4), it could be stated that the third runner can send more air to cylinder one. The air charging side to I3 IM is set from back of engine which is near third runner. The cylinder number and runner number are considered and numbered vice versa. This phenomenon is due to the fact that the first runner is closer than the others to the inlet. Figure 4 shows that the mentioned runner (third) has the most deviation among the other runners in all the 8 designs. Figure 5 also could prove that the third runner which is closer to air inlet has the lowest mass flow rate among the other runners.

Considering high intensity of turbulence, Figure 6 shows that the highest runners have the greatest value and third runner has the more quantity among other two runners. Figure 7 illustrates that the flow coefficient has sensitivity to L, so that increasing the L could affect flow coefficient in IM. This curve can prove that there will be an optimum value for IM's runner and plenum.

6. 2. Engine Performance Analysis In the previous section, detailed investigation on IM for different designs has been analyzed by CFD. In this section all those designs have been investigated in ESC in order to see the engine performance.

TABLE 6. The CFD objective functions applied in TOPSIS

Objective functions	Description
F1	CF_mean
F2	M_mean
F3	V_mean
F4	Tur_mean
F5	Deviation_ mean

TABLE 7. Different weights of objective functions and selected design based on TOPSIS

Objective function	Weight					
	case1	case2	case3	case4	case5	case6
F1	0.6	0.1	0.1	0.1	0.1	0.2
F2	0.1	0.6	0.1	0.1	0.1	0.2
F3	0.1	0.1	0.6	0.1	0.1	0.2
F4	0.1	0.1	0.1	0.6	0.1	0.2
F5	0.1	0.1	0.1	0.1	0.6	0.2
TOPSIS output	ST6	ST5	ST6	ST1	ST5	ST0

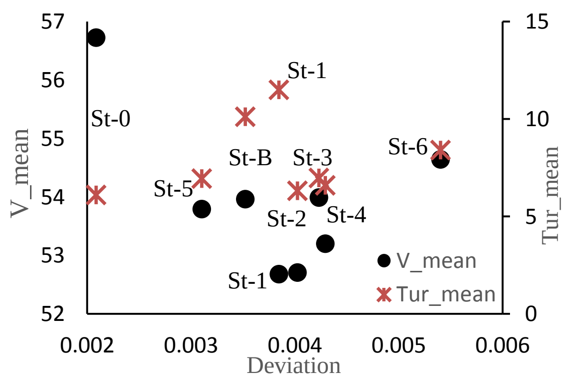

Figure 2. The variation of velocity vs deviation

TABLE 8. ESC output for different designs of IM

Performance	STB	ST0	ST1	ST2	ST3	ST4	ST5	ST6
Power	61.67	69.96	59.67	62.51	64.85	66.55	67.99	69.66
BSFC	337.89	327.26	343.10	339.7	337.06	334.76	332.14	327.46

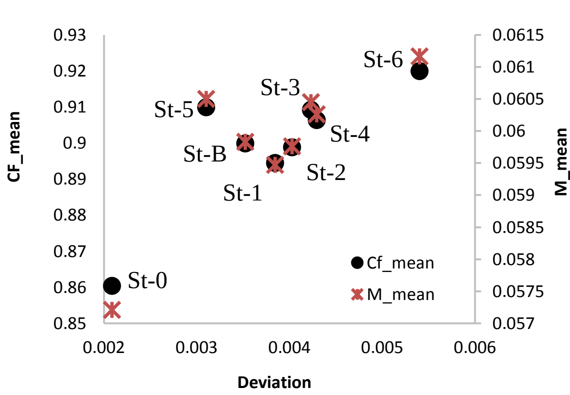

Figure 3. The variation of $CF_{\_mean}$ vs deviation

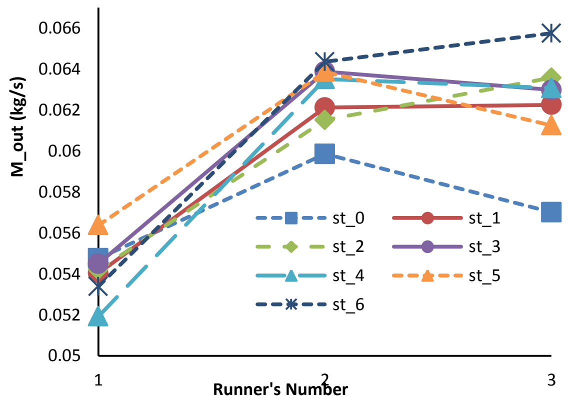

Figure 4. Mass flow rate in each design between runners in I3 IM

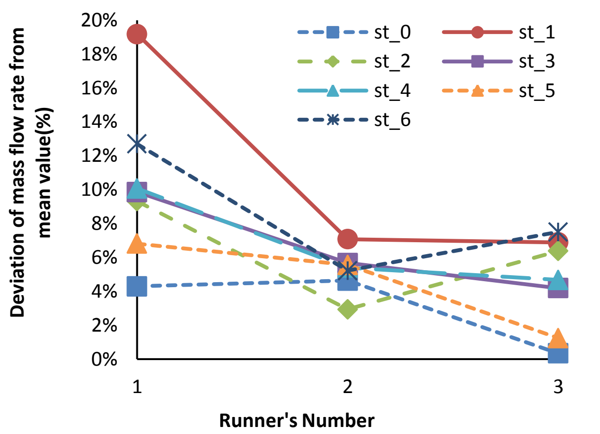

Figure 5. Mass flow rate dissipation from mean value in each design between runners in I3 IM

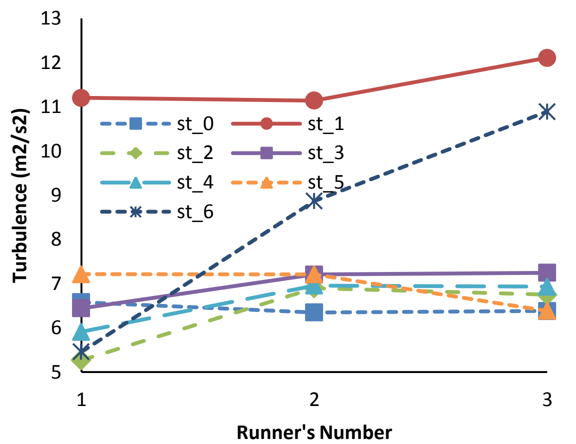

Figure 6. Turbulence intensity for each design between runners in I3 IM

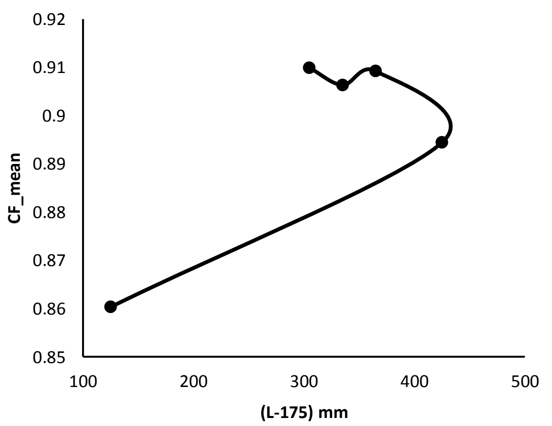

Figure7. Sensitivity of flow coefficient vs second part length of I3 IM

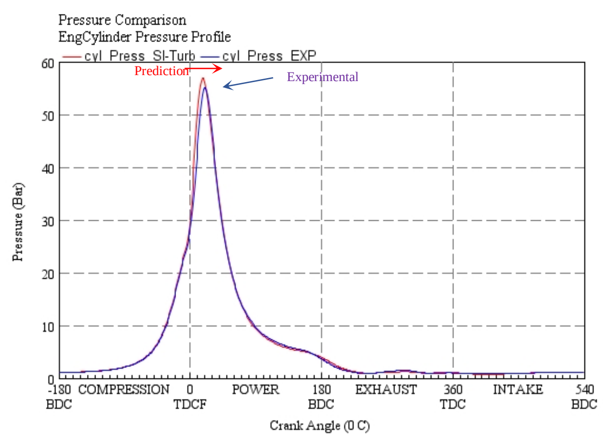

Figure 8. ESC validation in P-Teta combustion pressure

TABLE 9. Final results for hybridization of both technics in order to find BCD

ranking	CFD results	Design nominated based on ESC
1	-Highest runner end velocity - Minimum flow rate deviation	ST0
2	-Highest mass flow rate -Highest flow coefficient -Good turbulence intensity	ST6
3	-Good mass flow rate -Good runner end velocity -Low flow rate deviation	ST5

Through a steady state and Full-load CFD analysis which was discussed in detail at Section (5.1), it can be stated that, among the proposed eight design models of IM, Model ST5 with 18% reduction in volume of plenum chamber and 29% change in length of runners, had a good mass flow and high flow coefficient value with low deviation between the runners, compared to the other proposed models, and consequently could be nominated from 3D point of analysis. But, as all engine researchers know, the 3D calculation shall not be sufficient, and applying of ESC is necessary in this regards.

In this section all the designs have been applied to ESC and the engine' performance has been analyzed. Table 8 shows that from the engine performance point of view, the configurations ST0, ST6 and ST5 shall be nominated from 3D analysis. Final and summary of results of the present research are shown in Table 9.

7. CONCLUSION

In this study, the IM of an ICE was investigated by two powerful techniques in mechanical engineering. First, several designs were investigated in 3D analysis and then an ESC was applied to them.

In order to increase the efficiency of work, eight different geometries for intake manifold were modeled and imported to CFD software for numerical assessment. Performance of all the eight models were compared with each other in point of flow coefficient, flow rate, flow deviation between runners, flow velocity and turbulence intensity. Then all designs were imported to ESC and investigation of power and BSFC started. The following items could be summarized as the results of this research:

1- None of the techniques (i.e, CFD and ESC) are sufficient for design of engine air filling systems independently. The best method is following the research with hybridization of two methods in a closed looped way.

2- It was concluded that from the CFD calculation point of view, the best effective IM tends to get higher runner's length. But, considering the ESC output, lower length results in more power.

3- Although the high value of runner's length has better flow characteristics (i.e, intensity of turbulence, flow velocity and higher flow coefficient), but because of more deviation between the runners and wave tuning effects, it does not produce more power. So in ESC output point of view, the lowest runner length (i.e, 300mm) with 1.429l plenum would be nominated for I3 engine.

4- Hybridization and combination of these two methods in a closed looped solution has the best output for intake manifold. Therefore, it can be stated that the ST0 can satisfy the engine performance from the flow behavior point of view.

5- The difference between 3D analysis and ESC can be attributed to some unsteady conditions in cylinders causing the expansion waves which are back to the intake manifold. These waves can be reflected in dealing with the manifold and send positive pressure waves to cylinders and make more air charge filling.

8. REFERENCES

1. U. Kesgin, "Study on the design of inlet and exhaust system of a stationary internal combustion engine," ***Energy Conversion and Management***, Vol. 46, No. 13-14, (2005), 2258-2287.
2. M. A. Ceviz, "Intake plenum volume and its influence on the engine performance, cyclic variability and emissions," ***Energy Conversion and Management***, Vol. 48, No. 3, (2007), 961-966.
3. M. A. Ceviz and M. Akın, "Design of a new SI engine intake manifold with variable length plenum," ***Energy Conversion and Management,*** Vol. 51, No. 11, (2010), 2239-2244.
4. S. Potul, R. Nachnolkar, and S. Bhave, "Analysis Of Change In Intake Manifold Length And Development Of Variable Intake System", ***IJSTR*** Vol 3, No. 5, (2014).
5. S. Jafarmadar and R. Barzegar, "Three dimentional modeling of combustion process and emissions formation pre and main chambers of an indirect injection diesel engine," ***International Journal of Engineering,*** Vol. 25, (2012), 311-320.
6. J. Taylor, N. Fraser, D. Gurney, R. Dingelstadt, J. Stehlig, and V. Bruggesser, "Optimisation of Variable Length Intake Manifolds for a gasoline turbocharged downsized engine," in Internal Combustion Engines: Improving Performance, Fuel Economy and Emission, Imeche, Ed., ed: Woodhead Publishing, (2011), 27-39.
7. R. Tavakkoli-Moghaddam and S. M. Mousavi, "An integrated AHP-VIKOR methodology for plant location selection," ***International Journal of Engineering,*** Vol. 24, , (2011), 127-137
8. S. H. Zegordi, E. R. Nik, and A. Nazari, "Power plant project risk assessment using a FUSSY-ANP and FUZZY-TOPSIS

method", ***International Journal of Engineering***, Vol. 25, (2012), 107-120.

9. M. Velasquez and P. T. Hester, "An analysis of multi-criteria decision making methods," ***International Journal of Operations Research,*** Vol. 10, (2013), 56-66.
10. M. Yahyaei, M. Bashiri, and Y. Garmeyi, "Multi-criteria logistic hub location by network segmentation under criteria weights uncertainty (Research Note)," ***International Journal of Engineering,*** Vol. 27, (2014), 1205-1214.
11. B. Vahdani, M. Salimi, and S. M. Mousavi, "A new compromise decision-making model based on TOPSIS and VIKOR for solving multi-objective large-scale programming problems with a block angular structure under uncertainty", ***International Journal of Engineering,*** Vol. 27, (2014), 1673-1680.
12. Y.-J. Wang, "A fuzzy multi-criteria decision-making model based on simple additive weighting method and relative preference relation" ***Applied Soft Computing,*** Vol. 30, (2015), 412-420.
13. M. Aghajani Mir, P. Taherei Ghazvinei, N. M. N. Sulaiman, N. E. A. Basri, S. Saheri, N. Z. Mahmood, et al., "Application of TOPSIS and VIKOR improved versions in a multi criteria decision analysis to develop an optimized municipal solid waste management model," ***Journal of Environmental Management,*** Vol. 166, (2016), 109-115
14. R. Lourenzutti and R. A. Krohling, "A generalized TOPSIS method for group decision making with heterogeneous information in a dynamic environment," ***Information Sciences,*** Vol. 330, (2016), 1-18
15. Shojaeefard .M.H., Khalkhali .A., Firouzgan. A. "Experimental and numerical flow investigation of intake manifold and multi criteria decision making on 3-cylinder SI engine using TOPSIS", ***International Journal of Engineering,*** Vol. 28, No. 11, 1663-1670.

Intake Manifold Flow Assessment on a 3-cylinder Natural Aspirated Downsized Engine Using CFD and GT-SUITE

M. H. Shojaeefard, A. Khalkhali, A. Firouzgan

Automotive Simulation and Optimal Design Research Laboratory, School of Automotive Engineering, Iran University of Science and Technology, Tehran, Iran

P A P E R I N F O

Paper history:
Received 02 December 2015
Received in revised form 27 December 2015
Accepted 26 January 2016

Keywords:
Intake Manifold
Computational Fluid Daynamics
Intake Manifold Model
Engine Simulation Code
Volumetric Efficiency

چکیده

در این مقاله در نظر است طراحی چند راهه هوا بعنوان یکی از تاثیر گذارترین قطعه موتور جهت افزایش راندمان حجمی، با تاکید برتلفیق محاسبات جریان (CFD) و سیکل شبیه سازی موتور بصورت دقیق مورد بررسی قرار گیرد. هشت طرح بهمراه طرح موتور پایه (چهار سیلندر) در حجم و طول شاخه های متفاوت در نرم افزار CATIA طراحی شد. کلیه طرح ها بصورت مجزا و تنها با بکارگیری شبیه سازی عددی سه بعدی، محاسبات جریان می‌گردد. پنج تابع هدف در محاسبات جریان شامل، توزیع حداکثر دبی جرمی، حداقل انحراف درتوزیع دبی در شاخه ها، داشتن حداکثر ضریب جریان، حداکثر سرعت جریان و حداکثر شدت اغتشاشات در شاخه‌ها تعریف و با استفاده از تکنیک تصمیم گیری چند معیاره (MCDM) با بکارگیری وزن های مختلف، رفتار و مشخصه های جریان در هر شاخه به روش TOPSIS تحلیل می‌شوند. در ادامه از یک نرم افزار شبیه سازی چرخه ترمودینامیکی با مدل احتراق توربولانسی، که با نتایج فشار تجربی احتراق (بدست آمده در آزمایشگاه) صحه گذاری شده است، استفاده می‌گردد. کد شبیه سازی موتور دارای دقت ٦/١٥٪ بوده و می تواند در پیش بینی عملکرد موتور و یا هر گونه تغیرات در اجزاء موتور بکار گرفته شود. در این مقاله شبیه سازی سه بعدی جریان توسط نرم افزار فلوئنت و شبیه سازی موتور توسط نرم افزار GT-SUITE انجام شده است. نتایج نشان می‌دهد که تلفیق نتایج بدست آمده از دو روش با یکدیگر انتخاب درست و دقیق تری را برای طراح بدنبال خواهد داشت و شبیه سازی قطعه بدون ارزیابی عملکرد آن بر روی موتور دارای خطا خواهد بود. در پایان بهترین طرحها که دارای بهترین مشخصه های جریان و توزیع مطلوب که همراه با حداکثر توان باشند ارائه و آنالیز می‌شود.

doi*: 10.5829/idosi.ije.2016.29.02b.15*