Evaluation of CFD Sub-Models for the Intake Manifold Port Flow Analysis

M. M. Khan*, S. M. Salim
School of Engineering, Taylor’s University, Selangor, Malaysia

*Email: mohsenkhan82@yahoo.com

Abstract—The purpose of this research is to appraise and simulate computational fluid dynamics (CFD) models, with respect to computing time, which can predict the flow conditions in an Intake Manifold (IM) of naturally aspirated Spark ignition (SI) engine by using flow analysis numerical model from ANSYS CFD software (FLUENT 14.0). The simulations are carried out by using different steady state Reynolds Averaged Navier Stoke (RANS) turbulence models such as Standard k-epsilon (k-ε) Re-Normalization Group k-ε (RNG), Realizable k-ε, k-omega (k-ω), and Reynolds Stress Model (RSM). Initial Boundary conditions and validation of simulation results based on experimental data are used. It is observed that each model produces almost similar results when it comes to pressure drop inside plenum and inlet velocity. However, RNG k-ε accounts for swirl dominated flow, predicting better flow conditions inside the manifold. Therefore, RNG k- ε is best to capture intake manifold flow turbulent conditions.

Keywords— Fluent, Computational Fluid Dynamics (CFD), Intake Manifold, Realizable k-ε, k-omega (k-ω), Re-Normalization Group k-ε (RNG), Reynolds Stress Model (RSM), Turbulent flow.

1. Introduction

The engine of a car needs air for the combustion process in the cylinders. Air Intake Manifold (IM) has an important role in getting the desirable amount of air into automobile engine by improving the combustion efficiency and reducing air pollution [1]. The main purpose of an air IM is to provide the engine with constant amount of required clean air to burn in the combustion chamber. The air enters the combustion chambers through IM, in which plenum box distributes the flow uniformly into the respective ports which goes to the combustion chamber. To optimize the flow inside intake system, thorough understanding of flows, inlet velocity and pressure drop through the system is essential [2].

The main objective of this study is to analyze different Reynolds Averaged Navier Stoke (RANS) turbulence models for better flow prediction of Intake Manifold. The numerical model of Proton Satria Neo 1.6L Air IM (Fig. 1) is used for the simulations. FLUENT is used to analyze the internal flow of air intake manifold and to get the results which are validated from the data collected from experiment.

2. Research Methodology

Numerous studies on IM optimizations have been conducted. Kale et al. [3] analyzed the steady flow inside the IM of a S.I engine using STAR-CD but only considered Standard k-ε model to analyze optimum flow conditions inside IM. David et al. [4] studied the flow characteristic of flow inside the swirl helical IM port and concluded that CFD has high tendency on providing ways to design high swirl generating capacity intake ports which can help to reduce NOx emissions. However, little research has been done regarding the testing of CFD models which can analyze the flow inside IM accurately.

A three dimensional (3-D) geometry of IM consisting inlet, plenum and 4 outlet ports is developed. The mesh generated consists of 348,730 numbers of tetrahedral cells. To verify the adequacy of the grid, grid independent test was conducted [5]. Pressure based boundary conditions (from experiment, Fig. 2) are assigned to the inlet and outlets of IM. 3-D, steady state numerical simulations are carried out using Reynolds Averaged Navier Stoke (RANS) Standard k-ε, RNG k-ε, Realizable k-ε, k-ω and RSM. It is noted that RNG performed exceptionally well as it accounts for swirl dominated flows [6]. RSM under relaxation factor also shows desirable results but due to higher computing time (Table 1) and cost it is less feasible.

Table 1: Computational Time Taken To Run Each Model

<table>
<thead>
<tr>
<th>Models</th>
<th>Estimated Computational Time (Hour)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Standard k-ε</td>
<td>2.5</td>
</tr>
<tr>
<td>2 Realizable k-ε</td>
<td>3</td>
</tr>
<tr>
<td>3 RNG k-ε</td>
<td>3.5</td>
</tr>
<tr>
<td>4 k-ω</td>
<td>3</td>
</tr>
<tr>
<td>5 RSM</td>
<td>4.5</td>
</tr>
</tbody>
</table>

Experimental Setup (Fig. 2) is developed for the initial boundary conditions and validation of CFD simulations. Pressure sensor is placed at the exit of outlet ports and at the plenum of IM to measure the pressure difference (Placement indicated in Fig. 1). The signals from pressure sensor are recorded by Engine Control Unit (ECU) of the car which is capable of data logging up to 200 Hz. The plots of static pressure and velocity magnitude of each model and experimental data are drawn and the difference in the discretization of each model is shown in the Fig. 3.
3. Results and Discussion

The velocity magnitude and static pressure profiles of the IM plenum chamber of all the models are shown in Fig. 3a and Fig. 3b respectively. It is observed that velocity magnitude discretization of each model differs greatly while static pressure drop has almost the same results with minor fluctuations. The theoretically calculated inlet velocity (using the piston swept volume of 1.6L) is well predicted by RNG k-ε and Standard k-ε with up to 5 percent error while Realizable, k-ω and RSM shows discrepancies giving up to 10 percent error. For static pressure all model shows larger deviation in the pressure drop compared to experimentally calculated values. It is observed that the experimental data gathered also have uncertainty when it comes to sensor effectiveness and leakages. Furthermore, the piston movement causes fluctuating backflow velocities and pressure changes inside the ports, making it difficult to get accurate boundary conditions which are used to predict the flow inside the manifold using RANS steady state models.

The velocity vector and streamlines plots (Fig. 4a & Fig. 4b) obtained from the analysis shows the better prediction of the high turbulent flow inside the ports and plenum chamber of the IM, as found in other literature reviews [2] [3] [4] [5]. Beside the graphical results, vortices formed in RNG k-ε shows proper distribution of the flow in all the ports and plenum chamber. Hence making it better choice to analyze the flow inside intake manifold.

4. Conclusion

The investigation of the flow regime, within the intake manifold, using CFD analysis is possible. However, applying the correct parameters and CFD models, which can provide economical simulated results, is highly important. This study appraises different CFD turbulent models and the importance of realistic parameters to successfully predict the flow inside IM. It is noticed that each model shows variation with the experimental results. While, RNG and RSM shows promising results with lower margin of error compared to standard k-ε and k-ω.

For future studies, it is important to analyze each model using transient flow conditions for better accuracy and predictability of the flow inside IM with part throttle conditions. Furthermore, for future validations high calibrated sensors should be used to improve the accuracy of experimental data obtained.

References