Numerical simulation of in-cylinder tumble flow field measurements and comparison to experimental results

A. Mohammadebrahim¹, B. Shafiei², S. Kazemzadeh Hannani³*

¹Sharif University of Technology and Irankhodro Powertrain Company (IPCO), Tehran, Iran, m.ebrahim@ip-co.com
²Sharif University of Technology, Tehran, Iran, behshad@sharif.edu
³Sharif University of Technology, Tehran, Iran, hannani@sharif.edu

*Corresponding Author, Phone Number: +98-912-3018408

ARTICLE INFO

Article history:
Received: 22 September 2012
Accepted: 08 April 2013

Keywords:
Tumble
Intake port
Gas exchange
Flowbench test
Flow coefficient

ABSTRACT

This paper presents a comparison between measured and predicted results of the in-cylinder tumble flow and the flow coefficient generated by a port-valve-liner assembly on a steady-flow test bench. In this study, computational fluid dynamics (CFD) methods were employed to gain further insight into characteristics of an engine. The purpose was to advance understanding of the stationary turbulence process via computational techniques, at the same time. A simultaneous computer simulation was carried out to predict the in-cylinder flow field of the same engine under the same operating condition, using the Fluent software. The tumble ratio and the flow coefficient from both the numerical simulation and the experimental measurement were compared. A reasonably good level of the agreement has been achieved. Finally, this paper provides a useful validation study for a range of turbulence models for the in-cylinder flow and shows the rate of the tumble flow dispersion in the cylinder.

© Iran Society of Engine (ISE), all rights reserved.
1) Introduction

Improvements in computer processing power and fluid simulation codes have resulted in rapid advancements in computer-based engine simulation. The use of three-dimensional CFD codes allows for greater understanding of flow dynamics in the parts of gas exchange system, before any prototypes are ever manufactured [1]. In fact, many manufacturers are routinely utilizing CFD as part of their engine design process [2-3]. Steady-state CFD simulation comparison with steady-state flow bench results has been widely applied in academic and industrial research. Many researchers have focused on the improvements of volumetric efficiency, tumble ratio, and swirl ratio [4]. Guy et al. [5] used the commercial CFD package Fluent to complement the traditional optimization strategies. They studied a racing car's flow coefficient using both CFD simulation and experiment. A comparison of results between a CFD prediction and a flow bench measurement demonstrated the validity and usefulness of this approach.

Extensive investigations have been reported on the subject of validation of the CFD codes with the flow field measurements of internal combustion engines [6]. While experimental techniques can render acceptable parameter sensitivity results [7-9], such approach is exhaustive, expensive and time consuming. Both experimental and numerical methods still have a long way to be improved from the academic viewpoint. Nevertheless, for industrial applications, they are useful design tools, provided that the errors are within acceptable limits. The results of experimental investigations are global flow parameters, describing flow coefficient and swirl and tumble intensity. The ongoing progress for the development of CAE tools has led to the creation of new methodologies. CFD calculations provide insight into the details of port and in-cylinder flow, thus enabling efficient optimization.

In the present study, numerical simulations are done with tumble adaptor to have similarity between simulation and experimental test, while previous studies ignored this adaptor. However others just use one or two turbulence model(s), in this study, different turbulence models is applied and compared. Finally swirl/tumble are calculated and presented in some planes that will be useful to study swirl/tumble from top to piston surface.

2) Experimental apparatus

The flow-bench experiments are performed to quantify the in-cylinder tumble motion and engine volumetric efficiency by using the same cylinder head and valve. The experimental set up is shown in Figure 1 schematically. Special mechanisms and fixtures (1) are used to adjust valves lift. In standard tests on engines with four valves per cylinder, inlet or outlet valves are open simultaneously. Test is performed on cylinder head or flow box (2) and a dummy cylinder (3) is used with a diameter equal to engine bore. Pressure drop is measured with a pressure transducer (4) relative to atmospheric pressure. A barometer (7) is used to determine the pressure drop in orifice (8) and consequently to measure the volume flow rate. Desired differential pressures are supplied with a bypass valve (10) and airflow temperature is measured using a thermocouple (11) for air mass flow rate correction.
Generally, swirl meter measures rotational speed of paddle wheel (because of inlet air flow) as intensity of swirl or tumble (Figure 3(a)), or as shown in Figure 3(b) a torque can be applied from the air to a honeycomb to take into account the intensity of the swirl or tumble. If swirl meter is used with an adaptor like Figure 1, tumble flow is measured and if it is used in orientation like Figure 3, swirl flow is measured.

Figure 3: Swirl meter types (a) rotational speed measurement (b) torque measurement [11]

3) Governing Equations and Relations

Using air as the working flow and neglecting body force, the governing equation for homogeneous, compressible, Newtonian fluid include continuity, momentums and \( k - \varepsilon \) equations. These equations are fully described in the literature [12].

The flow and discharge coefficient are defined as the ratio of the experimentally obtained mass flow rate \( \dot{m}_{\text{meas}} \) to the theoretical mass flow rate \( \dot{m}_r \).

\[
C_f = \frac{\dot{m}_{\text{meas}}}{\dot{m}_r} \tag{1}
\]

If the flow is subsonic, the reference mass flow rate is given by

\[
\dot{m}_r = A_{\text{ref}} \frac{p_0}{\sqrt{RT_0}} \left( \frac{p_T}{p_0} \right)^{\frac{1}{\gamma}} \tilde{p} \tag{2}
\]

\[
\tilde{p} = \left\{ \frac{2\gamma}{\gamma - 1} \left[ 1 - \frac{p_T}{p_0} \right] \left( \frac{p_0}{p_T} \right)^{(\gamma - 1)/2} \right\}^{1/2} \tag{3}
\]

While, if the flow is choked, the mass flow is calculated as follows:

\[
\dot{m}_r = A_{\text{ref}} \frac{p_0}{\sqrt{RT_0}} \gamma^{1/2} \left( \frac{2}{\gamma + 1} \right)^{(\gamma + 1)/2} \tag{4}
\]

When the intake phase is analyzed, \( p_0 \) is the intake system pressure, \( p_T \) is the cylinder pressure, \( T_0 \) is the intake system temperature and \( A_{\text{ref}} \) is the reference area.

The difference between the discharge and flow coefficient lies in the definition of the reference area \( A_{\text{ref}} \).

For the discharge coefficient, this area is the valve curtain area and, therefore, it is a linear function of valve lift \( L_v \).

\[
A_{\text{ref}} = \pi D_v L_v \tag{5}
\]

For the flow coefficient, the reference area is given by the cylinder bore:

\[
A_{\text{ref}} = \frac{\pi B^2}{4} \tag{6}
\]

Finally, the flow coefficient is the most common method for assessing the flow efficiency of a port design. The flow coefficient is a non-dimensional parameter that essentially scales the flow rate obtained through a restriction by the theoretical maximum flow rate possible at the restriction’s cross-sectional area.

In order to compare the numerical and experimental results, the speed of each node was converted to the volume-average angular momentum:

\[
T = \int_0^a [U^2] U \rho x dx \tag{7}
\]

where \( T \) is tumble torque (N.m), \( U \) is velocity axial to the cylinder (m/s), \( \rho \) is density (kg/m\(^3\)), \( x \) is distance
from axis of rotation (m), $a$ is area of the plane (m$^2$) and $A$ is area (m$^2$).

4) Computer Simulation

Computer simulation of the steady flow test for the same engine (Table 1) was conducted on a workstation using Fluent 13. Because of symmetry in cylinder head, half model is used. The parameters used in the problem set-up are listed in Table 2. The boundary conditions for the model are selected to mimic the conditions of the steady-state flowbench. That is, the pressure gradient across the intake and outlet of the system were specified. The inlet flow into the mesh was confined to be normal to the boundary. Most of the results shown in this paper used a pressure drop of 5.0 kPa, which is commonly used for flowbench testing of conventional engines.

<table>
<thead>
<tr>
<th>Table 1: Specifications of the EF7-NA engine</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>General data</strong></td>
</tr>
<tr>
<td>bore (mm)</td>
</tr>
<tr>
<td>stroke (mm)</td>
</tr>
<tr>
<td>displacement (cc)</td>
</tr>
<tr>
<td>inlet port diameter (mm)</td>
</tr>
<tr>
<td><strong>Intake valve</strong></td>
</tr>
<tr>
<td>diameter (mm)</td>
</tr>
<tr>
<td>stem diameter (mm)</td>
</tr>
<tr>
<td>maximum lift (mm)</td>
</tr>
<tr>
<td>seat angle (deg)</td>
</tr>
<tr>
<td>inclination (deg)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2: Problem set-up</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Function</strong></td>
</tr>
<tr>
<td>solver</td>
</tr>
<tr>
<td>temporal discretization</td>
</tr>
<tr>
<td>pressure discretization</td>
</tr>
<tr>
<td>momentum discretization</td>
</tr>
<tr>
<td>pressure-velocity coupling</td>
</tr>
<tr>
<td>energy discretization</td>
</tr>
<tr>
<td>turbulence modeling</td>
</tr>
<tr>
<td>near wall treatment</td>
</tr>
<tr>
<td>fluid model</td>
</tr>
<tr>
<td><strong>Setting</strong></td>
</tr>
<tr>
<td>steady</td>
</tr>
<tr>
<td>implicit</td>
</tr>
<tr>
<td>2nd order upwind</td>
</tr>
<tr>
<td>2nd order upwind</td>
</tr>
<tr>
<td>PISO</td>
</tr>
<tr>
<td>2nd order upwind</td>
</tr>
<tr>
<td>RNG k-ε</td>
</tr>
<tr>
<td>sta. wall function</td>
</tr>
<tr>
<td>air (ideal gas)</td>
</tr>
</tbody>
</table>

It should be mentioned that a grid-independence study was performed on this model. Starting with the coarsest grid that could be meshed successfully, and examining progressively finer grids, the predicted $C_f$ for a valve lift of 8 mm were compared. Figure 4 shows that the 99500 tetrahedral cells produced very similar results to the finer grids. This mesh size was used for the remainder of the study. Steady-state computation was performed. The valve position was kept constant in the simulation. Reshaping of the grid structure near the interface between the valve seat and the dome cylinder head was performed. Figure 5 shows the final product of grid generation of the port-valve-liner-adaptor assembly with the dome cylinder head.

The grid is denser in the top region, because the flow is expected to be more complicated in this area.

In CFD calculations seven different turbulence models have been applied including RSM model. As it is shown in Figure 6, average vorticity magnitude in cylinder and mass flow rate at outlet are independent of model. These parameters are normalized with maximum values. Maximum difference is 1% in mass flow rate and 9% in vorticity. Also, for more exploration, 10 planes with 10 mm distance are defined (Figure 7) and swirl/tumble torques are calculated and compared versus different turbulence models (Figure 8). Finally, the RNG k-ε turbulence model which
performed more robust for this problem is used with default wall functions. The steady inflow and outflow rates need at least 15000 computational cycles to converge.

In summary, based on the computational results, this problem is independent of turbulence model employed.

5) Simulation Results and Comparison

A comparison of $C_f$ predicted with the CFD model to the $C_f$ measured on a flowbench is shown in Figure 9. The predicted and measured $C_f$ trends are quite similar which provides more confidence that the CFD model is able to replicate the main flow trends within the intake system.

A benefit of using CFD to analyze the intake flow is that it is now possible to examine the detailed behavior of the intake flow at any location within the port, head, or cylinder.
The simulation results are illustrated by velocity vector diagrams, as displayed in Figures 10 and 11. They are the cross sections on the YZ, and XY planes, respectively. Figure 10 reveals that there are two tumble jets inside the cylinder, one is the main jet with a large vortex guided by the dome chamber; the other is a counter rotating jet with much smaller scale. The maximum velocity is located around the inlet valve curtain. Note that the vectors point downwards at the BDC, because the piston was replaced by the outflow boundary. Figure 11 shows the velocity vector diagram in three horizontal planes. They located at 10 mm, 20 mm, and 30 mm beneath the TDC. With different scales, the flow patterns show similarities in different locations. The flow field tends to be more homogeneous as the flow further downwards.

Figure 10: Computed velocity vector diagrams in the YZ plane (symmetry plane and parallel to cylinder axis)

Figure 11: Computed velocity vector diagrams in XY planes, which are located at (b) 10 mm, (c) 20 mm and (d) 30 mm beneath TDC, respectively (Planes are parallel to piston plane and because of symmetry just half-planes are shown)
The tumble is shown in Figure 12. The comparison shows that it is in reasonably good agreement between the experimental and simulation results.

The intake flow behavior was found to vary significantly as a function of valve lift. Specifically, it was found that at low lifts the intake flow is relatively uniform around the periphery of the intake port, but at high valve lifts the flow into the cylinder is biased towards the top of the intake port. This results in a tendency to promote tumble at high valve lifts, but not at low valve lifts.

Test is repeated 10 times and error bars indicate uncertainty of each test (Figures 9 and 12). In the flow coefficient measurement, uncertainty is about 1% and in tumble measurement is 3%.

6) Conclusion

The flow bench measurement adopted in this research is able to provide accurate quantitative answers. The simulation and experimental results are in reasonable level of agreement. The discrepancy is acceptable to the engine designer. The accuracy is limited by the geometry modeling, CFD limitations, uncertainties of the turbulence modeling and some complex boundary conditions. This paper has validated the engine simulation code, Fluent, in the case of an engine with dome shape combustion CFD provides useful qualitatively information of the flow field in details.

The intake flow behavior was found to vary significantly as a function of valve lift. Specifically, it was found that at low lifts the intake flow is relatively uniform around the periphery of the intake port, but at high valve lifts the flow into the cylinder is biased towards the top of the intake port. This results in a tendency to promote tumble at high valve lifts, but not at low valve lifts.

Finally, this paper provides a useful validation study for a range of turbulence models for the in-cylinder flow and shows rate of tumble flow dispersion in the engine cylinder.

Acknowledgment

Support from CAE department of Irankhodro Powertrain Company (IPCO) is greatly appreciated.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>p&lt;sub&gt;0&lt;/sub&gt;</td>
<td>Intake system pressure</td>
</tr>
<tr>
<td>pr</td>
<td>Cylinder pressure</td>
</tr>
<tr>
<td>R</td>
<td>Gas constant</td>
</tr>
<tr>
<td>T&lt;sub&gt;0&lt;/sub&gt;</td>
<td>Intake system temperature</td>
</tr>
<tr>
<td>γ</td>
<td>Ratio of specific heat</td>
</tr>
</tbody>
</table>

References

فصلنامه علمی- پژوهشی تحقیقات موتور

www.engineerresearch.ir

تارنمای فصلنامه:

شیب‌سازی عددی اندازه‌گیری جریان گردبادی داخل استوانه و مقایسه آن با نتایج تجربی

ابوالفضل محمدابراهیم و همکاران، فصلنامه علمی- پژوهشی تحقیقات موتور، شماره 62 (بهار 1391)، صفحه 11-13

چکیده

در این مقاله، مقایسه‌ای بین نتایج اندازه‌گیری و بیش‌پیشینه شده ضریب جریان و جریان گردبادی با محور افقی انجام شد. نتایج در مجموعه‌ای از اجزای شامل راهگاه و کانالیتیک سیالات محاسباتی با استفاده از آزمون جریان پایا بدست آمد. روش‌های دینامیک سیال محاسباتی با استفاده از روش‌های محاسباتی بهتر از مشخصات جریان و قابلیت استفاده در روش‌های محاسباتی بهتر استفاده شده. با استفاده از روش‌های محاسباتی، میزان خروج و جریان گردبادی انجام شد. نتایج شیب‌سازی و آزمون تجربی، همخوانی مناسبی را با یکدیگر نشان دادند. در ادامه، با استفاده از داده‌های مختلف آزمونی، تأثیر آنها بر متغیرهای مورد نظر بررسی گردید و نهایتاً میزان جریان گردبادی در صفحات مختلف استوانه ارزیابی شد.

تمامی حقوق برای انجمن علمی موتور ایران محفوظ است.