

The Investigation and Application of Variable Tumble Intake System on a GDI Engine

He Changming and Xu Sichuan
Clear Energy Automotive Centre

ABSTRACT

The in-cylinder tumble intensity of GDI engine is crucial to combustion stability and thermal efficiency, required to be different for the different operation conditions. A new variable tumble system (VTS) applied to GDI engine was introduced to meet tumble ratio requirements in various situations. The transient gas exchange of four combustion systems all were investigated during both intake and compression strokes based on CFD simulation, namely (1) Case 1-Intake port B (with flap valve)/Spherical piston crown; (2) Case 2-Intake port B (without flap valve)/Spherical piston crown; (3) Case 3- Intake port A/Spherical piston crown; (4) Case 4-Intake port A/Dented piston crown. The simulated results of dynamic tumble ratio showed that during the whole intake process the dynamic tumble ratio of Case1 was obviously higher than other Cases with the same boundary conditions, and the maximum value was about 5~6 times higher. The crank angle range, in which the strong tumble motion fully developed, was merely about 70°CA, roughly from 410°CA to 480°CA. Given that the in-cylinder airflow could not form rotation motion around one axis that parallel to Y vector in this crank angle interval, with the intake valve opening gradually becoming smaller, it was impossible to generate higher tumble intensity again in the next crank angles. Moreover, for Case1 with a flap valve in intake manifold, the mean velocity and turbulence kinetic energy all were almost twice than those of other Cases when piston close to TDC. In terms of airflow streamlines or contours of in-cylinder flow field it was concluded that the geometric shape of intake port without flap valve should be modified to further increase tumble intensity, specially, in some locations that sensitive to tumble ratio. While the piston crown shape is not vital to tumble intensity change in terms of calculated results. Eventually for the simulated cases the corresponding prototypes of cylinder head were manufactured and processed to carry out a series of steady state flow testing. For tumble ratio there was significant difference among all Cases, which consistent with simulated results. The tumble ratio of VRS will mostly be changed from 1.56 to 2.8, as flap valve fully closed or open, respectively.

CITATION: Changming, H. and Sichuan, X., "The Investigation and Application of Variable Tumble Intake System on a GDI Engine," *SAE Int. J. Engines* 7(4):2014, doi:10.4271/2014-01-2885.

INTRODUCTION

The in-cylinder charge air motion for the Gasoline Direct Injection (GDI) engine has attracted so much attention to design engineer of internal combustion engine [1]. With the development and improvement of GDI technology, the various ways of airflow motion in cylinder successively emerged, such as piston top wall guidance, air charging motion of intake port and injector spray guidance, etc. [2]. The lean-mixture combustion system have fully developed, the typical lean combustion with in-cylinder direct injection are generally classified into swirl and tumble stratification [3,4]. The former was employed in D-4 GDI of Toyota, Audi 18TD and NEODI engine of Nissan, [5,6] etc. While the Mitsubishi 4G93 GDI and Ford-EcoBoost engines all are representatives of tumble stratification [7,8]. It's well known that for the latter the in-cylinder charge motion usually dominates the mixing process of fuel and fresh charge air [9]. The turbulence intensity will produce crucial influence on ignition stabilization and flame propagation velocity [10]. For a turbocharged gasoline engine

the charge motion has to meet various operation requirements compared to conventional naturally aspirated ones [11]. The basic advantages of higher tumble ratio exist in the following items: (1) Reduced fuel spray penetration, i.e. wall wetting. (2) To form well-distributed mixture. (3) Better combustion stabilization. (4) Even faster flame propagation velocity. However, too intense tumble will also bring about some problems, such as higher levels of combustion noise and increased pump loss for lower discharge coefficient [12]. Generally, as the GDI engine with larger mass flow rate during the air suction the in-cylinder tumble ratio will be increased accordingly compared to the lower speed or idle operation condition, due to the decreasing airflow velocity [13,14].

Based on the above reasons, it is required that for the relative lower or idle speed the tumble ratio of GDI engine is desired to be on higher level to produce more uniform mixture, to meet different requirements of engine operation condition. Therefore, a new variable tumble system should be applied to GDI engine.

In this paper, a special flap valve was designed and installed in all runners of intake manifold, regulated by an electronic control device. At the lower engine speed the flap valve will be fully closed or with small opening in order to enhance the airstream speed, so that promotes the formation of more uniform mixed gas with strong tumble intensity. Once the engine speed reaching to a certain value, the flap valve will totally opened and much more fresh charge are trapped to achieve more power output. Furthermore, the differences of the in-cylinder flow field, including the velocity, turbulence kinetic energy field and tumble ratio were also investigated. The transient flow simulation was carried out based on CFD simulation for four design schemes matched with various intake ports and piston crown shapes. Finally the experimental measurement on steady-state flow test rig for all cylinder head prototypes was performed to validate the simulated results for all Cases.

CAD MODEL AND MOVING MESHES

Geometric model taken as the simulated object should be consistent with physical solid as possible, and fillets or chambers will not be simplified. The CAD model is obtained by extraction surfaces of solid. The moving mesh can be generated by Fame Engine Plus tool. The majority of meshes are hexahedron, and the left often are prism and pyramids, etc. At some crankshaft angles the maximum mesh number mostly reaches to 1.5 million or so. Meanwhile, imported motion curves of intake valve, exhaust valve and piston all are used for the generation of volume meshes.

After completing the meshing of surface or edge elements, the overall moving meshes are established to simulate the intake and compression strokes. The transient simulation begins with the intake valve just is opening, which throughout the intake and exhaust valve overlap period, air charging and compression process. The simulated operation condition is corresponding to the engine running at the speed of 2000r/min.

Boundary Conditions

1. Inlet Boundary Condition: The boundary type is chosen as Inlet/Outlet, imported data of mass flow rate and temperature are basic data that based on 1D thermodynamics simulation or experimental testing, just as shown in [Figure 1](#).
2. Outlet Boundary Condition: The type of exhaust port is set to both pressure and temperature, one of which is displayed in [Figure 2](#).
3. Wall Boundary Conditions: The wall temperatures are also defined separately for valve seat and valve, intake and exhaust ports, valves, liner, and combustion chamber. The temperatures of these walls are referred to recommended value of 1D simulation or relevant empirical formulas.

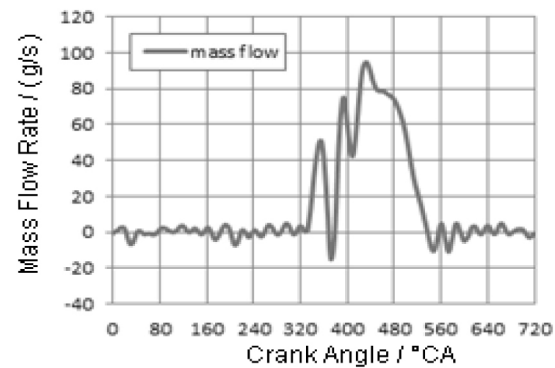


Figure 1. Mass Flow Rate on the Inlet of Intake Port

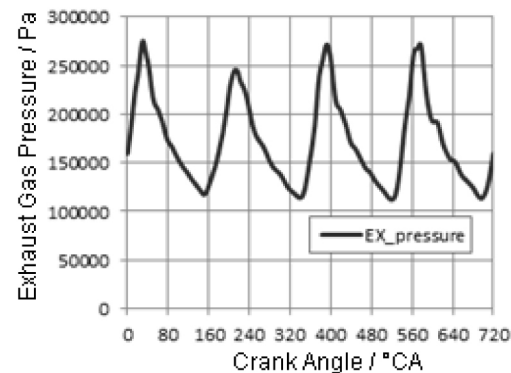


Figure 2. Exhaust Pressure on the Outlet

Initiation Conditions and Solver Setting

1. Before the outset of simulation calculation it needs to specify some given initial values for intake and exhaust ports, and in-cylinder flow field, generally as accurately as possible referred to calculated results from 1D thermodynamic or related test data, for accelerating the convergence process.
2. The Activated Equations: Momentum, Continuity and Energy Equations all are activated. The k-zeta-f model is selected for Turbulence Equation, higher accuracy with Four-Equation model, compared to k-e with only two ones[15].
3. Difference Scheme: Difference scheme of Continuity Equation is the central difference with second-order accuracy, blending factor is 1. Momentum Equation is set to Minmod scheme with second-order accuracy with 0.5 for blending factor, while the default upwind scheme with first-order accuracy are applied to TKE (Turbulence Kinetic Energy) and Energy Equation, respectively.

DESIGN SCHEMES SPECIFICATION

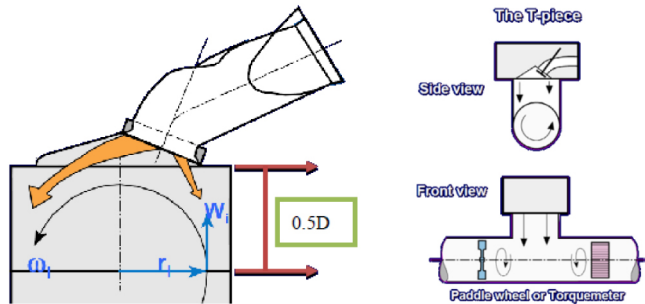


Figure 3. The Definition of Tumble Ratio and T-Pipe

The most critical dimensionless parameter of intake port is tumble ratio, and the schematic is illustrated in Figure 3, and D means the cylinder bore. The general form of tumble ratio also is expressed in Equation (1).

$$\text{Tumble Ratio} = \frac{\omega_{CFD}}{\omega_{Eng}} \quad (1)$$

$$\omega_{CFD} = \frac{\sum (\omega(i) \cdot r(i)^2 \cdot A(i))}{\sum (r(i)^2 \cdot A(i))}$$

and

where $\omega(i)$ is the angular velocity of one cell around the tumble ratio axis. $r(i)$ is the distance to tumble ratio axis. $A(i)$ is area of one cell. ω_{ENG} is frictional angular velocity.

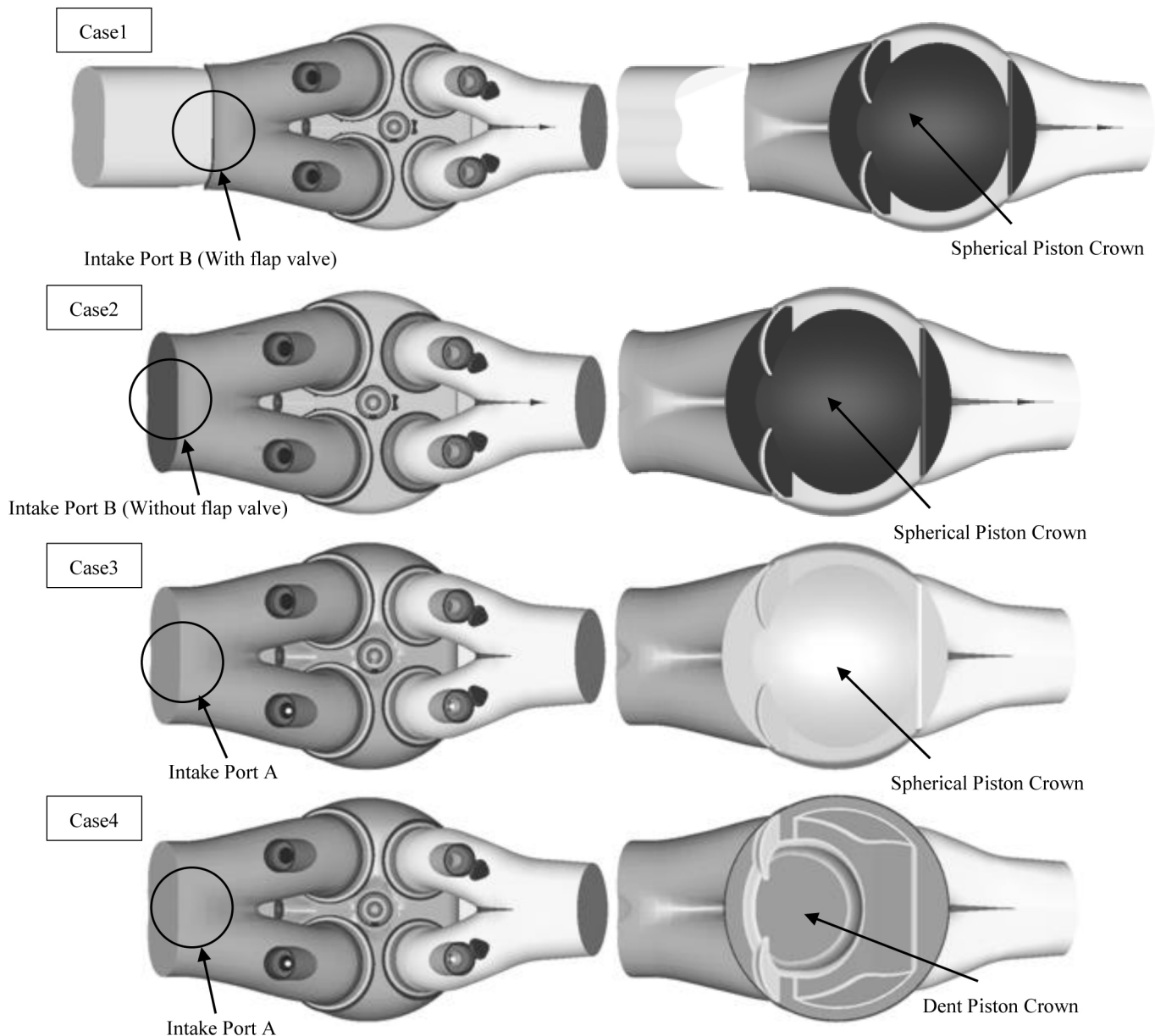


Figure 4. Schematic Diagram of Geometrical Structure for all Cases

The Cases used for contrast analysis totally consist of four combustion system models: (1) Case1: Spherical piston crown with intake port B (with flap valve). (2) Case2: Spherical piston crown with intake port B (without flap valve). (3) Case3: Spherical piston crown combined with intake port A. (4) Case4: Dent piston crown with intake port A. The geometric shape of exhaust port and the mounting positions of spark plug and injector all remain unchangeable, also with same compression ratio for all Cases. The geometric models of intake port and combustion chamber are compared for each Case, as shown in Figure 4. There are two types of intake port with different tumble ratio, namely intake port A and B. The steady flow calculation and rig test results all indicate that discharge coefficient of intake port B is relatively higher than that of intake port A, but the tumble intensity on the contrary.

SIMULATION AND CALCULATED RESULTS ANALYSIS

Visualization Planes

For the sake of displaying the in-cylinder flow field distribution it should define several cut planes. All these cut plots are available through AVL post-processing visualization tool. The comparison analysis of internal flow field is carried out for each case. In Figure 5 there are four cut planes displayed at different locations. Plane C and D exactly cut through the center point of intake valve seat parallel to the X axis, and plane E and F all are located in the center of spark plug, parallel to X and Y axis, respectively. From these cut planes all the physical-field information can be observed.

In-Cylinder Flow Field Attribution

For all Cases the transient flow simulations are performed, respectively. The transient flow field that changed with the crank angle can be observed in cylinder only through cut plots, because it is impossible to display detailed images for all physical fields, and the work load brought by post-processing is also unaffordable. For these reasons, in this paper it only presents the contrastive analysis of in-cylinder flow field for some critical moments, such as the tumble ratio reaching to the peak value (468°CA), and closing to the TDC (10°CA-BTDC and TDC), etc.

The velocity contour and airflow streamlines are shown in Figure 6 simultaneously. For Case1 the obvious tumble motion is formed in cylinder and all streamlines appear to revolve around one axis, perpendicular to the liner centerline. The main reason is the lower part of intake port is shielded by a flap valve, the air velocity significantly rises as the air flowing through the upper part of intake port, owing to sharply reduction of the effective flow sectional area (about by half). However, from Case 2 to 4, it is observed that there are several vortexes with opposite rotation direction based on airflow streamlines, and the continuity is disturbed, impossible to produce evident unidirectional rotation direction around one axis. The airflow is separated to two parts as flowing through inner valve seat, one revolving by clockwise direction, and the other on the contrary, causing no intensive tumble in cylinder. The maximum tumble ratio, velocity and average flow rate all are fully summarized in Figure 7. From Table 1 it can be observed that in Plane C and D, the maximum flow velocity and mean velocity for Case1 is about 2 or 4 times than Case 2, 3 and 4, as well as the maximum tumble ratio is 5~6 times. According to the above results, it is known that to intensify in-cylinder tumble should take relevant steps to decrease air flow rate on the left side of inner valve seat, accordingly in reverse on the right side. If so, the significant unidirectional and axial revolution motion will occur.

The in-cylinder velocity distribution and airflow streamlines all are investigated for each case as the tumble ratio up to maximum value. Presently taking Case1 as the baseline at 468°CA, the comparison for all Cases is presented in Figure 6. The flow velocity of Case1 is much greater than other Cases, and even backflow exists on the bottom left side of intake valve, because the airflow inertia action from left side completely dominates during whole fresh air charging. Nevertheless, for other Cases the two-side airflow enters into the cylinder with lower speed synchronously, so that it restricts the generation of large tumble. For Case3 and 4, the geometric shape of piston top is merely replaced, but with identical intake and exhaust ports, and the deviation of tumble ratio is quite small based on simulation results. At the speed of 2000r/min, given that flap valve being removed, the generated tumble intensity of Case 2 is drastically decreased compared to Case1. Therefore, reducing the flow velocity on the left side of intake valve is an effective way to increase in-cylinder tumble ratio.

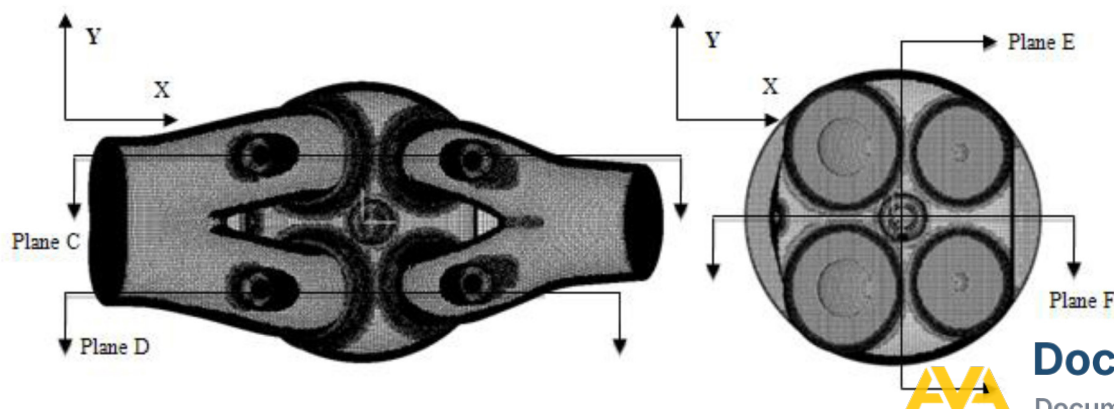


Figure 5. The Presentation of all Cut Plane Positions