Effects of different parameters on the flow field of peripheral ported rotary engines

Baowei Fan, Jianfeng Pan**, Wenming Yang, Hui An, Aikun Tang, Xia Shao and Hong Xue

School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China; Department of Mechanical Engineering, National University of Singapore, Singapore 117576, Singapore; School of Mechanical Engineering, California State Polytechnic University, Pomona, CA 91768, USA

(Received 24 October 2014; final version received 8 June 2015)

The performance of rotary engines is significantly influenced by the flow field. In this study, a detailed mathematical model was integrated into the simulation software FLUENT to investigate the gas flow field in a peripheral ported rotary engine by including a dynamic mesh model and a turbulent flow model. The models were also validated by experimental data. The basic flow mechanism in the combustion chamber was numerically studied. Meanwhile, the effects of the three major parameters on the flow field inside the combustion chamber, namely, rotating speed, intake shape, and intake angle, were also investigated. Results showed that a constantly changing swirl was formed in the combustion chamber during the intake and compression strokes as a result of the combined effects of the pocket of the rotor and the swirls in the combustion chamber. The swirl eventually broke into a unidirectional flow near the top dead center because of the significant decrease in combustion chamber volume. Furthermore, with the change in rotating speed, intake shape, and intake angle, significant differences in flow speed, inertia, and distribution were observed when the fluid entered the combustion chamber, which, in turn, led to obvious differences in the flow field, volume coefficient, and average turbulence kinetic energy in the combustion chamber.

Keywords: peripheral ported rotary engine; flow field; rotating speed; intake shape; intake angle; CFD simulation

1. Introduction

With increasing concerns about the energy crisis and environmental issues, a cleaner, more efficient, and environmentally adaptable energy system has become a pressing need. The highly advanced Wankel rotary engine is a promising energy system, which is an alternative to a reciprocating piston engine. The advantages of the Wankel rotary engine over a conventional reciprocating engine include a high power-to-weight ratio, a large specific power output from a high allowable engine speed, a simple and compact design resulting from less moving parts and multi-fuel capability, as well as low noise and vibration levels attributed to non-reciprocating components (Muroki, 1984; Shapovalov, 1999). As such, the Wankel rotary engine has been extensively used in various stationary and mobile applications (Antonelli & Martorana, 2012; Badr, Naik, O’Callaghan, & Probert, 1991). Nevertheless, the disadvantages of the Wankel rotary engine, which should not be ignored, include its sealing and leakage problems, lower efficiency, and higher unburned hydrocarbon emissions resulting from the flattened combustion chamber shape. Since the 1980s, several studies have attempted to improve the fuel efficiency and exhaust emissions of the Wankel rotary engine (Abraham & Bracco, 1989, 1991). Recently, some new apex seals were designed for the rotary engine to improve sealing (Warren & Yang, 2013).

A direct-injection and turbocharged system was also used to improve the efficiency of the engine (Kagawa, Okazaki, Somyo, & Akagi, 1993; Tashima, Taqtdokoro, Okimoto, & Niwa, 1991). Hydrogen was added to gasoline in the engine to improve its thermal efficiency and power output (Amrouche, Erickson, Park, & Varnhagen, 2014). Nevertheless, studies on the three-dimensional flow mechanisms of rotary engines, which are a determining factor of the combustion process and efficiency, remain insufficient. To our knowledge, fuel atomization, mixture formation, and combustion process are closely related to the flow field in the combustion chamber; thus, the importance of studies on the flow field is self-evident (Pariotis, Kosmadakis, & Rakopoulos, 2012; Taghavifar, Khaliliarya, & Jafarmadar, 2014). For studies on the flow field inside a rotary engine, the main approach has focused on experimental visualization. In addition, only a few studies have utilized numerical simulations. For the experiments, the reflection-type Schlieren method was used by Hasegawa and Yamaguchi (1993) to study the flow field in the center section of the combustion chamber of side-ported rotary engines, work which made a significant contribution to this area of flow field investigation. However, only the shape of the swirl in the rotor housing central plane was reported, and not the velocity and direction of the swirl. DeFilippis, Hamady, Novak, and Schock (1992) used laser Doppler velocimetry...
(LDV) to measure the flow field in the center section of the combustion chamber of a peripheral ported rotary engine. The velocity and direction of the swirl were obtained. However, with only a few measuring points, DeFilippis et al. could not determine the overall internal flow characteristics in the combustion chamber. The underlying reason why a swirl was observed in the rotor housing central plane in the experiment was unclear. Meanwhile, given the limitations of the experimental setup, only the flow field under low-speed conditions could be directly measured and the flow field under normal engine operating conditions (≥ 4000 rpm) could not be tested. Nevertheless, these valuable experimental data could still be used for validating the simulation. In terms of numerical simulations, some studies (Abraham & Bracco, 1988; T. I. Shih, Schock, Nguyen, & Stegeman, 1987) have used two-dimensional and three-dimensional numerical simulations to model the flow and combustion processes in rotary engines. However, given the computing power and limitations of the turbulence models developed in the 1980s, accurately predicting the large-scale swirl-changing process in the combustion chamber was impossible. Although various mathematical models have been significantly improved over the past few years, three-dimensional investigation coupled with a reasonable turbulence model on the flow process inside the combustion chamber of a rotary engine is still rare. The reason why a swirl was observed in the rotor housing central plane in the experiment remains unclear.

In the present study, we will integrate a three-dimensional moving grid and a reasonable turbulence model into the computational fluid dynamics (CFD) simulation software FLUENT. The computed results will be compared with the experimental data of DeFilippis et al. (1992) and our own experimental data to verify the turbulence model used in the rotary engines. In this manner, the variation of the flow field, average speed, and turbulent kinetic energy in a high-speed cold drag engine are reported, which are not easily obtained through experiments. The basic flow mechanism in the combustion chamber is numerically studied. The reason why a swirl was observed in the rotor housing central plane in the experiment in DeFilippis et al. is also determined. Meanwhile, the effect of the rotating speed, intake shape, and intake angle on the flow field are simulated and analyzed. The study on the three-dimensional flow mechanisms of peripheral ported rotary engines provides a theoretical foundation to improve and optimize the design of a rotary engine, with regard to such aspects as fuel atomization, mixture formation, and stratified combustion.

2. Dynamic mesh generation

2.1. Computational domain

In this study, the rotary engine used in the experimental study reported in DeFilippis et al. (1992) is selected as an object of study. A schematic of the test engine is shown in Figure 1. One cycle (including intake, compression, explosion, and exhaust strokes) corresponds to a crankshaft angle (CA) of 1080° in a rotary engine. As the rotor completes one rotation, during which time the eccentric shaft rotates through three revolutions, each chamber produces one power stroke. The engine specification is provided in Table 1.

| Engine parameters       | Value                        |
|-------------------------|------------------------------|
| Generating radius       | 105 mm                       |
| Eccentricity            | 15 mm                        |
| Chamber width           | 70 mm                        |
| Number of rotors        | 1                            |
| Intake stroke           | Advance angle, 447°; Delay angle, 530° |
| Exhaust stroke          | Advance angle, 192°; Delay angle, 336° |

Table 1. Engine specification.

2.2. Geometric model generation and meshing

Given that three combustion chambers are evenly distributed in the rotary engine, only one combustion chamber is simulated to save computational time. To our knowledge, a reasonable mesh size is significant in improving computational accuracy and saving central processing unit time. Considering the working characteristics of a rotary engine, the areas of the intake and exhaust channels are considered as a static mesh because they are not moving, whereas that of the combustion chamber is

| Grid size | 2 mm | 2.5 mm | 3 mm | 3.5 mm |
|-----------|------|--------|------|--------|
| Grid number at top dead center (TDC) | 131,077 | 74,346 | 47,361 | 27,632 |
| Grid number at bottom dead center (BDC) | 320,041 | 193,798 | 121,188 | 86,680 |

Table 2. Parameters of four different grid sizes.
considered as a dynamic mesh because of the movement of the rotor. For dynamic mesh generation, unstructured triangular grids are adopted. DEFINE_CG_MOTION was used to specify the motion of a particular dynamic zone in ANSYS FLUENT by providing the linear and angular velocities at each time step. For dynamic mesh generation, the more even the mesh size, the better the performance of the mesh movements. Therefore, the near-wall grid is not densified in this study.

To our knowledge, the grid size for engine simulation is typically set to 2 mm to 4 mm. Four different grid sizes are shown in Table 2, namely, 2, 2.5, 3, and 3.5 mm, to check grid sensitivity. The simulation results of the velocity profile under four different grid sizes are shown in Figure 2. Near the bottom of the pocket, which is marked with an arrow in Figure 2, the simulation results for 2 mm and 2.5 mm are better than that for 3 mm and 3.5 mm. That is, the grid sizes of 2 mm and 2.5 mm provide a more detailed description of flow near the rotor wall. Figure 3 shows the average velocity in the combustion chamber for four different grid sizes. As shown in the figure, the average velocity value in the combustion chamber for the grid size of 2.5 mm is close to that for the grid size of 2 mm within the range of 80° CA. However, the average velocity values for the grid sizes of 3 mm and 3.5 mm are significantly smaller than that for the grid size of 2 mm. Meanwhile, the computation time for the grid size of 2.5 mm can reduce calculation time significantly. As such, grid size is set to 2.5 mm after repeated comparisons. The three-dimensional mesh at 450° before top dead center (BTDC) is shown in Figure 4.

3. Mathematical models and validation

3.1. Mathematical models and boundary conditions

Air is set as the working medium for simulation. The gas flow in a rotary engine combustion chamber is a complex compressible viscous flow; the governing equations of mass, momentum, and energy are solved for unsteady compressible viscous flow. Meanwhile, the governing equations are discretized using the finite volume method and solved using ANSYS FLUENT Release 14.0 (ANSYS, 2011). Fluid density is calculated using the ideal law, and constant thermal conductivity and viscosity of the fluid are assumed. A coupled algorithm is used to deal with pressure–velocity coupling. The first-order upwind scheme is used to discretize the turbulent kinetic energy and turbulent dissipation rate terms. The second-order upwind scheme is used to discretize the energy, density, and momentum terms. Pressure inlet and pressure outlet are defined as the inlet and outlet boundary conditions, respectively. The engine is naturally aspirated; hence, the values of pressure boundary conditions are set to atmospheric pressure. Moreover, the combustion chamber and the rotor are set to be the wall boundary. Given that the
engine is motored, the temperature of the wall boundary is set to 300 K. Accurately emulating the effect of turbulence is the key for the flow field simulation because gas flow in a rotary engine combustion chamber is a complex compressible viscous flow and typically has a high turbulence scale. In this study, turbulence is modeled using a two-equation renormalization group (RNG) $k - \varepsilon$ submodel because it considers streamline bending, whirlpool, rotation, and fast-changing tension effects and has a higher potential for complex flow forecasting (Han & Reitz, 1995; T. H. Shih, Liou, Shabbir, Yang, & Zhu, 1995; Wang, Reitz, Pera, Wang, & Wang, 2013). The RNG $k - \varepsilon$ model is derived using a statistical technique called RNG theory (ANSYS, 2011). This model is similar in form to the standard $k - \varepsilon$ model. However, the effect of swirl on turbulence is included in the RNG model, which improves the accuracy for swirling flows and makes it suitable for complex flow field simulation inside a rotary engine.

3.2. Experimental results and model validation
The flow field was calculated under the experimental conditions (2000 rpm) described in DeFilippis et al. (1992) to validate the mathematical model. The calculated results were compared with the experimental data from DeFilippis et al., which were obtained by LDV measurement. The measurement locations, as shown in Figure 5, are all in the rotor housing central plane and equidistant from both end housings. The pocket shapes for the experimental test and the simulation were the same. Given that the pocket was located in the middle of the rotor surface, the camera could not photograph it during the experiment. As such, the pocket could not be observed in the experimental results shown in Table 3.

Table 3 shows the comparison of the simulation results with the experimental data in DeFilippis et al. (1992). As shown in the table, the calculated flow profile in the

Table 3. Velocity distribution at different times (m/s).

| CA (°) | Experimental results (DeFilippis et al., 1992) | Simulation results |
|--------|-----------------------------------------------|--------------------|
| 730    |                                               |                    |
| 820    |                                               |                    |
| 910    |                                               |                    |
| 1000   |                                               |                    |

Figure 5. Measurement locations of LDV (DeFilippis et al., 1992).
combustion chamber is similar to the experimental results in terms of the velocity of the flow field and the position of the swirl. For example, when the CA is at 730°, the velocities in the experimental and simulation results near the intake port are approximately equal to 20 m/s and the swirl locations are all located at the front of the combustion chamber. When the CA is at 1000°, the swirl is compressed into a unidirectional flow because of the decrease in combustion chamber volume, and the experimental and calculation results show that the velocity of the unidirectional flow is approximately 10 m/s.

An optical side-ported rotary engine test bed is built and particle image velocimetry (PIV) is used to test the flow field in the rotor housing central plane to further validate the turbulence model. Figures 6 and 7 show the schematic and photograph of the optical rotary engine and the PIV experimental setup in the laboratory, respectively. By controlling frequency and speed modulation, the motor is operated at 600 rpm. The numerical simulations are conducted under the same conditions as the experiment for fair comparison.

Table 4 shows the validity of the mathematical model and the numerical simulation. In most of the central regions, the simulated flow field in the combustion chamber is similar to that of the experimental results in terms of the velocity and position of the vortex. For example, a counterclockwise flow in the rotor housing central plane is observed in the experiment and simulation. Meanwhile, when the CA is at 250° BTDC, the magnitude of velocity for both the experiment and the simulation is approximately equal to 2 m/s and a swirl is located in the middle of the combustion chamber. Meanwhile, Table 5 shows the comparison results of velocity distribution in different turbulence models at 600 rpm (250° BTDC). The simulation result for the RNG $k-\varepsilon$ model can better reflect the experimental results than the standard $k-\varepsilon$ model and the realizable $k-\varepsilon$ model. As such, the RNG $k-\varepsilon$ model is proven to be accurate.

4. Simulation results and discussion
The change in the three-dimensional flow field in the combustion chamber with increasing rotating speed and varying intake shape and angle is difficult to obtain through experiments. In such a case, the three-dimensional dynamic simulation study becomes important. When the intake flow enters the combustion chamber in the intake stroke, it collides with the cylinder and rotor walls. After the collision, the intake flow turns around to form vortices. The vortex whose rotational axis is perpendicular to the cylinder head is called swirl, whereas the vortex whose rotational axis is parallel to the cylinder head is called tumble, in order to clearly express flow motion in the combustion chamber in the rotary engine. Meanwhile, in the streamline figures, high-speed flow through both sides of the intake port is denoted by A. The flow that comes...
4.1. Basic flow mechanism in the combustion chamber

The original engine is operated at the normal engine speed of 4000 rpm to illustrate the basic flow mechanism in the combustion chamber. Figure 8 shows the streamlines in the combustion chamber at different CAs.

During the early stage of the intake stroke, two tumbles are located at both sides of the combustion chamber. The tumbles are composed of two parts. One part is the high-speed flow through both sides of the intake port because when high-speed airflow enters the combustion chamber, it will hit the wall surfaces of the rotor and the combustion chamber to finally form the tumble. The other part is the flow that originates from the rear of the combustion chamber and moves toward the front of the chamber because of the thrust of the rotor. However, the high-speed
airflow from the intake port obstructs the flow from moving forward. When the flow hits the high-speed airflow from the intake port, it will bypass the high-speed airflow and move along both sides of the combustion chamber to form the tumble. In the middle of the intake stroke, a swirl is generated near the bottom of the pocket in front of the combustion chamber, as shown in Figure 8(b), because the two tumbles continue to move forward with the increase in combustion chamber volume. When the two tumbles move close to the bottom of the pocket, they do not readily continue to move forward because of the long and narrow volume in front of the pocket. As such, a swirl is formed when the confluence of the two tumbles moves close to the bottom of the pocket. As shown in Figure 8(c), when the swirl formed in the combustion chamber projects onto the rotor housing central plane, a swirl is also formed in the rotor housing central plane. This is the reason for the formation of the swirl detected in the rotor housing central plane in the experiment conducted by DeFilippis et al. (1992). All of the aforementioned analytical results indicate that swirl generation is the combined effect of the pocket of the rotor and the two tumbles in the combustion chamber. During the last stage of the intake stroke, the reverse flow phenomenon occurs from the combustion chamber to the intake port, as shown in Figure 8(d). As a result of the enlargement of the combustion chamber volume and the reverse flow phenomenon, the swirl begins to diverge and the radius of the swirl begins to increase.

During the early stage of the compression stroke, as shown in Figure 8(e), the original swirls have been broken up and the streamlines in cylinder are chaotic. During the last stage of the compression stroke, the flow in the rotor housing central plane eventually breaks into a unidirectional flow near the top dead center (TDC) because the volume of the combustion chamber has a minimum value, as shown in Figure 8(f). Considering the existence of the rotor pocket, the airflow from the rear parts of the combustion chamber flows through the pocket to the front of the combustion chamber.

4.2. Effects of the rotation speed on the flow field
The rotary engine has a special structure and operation mode. As its rotor turns one revolution, the output shaft turns three revolutions, which makes the rotary engine more suitable for operating at high rotation speeds than the reciprocating engine. In this study, the change in the flow field is analyzed when the engine speed is less than 30,000 rpm (rotor speed less than 10,000 rpm). Figures 9 and 10 show the change in the flow field at different CAs. In the figures, panels (a) to (d) show the streamlines under the rotating speeds of 4000, 8000, 16,000, and 26,000 rpm, respectively.

During the intake stroke stage, as shown in Figures 9(a) to 9(d), the flow patterns at the combustion chamber are similar under all four different rotating speeds. However,
the vorticity magnitude of the swirl increases with the increase in rotating speed. The largest vorticity magnitudes of the swirl are 7001.8, 15,627.6, 40,196.7, and 47,481 s\(^{-1}\) for 4000, 8000, 16,000, and 26,000 rpm, respectively. During the compression stroke stage, as shown in Figures 10(a) to 10(d), the original swirls under the 16,000 rpm and 26,000 rpm conditions still hold up well even when the original swirls under the 4000 rpm and 8000 rpm conditions have broken into several small swirls. This finding indicates that swirl breakup is postponed because the energy and inertia of the swirl increases with the increase in rotation speed. From the overall changes in the flow field during intake and compression strokes, the speed and inertia of the flow in the combustion chamber are increased and swirl breakup is postponed as the rotating speed increases.

Figure 11 shows that the volumetric coefficient initially increases and then decreases with the increase in rotation speed. The highest point is 16,000 rpm because as the rotation speed increases, the speed and inertia of the flow inside the combustion chamber also increase, but intake time constantly decreases under intake stroke. That is, the volumetric coefficient does not always increase with the increase in rotating speed, and 16,000 rpm is the turning point. Compared with the rotating speed of 2000 rpm, a 15.4% increase in the volume coefficient is observed when the rotating speed is 16,000 rpm. Figure 12 shows the average turbulence kinetic energy curves in the combustion chamber under different rotation speeds. To our knowledge, when the intake flow enters the combustion chamber, it collides with the walls of the rotor and the combustion chamber, which disturbs the flow in the combustion chamber. This disturbance increases as the rotation speed
increases; thus, the swirl motion and turbulent kinetic energy increase with the rotation speed.

4.3. Effects of the intake shape on the flow field

Four different intake shapes are compared to investigate the influence of intake shape on the flow field in the combustion chamber, as shown in Table 6. The shapes are round intake (the original intake), rectangular intake, regular trapezoid intake, and inverted trapezoid intake. The areas of all the four shapes are set to be the same to eliminate the effects of area.

The changes in the flow field with different intake shapes at 4000 rpm are shown in Figures 13 to 15. In the figures, panels (a) to (d) show the streamlines under the round, rectangular, regular trapezoid, and inverted trapezoid intake shapes, respectively.

During the early stage of the intake stroke, the area near the baseline is larger than that near the topline because of the difference in width of the trapezoid intake shape at different positions, as shown in Figures 13(a) to 13(d). This difference in width causes most of the fluid to flow into the combustion chamber through the baseline side. This area is called the main intake flow area. Correspondingly, the smaller part near the topline is called the secondary intake flow area. As shown in Figures 13(c) and 13(d), the main intake fluid area of the regular trapezoid intake shape is near the center of the combustion chamber. That is, the high-speed fluid from the intake port occupies most of the space in the center of the combustion chamber, which results in a small developing space for the two tumbles. As such, the swirl is squeezed into the rear of the combustion chamber, as shown in Figure 14(c). By contrast, the main intake area of the inverted trapezoid design is far from the

| Table 6. Sketches of the different intake shapes. |
|--------------------------------------------------|
| Round intake (the original intake) | Rectangular intake | Regular trapezoid intake | Inverted trapezoid intake |
|-------------------------------------|---------------------|--------------------------|--------------------------|
| ![Round intake](image1.png) | ![Rectangular intake](image2.png) | ![Regular trapezoid intake](image3.png) | ![Inverted trapezoid intake](image4.png) |

Figure 13. Streamlines in the rotor housing central plane with different intake shapes at 350° BTDC.
middle of the combustion chamber, which results in a large developing space for swirls, as shown in Figure 14(d). As such, a swirl is formed at the front of the combustion chamber and the center of the swirl is closer to the rotor wall compared with the round and rectangular intake shapes. During the later stage of the intake stroke, the reverse flow phenomenon occurs in all types of intake shapes. However, different shapes result in different intensities of reverse flow, which decreases as intake shape changes from regular trapezoid to rectangular, round (the original intake), and inverted trapezoid. Many remnants of the tumbles on top of the swirl for the inverted trapezoid design are observed, whereas no remnant tumbles are observed on top of the swirl for the regular trapezoid intake design.

From the overall changes in the flow field during the intake and compression strokes, we can conclude that the differences in the intake shapes result in different flow distributions when airflow enters the combustion chamber. The effect of reverse flow on the inverted trapezoid intake design is minimal compared with the other intake designs. Moreover, the swirl structure remains the best and swirl breakup is postponed.

The calculated results indicate that the volume coefficients of the four intake shape designs are 0.85293 (inverted trapezoid), 0.82699 (round), 0.81739 (rectangular), and 0.79521 (regular trapezoid). Thus, we can conclude that the volume coefficient of the inverted trapezoid intake design exhibits a 3.1% increase compared with the original intake design (round), primarily because the area and mass of the reverse flow for the inverted trapezoid intake design are minimal compared with those of the other intake designs. Moreover, the volume coefficient of the inverted trapezoid intake has the maximum value. Figure 16 shows the average turbulence kinetic energy in the combustion chamber of the different intake shapes. Evidently, the different intensities of the average turbulence kinetic energy decrease as the intake shape changes from rectangular to inverted trapezoid, round (the original intake), and regular trapezoid. Considering that the effect of reverse flow on the inverted trapezoid intake design is minimal, the intensity of the swirls for the inverted trapezoid intake design is stronger than that of the swirls for the round intake design, which results in a larger average turbulence kinetic energy for the inverted trapezoid intake design than for the round intake design.

4.4. Effects of the intake angle on the flow field

Given that the inverted trapezoid intake design has a better volume coefficient and that the average turbulence kinetic
energy in the combustion chamber also increases compared with the round intake design (the original intake design), the inverted trapezoid intake design is selected for further optimization. Table 7 shows the shape sketches of different intake angles, namely, 0°, 10°, 20°, and 30°, to analyze the effect of different intake angles on the flow field in the combustion chamber. Meanwhile, the areas of all the intake ports are set to be the same to eliminate the effects of area. As such, the topline lengths of the four types of inverted trapezoid are different.

The changes in the flow field with different intake angles under 4000 rpm conditions are shown in Figures 17 and 18. In the figures, panels (a) to (d) show the streamlines under the intake angles of 0°, 10°, 20°, and 30°, respectively.

With an increase in intake angle, the main flow region of the intake flow shifts away from the rotor wall and toward the combustion chamber wall during the intake stroke stage, as shown in Figures 17(a) to 17(d). Consequently, swirl formation at the front of the

Table 7. Schematic of different intake angles.

| Angle | Schematic of intake angles | Schematic of intake |
|-------|---------------------------|---------------------|
| 0°    | ![0° schematic](image)    | ![0° schematic](image) |
| 10°   | ![10° schematic](image)   | ![10° schematic](image) |
| 20°   | ![20° schematic](image)   | ![20° schematic](image) |
| 30°   | ![30° schematic](image)   | ![30° schematic](image) |

Figure 17. Streamlines in the rotor housing central plane with different intake shapes at 350° BTDC.
The calculated results indicate that the volume coefficients of the four intake angles are 0.8529 (0°), 0.8477 (10°), 0.8422 (20°), and 0.8312 (30°). This finding indicates that intake resistance is enhanced when the intake port has a certain tilt angle. That is, intake resistance increases with the increase in intake port angle, which results in the decrease in volume coefficient. Compared with the intake angle of 0°, a 2.5% decrease is observed in the volume coefficient when the intake angle is 30°. From the average turbulence kinetic energy curves in the combustion chamber of different intake angles shown in Fig. 19, the average turbulence kinetic energy in the combustion chamber decreases with the increase in intake port angle. The two reasons for this phenomenon are the decreasing volume coefficient and the increasing restraints of swirl motion.

5. Conclusions

In this study, a detailed mathematical model was integrated into the simulation software FLUENT by including a dynamic mesh and turbulent flow models. After validation by experimental test, further simulations were conducted to investigate the basic flow mechanism and the effect of various parameters on the flow field of a peripheral ported rotary engine. The following conclusions have been obtained:

1. The basic flow mechanism is that a constantly changing swirl is formed in the combustion chamber during the intake and compression strokes as a result of the combined effects of the pocket of the rotor and the swirls in the combustion chamber. Near the TDC, the swirl breaks into a unidirectional flow because of the significant decrease in combustion chamber volume.

2. When the swirl formed in the combustion chamber projects onto the rotor housing central plane, a swirl is also formed in the rotor housing central plane. That is the formation reason of the swirl detected in the rotor housing central plane in the experiment conducted by DeFilippis et al. (1992).

3. With an increase in rotation speed, swirl breakup is postponed as the flow speed and average turbulence kinetic energy in the combustion chamber increase. Moreover, the volumetric coefficient initially increases and then decreases with the increase in rotation speed. The highest point is 16,000 rpm. Compared with the rotating speed of
2000 rpm, a 15.4% increase in the volume coefficient is observed when the rotating speed is 16,000 rpm.

(4) For the inverted trapezoid intake design, the effect of reverse flow is minimal compared with those for the other intake designs. Consequently, the swirl structure remains the best and swirl breakup is postponed. Meanwhile, the average turbulence kinetic energy increases compared with that of the round intake design, and the volume coefficient is at maximum among all intake designs. Compared with the original intake design (round), a 3.1% increase in the volume coefficient is observed.

(5) With an increase in the intake port angle, more space at the front of the combustion chamber is occupied by intake flow. In addition, the space left for the tumbles to develop decreases. Consequently, swirl formation becomes weaker. Moreover, the volume coefficient and the average turbulence kinetic energy in the combustion chamber decrease constantly. Compared with an intake angle of 0°, a 2.5% decrease in the volume coefficient is observed when the intake angle is 30°.

Disclosure statement
No potential conflict of interest was reported by the authors.

Funding
This work was supported by the National Science Foundation of China [51376082]; the Natural Science Grant of Jiangsu Province [BK20131253]; the Jiangsu ‘Six Personnel Peak’ Talent-Funded Project [2011-ZBZZ-27]; and the Postgraduate Scientific Research and Innovation Project in Jiangsu Province [CXZZ13-0671].

References
Abraham, J., & Bracco, F. V. (1988). Comparisons of computed and measured mean velocity and turbulence intensity in a motored rotary engine (SAE Paper 881602).
Abraham J., & Bracco, F. V. (1989). Fuel-air mixing and distribution in a direct-injection stratified-charge rotary engine (SAE Paper 8903229).
Abraham J., & Bracco, F. V. (1991). 3-D computations of premixed-charge natural gas combustion in rotary engines (SAE Paper 910625).

Amrouche, F., Erickson, P., Park, J., & Varnhagen, S. (2014). An experimental investigation of hydrogen-enriched gasoline in a wankel rotary engine. International Journal of Hydrogen Energy, 39, 8525–8534.
ANSYS Inc. (2011). ANSYS Fluent 14.0: theory guide. Canonsburg, PA 1531.
Antonelli, M., & Martorano, L. (2012). A study on the rotary steam engine for distributed generation in small size power plants. Applied Energy, 97, 642–647.
Badr, O., Naik, S., O’Callaghan, P. W., & Probert, S. D. (1991). Wankel engines as steam expanders: Design considerations. Applied Energy, 40(3), 157–170.
DeFilippis, M., Hamady, F., Novak, M., & Schock, H. (1992). Effects of pocket configuration on the flow field in a rotary engine assembly (SAE Paper 920300).
Han, Z., & Reitz, R. D. (1995). Turbulence modeling of internal combustion engines using RNG $k – \varepsilon$ models. Combustion Science and Technology, 106, 267–295.
Hasegawa, Y., & Yamaguchi, K. (1993). An experimental investigation on air-fuel mixture formation inside a low-pressure direct injection stratified charge rotary engine (SAE Paper 930678).
Kagawa, R., Okazaki, S., Somyo, N., & Akagi, Y. (1993). A study of a direct-injection stratified-charge rotary engine for motor vehicle application (SAE Paper 930677).
Muroki, T. (1984). Recent technology development of high-powered rotary engine at mazda (SAE Paper 841017).
Pariotis, E. G., Kosmadakis, G. M., & Rakopoulos, C. D. (2012). Comparative analysis of three simulation models applied on a motored internal combustion engine. Energy Conversion and Management, 60, 45–55.
Shapovalov, V. (1999). The two-stroke rotary diesel engine (SAE Paper 1999–01–2888).
Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z. G., & Zhu, J. (1995). A new $k – \varepsilon$ eddy viscosity model for high Reynolds number turbulent flows. Computers & Fluids, 24(3), 227–238.
Shih, T. I. P., Schock, H. J., Nguyen, H. L., & Stegeman, J. D. (1987). Numerical simulation of the flow field in a motored two-dimensional wankel engine. Journal of Propulsion and Power, 3(3), 269–276.
Taghavifar, H., Khalilarya, S., Jafarmadar, S. (2014). Engine structure modifications effect on the flow behavior, combustion, and performance characteristics of DI diesel engine. Energy Conversion and Management, 85, 20–32.
Tashima, S., Taqodokoro, T., Okimoto, H., & Niwa, Y. (1991). Development of sequential twin turbo system for rotary engine (SAE Paper 910624).
Wang, F., Reitz, R. D., Pera, C., Wang, Z., & Wang, J. X. (2013). Application of generalized RNG turbulence model to flow in motored single-cylinder PFI engine. Engineering Applications of Computational Fluid Mechanics, 7(4), 486–495.
Warren, S., & Yang, D. C. H. (2013). Design of rotary engines from the apex seal profile (Abbr.: Rotary engine design by apex seal). Mechanism and Machine Theory, 64, 200–209.