THE VICE-VERSE MOVEMENT OF THE REVERSE TUMBLE CENTER OF ROTATION IN A PARTICULAR COMBUSTION CHAMBER

In this paper some initial results concerning the effect of characteristic vice-verse movement of the reverse tumble center of rotation in a particular combustion chamber in spark ignition engine were presented. The combustion chamber considered in this paper consists of a flat head with two vertical valves and cylindrical bowl. Results were obtained by dint of 3D numerical modeling technique.

The fluid flow pattern, represented as vectors, in all three planes at the very beginning of induction (15° ATDC and valve lift H=0.35 cm) is shown in fig. 1. Obviously, the combustion chamber is in form of cylinder (the simplest shape) with vertical valves. The basic block data sheet consists of bore/stroke ratio = 9.55/9.843 cm, diameter of cylinder (the simplest shape) with vertical valves. The basic block data sheet consists of bore/stroke ratio = 9.55/9.843 cm, diameter of both intake and exhaust valve Dv=3.38 cm, squish gap SG=0.2 cm, engine speed RPM=2000 min⁻¹, volumetric efficiency=0.72 and mixture quality λ=1. It should be stated that maximum valve lift is Li=0.82 cm (at 90° ATDC) while the other geometrical data (location, valve and port shape) could be seen in fig. 1. The commencement of intake valve opening was defined at 10° BTDC and its closure at 1900 ATDC.

Fig.1. Perspective view and cross-section in x-z plane of the combustion chamber geometry layout

The fluid flow pattern, represented as vectors, in all three planes at the very beginning of induction (15° ATDC and valve lift H=0.35 cm) is shown in fig. 2.

2. THE MODELLING OF NONREACTIVE FLOW

The full 3D conservation integral form of unsteady equations governing the turbulent motion of nonreactive mixture of ideal gas is solved on finite computational grid with moving boundaries (piston and valves) in physical domain (42,000-82,000 cells) by dint of finite volume ALE (Arbitrary Lagrangian-Eulerian) method /6, 7, 8/;

- The block-structured grid was adopted for valve/port assembly;
- The phenomenological model of turbulence was applied (standard k-ε model of turbulence);
- Diffusion velocities follow Fick’s law;
- The turbulent “law of the wall” was applied for the calculation of stress tensors and heat fluxes, i.e. the boundary conditions were applied in the vicinity of the wall;
- Heat fluxes were calculated by Reynolds analogy;
- Valves are treated as internal obstacles on the grid;

The calculations were carried out for induction and compression stroke only.

3. RESULTS

The analysis of fluid flow pattern during induction and compression was based on fairly complicated geometry layout presented in fig. 1. Obviously, the combustion chamber is in form of cylinder (the simplest shape) with vertical valves. The block-structured grid was adopted for valve/port assembly;

- The phenomenological model of turbulence was applied (standard k-ε model of turbulence);
- Diffusion velocities follow Fick’s law;
- The turbulent “law of the wall” was applied for the calculation of stress tensors and heat fluxes, i.e. the boundary conditions were applied in the vicinity of the wall;
- Heat fluxes were calculated by Reynolds analogy;
- Valves are treated as internal obstacles on the grid;
- The calculations were carried out for induction and compression stroke only.

The calculation of reverse tumble center of rotation in a particular combustion chamber in spark ignition engine was presented. The combustion chamber considered in this paper consists of a flat head with two vertical valves and cylindrical bowl. Results were obtained by dint of 3D numerical modeling technique.

The fluid flow pattern, represented as vectors, in all three planes at the very beginning of induction (15° ATDC and valve lift H=0.35 cm) is shown in fig. 2.
As can be seen from Fig.2 the intake flow hits the piston crown, curls and commences to form the vortex flow around the y-axis in clockwise direction with regard to the left side provided that it is stipulated as such. Nonsymmetrical fluid flow pattern is obvious due to different distances from valve axis to cylinder wall. The major part of intake flow, as it can be seen from the representation of the fluid flow pattern in y-z plane, emanating under certain elevation from valve assembly hits the piston crown and as a result some sort of fluid flow separation is encountered. This fluid flow separation and subsequent curling of the flow promotes the formation of four vortices, two around the perimeter of valve face and two in the zone between piston crown and cylinder head. The existence of these vortices and fluid flow separation are clearly legible in x-y plane (vectors of higher intensity within the separation zone).

The further movement of the piston downwards (75° ATDC) and the increase of the valve lift (Hv=0.77cm) yielding the increase of intake flow enables the appearance of reverse tumble with its center of rotation in the zone beneath exhaust valve (Fig.3).

At the same time vortex flow located in the zone between cylinder wall and intake valve face is increased yielding fluid flow separation and the change of its direction. Such a direction coincides with reverse tumble and therefore enhances its intensity. Low intensity corner vortices persist while the separation of fluid flow is also encountered (low intensity vectors in the uppermost left corner). The well formed vortices around x=const axes contribute to the global extension of the zone with high turbulent intensity.

The higher intensity of reverse tumble promotes the destruction of all vortex flows except those around z-axis. Namely, the vortex flow beneath the intake valve is broken up entirely as well as those vortices located on the flanks and in the vicinity of the cylinder wall.

The reverse tumble affects the spatial distribution of kinetic energy of turbulence as well. In comparison with case at 180° ATDC the zone with maximum kinetic energy of turbulence at 210° ATDC is displaced from cylinder wall to the location under intake valve. It is the result of vortex flow destruction in the vicinity spreaded out over the central part of the combustion chamber. The center of its rotation is shifted to the zone under the intake valve pursued by broader zone with high turbulence intensity. The role of small vortex flow beneath the intake valve, with its axis of rotation just adjacent to the valve face, is nearly neglectable and reduced to the close proximity of intake valve face.

Fluid flow patterns at BDC (180° ATDC, Hv=0.15cm) just before intake valve closure are shown in Fig.5 while the spatial distribution of kinetic energy of turbulence in x-z plane, y=0, is shown in Fig.6.
of cylinder wall and the higher intensity of reverse tumble. This phenomenon is shown in fig.7.

It should be stressed that the average value of kinetic energy of turbulence during compression shows abrupt decrease (3.04 $10^5$ cm$^2$/s$^2$ at 180° ATDC and 1.82 $10^5$ cm$^2$/s$^2$ at 210° ATDC). Further ahead, during compression fluid flow pattern in x-z plane pertains its general shape up to 300° ATDC. Namely, as can be seen in fig. 8 the location of reverse tumble center of rotation is nearly fixed to the left of the intake valve axis. Larger piston velocities and the reduced volume of the combustion chamber are the main reason for the apparent intensification of reverse tumble and its spreading through the entire chamber.

Approximately at 300° ATDC begins the stretching of the reverse tumble and its occupation of the zone under the exhaust valve. This entirely complies with “spin-up” theory. The reverse tumble is subjected to compression by piston movement and slowly squeezed out from the intake valve zone. In that way the reverse tumble center of rotation begins to move towards the exhaust valve.

This squeezing out of reverse tumble continues up to the 360° ATDC when there is no reverse tumble in the intake valve zone. On the contrary in the exhaust valve zone the reverse tumble exhibits the certain effect as can be seen in fig.8.

The gradual displacement of reverse tumble center of rotation from intake valve zone into exhaust valve zone during compression is followed with subsequent displacement of maximum kinetic energy of turbulence as is shown in fig.9 and 10.

4. CONCLUSIONS

- In the case of valve/port assembly considered the fluid flow pattern is complex and entirely 3D;
- The appearance of reverse tumble was observed to the extent responsible for the prime characterization of the fluid flow pattern during induction and compression;
- The reverse tumble intensity increases during induction;
- During induction the reverse tumble center of rotation moves gradually from exhaust valve zone into intake valve zone
- At the very beginning of compression the zone of high turbulence intensity is located beneath the intake valve;
- During compression the reverse tumble increases its intensity and breaks up all vortex flows established during induction except those around z-axis;
- During compression the reverse tumble center of rotation moves gradually from intake valve zone into exhaust valve zone;
- During compression the zone with maximum kinetic energy of turbulence strictly follows the movement of reverse tumble center of rotation.

5. REFERENCES

/1/ Z.Jovanović, S.Petrović “The mutual interaction between squish and swirl in s.i. engine combustion chamber”, p.72-86, MVM, vol 23, No.3, 1997


/3/ Z.Jovanović “The modification of the combustion chamber geometry layout on the basis of fluid flow
pattern criteria”, Proceedings, p.31-37. “MOTOAUTO 99” Conference, Plovdiv (Bulgaria), 1999


/7/ A.A.Amsden “KIVA3V, Rel.2-Improvements to KIVA3V”, LA-UR-99-915, 1999

/8/ A.A.Amsden “SALE3D: A simplified ALE computer program for calculating 3D fluid flows”, NUREG-CR-2185, 1982