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

ПРОТИВОПОСОЧНО ДВИЖЕНИЕ НА ОСТА НА НАПРЕЧНО ВИХРОВО ДВИЖЕНИЕ С ПРОТИВОПОТОК В СПЕЦИАЛНА ГОРИВНА КАМЕРА

JOVANOVIC Z. S. Serbia Consultant, Dr. Sc., Institute of Nuclear Science, Vinca, Belgrade
MASONICIC Z. M. Serbia Research Eng, Bsc. Institute of Nuclear Science, Vinca, Belgrade
TOMIC M. C. Serbia Professor, Dr. Sc University of Belgrade, Fac. Mech. Engr.

In this paper some initial results concerning the evolution of the fluid flow pattern in the combustion chamber of i.c.engine imposed by fluid ingress through intake port/valve assembly were presented. The results were obtained by dint of multidimensional modeling of non reactive flow in arbitrary geometry with moving boundaries. Bunches of results were obtained (app. 2400 plots) and therefore only a few relevant were selected. The fluid flow pattern is extremely complicated and entirely three-dimensional. Some interesting results concerning reverse tumble and its center of rotation shifting from exhaust valve zone to intake valve zone during induction stroke and vice versa from intake valve zone to exhaust valve zone during compression were encountered. In the vicinity of BDC the maximum turbulence intensity is located beneath the intake valve while during compression it pursues entirely the displacement of reverse tumble axis of rotation.

1. INTRODUCTION

In this paper some results concerning the effect of charcteristic 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 cyindrical bowl. Results were obtained by dint of 3D numerical modeling techique.

The bunch of results presented in this paper are only a part of the broader research concerning the analysis of the effect of various organized flows variations onto flame front shape and its propagation through unburnt mixture. Some results related to the isolated effect of squish and swirl on flame front shape and its propagation through various combustion chambers are already published /1, 2, 3/ but the effect of the third type of organized flow (tumble) is fairly unclarified. Tumble flow, according to some theoretical and experimental results /4, 5/, is very important for specific power and fuel economy increase.

Sooth to say, the theory of turbulence and so called “spin-up” effect, when the vortex filament subjected to compression reduces its length and promotes rotation around its axis rendering the movement on the larger scale thereafter, it can be presumed that tumble flow pursues the same rule. In the other words, destruction of well formed tumble during compression generates the higher turbulence intensity and larger integral length scale of turbulence in the vicinity of TDC. This increase of turbulence flow in the vicinity of TDC has a great influence on the flame kernel formation period reduction and faster flame propagation. The aforementioned logic imposes the conclusion that the most beneficial fluid flow pattern in the vicinity of BDC is well shaped high intensity tumble. The principal goal of this paper is qualitative and quantitative characterization of fluid flow pattern and the analysis of the valve/port assembley from the point of compliance with presumed ideal fluid flow pattern.

In lieu of the fact that the application of multidimensional numerical modeling is fairly complicated and implies some asumptions and simplifications, it is the only technique that encompasses the valve/port geometry in an explicit manner yielding the below listed assumptions indispensible.

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 fine computational grid with moving boundaries (piston and valves) in physical domain (42.000-82.000 cells) by dint of finite volume ALE (Arbitraty Lagrange-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 intenal 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 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 90^0 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 10^0 BTDC and its closure at 190^0 ATDC.

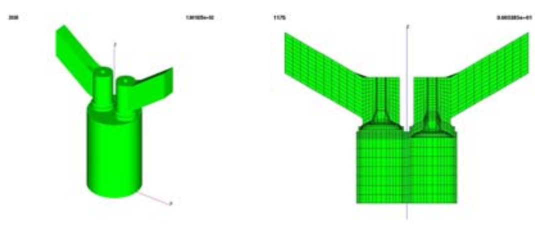

Fig.1. Perspective view and cross-section in x-z plane of the combustion chamber geometry layout

The fluid flow pattern, represented as vectors, in all three planes at the very beginning of induction (15^0 ATDC and valve lift Hv=0.35 cm) is shown in fig. 2.

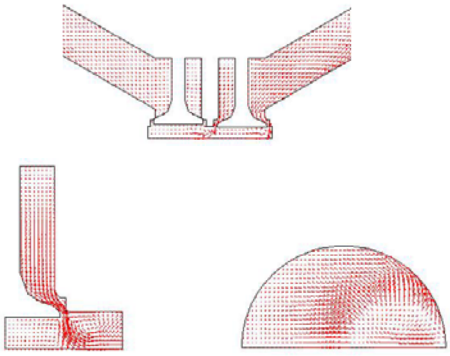

Fig.2. Fluid flow pattern in x-z (up), y-z (left) and x-y (right) plane at 15^0 ATDC

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 vorteces 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 (75^0 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).

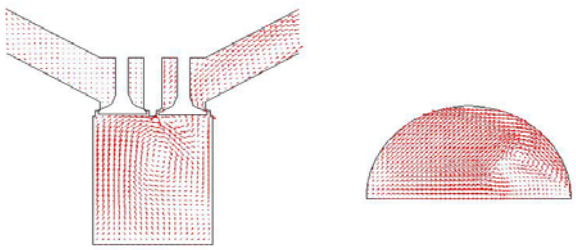

Fig.3. Fluid flow pattern in x-z (left) and y-z (right) plane at 75^0 ATDC

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.

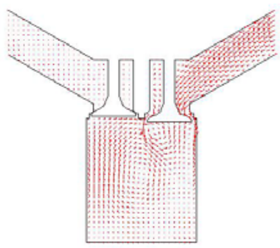

Fig 4. Fluid flow pattern in x-z plane at 150^0 ATDC

The same trend is preserved at 150^0 ATDC (Hv=0.5cm) as well. Namely the intensity of the reverse tumble is increased and 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.

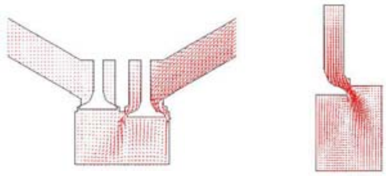

Fig.5. Fluid flow pattern in x-z (left) and x-y (right) plane, y=0, at 180^0 ATDC

Fluid flow patterns at BDC (180^0 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.

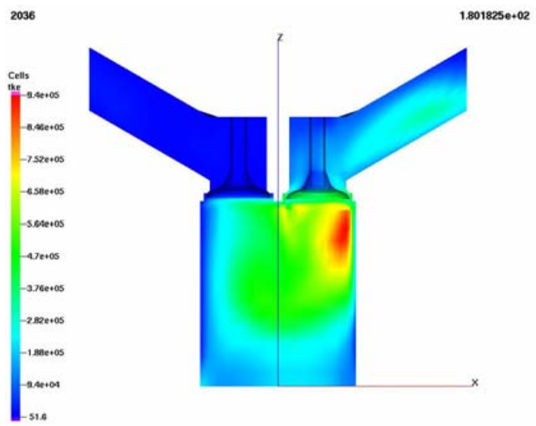

Fig.6. Spatial distribution of kinetic energy of turbulence in x-z plane, y=0, at 180^0 ATDC

It can be seen in fig.5 (BDC) that fluid flow pattern is dominated by well formed high intensity reverse tumble occupying nearly the entire combustion chamber. Namely, the gradual displacement of reverse tumble center of rotation towards the zone located under the intake valve and subsequent increase of its intensity cause the deflection of flank vortex flow and their redirection to the cylinder wall. It is elucidated in fig.5 where in addition to the intensive vortex flow around y=const axis (low intensity vectors in the zone located in the vicinity of intake valve, to the right of its axis) high intensity vortex flow around z-axis is encountered. The sustained action of small vortex flow located to the left of intake valve axis and just beneath the intake valve face is subjected to permanent compression by reverse tumble promoting its rotation. Obviously, the vortex flows around all three axes exist in the zone located under intake valve face but the effect of reverse tumble is of crucial importance. Such a conclusion is validated by dint of spatial distribution of kinteic energy of turbulence shown in fig.6.

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 comparision with case at 180^0 ATDC the zone with maximum kinetic energy of turbulence at 210^0 ATDC is displaced from cylinder wall to the location under intake valve. It is the result of vortex flow destruction in the vicinity of cylinder wall and the higher intensity of reverse tumble. This phenomenon is shown in fig.7.

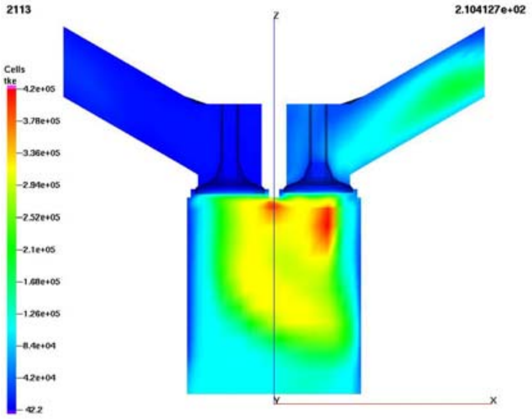

Fig.7 Spatial distribution of kinetic energy of turbulence in x-z plane, y=0, at 210^0 ATDC

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 180^0 ATDC and 1.82 10^5 cm^2/s^2 at 210^0 ATDC). Further ahead, during compression fluid flow pattern in x-z plane pertains its general shape up to 300^0 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.

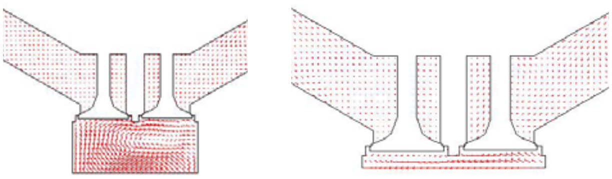

Fig.8. Fluid flow pattern in x-z plane at 300^0 (left) and 360^0 (right) ATDC

Approximately at 300^0 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 360^0 ATDC when there is no reverse tumble in the intake valve zone. On the contrary in the exhaust valve zone the reverse tumble exibits 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.

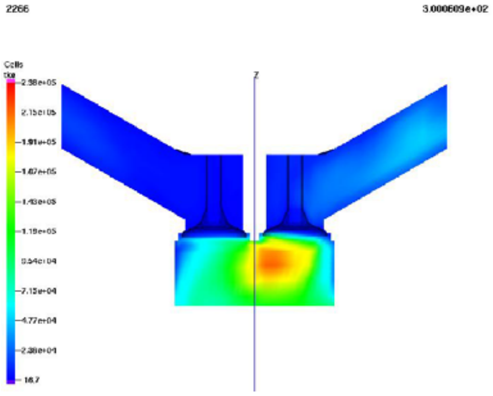

Fig.9. Spatial distribution of kinetic energy of turbulence in x-z plane, y=0, at 300^0 ATDC

It is obvious that at 300^0 ATDC (fig.9.) the maximum value of kinetic energy of turbulence, located in the zone enbounded by intake valve axis and cylinder axis, coincides with the location of reverse tumble center of rotation.

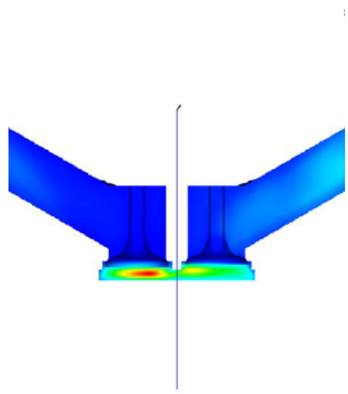

Fig.10. Spatial distribution of kinetic energy of turbulence in x-z plane, y=0, at 360^0 ATDC

The same situation is observed at 360^0 ATDC (fig.10.) i.e. the reverse tumble center of rotation coincide with maximum value of kinetic energy of turbulence.

It should be stated that valve/port assemly considered is well designed due to fact that that the reverse tumble center of rotation as well as the zone with maximum kinetic energy of turbulence at 327^0 ATDC (ignition) coincide with spark location i.e. between tha valves.

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

/2/ Z.Jovanović, I.Zeljković “The modeling of fluid flow, combustion and emission- multidimensional approach”, (in Serbian), p.281-316, in “Technolgy equipment for the reduction of emission from stationary and mobile sources”, Belgrade, Nov.1997, ISDN-86-7083-311-5, Faculty of Mech.Engr., Belgrade

/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

/4/ S.Nadarajah, M.J.Tindal, M.Yianneskis "Swirl centre precession under steady flow conditions", Proceedings IME, C413/065, p.103-108, 1991

/5/ A.D.Gosman "Flow processes in cylinders", p.616-772, The Thermodynamik and Gas Dynamics of I.C.Engines, vol. II, Oxford Science Publications, ISBN 0-19-856212-8, 1986

/6/ Z.Jovanović "Multidimensional modeling of combustion in Otto-cycle engines", (in Serbian), p. 153-186, ISBN 86-7083-262-3, University of Belgrade, 1995

/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