THE EFFECT OF INTAKE FLOW MODELLING ON FLAME FRONT SHAPE AND ITS DISPLACEMENT IN CYLINDRICAL COMBUSTION CHAMBER

Dr.ing.Z.JOVANOVIC
Institute of Nuclear Science-Vinca, Belgrade, Yugoslavia
Prof.Dr.ing.S.Petrovic
University of Belgrade, Yugoslavia

ABSTRACT: In this paper some results concerning the effect of intake flow modeling obtained by 3D numerical modeling of combustion on flame front shape and its displacement through combustion chamber geometry of s.i .engine consisting of flat head with two vertical valves and cylindrical bowl were presented. These results obtained with KIVA3V code were compared with results of flame front shape and its displacement obtained for the case with no valves i.e. without intake flow modeling .In order to alleviate the application of KIVA3V code and to enhance its flexibility two additional computer codes were applied as well, i.e. AVL TYCON code intended for cam design calculations and calculations of the dynamic behavior of timing drives and gear transmission units and AVL BOOST code intended for engine cycle calculations, including 1.5D fluid flow calculations through pipelines. The first code was used for the calculation of valve lift curve while the other was used for the calculation of relevant data set in valve regions. It was found that for particular combustion chamber shapes considered the entirely different flame front shapes and propagation velocities were encountered for these two cases ensuing primarily from the entirely different fluid flow patterns in the vicinity of TDC.

1. INTRODUCTORY REMARKS

It is known for a long time that various types of organized flows in combustion chamber of i.c engines are of predominant importance for combustion particularly with regards to flame front shape and its propagation [1, 2]. Some results related to synergic effect of squish and swirl on flame front shape and its propagation through various combustion chamber layouts are already analyzed [3, 9] but the isolated or combined effect of the third type of organized flow i.e. the effect of tumble or "tumble-like" intake flow on flame front shape and its displacement is not sufficiently clarified ensuing partly from the ambiguity concerning the exact definition of tumble flow. Namely, in spite of the fact that tumble flow is inherent to multi-valve engines some two valve engines exhibit some characteristics similar to tumble flow [4, 8] and therefore in the rest of the paper , in lieu of tumble or tumble-like flow we are operating more general intake flow. In addition, one of the reason is also CFD with all the difficulties ensuing from 3D grid generation of real multi-cylinder engine and experimental verification of numerical results. For that reason in this paper, as a part of broader research concerning the effect of intake port/valve geometry and maximum valve lift variations on flame parameters the combined effect of squish and intake flow on flame front shape and its propagation was analyzed. These results were compared with results of flame front shape and its displacement for the case with no intake flow modeling (no valves) indicating the isolated effect of squish thereafter.

2. COMBUSTION MODELING

The analysis of this type is inherent to multidimensional numerical modeling of reactive flows in complicated geometry therefore it was quite logical that such a technique (KIVA3V code [5]) was applied for the analysis of the effect of intake flow modeling on flame parameters particularly due to fact that it is the only technique that encompasses the valve/port geometry in an explicit manner. Taking into account various options contained in aforementioned code the following assumptions were adopted:

-flame propagation is controlled by turbulent diffusion modeled via k-ε model of turbulence;
-chemistry was modeled with quasi global irreversible reaction of fuel oxidation (C_8H_{18}) followed by two group of reactions, those which proceed kinetically and those which reduce the chemical heat release;
-the ignition was not modeled but rather bypassed as well as the entire ignition delay period by energy deposition in spark cells (artificial rise of temperature in spark cells);
-the commencement and the length of energy deposition was adjusted with reference to the location of peak cylinder pressure determined with BOOST code [7];
-in the case with valves the relevant set of data on open boundaries were calculated with BOOST code as well;
-valve lift curves and valve timing were calculated through series calculations with TYCON code [6].

3. RESULTS

3.1 The isola ted effect of squish

The analysis of the isolated effect of squish obtained through squish area variation (SA) from 23% to 63%, as related to bore cross-section, on fluid flow pattern, flame front shape and its displacement was carried out for combustion chamber geometry layouts shown in a form of perspective plots in fig. 1, 2 and 3.

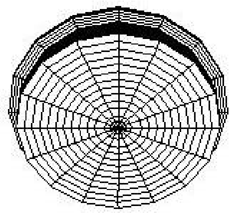

Fig.1 Perspective plot of combustion chamber with SA=23% (no valves)

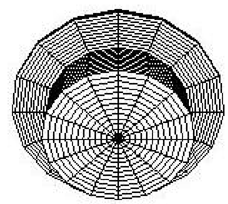

Fig.2 Perspective plot of combustion chamber with SA=54% (no valves)

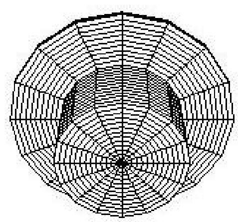

Fig.3 Perspective plot of combustion chamber with SA=63% (no valves)

It can be seen that combustion chamber is in form of cylindrical bowl with variable diameter and depth ranging from (2.533, 2.806, 3.071, 3.34, 3.609) to (1.383, 1.155, 0.9653, 0.815, 0.698) respectively (as a prerequisite condition for constant compression ratio of 9.5). Supplementary data to be taken account are bore/stroke ratio=8.25/9.2, total volume=550cm3, volumetric efficiency ?v=0.82, mixture quality ?=1 and engine speed n=2000min^{-1}. The fluid flow pattern during compression, for all five cases, is mainly controlled by piston motion and is similar except in the vicinity of TDC. The flow field pattern represented as vectors in x-z plane for the case with SA=23% is shown in fig.4

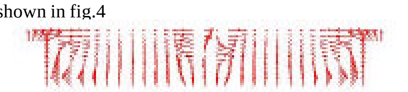

Fig.4. Flow field pattern in x-z plane, y=0, at –9.78 deg. ATDC (SA=23%, no valves, extreme case)

It can be seen that due to ingress of fluid from squish zone small vortices in clockwise direction with regard to the left side, provided that it is stipulated as such, are formed around the perimeter of cylindrical bowl (fig.4). Under the influence of flame arrival (larger velocities in front of the flame front) these vortices are reducing their size and the entire flow field is is generally flame dominated. The fairly similar situation is obtained for SA=34%. On the contrary the situation is slightly different for case with SA=44% and SA=54% (shown in fig.5).

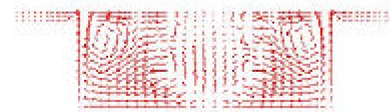

Fig.5. Flow field pattern in x-z plane, y=0, at –0.01 deg. ATDC (SA=54%, no valves, real case)

No one quiescent zone exists anywhere except in the soothing zone behind the flame front. Vortex flow around the perimeter of the bowl is well formed and strong enough to reach the zone in front of the flame front but not to the extent to deteriorate it. Namely, the smooth coinciding of the flow is encountered with unconstrained flame propagation thereafter.

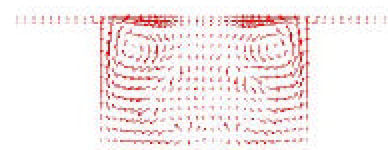

Fig.6. Flow field pattern in x-z plane, y=0, at 0.06 deg. ATDC (SA=63%, no valves, extreme case)

In the case with SA=63% (fig.6) the flow field and the flame propagation are entirely squish dominated instead of flame dominated as it was the case with SA in the range from 23% to 44%. The vortex flow around the perimeter of the bowl is excessively strong and protrudes inwards to the extent that causes the detention of the flame propagation particularly in the upper part of the combustion chamber. It is interesting to note that in the case of flame dominated flows the maximum kinetic energy of turbulence is located in the central part of the chamber indicating that the effect of squish flow is negligible.

The flame propagation through combustion chamber considered is represented in form of isocontours of temperatures ranging from minimum to maximum and is shown for corresponding SA cases in fig. 7, 8 and 9.The exact location of the flame front is determined in the zone of maximum line densities due to equal temperature difference of isocontours. Spark plug was located on z-axis in all five cases. Flame propagation through unburned mixture is controlled, excluding macro flows, by turbulent diffusion i.e. by high intensity of turbulence and cascade process of tearing or breaking up large vortices into smaller ones fully complying with assumptions imposed by k-e model of turbulence. Anyhow, at least five conflicting mechanisms, acting in concert, described elsewhere [1, 2] are of prime importance for the determination of flame front shape and its displacement. These are, inter alia, flame generated turbulence, compression by the flame, increase of the viscosity behind the flame front, the sign and the magnitude of the density gradient across the flame front and the effect of large heat release due to chemical reactions.

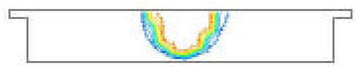

Fig.7. Spatial distribution of isocontours of temperature in x-z plane, y=0, at 0.06 deg. ATDC (SA=23%, no valves, extreme case)

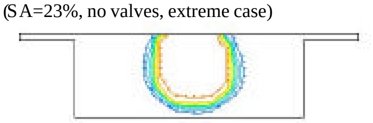

Fig.8. Spatial distribution of isocontours of temperature in x-z plane, y=0, at 0.06 deg. ATDC (SA=54%, real case)

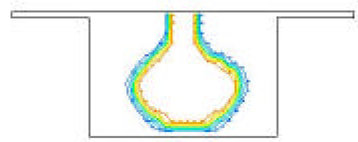

Fig.9. Spatial distribution of isocontours of temperature in x-z plane, y=0, at 0.06 deg. ATDC (SA=63%, no valves, extreme case)

As can be seen in fig.7 for the case with insufficient squish effect (flame dominated flows) the flame front shape is widen, its propagation slow and controlled solely by aforementioned mechanisms. For the case with SA=54%, shown in fig.8, the full support of macro flows is encountered.

The flame front shape is nearly spherical and it propagates faster than in previous cases. The larger velocities in front of the flame front coincide with vortex flow around the perimeter of the bowl causing faster flame propagation thereafter.

For the case with SA=63%, shown in fig.9, all negative aspects concerning squish dominated flame are clearly legible. Namely, we are nearly faced with the break up of the flame front in the upper part of the chamber. In addition, the tip of the flame front, being expelled downward, along the z-axis, forms the mushroom shape yielding the appearance of pockets of unburned mixture in the upper part of the combustion chamber in the later phase.

3.2. The combined effect of squish and intake flow

The analysis of the combined effect of squish and intake flow on flame front shape and its displacement was carried out for identical combustion chamber geometry layouts as in 3.1. i.e. for SA in the range from 23% to 63%, shown in fig. 10, 11 and 12..

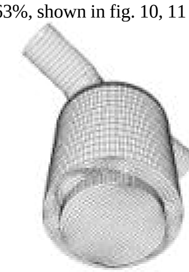

Fig.10. Perspective plot of the cylindrical combustion chamber (SA=44%) with valves

According to the results, presented in 3.1, it was quite obvious that two insufficient squish cases (flame dominated flows) were entirely irrelevant (SA=23%, SA=34%) and therefore excluded from the analysis. In these two cases flame propagation behaves as in the case with flat piston.

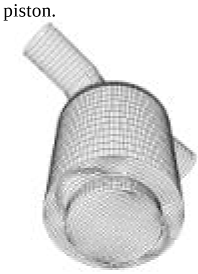

Fig.11 Perspective plot of the cylindrical combustion chamber (SA=54%) with valves

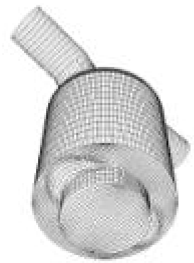

Fig.12 Perspective plot of the cylindrical combustion chamber (SA=63%) with valves

In the case with valves, combustion chambers, shown in fig. 10, 11 and 12, were treated as an integral part of the complete engine shown in fig.13.

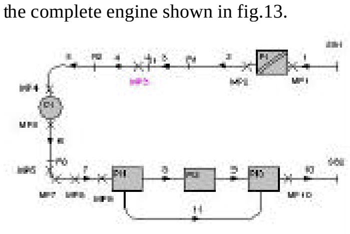

Fig.13. Schematic presentation of the complete engine

As can be seen in fig.13, in addition to the mono cylinder (CYL), various components such as air cleaner (F1), plenum chambers (P11, P12, P13), bending (5), restrictions (R1 -R3) and pipes of definite lengths and diameters are included (1-11). The distance of two reference points (MP5 and MP6) were specified in such a way to correspond to the location of open boundaries in fig. 10, 11 and 12. Each element requires pertinent set of specifications. Namely, for a cylinder it implies, in addition to the valve/port specifications (port surface areas, port wall temperature, geometry of valve assembly, clearance, valve lift curves, valve timings, etc.) the geometry of the chamber (Heron, no offset, as shown in fig. 10, 11 and 12), spark plug location, ignition timing, etc. The valve lift curves for mono cylinder, shown in fig.14, were calculated with TYCON code [6], spread out along crank angle, appropriately shifted and included in both BOOST [7] and KIVA3V [5] codes.

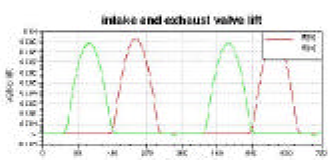

Fig.14. Calculated valve lift curves versus cam angle

For the purpose of ignition and valve timing optimizations and in order to obtain the appropriate set of requisite values for KIVA3V code such as pressures (fig.15), temperatures (fig.16), velocities, species densities and turbulence parameters, a large number of preliminary engine cycle calculations (transients, traces, series) were carried out with BOOST code.

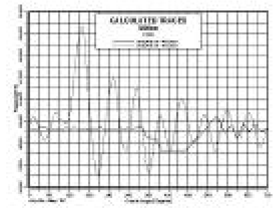

Fig.15. Pressure versus crank angle at reference points MP5 and MP6 for SA=54%

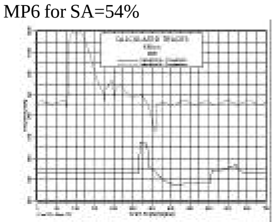

Fig.16. Temperature versus crank angle at reference points MP5 and MP6 for SA=54%

The slightly different values obtained for different SA were appropriately included in KIVA3V code.
The evolution of the fluid flow pattern for combustion chamber geometry layouts with valves is shown in fig. 17, 18 and 19.

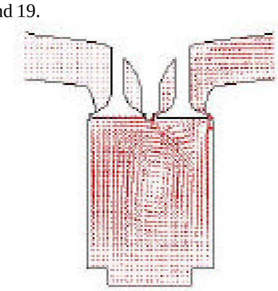

Fig.17 Fluid flow pattern in xz plane, y=0, at 180 deg. ATDC (SA=54%)

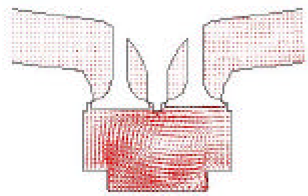

Fig.18 Fluid flow pattern in x-z plane, y=0, at 300 deg. ATDC (SA=54%)

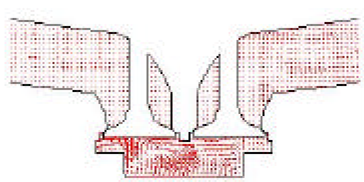

Fig.19 Fluid flow pattern in x-z plane, y=0, at 350 deg. ATDC (SA=54%)

During opening of intake valve the intake flow hits the piston crown, curls and commences to form the vortex flow around y-axis with its center of rotation in the zone beneath the exhaust valve. At the same time the intake flow strikes upon the cylinder wall, rebounds, hits the piston crown, curls and hits the face of intake valve promoting the fluid flow separation and forming of two vortices, the first one located beneath the left side of intake valve face coinciding with vortex flow around y-axis and the second located beneath the the right side of intake valve face rotating in the opposite direction. During induction the intensity of vortex flow around y-axis is increasing and its center of rotation gradually shifting to the central part of combustion chamber. In addition, the vortex flow due to separation, being squeezed by vortex flow around y-axis reaches the narrow zone in the close proximity of intake valve affecting that zone only while the other vortex flow due to separation is pushed to the cylinder wall (fig.17). Obviously, in the case with valves the fluid flow pattern at the very beginning of compression is entirely different in comparison with zero velocity field. The higher intensity of vortex flow around y-axis during compression promotes the destruction of all vortex flows, except those around z-axis and at 300 deg. ATDC its center of rotation is entirely displaced in the zone beneath the intake valve. At that angle commences the stretching of the vortex flow around y-axis i.e. the vortex flow around y-axis is subjected to compression by piston movement and slowly squeezed out from the intake valve zone (fig.18). The squeezing out of vortex flow around y-axis continues up to 350 deg. ATDC when there is no vortex flow in the intake valve zone. On the contrary, in the exhaust valve zone the vortex flow around y-axis exhibits the certain effect (fig.19). Namely, in the case with valves during induction and compression the center of rotation of vortex flow around y-axis is gradually displaced from exhaust valve zone to intake valve zone and vice verse. The existence of vortex flow around y-axis in the exhaust valve zone in the vicinity of TDC dominantly affects the flame front shape and its propagation. In the case with no valves such a vortex flow was not encountered.

In the case with valves flame front shape and its propagation for appropriate angles is show in fig. 20, 21 and 22.

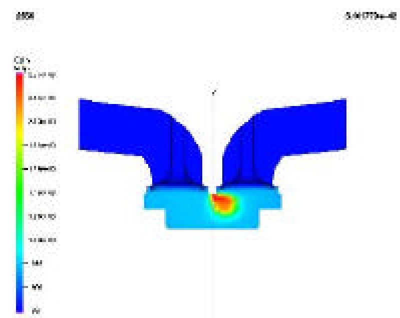

Fig.20 Spatial distribution of isocontours of temperatures in x-z plane, y=0, at 340 deg. ATDC (SA=54%)

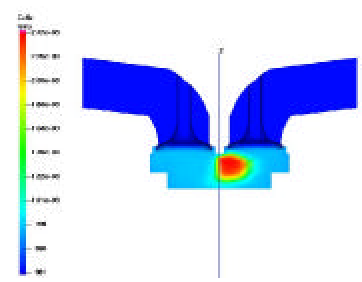

Fig.21 Spatial distribution of isocontours of temperatures in x-z plane, y=0, at 350 deg ATDC (SA=54%)

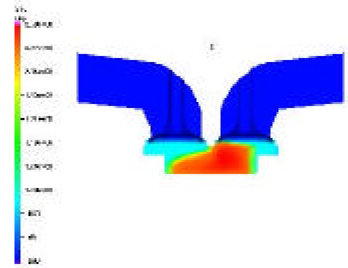

Fig.22 Spatial distribution of isocontours of temperatures in xz plane, y=0, at 360 deg. ATDC (SA=54%)

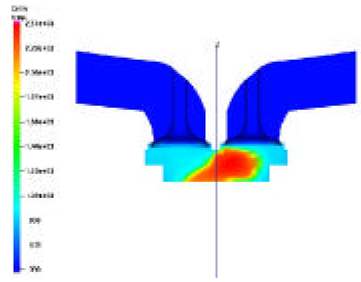

Fig.23 Spatial distribution of isocontours of temperatures in xz plane, y=0, at 360 deg. ATDC (SA=44%)

As can be seen in fig.20 the spark plug is centrally located and the flame front shape is extremely irregular and characterized with non uniform spatial propagation. Namely due to persistence of vortex flow around y-axis in the zone beneath the exhaust valve the deflection of the flame front is fairly legible. Such a trend continues ahead (fig.21 and fig.22) and yields the appearance of detention zone beneath the exhaust valve. It should be noted that that fairly similar situation is encountered for SA=44% (fig.23) adn SA=63% indicating that the effect of squish for the whole range of SA is negligible.

4. CONCLUSIONS

In the case with no valves the squish plays an important role in flame front shape and its displacement. For particular combustion chamber with no valves three entirely different forms are encountered i.e. the insufficient squish (SA=23%-44%) yielding flame dominated flows, moderate squish (SA=44%-54%) yielding nearly spherical flame and excessive squish (SA=63%) yielding squish dominated flame. For particular geometry considered with valves flame front shape and its displacement are fully controlled by intake flow. The strong vortex flow around y-axis beneath the exhaust valve in the vicinity of TDC generated by intake flow annihilates entirely any effect of squish ranging from insufficient to excessive one and causes the detention of the flame propagation in that zone thereafter.

5. REFERENCES

[1] Z.Jovanovic "The role of tensor calculus in numerical modeling of combustion in i.c.engines", p.457-541, in "Computer Simulation for Fluid Flow, Heat and Mass Transfer, and Combustion in Reciprocating Engines", ISBN 0-89116-392-1, Hemisphere Publishers, 1989

[2] S.Petrovic, R.Jankov, M.Tomic, Z.Filipi, Z.Jovanovic "Modeling of Combustion in Otto-cycle engines", University of Belgrade, ISBN 86-7083-262-3, 1995 (in Serbian)

[3] Z.Jovanovic, S.Petrovic "The mutual interaction between squish and swirl in s.i.combustion chamber", p.72-86, MVM, vol.23, No.3, 1997

[4] A.Chen, K.C.Lee, M.Yianneskis "Velocity characteristics of steady flow through a straight generic inlet port", International Journal for Numerical Methods in Fluids, vol.21 ,1995

[5] A.A.Amsden "KIVA3V: A Block-Structured KIVA Program for Engines with Vertical or Canted Valves", LA-13313-MS, Los Alamos, 1997

[6] AVL TYCON, Users Manual, ver.4.3, 1998

[7] AVL BOOST, Users Manual, ver.3.3, 2000

[8] Z.Mahmood, A.Chen, M.Yianneskis "On the structure of steady flow through dual intake engine flow", Kings College (internal report)

[9] Z.Jovanovic, S.Petrovic "3D Fluid Flow in i.c.engine combustion chamber of arbitrary geometry", p.31-37, Proceedings, Vol. II, Russe, ISBN 954-90272-2-8, 1997