Numerical study of hydrodynamic perturbations caused by filiform spark discharge near wall

E V Dolgov and A A Firsov*
Joint Institute for High Temperatures (JIHT RAS), 125412, Moscow, Russia

*af@jiht.org

Abstract. A numerical simulation of the high-speed jet caused by the decay of the thermal cavern after cylindrical 10 mm long spark discharge located on the wall in ambient air is performed. It is shown that after the shock wave retreats, two counter-flowing jets are formed in a low-density hot gas, moving inside the thermal cavern from the electrodes to the center of the area. A strong distortion of the hot gas region is observed and a cold gas suction into the heat cavern is appeared. At the next phase, the formed hot gas jets collide, and the flow is directed radially from the axis of the initial discharge channel. Finally, the hot low-density gas takes the form of half the toroid, which size increases in time. The central part of the flowfield is occupied with the cold gas. The jet velocity and momentum are simulated at variation of the discharge energy (5-500mJ). The simulation results are considered to be consistent with the experimental data previously published by the authors.

1. Introduction
The spark electrical discharge has been thoroughly investigated [1] and is actively used in various fields of science, including ignition in internal combustion engines [2] and generation of NOx for biomedical applications [3], etc.

Recently, an interest was arisen for a long (5-50mm) pulsed (100ns-5μs) spark discharge due to its ability in generation of gasdynamic perturbations potentially beneficial for a flow control. In several publications, the results of studies on the features of the spark discharge thermal cavern decay were presented: it was shown that large-scale jet flows are formed in the direction perpendicular to the channel due to the spark channel bends during the decay of the long spark discharge [4]. The effect of discharge shape (bending) on pattern of the gasdynamic perturbations was shown by using additional electrodes to distort the shape of the discharge [5], and also by numerical simulation [5, 6]. Later, using an 8-frame IIT framing camera, direct experimental proof of the described effect was obtained [7].

It was proposed to use these specific properties of the spark discharge as a tool that rapidly increases the mixing degree of the injected fuel with an incoming high-speed (including supersonic) flow [8]. It was shown that the long spark discharge arranged transversely to the flow and passing through the core of the stream is localized along the jet of fuel and introduces significant perturbations into the area of fuel jet. This behavior of the discharge was supposed to promote a stretching the interface between the fuel and the oxidizer that might lead to intensification of mixing. However, the method used in the study - schlieren visualization of the flowfield - did not allow exact determination of the interface between the jet and the flow under the influence of the discharge. When fuel is injected from the wall-mounted nozzle into the supersonic flow, the fuel is localized in a near-wall layer and doesn’t penetrate far into the flow. In this case, the need to use a discharge that crossed the
core flow is eliminated. It appears that the use a spark discharge right on the wall might be a more appropriate approach. In paper [7], using numerical simulation it was shown that spark discharge near the surface in the region of fuel injection into a supersonic flow leads to a substantial increase in the interface between the fuel and the oxidizer, thereby accelerating the mixing of the components.

Another direction of the use of gasdynamics caused by a spark discharge is development of plasma actuators for suppression of a transonic buffet - the normal shock wave oscillations that takes place in transonic flow around a wing at high Mach number or high angle of attack [9-12]. The spark discharge acted as a vortex generator in the boundary layer on a flat surface [9-10, 12], and in the work published in paper [11] the spark discharge was combined with protrusions on the wing surface to create a directed transverse action. In paper [9], it was shown that the spark discharge near the surface leads to the formation of a jet of a rather complicated shape: after the shock wave has run away, the thermal cavern is significantly modified in geometry - a gas flows from the ends of the discharge channel to the center. Then the region of the low-density hot gas moves into plane perpendicular to the initial position of the spark, into a wide sector of about 180 degrees in the form of a half of toroid, the center of which takes a gas from the surrounding area. In paper [12], it was shown experimentally that such a discharge on the surface of the wing can lead to decrease the buffet amplitude. By the numerical simulation it was shown an entire suppression of the oscillations of the shock wave upon action by a pulse-periodic spark discharge.

There is of a practical interest to study a very short nanosecond (~ 100ns) discharges due to potential applications for the airflow control, including applied aerodynamics problems (flow around various bodies) and problem of combustion/ignition of lean mixtures. In paper [13], it was shown that the decay of a thermal cavern of a nanosecond spark discharge with a spark gap of 1 mm results in the transfer of active particles into the surrounding volume by a distance of ~ 10 mm from the discharge axis. To use the discharges for aerodynamic applications, it is important to optimize the discharge parameters in terms of the flow control authority, the effect on the airflow pattern depending on the discharge parameters, such as interelectrode distance and energy deposition [14]. In this paper, the major focus is made on the parametric study of structure and dynamics of gas perturbations caused by the long spark discharge.

2. Description of simulation model

Simulation of the gasdynamics caused by the decay of a thermal cavern arising after a breakdown of a long spark discharge was performed in the software package FlowVision 3.10.02. The simulation is based on the solution of the three-dimensional unsteady system of Navier-Stokes equations supplemented by the k-ε model of turbulence. Our previous simulation of spark in ambient air was performed without turbulence model [5], but real applications of spark for flow control or mixing corresponds to the turbulent flow. Therefore, in order to better reproduction the result of this work under turbulent flow conditions the turbulent model was used in current simulation. Gas density was calculated using ideal gas law. For description of transport gas properties tables with dependencies of viscosity, thermal conductivity and specific heat on temperature (up to 30kK) at constant pressure were used [15]. The simulation domain (shown in figure 1) was the one-eighth of the sphere with a radius of 30 mm due to the symmetry of the region: on the planes XY and YZ, the symmetry boundary condition was chosen; on the XZ plane, the adiabatic wall boundary condition with no-slip and a wall function for modeling the boundary layer in the first cell was set. On the remaining convex surface, the "input/out" boundary condition was set with the fixation of the total pressure 101325 Pa and the temperature 273K. Due to the fact that this condition leads to reflection of the shock wave back to the simulation domain when a shock wave passes through the boundary, shortly before the interaction of the shock wave with the boundary of the region, and for some time after it leaves the calculation domain, this boundary condition was replaced by the Riemann nonreflecting boundary. It was done in order to allow the shock wave leave the considered calculated domain without reflection. Unfortunately, this condition in FlowVision can be used with zero gas velocity indication on it only for a short time, otherwise unphysical vortex flows begin to form on the boundary. Therefore, this condition cannot be used during the whole calculation.
The 10-mm spark discharge was simulated by a volumetric heat source and was placed in the center of the sphere underlying the calculation domain. The shape of the volumetric heat source was a half of cylinder with rounded ends, recessed into the surface (figure 1). The cylinder radius was 0.3 mm, and its length was 5 mm due to the symmetry of the region. The spatial profile of the energy input was not applied, it was accepted that the energy input is uniform in length and radius. The dependency of heat power on time during the volumetric heat source operation was also not taken in the account, the energy input duration was set to 200 ns. Calculations were carried out for energy release into in the discharge of 5, 20, 60, 200 and 500 mJ.

![Figure 1](image_url)

**Figure 1** The calculation domain. 1 – symmetry condition, 2 – adiabatic wall condition, 3 – outlet with fixed total pressure, 4 – volumetric heat source, 5 – approximate location of electrode for such shape of discharge

In simulation, a variable time step was used. At the time of the volumetric heat source operation, the time step was 1 ns, and after that the time step was set by the Courant number $CFL = 1$. The calculation was performed on a reconfigurable rectangular grid using local adaptations in the specific area. Before the activation of a volumetric heat source, the grid in discharge region and in the space at a distance of 2.7 mm around the heat source was adapted, resulting in a spatial resolution of 54 cells per 1 mm. Periodic adaptation to the pressure gradient was used to accompany the shock wave by fine mesh, and periodic adaptation of the density field in the interval from 0 to 1.2 kg/m$^3$ was used to accompany the expanding heat cavern area. Moreover, in both cases, a space with a thickness of 6-20 cells around the indicated regions was additionally adapted to prevent the movement of gasdynamical features of considered flow between adaptations performed once per 5 time steps. Also in this procedure, outside discussed area, smaller cells were united into larger one if a shock wave or a low density gas left the given location of the computational domain. Adaptation made it possible to reduce the number of computational cells and the calculation time while maintaining sufficient spatial resolution in the area of the shock wave and thermal cavern. Periodic use of adaptation also reduced the calculation time, because one calculated step with the adaptation requires substantially more time than the step on which the grid was not rebuilt. Subsequently, with the motion of the shock wave to the boundary of the computational domain, the spatial resolution of the shock wave by the calculated grid was reduced due to the decrease of the influence of the shock wave on the thermal cavern. Starting from 70 μs, the spatial resolution of the computational grid in the region of thermal cavern was reduced to 14 cells per 1 mm, and was retained until the end of the calculation. As a result of the calculation, it was possible to sufficiently resolve by a grid and time step the gasdynamical features of the process under investigation and keep the number of cells at a level no greater than $5e+06$, and on average about $5e+05$.  

3
3. Results of simulation
The results of the numerical calculation are periodical (in increments of 10 ns to 5 μs) complete data saves of all gasdynamic quantities in volume under consideration, what makes possible after completion of the calculation to carry out a full analysis of gasdynamics of the thermal cavern decay. The gasdynamics of the thermal cavern passes through the following stages (see figure 2). First, within 200 ns, the region occupied by the volumetric heat source is heated, the thermal cavern rapidly expands and a shock wave leaves this region. After the retreat of the shock wave, high-speed jets are formed in the low-density hot gas, moving inside the thermal cavern from the electrodes to the center of the region (center of sphere). The position of one electrode matches to the left end of volumetric heat source in 1, and second one is located symmetrical about the dark blue (YZ) plane. As a result, deformation of the heat cavern occurs and cold gas suction is formed inside the heat cavern. After a while (at 80-180 μs, depending on the energy), the jets collide and a new jet flow arises in the YZ plane, directed outward from the center to the angle about 180°. As a result, the hot low-density gas takes the form of half the toroid, whose size increases in time, and the central region is occupied by the cold gas. This development of the gasdynamics caused by thermal cavern decay at the surface is in good agreement with the results of the experimental studies that presented in paper [6]. Typical experimental schlieren images of spark discharge cavern decay in YZ plane from paper [6] are presented in figure 3.

![Figure 2](image-url) The density field (kg/m³) at the decay of the thermal cavern, energy 0.5J, time moments 10, 80, 140, 200, 500 and 900 μs.

![Figure 3](image-url) Schlieren visualization of the gasdynamics of the spark discharge cavern decay in the YZ plane from paper for 0.7 J [6].
In addition to the qualitative comparison, a quantitative comparison of the results of the simulation with previous experiments is performed. Dependences of the coordinate of the thermal cavern boundary (the boundary of the density gradient) on time is obtained, typical position of boundary at time moment 100 μs were 3.6 mm at 60 mJ, 5.2 mm at 200 mJ, and 7.1 mm at 500 mJ. The initial velocity of expansion of the thermal cavern in the first 10 microseconds was 300-500 m/s, and at a later stage the velocity of the hot gas boundary did not exceed 30 m/s. The data obtained in the simulation correlate good enough with the data obtained in the experimental work [6], as it is shown in figure 4.

During the analysis of modeling data, it was found that the gas velocity is different from the velocity of motion of the thermal cavern boundary displacement. The gas velocity at 60 mJ energy release in volumetric heat source was 50-100 m/s, and for large values of the energy release it was in the range of 100-200 m/s for a long time about 200 μs. The dependences of the maximum velocity module and velocity components on time are presented in figure 5. It is clearly seen from the curves that the velocity modulus has a second local maximum in the time range of 80-160 μs, depending on the energy. The second local maximum corresponds to the time moment when the jets moving to the center of the spark collide, and a jet flow arises in the YZ plane, directed outward from the center of discharge. As a result, the maximum values of the velocity do not longer correspond to the x-component. In the calculations performed, typically, the maximum value of Vz at a given time is slightly greater than the maximum of Vy. The velocity fields for the planes XY and YZ for two different times - before and after the jet turn - are shown in figure 6.

![Figure 4](image1.png)

**Figure 4** The coordinate of the thermal cavern boundary along the Y axis as a function of time.

The results of the experiments were taken from paper [6].

![Figure 5](image2.png)

**Figure 5** The time dependences of the maximum velocity of the jet at different energy release
Figure 6 Changing the direction of the jet motion: the velocity fields (m/s) in the XY and YZ planes for the energy release of 60 mJ at the moments of 85 μs and 98 μs

During the analysis the calculation results, it was found that the maximum gas velocity is observed in the low-density region of the thermal cavern. On the other hand, the formation of jet in the hot region also results in the movement of the cold gas, which is sucked into the thermal cavern. As a result, the region of maximum momentum is shifted relative to the region of maximum velocity. To illustrate this effect, fields of the velocity magnitude $|V|$, the density $\rho$ and the specific mass velocity (momentum) of the gas $V\rho$ for the energy release of 200 mJ at the time moment 80 μs are presented in figure 7. For other energy inputs, structures are similar, but are shifted in time. From the presented velocity and density distributions it is clearly seen that they have opposite gradients along $X$, as a result of that the momentum distribution is uniform enough.

Figure 7 (a) velocity field (m/s), (b) density (kg / m³) and (c) momentum (kg / (m²s)) in the XZ plane for a 200 mJ energy input and 80 μs time point. Black line – wall, grey line – YZ symmetry

Figure 8 Dependence of jet’s maximum specific mass velocity (momentum $M$) and its components on time
The study of gasdynamics after spark discharge is caused by an interest to use this type of discharge as an alternative to mechanical and jet vortex generators for solution of problems of flow detachment control and mixing intensification in high speed flows. Therefore, such a characterization as a momentum is of much greater interest than the gas velocity. To estimate the influence of the energy release on the momentum, let us discuss the dependences of the specific mass velocity \( \rho V \) on time presented in figure 8. The graph shows the maximum values of the magnitude and the components of this quantity in space. It can be seen that the maximum local momentum is achieved not at the maximum energy considered 500 mJ, but at lower values of 60-200 mJ. This can be explained by overheating of the heat cavern: high velocity does not compensate the decrease of the gas density. Thus, with the help of numerical simulation and subsequent analysis of the results it was found that the spark with energy about 60 mJ per 10mm in ambient air should be the most efficient vortex generator for discussed spark geometry and duration of the energy release.

Conclusions
The simulation of the gasdynamic perturbations caused by the decay of a hot channel after the 200 ns pulsed 10 mm long spark discharge (5-500 mJ per 10 mm) arranged near the surface in the ambient air at laboratory conditions was performed using CFD-software package FlowVision 3.10.02. Using numerical simulation, it was shown that pulsed heating of the gas first leads to a rapid expansion of the thermal cavern and subsequent formation of high-speed jets directed from the ends of the plasma channel to its center with the suction of cold gas into the cavern. It was found that, in the time interval 80-180 \( \mu \)s after the discharge, a local maximum of the gas velocity occurs due to the collision of the jets moving to each other and the formation of a radial jet stream directed away from the center of the spark in a plane perpendicular to the axis of the discharge takes place. As a result, the hot gas takes the form of a half of toroid, increasing in time, and the cold gas occupies the central area. The qualitative picture of the hydrodynamics of the hot channel decay as well as the value of velocity of the thermal cavern boundary motion, obtained in the simulation, correlate well with the data obtained in previous experiments. Analyzing the results, it was found that the region of maximum velocity is shifted relative to the region having the maximum gas momentum, since the maximum velocity takes place in a region with a low density. It is shown that, in terms of the generated momentum, the most effective vortex generator is a spark with energy of the order of 60 mJ per 10 mm at discussed conditions.

Acknowledgments
The study of gasdynamic perturbations caused by spark discharge was initiated in JIHT RAS by Prof. Sergey Leonov and authors would like to thank Prof. Leonov for constructive discussions and important comments on the results obtained.

This work is supported by grant of Russian Scientific Foundation № 18-19-00740.

References
[1] Bazelyan E M, Raizer Yu.P 1998 Spark Discharge (CRC Press)
[2] Alvarez C E C, Couto G E, Roso V R, Thiriet A B, Valle R M 2018 A review of prechamber ignition systems as lean combustion technology for SI engines Appl. Therm. Eng. 128 107–20
[3] Janda M, Martišovitš V, Hensel K and Machala Z 2016 Study of transient spark discharge focused at NOx generation for biomedical applications J. Phys.: Conf. Ser. 768 012009
[4] Leonov S B, Isaenkov Y I, Firsov A A, Nothnagel S L, Gimelshein S F and Shneider M N 2010 Jet regime of the afterspark channel decay Phys. Plasmas 17 5 053505
[5] Shurupov M A, Leonov S B, Firsov A A, Yarantsev D A and Isaenkov Yu I 2014 Gasdynamic instabilities during decay of the submicrosecond spark discharge channel High Temp. 52 2 169-78
[6] Firsov A A, Shurupov M A, Yarantsev D A, Leonov S B and Shneider M N 2013 Mixing Actuation by Unstable Filamentary Discharge 51st AIAA Aerospace Sciences Meeting, Texas, USA, 07-10 January 2013, AIAA 2013-1188
[7] Firsov A A, Dolgov E V, Rakhimov R G, Shurupov M A and Leonov S B 2018 Mixing enhancement by electrical discharge in supersonic airflow *AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, AIAA 2018-1195

[8] Leonov S B, Isaenko Y I, Firsov A A, Yarantsev D A and Shneider M N 2010 High-Power Filamentary Pulse Discharge in Supersonic Flow *48th Aerospace Sciences Meeting, January 2010, Orlando, USA*, AIAA 2010-259

[9] Firsov A A, Isaenko Y I, Shurupov M A and Leonov S B 2016 Development of plasma actuator based on surface sparks for a buffet control *54th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, AIAA 2016-1690

[10] Firsov A, Isaenko Yu, Leonov S, Moralev I and Soudakov V 2017 Buffet suppression by submicrosecond spark discharge *7th European Conference for Aeronautics and Space Sciences (EUCASS), 3-6 July 2017*, paper EUCASS2017-291

[11] Sidorenko A A, Budovsky A D, Vishnyakov O I, Polivanov P A and Maslov A A 2017 Transonic buffet control by plasma actuator with spark discharge *7th European Conference For Aeronautics And Space Sciences (EUCASS), 3-6 july 2017*, paper EUCASS2017-345

[12] Firsov A A, Moralev I A, Isaenko Yu I, Leonov S B and Soudakov V G 2018 Suppression of transonic buffet phenomenon by spark plasma actuator *AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, AIAA 2018-2050

[13] Stepanyan S, Hayashi J, Lovascio S, Stancu G D and Laux C O 2017 Hydrodynamic effects induced by nanosecond sparks in air and air/fuel mixtures *55th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, AIAA 2017-1581

[14] Singh B, Rajendran L K, Bane S P, Vlachos P 2018 Characterization of Fluid Motion Induced by Nanosecond Spark Plasmas: Using Particle Image Velocimetry and Backgound Oriented Schlieren *AIAA Aerospace Sciences Meeting, AIAA SciTech Forum*, AIAA 2018-0680

[15] D'Angola A, Colonna, G, Gorse C et al. 2008 Thermodynamic and transport properties in equilibrium air plasmas in a wide pressure and temperature rang *Eur. Phys. J. D* 46 129