2D simulation of interaction between separation region and gasdynamic perturbation caused by rapid heat release in supersonic airflow

Saveliev A.S.
Joint Institute for High Temperatures of the Russian Academy of Sciences, Izhorskaya 13 Bldg 2, Moscow, 125412 Russia

E-mail: fisteh@mail.ru

Abstract. 2D modeling of rapid local release of thermal energy (0.1–7 J/m) into the supersonic airflow (Mach number $M=2$) was carried out in order to change the flow pattern around the wing model. It was shown that the results of numerical simulation of the process of a single instantaneous local heat release on the surface of the model are in good agreement with the experimentally obtained patterns of the flow around the wing model when energy was released using long spark discharge. The simulation of the process of local rapid energy deposition in a pulse-periodic mode was carried out and it was shown that, on the average over time, it is possible to significantly reduce the separation region size. In this case the calculated aerodynamic coefficients of the model change by several percent. It was found that the right choice value of frequency is important, since at an inappropriate value, the effect may be the opposite. It was shown that the stability of the repetition rate of local energy inputs is also important for the shift of the separation point downstream. The mechanisms of the effect of rapid local energy input on the separation region at relatively low and high values of thermal energy are discussed.

1. Introduction
Due to the increase in passenger traffic in civil aviation and the requirements strengthening for ensuring safety, environmental friendliness, lost cost of flight, researchers are looking for ways to improve the parameters of the airflow around the surface of the aircraft. This problem extends to the flights of supersonic aircraft, although civilian supersonic aircraft have not been operated recently due to difficulties arising, including for the reasons mentioned above. A decrease in drag, an increase in lifting force, an increase in the critical angle of attack, an increase stall resistance near the leading edge, and a delay in separation of the boundary layer are the desirable improving of flow parameters. The last phenomenon of flow separation can occur not only on the outer surface of the aircraft, but also on the inner surface of its power device (engine), for example, inside a supersonic inlets, where flow separation from the streamlined surface can occur, for example, during the interaction of the boundary layer and the shock (because of area of increased pressure behind its front). Such interaction may also occur near the trailing edge of the wing streamlined by a supersonic gas flow on its upper surface at a positive angle of attack. At this point, a relatively developed (thick) boundary layer and a region of increased pressure behind the front of the oblique shock wave, starting at the trailing edge, are simultaneously present. In this paper, the process of interaction of a gasdynamic perturbation
caused by the rapid heat release into the incident supersonic gas flow flowing around the wing model and the separation region arising near the trailing edge is investigated. Previously, experimental studies of this process were carried out [1–3], which showed that rapid heat release using a submicrosecond long spark discharge extended across the incoming flow, initiated near the streamlined surface upstream of separation region, can lead to a short-term shift of the separation point downstream. In these studies, a model of a NACA-airfoil with a built-in discharger for initiating a spark discharge was placed in a stationary supersonic airflow with Mach number \( M = 2 \) at a positive angle of attack. A separation flow was formed near the trailing edge, and a rapid local heating of the gas occurred upstream of the separation point, which led to the appearance of a gasdynamic perturbation affecting the separation region. In [1], the study was performed using the shadow (schlieren) method of visualization of flow pattern around the model, and in [2–3] using the Particle Image Velocimetry (PIV) method.

In other studies the authors also investigate a similar interaction of a gasdynamic perturbation with the separation region caused by a rapid turn of the flow on the ramp. In [4–5] the initiation of spark discharges in a pulse-periodic mode in caverns located upstream of the separation region was proposed. In [4], the flow was visualized above the ramp surface using the shadow (schlieren) method, and in [5], using the PIV method. It was shown that during the formation of a jet from a cavity after discharge, an unsteady flow arises, forming a swirling motion near the surface of the ramp, which leads to momentum transfer from the main flow to the boundary layer. As a result, the with the help of an electric discharge authors managed to reduce the separation region in front of the streamlined angle and, thus, to shift the separation point downstream. The authors note that the use of a discharge in a cavity has a limitation associated with choking of a supersonic flow in channels. It should also be mentioned that due to the peculiarities of gasdynamic flows in the channel, there is a limitation on the frequency of discharge initiation, due to the fact that the process of filling a cavity with gas after the jet exits has a time determined by the characteristic size of the channel and the speed of sound in the incoming gas. It should also be noted that constant (over time) heating of the boundary layer on the ramp surface leads the separation region to increase in the size [6].

In [7] the interaction of the separation region on a ramp with the surface plasma of a direct current discharge was studied at a freestream Mach number \( M = 2 \). It is shown that when the discharge is turned on, the force acting on the surface of the model can be significantly reduced. In another work [8], a spark discharge was used in a pulsed mode at a frequency from 5 kHz to 100 kHz with a relatively small average power up to 80 W. The flow velocity was corresponding to the Mach number \( M = 4.5 \). It was shown that the initiation of a discharge in the pulse-periodic mode leads to a change in the frequency spectrum of the pressure, perturbation of the flow pattern near the separation region. In [9], in which cases of sub- and supersonic flow around models with separation from the surface are considered, the authors note the fact that at present there is only a qualitative understanding of the effect of pulse-periodic energy release using plasma, namely, that it is necessary to generate gasdynamic perturbations of a sufficiently large pressure amplitude and with a “correct” frequency corresponding to flow of certain parameters. It is also noted that shock waves arising upon the initiation of a dielectric barrier discharge are not in themselves capable of significantly affecting the flow pattern because of their weak intensity and too short characteristic time of their existence.

The previously proposed methods of uncontrolled [1–2] and controlled [3] initiation of long spark discharge of submicrosecond duration near the model surface showed the possibility of heat generation in a wide range of specific discharge electrical energy in range \( 1–100 \, \text{J/m} \), which means the possibility of generating shock waves with an intensity of wide range of values. In contrast to a discharge in a cavity, gas that streamlines model’s surface takes part in the discharge directly, therefore, the initiation period is not limited by the filling time of the cavity. Moreover, there is no mass transfer process through the model’s surface, and the surface itself is practically devoid of edges on which weak shock wavers form. Since the dielectric surface does not directly contact the discharge plasma, it is not so susceptible to destruction due to high temperature. On the surface of the dielectric there are no high values of electric field strength that can cause its breakdown and destruction. The
geometric configuration of experiments with long spark discharge allows one to perform two-dimensional computer simulation of the process without involving significant computational capacity. Nevertheless, it should be noted the relatively low efficiency of converting electric energy to heat (up to 10% in the controlled mode [10, 11] and up to 20% in the uncontrolled mode [12]) for long spark discharge, while for other types of surface discharges this the value can reach 50% ([13, 14]). In [10–13], the efficiency was determined by comparing gasdynamic patterns (the position of the shock wave or the velocity of the induced unsteady gas flow) obtained by numerical simulation and in experiments.

The aim of this work is a 2D numerical simulation of the process of rapid heat release under geometric and initial conditions corresponding to experiments [1–3], but at a discharge repetition rate that is currently unattainable in the experiment.

2. Numerical simulation

Numerical simulation of the process was performed in the same way as in [10, 12]. A two-dimensional process of rapid heat release on the upper surface of a wing model (NACA profile of 12% thickness and a chord length of 100 mm), streamlined by a laminar supersonic air flow with a Mach number $M=2$, total pressure $p_0=1$ bar, and static pressure $p_e=0.15$ bar is considered, the total temperature $T=300$ K and the static temperature $T_e=160$ K. The air viscosity was calculated using the Sutherland formula. These values correspond to the parameters of a supersonic jet formed on a ST-4 supersonic atmospheric-vacuum wind tunnel [15]. Numerical simulation was carried out using the CFD package OpenFOAM v.5, solver sonicFoam [16], based on the calculation of pressure. The algorithm of this solver is based on solving the equations of continuity, momentum conservation (Navier-Stokes equation) and energy, using the PISO method. The use of PISO makes it possible to somewhat soften the condition on the value of the Courant number, which in turn allows one to carry out calculations on a more detailed grid with an acceptable value of the time step. The grid was built using the utilities included in the package, and the visualization of the data was carried out using the ParaView and gnuplot programs.

The calculation was carried out in the area divided into cells that make up the computational grid. The coordinate system was always chosen in such a way that the $x$ axis starts at the leading edge of the wing model and runs along its chord, and the $y$ axis also starts at the leading edge and is perpendicular to the $x$ axis (see figure 3a). The mesh is designed in such a way that its characteristic size is approximately one order of magnitude greater than the length of the chord of the wing model. The grid consisted only of hexahedral cells, and the characteristic size of the computational cell was varied in such a way that it had the least value near the model surface. In fact, the grid had two regions; in one, a practically inviscid gas flow was simulated, and in the other—viscous flow in the boundary layer. In the second region the grid has a thickening towards the surface of the wing model. The thickness of the region varied depending on the distance from the leading edge according to the formula corresponding to the thickness of the boundary layer. Figure shows a general view of the grid and its thickening to the surface in the boundary layer region.

The number of cells in the region of the boundary layer and in the region of inviscid flow could be programmed. In the field of viscous flow modeling 400 cells along the boundary layer and 40 cells across it were usually used. The total number of grid cells in the calculations was usually equal to 240 000. In [12], grid convergence was shown by the parameter of distance from the trailing edge of the wing to the point of separation of the boundary layer ($x_{SP}$, see figure 3a). A study was also made of the grid convergence in terms of the aerodynamic coefficients $C_l$ and $C_d$:

$$C_l = \frac{2f_l}{Ap_n u_n^2}, \quad C_d = \frac{2f_d}{Ap_n u_n^2},$$

where $f_l$ is the lifting force, $f_d$ is the drag force, $A$ is the surface area of the airfoil, $u_n$ is the freestream velocity, $p_n$ is the static freestream density. It should be noted that in values of $C_d$ and $C_l$ the forces caused by the shear stress in the laminar flow in the boundary layer are taken into account. It was
shown that for angles of attack of 0° and 10° the deviation of the calculated values (the position of the separation point \(x_{sp}\), \(C_d\), \(C_l\)) changes by no more than 1% when the number of cells changes from 240 000 to 300 000. In the study of grid convergence, grids with a number of cells from 80 000 to 300 000 were used.

![Figure 1. General view of the computational grid (left) and thickening to the top of the model surface (right): gray color is airfoil, blue color is incoming supersonic airflow, red color is region (width is 2 mm) of energy deposition. Here and below the direction of movement of the supersonic flow is from left to right.](image)

The validation of computer calculation methods in stationary and non-stationary cases was carried out. In the stationary case, the dependences (on the angle of attack) were compared: the positions \(x_{sp}\) of the separation point, the positions of the bow shock wave relative to the leading edge, and the velocity above the upper surface near the place of rapid heat release,—obtained in the simulation and in the experiment. The best agreement for the position of the bow shock wave is shown. It should be noted that the measured and calculated positions of the separation point are consistent with each other, but still differ: from 0% to 10% in the range of angles of attack from 0° to 14°, and in the calculation, the position of the separation point is downstream relative to one measured in experiment. This may be due to the fact that the real flow process that is realized in the experiment is not two-dimensional. Additionally, a study was carried out with the k-omega SST turbulence model turned on. In this case, the position of the separation point shifts downstream significantly, so that the size of the separation region differed from the experimentally measured one by 60% at an angle of attack of 10°. It should be noted that the calculated Reynolds number per chord length was about 500 000, which corresponds to the transitional flow regime in the boundary layer. So the study of turbulent phenomena was not carried out as part of this investigation; therefore, all data were obtained under the assumption of a laminar flow around the wing model.

The calculation was carried out as follows. First the stationary flow pattern was calculated. The obtained fields of pressure, temperature, and components of the velocity vector were used further in non-stationary calculation. For this, the \(p_d\) and \(T_d\) values, respectively, calculated by the formulas \([10, 12, 13]\) were set on the pressure and temperature fields in the region of the initiation of the discharge:

\[
p_d = p_\infty + \left(\gamma - 1\right) \frac{\eta E}{V_d},
\]

\[
T_d = T_\infty \frac{p_d}{p_\infty},
\]

where \(\gamma\) is the heat capacity ratio for air (assumed to be constant and equal to 1.4), \(\eta\) is the fraction of electric energy that goes into heating the gas, \(E\) is the stored electric energy corresponding to the value measured in the experiment and spent on maintaining the spark discharge, \(V_d\) is the volume of gas involved in spark discharge (volume of the instantaneous heating area). The value of \(\eta\) depends on the
method of initiating the discharge; it was taken equal to 0.2 for the uncontrolled method of initiating the discharge [10] and 0.07 for the controlled one [12]. The value of $V_d$ was calculated as the product of the transverse area of the glow of the discharge plasma visible in the shadow (schlieren) patterns ([1]) by the length of the discharge channel $L_d$. The place of heat release corresponds to the place of occurrence of the spark discharge in the experiment (which corresponds to the coordinate $x_d=63 \text{ mm}$ in the coordinate system of the computational domain). Figure 2 shows the pictures corresponding to the stationary flow at angle of attack $14^\circ$, the moment of heat energy release of $\eta E/L_d=7 \text{ J/m}$ and $10 \mu s$ after the discharge.

![Figure 2. Examples of the pressure field (Pa) in the stationary case (left), at the time of discharge (in the center) and 10 \mu s after the discharge (on the right).](image)

3. Obtained results

3.1. Simulation of rapid heat release in a single mode

As in experiments [1–3], it was found that the rapid heat release with an electric discharge leads to the formation of a quasi-cylindrical shock wave (see figure 2), which, propagating downstream, leads to deformation of the separation region and to a shift of the separation point downstream. Figure 3 shows the calculated fields of the $x$-component of the velocity ($u_x$) at various points in time after discharge: 0 $\mu s$, 10 $\mu s$, 75 $\mu s$, 110 $\mu s$. The mapping $u_x$ is illustrative when examining the position of the separation point and finding the separation region, since in this region in the laminar case there are reverse flows with a change in the sign of $u_x$. In other words, a return flow of gas occurs where a colder region is visible. It is seen that within about 100 $\mu s$ after the discharge, the gasdynamic disturbance caused by the spark discharge interacts with the separation region and the separation point is shifted downstream. Then, when this disturbance shifts downstream by incoming airflow, the separated flow begins to form again (see the last flow pattern). After even more time, the separation region grows and recovers in the form in which it was before the moment of rapid energy release. It should be noted that at the initial time after the discharge, the quasi-cylindrical shock wave has sufficient intensity (i.e., pressure behind the front) that can cause the separation of the boundary layer upstream. Figure 3b shows a small area of blue color corresponding to the separation of the boundary layer with subsequent reattachment. Then after several tens of microseconds the intensity of the shock wave decreases, and the separation with the reattachment disappears.

To accurately determine the position of the separation point relative to the trailing edge, the dependences of $u_x(x)$, where $x$ is the coordinate along the chord of the model (see figure 3a) were analyzed. As a result, it was found that the gasdynamic disturbance splits the separation region into several parts. This phenomenon is especially clearly seen at relatively low values of the energy input into the flow. The effect of fragmentation of the separation region is seen in figures 3b and 3c, where a separation region with reattachment and a region of two vortices arise respectively. Multiple parts of the separation region are also observed at relatively high values of released energy. Figure 4a shows an example of the calculated distribution of the modulus of the velocity vector $u$ along the $x$ axis (taking into account the sign of the projection $u_x$ on this axis) at a thermal energy $\eta E/L_d=0.57 \text{ J/m}$ and at a time of 50 $\mu s$ after the discharge. This distribution was calculated from a distance of 1 $\mu m$ from the surface of the wing model. It is seen that the separation region is fragmented into several parts. This is shown in figure 4b, where the filled and numbered areas correspond to the boundaries of the
reverse flows. In other words, if we draw a vertical line with the equation \( t = 50 \, \mu s \) in figure 4b, then the \( x \)-coordinates of its intersection with the shaded areas will correspond to the boundaries of the red areas in figure 4a. It should be noted that further we will operate with the value \( x_{SP} \) — the coordinate along the \( x \) axis relative to the trailing edge, i.e. \( x_{SP} = 100 - x \). Using the \( x_{SP} \) value is more convenient since it is equal projection of the width of the reverse flow region onto the \( x \) axis. The smaller this value, the smaller the separation region.

**Figure 3.** Examples of the obtained unsteady flow patterns at the value of heat energy of 7 J/m at various times \( t \) after the discharge: a) \( t = 0 \, \mu s \), b) \( t = 10 \, \mu s \), c) \( t = 75 \, \mu s \), d) \( t = 110 \, \mu s \). a) shows the location of the initiation of heat generation and some designations used.

It should be drawn to region 2 in figure 4b, which, as noted earlier, arises due to the propagation of a quasi-cylindrical shock wave upstream, at the front of which there is a positive pressure gradient. However, with relatively small amounts of energy input, this separation region with reattachment does not exist for long time. However, the time of its existence, as will be shown below, affects the choice of the frequency of discharge initiation. The effect of fragmentation of the separation region upon interaction with gasdynamic perturbation caused by rapid local heat release was obtained by other authors earlier [17].

To validate the method of numerical calculation in the non-stationary case, we compared the data on the position of the separation point obtained in the experiment and in the simulation. Figure 4b shows an example of such a comparison at a specific thermal energy of 0.57 J/m in the calculation and 0.44 J/m in the experiment and at an angle of attack of 8°. The distance from the trailing edge to the initial position of the separation point obtained in the calculation and measured by the shadow (schlieren) method [1] differ by about 10%. At about 30 \( \mu s \) after the start of the discharge, it can be seen that the main separation region (region 1) begins to decrease in size, i.e. the separation point shifts downstream. But at the same time, new regions of the return flow (regions 3, 4) are formed slightly upstream, which also shift downstream at a speed close to the displacement velocity measured in the experiment.

Also, an important conclusion follows from figure 4b, which explains the experimentally observed phenomenon [1] of the almost instantaneous return of the separation point to its initial position (see the solid curve in figure 4b at \( t = 60 \, \mu s \)). It can be seen in the calculation that at a moment of time about 60 \( \mu s \), the separation region arises and begins to grow almost at the place of the initial separation position \( x_{SP0} \) (region 8). By the time moment of about 120 \( \mu s \), the separation region almost completely returns to the initial size.
On the whole, the comparison of the data obtained (the position of the separation point relative to the trailing edge at different times $x_{SP}(t)$) in the experiment and in the simulation with a single energy release gives qualitative agreement. The best agreement between experiment and calculation is obtained by comparing the shadow (schlieren) patterns obtained in [1] and the calculated flow patterns. When comparing the velocity fields obtained using the PIV method ([2–3]) and using numerical simulation, they are less consistent. Each of the methods has its advantages and disadvantages, but the important differences between these two experimental methods used in experiments [1–3] are:

- in the case of shadow (schlieren) visualization, a flow pattern is obtained, which is averaged across the stream, and in the case of PIV is obtained in the plane of the laser sheet (in the experiments passed through the middle of the incident stream);
- the shadow (schlieren) method shows the boundaries of a sharp change in gas density (for example, the boundaries of the separation region or the front of the shock wave), while the PIV method gives the spatial distribution of two components of the velocity vector in the plane of laser sheet, including showing the regions of the return flow;
- PIV method, ceteris paribus, has spatial and temporal resolutions one order of magnitude greater than these for shadow method.

The first indicated circumstance has the greatest influence on the discrepancy between the experimental data (flow patterns) obtained by these two methods, because the supersonic flow used in the experiment is not strictly two-dimensional.

Figure 4. An example of the results of a non-stationary simulations at an angle of attack of 8°: a) the distribution of the velocity vector length along the wing model surface taking into account the sign at a time moment of 50 μs and at a thermal energy of 0.57 J/m, b) the dependence of the coordinates $x_{SP}$ of boundaries of the reverse flow regions on time after the discharge at thermal energy 0.57 J/m, as well as comparison with experiment (solid curve, [1]). $x_{SP0}$ (dashed line)—the coordinate (relative to the trailing edge) of the position of the separation point in the stationary case without energy release. The red star marks the time of energy release and its coordinate relative to the trailing edge.

3.2. Simulation of heat release in a pulse-periodic mode

Since it was shown above that there is a correspondence (either qualitative or quantitative) between the data obtained in the simulation and in the experiment, the next step in the studies was modeling the heat release into the flow in a pulse-periodic mode. In this case, as will be shown below, the discharge repetition rate was a value unattainable under the experimental conditions [1–3]. In [1], the initiation of a spark discharge with a frequency of about 10 kHz was shown, which corresponded to an average power consumption of 1 kW, which is the power limit for the high-voltage source used. With increasing energy or frequency, the necessary electrical power will be proportionally increased. In
simulation this periodic process was modeled as follows. First, an additional pressure and temperature field was applied to the stationary flow pattern, as described above. After a time, which can be called the period of initiation of the discharge, the same field of increased pressure $p_d$ and temperature $T_d$ was already superimposed on the unsteady picture and so on. As a result, after several iterations, the flow pattern on average ceased to change. Figure 5 shows the dependence of the position of the separation point on time for the thermal energy of the discharge $\eta E/L_d=3.2 \text{ J/m}$, an angle of attack of 14°, and a period of initiation of the discharge of 50 μs, calculated by diagram analysis similar to the diagram in figure 4b. Red stars indicate the place and time of energy release, and the place of energy deposition almost coincided with the position of the separation point $x_{SP0}$ in the stationary case.

![Figure 5. Example of the dependence of the position of the separation point on the time $SPP(t)$ for the thermal energy of the discharge of 3.2 J/m, the angle of attack of 14°, and the period of initiation of the discharge of 50 μs. $SPP_0$—position of the separation point in the stationary case without energy release; $SPP_{av}$—time-averaged $SPP$ value.](image)

It can be seen that the position of the separation point has shifted downstream, and after only the third pulse of the energy release into the stream, the flow pattern has became quasi-stationary. On average over time, with such a strategy for initiating a discharge, the position of the separation point is shifted downstream by 12 mm under these conditions. In this case, the average input electric energy is about 65 kW, which is not achievable in the experiment. It should be noted that such a high value of the required electric power is necessary because of the low efficiency of converting electric energy into heat (in this case, 7% [10]). With an increase in this efficiency to 50% (as, for example, in [14]), the required electric power will decrease by almost one order of magnitude.

Since when energy is released on the upper surface of the model, a region with increased pressure is formed, obviously, a decrease in the magnitude of the lifting force should be expected. On the other hand, with a positive angle of attack, the force caused by a local increase in pressure on the upper surface of the model will act in the direction of motion, which will reduce the drag force. Figure 6 shows the time dependences of the relative deviations of the $C_d$ and $C_l$ values for the simulation corresponding to figure 5. Therefore rapid heat release leads to a short-term increase in pressure on the upper surface of the model, which reflects a sharp decrease in the coefficients. This decrease lasts less than 10 μs, and then by the time of the next heat release, the aerodynamic coefficients restore their value to 2–3% of the initial one. Although it can be seen that the time-average value of the coefficient of lift decreases by 6% from the initial value, approximately the same relative value decreases the average drag coefficient.

On the whole, the selected parameters of heat release (discharge energy and its initiation frequency) should be considered as successfully selected, since on average the separation point shifts downstream, and the aerodynamic coefficients on average do not change dramatically. Figure 7 shows an example of poorly selected parameters for initiating a discharge at angle of attack of 10°: the dependence of the position of the separation point on time and the deviation from the initial value of the lift coefficient. The place of initiation of the discharge was the same, the discharge was initiated
every 20 μs, and the thermal energy of one discharge was 0.14 J/m (which corresponds to the average electric power in the experiment of 2.5 kW). Due to the fact that the average input into the flow is smaller, the average drop in the lift coefficient is only 3%, but the average position of the separation point is almost indistinguishable from the initial one. Also, the instantaneous positions of the separation point turn out to be noticeably higher upstream from the initial position before the discharge was initiated, and after about the fortieth act of heat release the position of the separation point is almost always higher than the initial one.

Figure 6. Time dependences of the relative deviation of the values of the coefficient of lift $C_l$ (left) and drag $C_d$ (right) from the corresponding initial values for the case corresponding to figure 5. $<C_l>$ and $<C_d>$ are the time-average values of the coefficients of lift and drag, respectively. $C_{l0}$ and $C_{d0}$ are their initial values calculated in a stationary case without heat release.

Figure 7. Time dependencies of the position of the separation point (left) and the relative deviation of the lift coefficient from the initial value (right).

Earlier in [1], it was shown that when a discharge is initiated in an uncontrolled manner, it leads to a significant fluctuation in the time period between spark discharges. Since the computer simulation technique provides such an opportunity, an additional study was conducted of the influence of the stability of the frequency of initiation of heat generation into the stream. For this, in the simulation corresponding to figures 5 and 6, in the case of successfully selected parameters, heat was generated once at a random moment in time (417 μs). It can be seen that after this the position of the separation point began to oscillate in a position upstream, i.e. on average, the flow pattern “worsens”.
4. Discussion

The technique is proposed for numerical simulation of rapid heat release on the surface of a model streamlined by supersonic gas flow in the presence of a separation region. The calculation results obtained for the conditions corresponding to the experimental ones ([1–3]) for stationary flow and for pulsed heat generation using long spark discharge of submicrosecond duration are compared. The simulation and experiment were carried out at angles of attack of 8°–14° to the incident supersonic airflow with a Mach number $M=2$. The thermal energy of one discharge released into the incident flow varied from 0.1 J/m to 7 J/m. The reasons for the discrepancy between the results obtained using different experimental techniques in stationary and non-stationary cases are indicated. The reasons for the discrepancy between experiment and calculation are also indicated. It is shown that the value of thermal energy matters, because at a relatively large value a separation region arises upstream of discharge place, worsening the flow pattern. Moreover, the lifetime of this separation region with reconnection also depends on the discharge energy. This time limits the spark initiation period from below. The period of initiation of heat release into the incident flow is limited from above, most likely, by the time of restoration of the flow pattern to one as in steady case. A suitable period for initiating heat generation will lie in this range. It is shown that the stability of the time period of initiation of the discharge affects the quasistationary flow pattern. So, with a random additional initiation of the discharge, this picture may eventually deteriorate significantly, while returning to the previous form (under stable period of initiation) may take much time.

As for the mechanism of the effect of rapid heat release on the separation region, two possibilities are presented here [3]. At relatively small values of the energy deposition, a swirling flow (trailing vortex structures, caused by discharge) is formed near the model surface, which drifts downstream, while leading to a more intense transfer of momentum from the main flow to the boundary layer and separation region, thereby flow tends to keep attached to the surface. A similar result was obtained by the authors when studying the separated flow on a ramp [5] and on wing model [1, 17]. At relatively large values of the discharge energy, the intensity of the quasi-cylindrical shock wave is significant, so that the pressure behind its front may be greater than the pressure in the separation region. In this case, the gas velocity behind the shock wave front can reach values of the order of tens of percent of the speed of the incident supersonic gasflow [3]. This unsteady flow affects the velocity inside the boundary layer, although for a short time. With the parameters considered in this paper (specific heat release energy 0.1–7 J/m), this time is limited to 100 μs, then the shock wave intensity decreases, and the gas velocity behind its front becomes close to 0 m/s (the case of propagation of acoustic waves).

It was shown that not only the magnitude of the energy plays a role when acting on the separation region, but also the initiation frequency. This qualitative result as a whole coincides with the

![Figure 8. Example of a simulation corresponding to figure 5, with an additional energy input at a random point in time (417 μs).](image)
conclusion in [9]. Nevertheless, according to the data obtained in this work on the dependence of the position of the separation point on time, it can be concluded that the initiation period of local heat release is limited from below by the existence time of separation with reconnection caused by the front of the quasi-cylindrical shock wave upstream, and also limited by the recovery time initial flow pattern. Apparently, with the correct choice of the energy of the spark discharge, the period of its initiation lies in the indicated time range.

The initiation energy affects not only the efficiency of displacing the separation region downstream, but also on the lifetime and size of the separation region with reconnection. A qualitative explanation is that a quasi-cylindrical shock wave has an intensity that decreases in time, because the wave is fading. Accordingly, over time, the velocity of propagation of the shock front decreases to sonic. Since the wave propagates through a gas moving at a supersonic speed, it begins to shift downstream at a certain time moment after the discharge, and the separation region with reconnection disappears. As was noted, both the intensity (pressure drop at the front causing separation of the boundary layer) of the wave and the velocity of its front monotonously depend on the amount of heat released in the spark channel. The theory of a strong explosion gives a dependence of the intensity and velocity of a wave on time and energy, and these quantities depend on energy in the form of a power law with an exponent of less than 1 [18]. A similar result was obtained earlier using numerical modeling [10], which was also used in this work, where the exponent at the discharge energy for the gas velocity behind the front of the cylindrical shock wave was 0.25. Such a “weak” dependence explains the fact that in the experiment [1], with an increase of 4 times the electric energy spent on maintaining the spark discharge, the dynamics of the displacement of the separation point downstream practically does not change.

Although in the experiment the energy value is limited by the maximum of output power of the high voltage source and the properties of the dielectric material (thermal stability), in the simulation model this value can be changed in any range. As a result of the calculation, it was found that a 20-fold increase in energy causes significant changes in flow pattern. With increased values of the thermal energy deposited in the flow near the surface of the model, the velocity behind the front of the shock wave is significant and comparable to the velocity inside the boundary layer. This increase in momentum for the boundary layer is sufficient to lead to a shift of the separation point downstream, as noted above.

This work does not provide quantitative data on the parameters of the initiation of a spark discharge, such as energy, repetition rate, and the place of initiation. This is due to the fact that it considers only one specific model streamlined by a supersonic air flow with specific parameters that repeat the conditions of previous experiments. These conditions may correspond to movement in air intake devices, near the surface of the controls of supersonic aircraft or small shells or missiles, although the use of high-voltage devices in last mentioned case is hardly feasible. When considering the real bearing surfaces of aircraft, it is necessary to take into account the three-dimensionality and turbulent nature of the flow, which was not taken into account in this work. However, such simple reasoning can be applied when first considered in a situation more appropriate for modern supersonic aircraft, when simulating fast heat generation using computer simulation.

The next stage of research by numerical simulation may be the case of a non-constant position of the place of initiation of heat release, which is also difficult to provide in the experiment, because on a real model the discharge device is usually rigidly connected to its surface. Also a change in the place of heat evolution from discharge to discharge is not only of interest, but also a simultaneous change in the period of initiation of the discharge and its energy. In this case the discharge energy may be unstable, for example, increase as the discharge initiation site shifts downstream. The continuation of this work may be a study by numerical simulation of the interaction of the shock wave with the boundary layer on the ramp, where rapid heat is generated in front of the separation region. Then, in this case, the problem can be considered with dimensionless parameters, such as the Mach number, the angle of rotation of the flow on the ramp, the ratio of pressure in the region of heat generation and pressure-plateau in the separation region, etc.
5. Conclusions
A numerical simulation of the two-dimensional process of rapid heat release into the oncoming supersonic gas flow in a pulsed and pulse-periodic mode is performed. The obtained data are compared with the experimental ones obtained earlier. It is shown that the initiation of heat upstream from the separation region causes the formation of an unsteady gasdynamic perturbation (quasi cylindrical shock wave), the propagation of which leads to a short-term shift of the separation point downstream. It is assumed that in this case two mechanisms are possible. The first is characteristic of relatively small values of the discharge energy and weak shock waves. Arising in the near-wall swirling flow lead to a more intense transfer of momentum from the main stream to the boundary layer and separation region. At relatively high discharge energies, a shock wave of higher intensity arises, and the parameters (pressure, velocity) behind its front turn out to be comparable (or even more) with the parameters in the separation region. The interaction of a shock wave of high intensity with the separation region also leads to a shift of the separation point downstream. The generation of shock waves of even greater intensity is undesirable, because this can lead to the formation and prolonged existence of a separation region upstream of place of initiation of heat release. This can lead to a large changes in values of the aerodynamic coefficients of the model. The simulation results in the case of a pulse-periodic mode showed that with correctly selected parameters for the initiation of heat release (the place of the discharge, its energy and the initiation period), it is possible to achieve a significant shift in the position of the separation point in average. In this case, it is important to provide energy release at appropriate time moments, for example, with a constant period corresponding to the recovery time of the separated flow after the discharge. As in the case of shock waves of too high intensity, aperiodic initiation of heat generation can lead to a deterioration of the flow pattern and values of aerodynamic coefficients.

References
[1] Golub V V, Saveliev A S, Sechenov V A, Son E E and Tereshonok D V 2010 High Temp. 48(6) 903–9
[2] Saveliev A S 2018 J. Phys.: Conf. Ser. 1112 012014
[3] Saveliev A S 2019 J. Phys.: Conf. Ser. 1394 012030
[4] Hongyu W, Jun L, Di J, Zhibo Zh, Mengxiao T and Yun W 2017 Int. J. Heat Fluid Flow 67(Part A) 133–7
[5] Mengxiao T, Yun W, Hongyu W, Shuangguang G, Zhengzhong S and Jiaming S 2018 Exp. Therm. Fluid Sci. 99 584–94
[6] Glushniva A V, Saveliev A S, Son E E, Tereshonok D V 2014 High Temp. 52(2) 220–4
[7] Watanabe Y, Houpt A and Leonov S B 2019 Aerospace 6 35
[8] Hedlund B, Houpt A, Gordeyev S and Leonov S 2017 Proc. 55th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum (Grapevine TX) AIAA 2017-0154
[9] Adamovich I V, Leonov S B, Frederickson R, Zheng J G, Cui Y D and Khoo B C 2017 Proc. 55th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum (Grapevine TX) AIAA 2017-1339
[10] Saveliev A S 2019 J. Phys.: Conf. Ser. 1394 012011
[11] Elias P Q and Castera P 2013 J. Phys. D: Appl. Phys. 46 365204
[12] Saveliev A S 2019 Vestnik OIWT RAN 3 31–6
[13] Znamenskaya I A, Latfullin D F, Lutsy A E Mursenkova I V and Sysoev N N 2007 Tech. Phys. 52 546–54
[14] Aleksandrov N L, Kindysheva S V, Nudnova M M and Starikovskiy A Yu 2010 J. Phys. D: Appl. Phys. 43 255201
[15] Son E E 2016 Physical Mechanics: Laboratory Workshops (Begell House Publishers)
[16] Greenshields C J, Weller H G, Gasparini L and Reese J M 2010 Int. J. Numer. Methods Fluids 63 1–21
[17] Tereshonok D and Son E E 2012 Europhys. Lett. 99 15002
[18] Sedov L I 1993 Similarity and Dimensional Methods in Mechanics (CRC Press)