Numerical and experimental modelling of MHD interactions at hypersonic flow around blunted body

T.A Korotaeva, V.P. Fomichev, M.A. Yadrenkin, E.K. Shipko
Khristianovich Institute of Theoretical and Applied Mechanics, 4/1 Institutskaya str., Novosibirsk 630090, Russia

Abstract. The problem of the reducing heat fluxes to the surface of the re-entry capsules when they enter the dense layers of the atmosphere remains an urgent challenge. The purpose of the numerical and experimental study presented in this paper is to check the possibility to decelerate the descent vehicles and reduce the thermal load on their surface in the upper atmosphere using the magnetohydrodynamic (MHD) method. The problem of the flow around a blunt body with a hypersonic air flow (M = 6) is considered when a local MHD interaction is realized near a stagnation point. An experimental study has been carried out at the MHD test rig. The test rig is designed to study the fundamental and applied problems of magnetoplasma aerodynamics. At the experimental simulation the free-flow ionization has been implemented using an electric high-voltage discharge, as distinct from the natural conditions when thermal ionization takes place behind shock waves at the flow hypersonic speeds. Numerical simulation of the flow around a blunt body has been carried out in the framework of Navier-Stokes equation with an approximate specification of the sources of force and heat in the region of MHD interaction. The flow parameters have been defined based on the results of the analysis of experimental data. The Stuart number has been obtained in all experiments that made it possible to establish a correlation between the calculated and experimental data. As a result of the studies, the possibility of changing the flow pattern around the model, the motion of the bow shock wave towards the free flow, and the heat fluxes decrease to the body surface have been demonstrated. It has been shown that, as a result of MHD interaction, the pressure and heat flux decrease about 2 times in the vicinity of the stagnation point where recirculated gas flows occur.

Introduction
Development of an effective thermal shield for reentry capsule is one of the most important problems in the advancement of space transportation systems. At high altitudes, capsule speed changes insignificantly. At the same time, shock waves are close to the surface of the vehicle causing the high heat fluxes. Magnetohydrodynamic (MHD) method of heat load reduction and deceleration enhancement is one of the possible methods to control the flow at hypersonic speeds in the upper atmosphere (70–30 km).

MHD methods used to increase the drag of bodies moving with hypersonic speeds and reduce the heat fluxes to their surface has been considered in [1–2]. Theoretical estimation of the altitude-velocity conditions for the realization of strong MHD interaction are presented in the works mentioned. There are many theoretical works devoted to the study of the magnetic field influence on the flow around blunted...
bodies and heat transfer processes \cite{3,4}. In the paper \cite{5}, the authors presented the results of theoretical studies of flight control of a high-speed vehicle using a magnetic field source in the forebody. It has been shown that the critical part of the typical flight trajectory of the re-entry capsule lies within the range of the MHD control implementation modes. There are works address the problem of using MHD systems on real re-entry capsules. The work \cite{6} is devoted to the numerical simulation of the flow control for the OREX orbital experimental re-entry capsule using the MHD system on the board. The parameters of the free flow and the heating temperature of the model shell were used based on the data obtained during the real flight experiment. It is noteworthy that it is possible to reduce the heat flux at the critical point by about 15\%, and reduce the total aerodynamic heating to 40\%, depending on the flight altitude of the vehicle at the moderate magnetic fields generated up to 0.5 T. Results of simulation of MHD deceleration of the re-entry capsule Stardust for various modifications of the magnetic system are presented in \cite{7}. It has been shown that is possible to reduce the heat flux to the surface of the vehicle by 2–3 times at the critical point with the total aerodynamic drag body increasing by almost an order of magnitude. Electrodynamic air deceleration of the re-entry capsule Hayabusa released into the atmosphere has been studied in \cite{8}. Total resistance of the body increased by 5\%, and the heat flux decreased 15\% at the magnitude of magnetic induction $B = 1$ T.

However, there are very few experimental works confirming the effects considered. Results of a study of the influence of a magnetic field on a shock-wave structure at a supersonic flow around a cone with a nitrogen flow and on the change of the heat flux to the model surface (cone-cylinder) are presented in \cite{9}. Ionization of the gas has been implemented using an electric discharge. It has been found experimentally that at the discharge current and the magnetic field induction increasing the thermal load on the model decreases.

That is difficult to implement the flow parameters corresponding to the thermal gas ionization that causes the phase transition at the bow shock wave at the phenomena modeling in aerodynamic test rigs. Under experimental conditions with a lower enthalpy of flow than under natural conditions artificial sources of gas ionization, for example, high-voltage electric discharges are used. Despite the fact that the ionization parameters obtained at the discharge using are not the same as at the natural conditions; and the discharge plasma is significantly nonequilibrium, one can claim the result of the simulated MHD-interaction would give reliable force acting on the flow and the flow pattern.

In the present work we study numerically and at testing the effect of MHD-deceleration of hypersonic flow ($M = 6$) in front of a blunt body in the region located between the detached bow shock wave and the surface of the model blunt part. In this zone, the MHD-force is determined by the current density and magnetic induction, i.e. value $\mathbf{j} \times \mathbf{B}$. This value can be used for definition of the parameter of the hydro magnetic interaction or Stuart number. Numerical modeling is carried out at the same Stuart numbers and flow parameters. At the numerical modeling the flow around the body, the region of MHD-interaction can be simulated by a zone with sources of force or force and energy specified. Addition of the force term in the equation of conservation of momentum and thermal terms in the energy equation permits on the one hand to exclude Ohm’s law from consideration, on the other hand to evaluate the possible MHD effect on the flow structure and thermal loads acting on the surface models.

**Problem statement and techniques**

The purpose of this study is to check the concept of the MHD parachute under the test conditions. Compared to the natural conditions where the thermal ionization takes place at the hypersonic flow deceleration, at the testing the free-stream ionization has been implemented using an electric high-voltage discharge. Figure 1 shows the schematic of the experiment (figure 1, a) and the problem statement for numerical simulation.
1.1. Experiment statement
An air flow with the Mach number $M = 6$ flows around the body under the conditions simulating a vehicle motion at the altitude of 28-30 km. The arrow shows the flow direction in figure 1. The magnetic field direction is orthogonal to the direction of the electric discharge current ionizing the flow. The region of the MHD interaction is organized in such a way that a decelerating force $\vec{j} \times \vec{B}$ arises in front of the blunt part of the model. The volumetric electromagnetic force is directed towards the free flow.

1.2. Numerical study statement
At the numerical modeling the flow around the body, the region of MHD interaction can be simulated by a zone with sources of force or force and energy specified. In figure 1, b, showing the schematic of the problem for the numerical solution, this zone is indicated by red color. The magnitude of the force $\vec{j} \times \vec{B}$ specified at the calculations can be determined from using value of the Stuart number (1).

$$ S = \frac{j \times B l}{\rho \infty v \infty ^2} $$

The number has been obtained in all experiments that made it possible to establish a correlation between the calculated and experimental data.

1.3. Test rig and experimental techniques
Experimental modeling has been carried out at the hypersonic air flow $M = 6$ using the pulsed aerodynamic MHD-test rig [10] based on a shock tube. Test rig is designed to study the fundamental and applied problems of magneto plasma aerodynamics. Electromagnet make it possible to change the magnitude of the magnetic field up to 2.5 T. Ionization of the flow in the vicinity of the stagnation point on the model has carried out using high-voltage electric discharge with the duration of 120 μs and value of about 120 A depending on the experimental conditions. Static pressure is of 1300–1600 Pa, flow density is of about 0.02 kg/m3, temperature is about 300K, flow velocity is about 2000 m/s. After the bow shock wave the static pressure increases by 4 times. At these conditions an electric discharge ionizing flow should be characterized as an arc discharge of low pressure with thermal ionization of air particles.

The stagnation pressure on the model at the critical point has been measured by a high-frequency sensor. Optical diagnostics has included schlieren photographs of the bow shock wave deformation and photographs of the discharge plasma glow. The observation has been carried out at the frequency of 80 kHz with exposure of 1 μs.

The photographs have been used to determine the length of the electric-discharge plasma in the direction along the magnetic field, which made it possible to estimate the magnitude of the discharge current density and the magnitude of the hydromagnetic interaction parameter $S$. The parameter has been also defined by the magnitude of the pressure change at the critical point as a result of the MHD interaction. These values are in good agreement with ones obtained at numerical simulation, although not in the entire range of parameters.
Experimental results

Measurements of the pressure in stagnation point of the blunted model and current oscillogram have shown the process of flow statement and MHD-influence. A typical pressure oscillogram is given in figure 2. One can see the main stages of the process: the flow settling near the model (I), quasi-stationary mode of the test rig (II) and the moment of the initiation of electric discharge in the magnetic field (III) leading to the gas pressure decreasing at the critical point of the model.

![Oscillogram of the discharge current (top) and oscillogram of the pressure sensor (bottom).](image)

Discharge burning occurs behind the bow shock wave where on the axis of the flow the pressure of about 52 kPa under the experimental conditions. Given the high value of current and relatively low energy deposition, one can presumably assume a thermal model of the discharge plasma, in which the separation of the electron temperature from the gas temperature is not significant. Thus, it is possible to estimate the zone of effective gas conductivity and current density using the Schlieren pictures. Assessment of the Stuart number is based on the assumption that the problem under study can be considered in a two-dimensional approximation since the electric current creating the volumetric electromagnetic force $\mathbf{j} \times \mathbf{B}$ flows predominantly in a transversal direction normal to the image plane.

Figure 3 shows the typical photographs of the flow and discharge plasma without and with magnetic field. The pictures are taken with frequency 80 kHz.

![Shock-wave structure with electric discharge and without B-field.](image)

One can see on figure 3 that the hypersonic flow blows of the discharge initiated in the vicinity of stagnation point. So the discharge plasma does not influence on the bow shock structure. Figure 4 shows the electrical gas discharge between the electrodes (figure 4,b), the counter flow movement of the discharge region as effect of action of the electromagnetic forces (figure 4,c-e) and the transformation of the departed bow shock wave before the MHD interaction zone (figure 4,c-e).
Numerical modelling

Numerical simulation of the problem has been performed using the academic version of ANSYS package. The stationary Reynolds-averaged Navier-Stokes equations have been solved for the axisymmetric problem statement. Turbulence model \( k-\omega \) SST has been used at simulation of turbulent regime. In the framework of the problem, a 'density-based' solver has been exploited for an implicit second-order accuracy scheme of Roe.

Figure 4.a shows an example of a two-dimensional computational mesh built using ANSYS ICEM CFD (grid generator). The mesh consists of about 200 000 rectangular cells. It is finer in the boundary layer zone (figure 5,a). One can see that two cell zones have been defined. The first one is 'fluid' noted 'I' with air property given, the second one ('II') can contain sources of the force and/or energy.

Figure 4. Shock-wave structure evolution at the strong MHD interaction between the flow and discharge plasma in the magnetic field.

Figure 5. Mesh (a) and boundary condition (b):
I – cell zone-'fluid', II – cell zone-'fluid1', 1 – ‘wall’, 2 – ‘axis’, ‘interior’, 4 – ‘pressure-far-field’, 5 – ‘pressure-outlet’.

Figure 5.b shows the schematic for assignment of the boundary conditions. The type ‘pressure-far-field’ is used as the boundary conditions on the external boundary of the computational domain, (figure 5,b), i.e. static pressure, the Mach number, and the static temperature have been set. On the model surface, the boundary non-slip condition ‘wall’ with the wall temperature \( T_w = 300^\circ K \) is used. The experiment has been a pulsed, with short-duration of only 120 \( \mu s \); this allows us to simulate the MHD interaction at a constant surface temperature of the body.

At the output boundary, the boundary type 'pressure-outlet' is used. The lower boundary of the computational domain for the asymmetrical formulation is the symmetry axis and the condition ‘axis’ is specified on it. The boundary between zones I and II is defined as ‘interior’. The air parameters are follows: the ‘ideal gas’ option is set as the equation of state, the heat capacity \( C_p \) is assumed to be constant, the thermal conductivity is determined by the kinetic theory, the viscosity is described by Sutherland's law. To account for electromagnetic forces in the frame of ANSYS FLUENT, a zone has been defined near the model that is either localized as at testing or occupying the entire blunt part...
(figure 5,a), in which the standard means of FLUENT have been used to set the source of distributed force and energy. Fundamentally, source terms are given for the equations of conservation of motion and energy. We present these equations in the form [11]:

\[
\rho \left( \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right) = -\nabla p + \vec{F} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z},
\]

\hspace{1cm} (2)

where the source of pondemotor force \(\vec{F}\) under the assumption of electric neutrality of the gas can be written as \(\vec{F} = j \times \vec{B}\). This force can be determined using formula (1). The energy equation must include the power determined by the pondemotor force \(\vec{F}\), and the Joule heating \(\vec{H}\).

\[
\rho \left( \frac{\partial H}{\partial t} + (\vec{v} \cdot \nabla) H \right) = \frac{\partial p}{\partial t} + \vec{F} \cdot \vec{v} + \text{div} \vec{q} + \frac{\partial \tau_{xx}}{\partial x} \vec{v} + \frac{\partial \tau_{yy}}{\partial y} \vec{v} + \frac{\partial \tau_{zz}}{\partial z} \vec{v} + \frac{j^2}{\sigma}
\]

\hspace{1cm} (3)

The velocity \(\vec{v}\) is determined as a value averaged over region I at the numerical simulation of the flow around the body taking into account only the source of force. Joule heating is determined using conductivity and current density determined by experimental studies.

In this paper, these source of force and energy for the mentioned flow parameters with the Stuart number of 0.1, for example, is of the order of 0.8 N/cm\(^3\) and 500 W/cm\(^3\), correspondingly.

Numerical simulation has been carried out under experimental conditions but it has not exactly repeated the experiment. The Stuart number, varying from 0.1 to 0.3, allowed finding a correlation between the results of experiments and calculations.

Figure 6 shows the heat flux distribution along the generatrix of the body of revolution. The solid black line corresponds to the uniform flow around the body. The other lines correspond to the cases of the MHD interaction simulation at the different Stuart numbers.

In figure 6 one can see an example of the results obtained numerically for a thin, “smeared” on the blunt part of the body imitating the MHD interaction region. Streamlines, pressure pattern and schlieren visualization obtained in one of the experiments described above are shown. A vortex zone in front of the blunt body can be clearly seen in the figure. The zone is significantly smaller in the case when only the force source is taken into account, figure 6,a. Figure 6,b shows that addition of thermal source leads to the significant stand-off increasing. Numerical flow pattern is close to the experimental one when both force and energy sources are “switched on”. Numerical simulation has shown that the region of MHD interaction leads to heat flux decreasing in to the blunt part of the model.

Figure 7 shows the heat flux distribution along the generatrix of the body of revolution. The solid black line corresponds to the uniform flow around the body. The other lines correspond to the cases of the MHD interaction simulation at the different Stuart numbers.
Figure 7. Heat flux distribution along the body generatrix at the uniform flow around the model (black line) and flow with force source corresponding to Stuart number $S \approx 0.1$ (red line) and $S \approx 0.2$ (green line).

The result obtained agrees with the conclusions of the work [7], where authors showed that that is possible to reduce the heat flux to the surface of the apparatus at its critical point by a factor of 2-3 at certain regimes. There is, practically, a complete deceleration of the flow that leads to the appearance of a vortex zone in front of the body and decrease of the heat flux in it.

The heat flux to the body decreasing is determined by the decrease in the recovery temperature of the flow decelerated in the magnetic field.

In figure 8 one can see an example of the results obtained numerically for a region imitating the region of MHD interaction in the case when it is localized in front of the blunt part of the model, as that is at the testing. Streamlines, pressure isolines and schlieren (flow visualization obtained in one of the experiments) are shown. The position and size of the region can be estimated from the solid black line over the pressure isolines (figures below).

![Figure 8. Experimental and numerical flow visualization (S = 0.2): (a) numerical data, (b) – Schlieren visualization compared with numerical results](image)

Discussion and conclusions
The problem of the flow around a blunt body with a hypersonic air flow ($M = 6$) is considered when a local MHD interaction is realized near the stagnation point. An experimental study has been carried out at the MHD test rig. At the experimental simulation the free-flow ionization has been implemented using
an electric high-voltage discharge. Numerical simulation of the flow around a blunt body has been carried out in the framework of the stationary Reynolds-averaged Navier-Stokes equations with an approximate specification of the sources of force and heat in the region of MHD interaction.

The flow parameters have been defined based on the experimental data analysis. The Stuart number has been obtained in all experiments that made it possible to establish a correlation between the calculated and experimental data. As a result of the studies, the possibility of changing the flow pattern around the model, the motion of the bow shock wave towards the free flow, and the heat fluxes decrease to the body surface have been demonstrated. It should be noted that only force effect taking into account (Stuart number) is not enough for adequate simulation of the MHD-influence on bow shock structure.

It has been shown that the MHD interaction leads to the pressure and heat flux decrease about 2 times in the vicinity of the stagnation point where recirculated gas flows occur.

The results of experiments and calculations prove that it is possible to control the shock-wave structure in the high-speed air flow in front of the blunt body with external ionization of the flow in the wide range of external magnetic field induction while reducing thermal loads on the surface of the capsule.

Acknowledgments
The research has been (partly) carried out within the framework of the Program of Fundamental Scientific Research of the state academies of sciences in 2013-2020 (project No. 0323-2018-0006 and 0323-2018-0011) and the grant from the Russian Science Foundation (project No. 17-19-01375).

References
[1] Rosa, R.J., Engineering Magneto-Hydrodynamics Ph. D. Thesis, 1956.
[2] Resler, E.L., Sears, W.R., The Prospects for Magneto-Aerodynamics J. Aeronaut. Sci., Vol. 25, April 1958, pp. 235-245, 258.
[3] Bush W.B., The Stagnation-Point Boundary Layer in the Presence of an Applied Magnetic Field, J. Aerospace Sci., Vol. 28, No. 8, Aug. 1961, pp. 610–611.
[4] Kemp, N.H., On hypersonic stagnation-point flow with a magnetic field, J. Aeronaut. Sci., 1958, vol. 25, No. 6, pp. 405-407.
[5] Ericson, W.B., Maciulaitis, A., Investigation of Magnetohydrodynamic Flight Control, J. Spacecraft, Vol. 1, No. 3, 1964, pp. 283-289.
[6] Fujino, T., Ishikawa, M., Numerical simulation of control of plasma flow with magnetic field for thermal protection in earth reentry flight, IEEE Trans. on Plasma Sci., 2006. Vol. 34, No. 2. pp. 409-420.
[7] Bityurin, V.A., Bocharov, A.N., Features of electromagnetic thermal protection of space capsule/doi: 10.1134/S106378501105004X
[8] Katsurayama H. et al., Numerical Study of the Electromagnetic Control of a Weakly Ionized Flow around a Blunt Body: Role of a Insulative Boundary in the Flow, AIAA Paper 2007-4529.
[9] Bobashev, S.V., et al., Experimental investigation of magnetohydrodynamic action on a heat flux toward the surface of a model, Technical Physics, doi: 10.1134/S106378421012008X
[10] I. P. Ginzburg “Aerogazodinamika (Kratkii Kurs)” Moscow, Izd. Vyshaya Shkola, 1966, p 1–404.
[11] Golovnov, I.A., Pozdnyakov, G.A., Pravdin, S.S., Podzin, V.E., Pushkareva, T.I., Fomin, V.M., Fomichev, V.P., Yakovlev, V.I. A Test rig for experimental simulation and study of the MHD-interaction in a hypersonic flow, Preprint № 7-2003, ITAM SB RAS, 2003.