Calculation analysis of the experimental data of HIFiRE-I using the computer code NERAT-2D

S T Surzhikov
Ishlinsky Institute for Problems in Mechanics RAS, Vernadsky prospekt 101(1), Moscow, 119526, Russia
Moscow Institute of Physics and Technology (State University), 9 Institutskiy per., Dolgoprudny, Moscow Region, 141701, Russia
Dukhov All-Russian Scientific Research Institute of Automatics, Moscow, Russia
surg@ipmnet.ru

Abstract. A computer simulation of the densities of convective heat fluxes to the surface of a blunted circular cone has been performed at hypersonic flight speeds. For comparison, the conditions implemented in the scientific program HIFiRE-I were used. Calculation studies were performed using the author's computer code NERAT-2D and the algebraic models of turbulent mixing of Baldwin–Lomax and Prandtl. A satisfactory agreement between the calculated and experimental data is shown.

1. Introduction
The international research program HIFiRE (Hypersonic International Flight Research and Experimentation) [1–3] was created with the aim of conducting flight experiments in the field of hypersonic technology with an emphasis on the study of aerothermodynamics and thermogasdynamics problems that are extremely difficult or impossible to solve using aerodynamic stands.

The main executors of the scientific programs were the US AFRL (United States Air Force Research Laboratory) and DSTO (Australian Defense Science and Technology Organization). Significant participation in the program was made by Boing and the University of Queensland in Australia.

With the development of hypersonic technology, it became clear that one of the main problems is the extremely high thermal and accompanying force impact on the aircraft. For the design of new devices, a detailed knowledge of these influences is necessary in conditions as close as possible to the real ones. A wide range of ground studies performed using aerodynamic tubes showed that the specific features of hypersonic aerophysics are not fully modeling on ground stands. There was an urgent problem of conducting flight experiments and filling the database on the results of hypersonic research both on ground stands and in real flight.

Obviously, such a work should be closely linked with the use of modern computing and information computer technologies. One should pay attention to the fundamental difference between the HIFiRE program and other hypersonic programs, for example, the X-41 and X-51 [4,5], the main purpose of which was to demonstrate the technological possibilities of implementing a controlled hypersonic flight. The HIFiRE program focused on the study of the fundamental scientific problems of hypersonic flight.
The main objects of research were:
1) Aerodynamics and aerothermodynamics;
2) Thermogasdynamics of scramjets;
3) Integration of the propulsion system with a high-speed glider;
4) Flight management;
5) High-temperature materials and structures;
6) Organization of measurement of high temperatures and force impacts;
7) System research in the field of organization of hypersonic flights.

Another important element of the HIFiRE program was the organization and mutual coordination of flight, bench and computer studies of the above objects.

Nine flight experiments were planned in the scientific program HIFiRE, not all of which were realized:
1) HIFiRE-0: testing of control systems, instrumentation and software for research launches;
2) HIFiRE-1 and 5: investigation of hypersonic boundary layer features near the surface of circular and elliptical blunted cones;
3) HIFiRE-2, 3 and 7: are directed to the study of the thermogasdynamics of a scramjet;
4) HIFiRE-4 and 6: research problems of control of hypersonic flow;
5) HIFiRE-8: study of integration problems of the propulsion system with a hypersonic glider.

Flight experiments were performed using rocket accelerators. Most of the experiments were performed on the descending part of the ballistic parabolic trajectory. The HIFiRE-1 experiment was implemented on March 22, 2010. The main purpose of this experiment was to study the aerodynamic heating of a blunted circular cone. This launch was preceded by intensive ground-based experiments and calculated studies using computer codes [6, 7]. Despite a number of technical problems in the performance of this flight experiment, some experimental data were obtained and published in [3]. The HIFiRE-5 experiment was realized on April 23, 2012. The main task of this experiment was to study laminar and turbulent junction on an elliptical cone.

As already noted, the experiment HIFiRE-1 was preceded by bench tests and calculated studies of the flight conditions of the experimental blunted circular cone. In [6, 7] a detailed description of experiments performed on the shock tubes of the Calspan University at the Buffalo Research Center (CUBRC) under the leadership of M. Holden is given. In particular, a detailed analysis of the experimental and calculated distributions of the density of convective heat fluxes along the surface of a blunted cone with fixation of a section of a laminar-turbulent transition is given in [7]. Two experiments were thoroughly investigated: flow around the cone with the Mach number \( M = 6.58 \) (test # 4) and at \( M = 7.16 \) (test # 30). The results of numerical simulation using computer code DPLR [8] and different models of turbulent mixing are also given: the Baldwin-Lomax model [9], the Spalart–Allmaras model [10], and SST model [11]. In the numerical studies, the STABL code was also used [12, 13].

In this paper, we present the results of numerical simulation of convective heating of the experimental HIFiRE-I cone for the conditions at \( M = 6.58 \) and \( M = 7.16 \) using the author's NERAT-2D computer code. This code is based on finite difference method, the integration of a system of Reynolds-averaged Navier–Stokes equations on multi-block structured grids. Various versions of this code were used to analyze the flight and calculated data of Fire-2 [14], Apollo-4 [15], RAMC-II [16], as well as experiments [17]. For the modeling of turbulent mixing, the algebraic models of the Prandtl mixing path (PMM) [18] and the Baldwin-Lomax model (BLM) [9] were used.

2. Results of numerical simulation
The initial numerical simulation data are presented in table 1. Location of the laminar-turbulent transition was determined by input of the critical Reynolds \( Re_c = \frac{\rho,V_e x}{\mu_e} \) number, where \( \rho_e, \mu_e \) are the density and dynamic viscosity in the oncoming stream, \( V_e \) is the velocity in the oncoming stream.
Table 1. Initial data for calculation of convective heating of the experimental model HIFiRE-1.

| Variant | $p_\infty$ (erg/cm$^3$) | $\rho_\infty$ (g/cm$^3$) | $T_\infty$ (K) | $V_\infty$ (cm/s) | M |
|---------|--------------------------|---------------------------|---------------|------------------|---|
| A       | $7.33 \times 10^4$       | $0.124 \times 10^{-3}$    | 214.4         | $1.927 \times 10^5$ | 6.58 |
| B       | $4.62 \times 10^4$       | $0.720 \times 10^{-4}$    | 231.7         | $2.183 \times 10^5$ | 7.16 |

Since the experimental data are presented in [7] only for the conical and cylindrical part of the experimental blunted cone HIFiRE-1, the closing cone of the real experimental model was not taken into account.

Figure 1 shows two types of computation grids, which were used in this paper. Four blocks of the grid were used. The first type of structured nonhomogeneous computational grid (figure 1, a) had the following number of nodes in four blocks:

1st block (near blunted nose): $NI \times NJ = 401 \times 81$, where $NI$, $NJ$ is the number of nodes along the normal to the surface and along the surface;

2nd block (along the conical and cylindrical parts of the model): $NI \times NJ = 401 \times 401$;

3rd block: $NI \times NJ = 401 \times 97$;

4th block: $NI \times NJ = 41 \times 97$.

Note that in the fourth block the index of the grid $J=1, ..., NJ$ varies along the normal to the surface.

Figure 1. Geometry of the experimental cone HIFiRE-1 and two types of computational grids.
The second type of computation grids (figure 1, b) was significantly more economical:

1st block: $NI \times NJ = 101 \times 21$;  
3rd block: $NI \times NJ = 101 \times 55$; 
2nd block: $NI \times NJ = 101 \times 201$;  
4th block: $NI \times NJ = 21 \times 55$.

However, in the first cell resting on the surface, an additional grid of 20 nodes was inserted (figure 1, b).

Figures 2 and 3 show numerical simulation results for the densities of convective heat fluxes on surfaces obtained using a grid of the 1st type. In this case, the minimum value $y^+ = y \frac{\rho_u}{\mu_u} \left( \frac{\partial u}{\partial y} \right)_w = 20$, where $u$ is the velocity along the surface at a distance $y$ from it.

Figure 2 shows the results of calculations using the Prandtl mixing model (PMM) [18]. Here we demonstrate the effect of the critical Reynolds number and the turbulent Prandtl number on the numerical simulation results for experimental variant A (see table 1).

![Figure 2](image.jpg)

**Figure 2.** Density of convective heat fluxes along the surface in the experimental variant A. Model of turbulent mixing PMM.

The first type of grid.

Figure 3 compares the calculated data obtained with the Baldwin–Lomax model (BLM) and the PMM model for a fixed Reynolds number $Re_*$ for experimental variant B.

The results of similar calculations on a much coarser grid, but with a sublayer in the first cell, are shown in figure 4 (variant A) and in figure 5 (variant B). In this case, the minimum value is $y^+ = 0.5$.

Presented computational data allow us to confirm the well-known results of the simulation of heat transfer in turbulent flow about the fundamental importance of choosing a detailed calculation grid near the surface. Note that a coarse grid along the surface also leads to significant errors in determining the density of heat fluxes on a cylindrical surface ($x > 120 \text{ cm}$).

### 3. Conclusion

Validation of the NERAT-2D computer code together with the Prandtl and Baldwin-Lomax turbulent mixing algebraic models was performed using the experimental data of HIFiRE-I.
A satisfactory description of the experimental data is shown. The importance of using a detailed grid for simulating turbulent heat transfer and good agreement with experimental data was confirmed.

**Figure 3.** Density of convective heat fluxes along the surface in experimental version B. Models of turbulent mixing BLM (1) and PMM (2). The first type of grid.

**Figure 4.** Density of convective heat fluxes along the surface in the experimental version A. Models of turbulent mixing BLM (1) and PMM (2). The second type of grid.
Figure 5. Density of convective heat fluxes along the surface in experimental version B. Turbulent mixing models BLM (1) and PMM (2). The second type of grid.

Acknowledgments
The work was performed within the framework of the Government program of basic research of the Russian academy of sciences (contract # AAAA–A17–117021310372–6) and partially under support of the RFBR grant # 016–01–00379.

References
[1] Kimmel R L, Adamczak D and Brisbane DSTO-AVD Team 2011 HIFiRE-1 Preliminary Aerothermodynamic Experiments AIAA 2011-3413
[2] Li F, Choudhari M, Chang C, Kimmel R and Adamczak D, 2011 Transition Analysis for the HIFiRE-1 Flight Experiment AIAA 2011-3414
[3] Stanfield S A, Kimmel R, and Adamczak D 2012 HIFiRE-1 Data Analysis: Boundary Layer Transition Experiment During Re-entry AIAA 2012-1087
[4] McClinton C R X-43 2006 Scramjet Power Breaks the Hypersonic Barrier Dryden Lectureship in Research for 2006 AIAA 2006-0001
[5] Hank J M, Murphy J S and Mutzman R C 2008 The X-51A Scramjet Engine Flight Demonstration Program AIAA 2008-2540
[6] Wadhams T, Mundy E, MacLean M and Holden M 2008 Ground Test Studies of the HIFiRE-1 Transition Experiment. Part 1: Experimental Results J. of Spacecraft and Rockets 45 (5) pp 1134–1148
[7] MacLean M, Wadhams T, Holden M and Hohnson H 2008 Ground Test Studies of the HIFiRE–1 Transition Experiment. Part 2: Computational Analysis J. of Spacecraft and Rockets 45 (5) pp 1149–1164
[8] Wright M J, Bose D and Candler G V 1998 A Data Parallel Line Relaxation Method for the Navier–Stokes Equations AIAA J. 38 (9) pp 1603–1609
[9] Baldwin B S and Lomax H 1978 Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows AIAA Paper 78–0257
[10] Spalart P R and Allmaras S R 1992 A One-Equation Turbulent Model for Aerodynamic Flows
AIAA J. 32 (8) pp 1598–1605

[11] Menter F L 1994 Two-Equation Eddy-Viscosity Turbulent Models for Engineering Applications
AIAA J. 32 (8) pp 1598–1605

[12] Johnson H and Candler G 2005 Hypersonic Boundary Layer Stability Analysis Using PSE-Chem
AIAA 2005-5023

[13] Johnson H and Candler G 2006 Analysis of Laminar-Turbulent Transition in Hypersonic Flight Using PSE-Chem
AIAA 2006-33057

[14] Surzhikov S T 2016 Radiative Gas Dynamics of the Fire-II Superorbital Space Vehicle
Technical Physics 61(3) pp 349–359

[15] Surzhikov S T 2016 Radiative Gas Dynamics of Apollo Command Module at Angle of Attack
AIAA paper 2016-4022

[16] Surzhikov S T and Shang J S 2014 Three Dimensional Simulation of Shock Layer Ionization for RAMC-II Flight Tests
AIAA paper 2014-1078

[17] Surzhikov S T 2013 Convective heating of small-radius spherical blunting for relatively low hypersonic velocities
High Temperature 51 (2) pp 231–245

[18] Tannehill J C, Anderson D A and Pletcher R H 1997 Computational Fluid Mechanics and Heat Transfer
Taylor & Francis 792 p