International Journal of Engineering Research and Development
***e**-ISSN: 2278-067X, **p**-ISSN: 2278-800X, www.ijerd.com*
Volume 3, Issue 4 (August 2012), PP. 52-57

Flow Analysis of an Atmosphere Reentry Vehicle

Dr.B.Balakrishna[1], S. Venkateswarlu[2], Dr P. Ravinder Reddy[3]
[1]*Associate Professor, Department of Mechanical Engineering, University College of Engineering, JNTU Kakinada, A.P, India.*
[2]*PG Student, Department of Mechanical Engineering, University College of Engineering, JNTU Kakinada, A.P, India.*
[3]*Professor&Head, Department of Mechanical Engineering, Chaitanya Bharathi Institute of Technology, Hyderabad,AP,India.*

Abstract—The main thrust of the paper is the evaluation of the aero thermodynamic analysis of reentry trajectory. Computational fluid-dynamics results are presented to show the flow field around a blunted cone-flare in hypersonic flow. This is of particularly interest since it features the hypersonic flow around planetary re entry vehicles, the reason between the cone and the flare is particularly critical with respect to the evaluation of the surface heat flux. Indeed, flow separation is induced by the shock wave boundary layer interaction, with subsequent flow re attachment, that can dramatically enhance the surface heat transfer. The exact determination of the extension of the recirculation zone is particularly delicate task for numerical codes. A numerical approach has been adopted to study the flow field that develops around the EXPERTS capsule. Laminar flow computations have been carried out using a full Navies- Stokes solver, with free stream condition with two different Mack numbers. Model created in CATIA, and meshing in GHAMBIT 2.4.6 finally solution is by FLUENT 6.3 solver.

Keywords—Re-entry vehicle, Hyper sonic flow, Heat flux, Navies strokes formula, Laminar flow.

I. INTRODUCTION

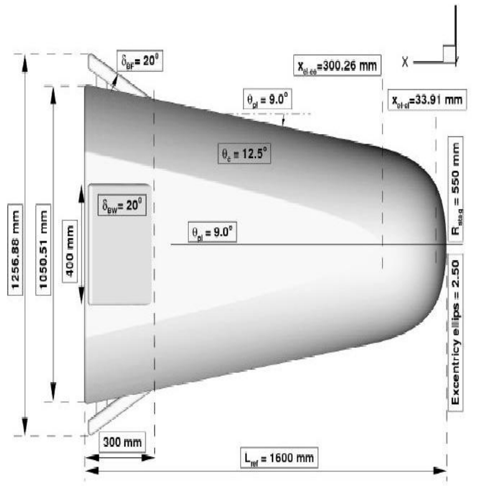
The general objective of the European Space Agency FLPP program (Future Launchers Preparatory Programmed, [1]) is placing Europe inside the worldwide strategic area of atmospheric reentry for future international transportation, exploration and scientific projects. Several studies on experimental vehicle concepts and improvements of critical reentry technologies have been undertaken in recent years by ESA (ARD), France (Pre-X), Germany (Phoenix) and Italy (USV), in order to consolidate their worldwide position in this strategic field ([1]). The aero thermodynamic studies can support and address both aero shape consolidation and mission analysis [2], whose goal is to minimize the heat fluxes to the windward part of the vehicle (nose cap) and control surfaces (flaps). Body-flap efficiency prediction and environment characterization have been carried out in recent past with both thermo-chemical equilibrium and non equilibrium flow assumption ([3],[4]). A preliminary benchmarking phase [5] has been performed in order to assess the numerical strategy to be adopted for the scheduled activity concerning both laminar and transitional simulations to be performed with different flow field hypotheses (chemical equilibrium and non equilibrium), and also accounting for the effects of the angle of attack, the angle of sideslip, the Mach number (i.e. total enthalpy) and the symmetric and asymmetric flap deflection.

The numerical code used to carry out the aerothermodynamics analysis of the IXV vehicle is the CIRA code H3NS [6] that solves the Reynolds Averaged Navier Stokes equations in a density-based finite volume approach with a cell centered, Flux Difference Splitting second order ENO-like upwind scheme for the convective terms. The need for a safer access to space dictates the review of operational capabilities and hence of design approach for manned reentry vehicles of next generation [7–9]. Research has shown that reentry vehicle designs with high L/D could be designed to take advantage of aerodynamic lift during reentry. Higher L/D is desirable because it increases the area from which a re-entering vehicle can be recovered (e.g. reentry window) [10]. Keeping constant the temperature of the nose stagnation in the radiation equilibrium conditions, restricting the g peak experienced to less than a tenth above normal ground-level values and, finally, with a wider than usual "reentry window" that would permit landing at any one of the many choices of airfields [11]. The results, provided in this paper, consider a Mars entry scenario compliant with an approach to the red planet both by elliptic and hyperbolic orbit [12], [13]. These results may be used to provide numerical data for understanding requirements for the human exploration of Mars

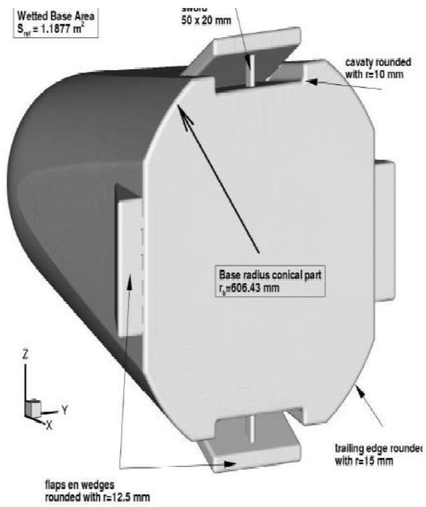
From the point of view of approach strategies, the different values of velocity at entry interface (given the entry angle) will characterize the MBS design by means of mechanical loads (i.e. pressure and acceleration), thermal loads (i.e. heat flux peak and integrated heat load), and landing dispersion [16], [15]. The flow around the capsules was computed by the DSMC method. The number of collisions was calculated by the majorant frequency technique [16,17]. The collisions of molecules were computed using the variable hard sphere (VHS) model. The internal degrees of freedom were taken into account by the Larsen – Borgnakke model [18]. The SMILE software system was used for computations [19]. A number of papers have been already written about EXPERT with different aims, from the evaluation of the aerodynamic behavior to the description of tests and experiments to be made during the re-entry (see [20] and related references). Preliminary computations of aero-thermo-dynamic data base at high altitudes were provided by approximate engineering methods or bridging formulae [20,21]. The aim of the present work is making an additional analysis and, hopefully, a better characterization of the aero-thermo-dynamic data base in rare field regime.

II. MODELING AND MESHING OF EXPERT CAPSULES

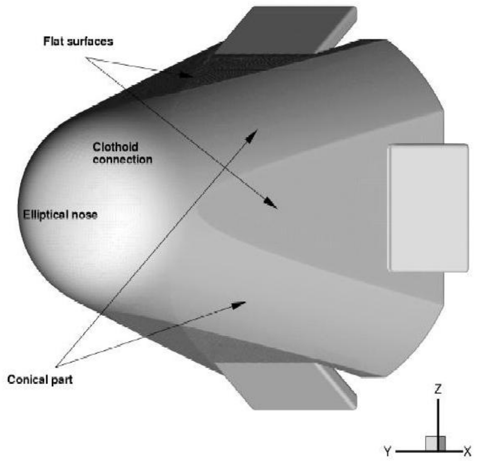
The geometry of the EXPERT capsule is quite complex and the solid modeling is carried by Catia 3D modeling tools. The dimensions taken for the EXPERT capsule as for the figures 1.The solid model was imported from CATIA to GAMBIT. The model of the EXPERT Capsule is modified slightly to do the flow analysis. Here the analysis the flow over the capsule so requires the flow domain for the flow analysis. Therefore for flow analysis, a flow domain is created as for the dimensions required. Here creating the cylindrical shape domain with diameter 4L and length 7L.where L is the length of the Capsule. Then subtract the main body form the flow domain. Before starting the mesh need to create the boundary layer around the capsule body.



a Side view



b. Base view



c. Perspective view

Fig1. Dimensions of an EXPERST capsule

And then mesh the faces of the body by using tri pave mesh. To create 3D mesh of the domain the tetrahedral pave elements are used. Check the mesh of the domain for convergence. In this the flow domain selected as the

PRESSURE-FAR –FIELD and the Capsule body as a WALL. The meshed capsule with flow domain is shown in figure 2.

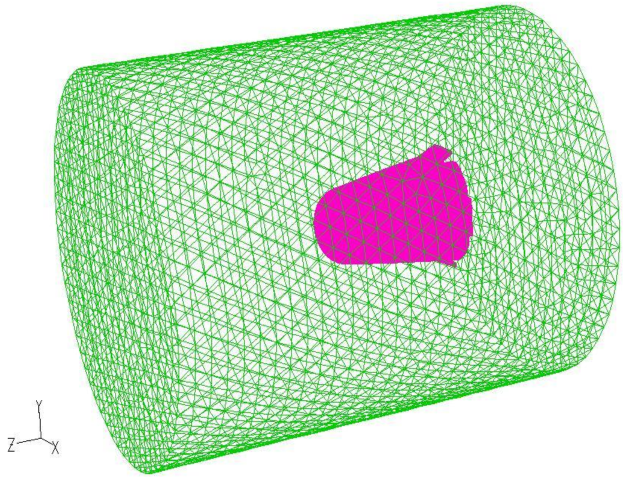


Fig2. Meshed capsule within the flow domain

III. FLOW ANALYSIS USING FLUENT 6.3.26

After the mesh of the capsule in Gambit then it is imported to FLUENT 6.3.16 software for the flow analysis with Mach number Ma=,7,9,11,13**,** the altitude is 45.7 *Km*, the static pressure is P_∞ = 130.5Pa, the stagnation pressure is P_{stang}=39,000Pa, the static temperature is T_∞ = 267K, the flow has an angle of incidence of three degree $\alpha = 3$. After importing of the mesh file in to the FLUENT .we are checking the mesh for the accurate solution. The model type solver pressure based with viscous model as Spalart-Allmaras. Ideal-gas law to determine the air density, while Sutherland's law was used to calculate the air viscosity. The operating pressure was set to 0 Pa, to decrease the chance of numerical error due to the low pressures resulting from the solution. For this problem we are giving the boundary conditions from the Pressure-Far-Field. The boundary conditions given to Far-field are Gauge Pressure, and Mach number.

IV. RESULTS AND DISCUSSIONS

Table1 shows the variation of temperature, pressure, velocity and turbulence with Mach number of atmospheric reentry vehicle. It can observed from the obtained results that the maximum temperature of 2460^0 k is obtained at mach number 5 from there it gradually decreases until a minimum temperature of 1890^0 k at mach number 7 and then it increases to 1100^0k at mach number 11 and then gradually decreases to 10800^0 k at mach number 13.

Table1 Variation of temperature, pressure, velocity and turbulence with respect to Mach number of the atmospheric reentry vehicle.

S.No	Mach number	Temperature variation,^{0}K	Pressure variation, Pa	Velocity variation, m/s	Turbulence Variation, m/s²
1	1	1.21e+02	1.77e+03	1.03e+00	3.38e+01
2	3	5.09e+02	2.09e+03	3.02e+00	3.83e+01
3	5	2.46e+03	2.09e+03	5.03e+00	7.98e+01
4	7	1.89e+03	1.43e+03	7.02e+00	9.64e+01
5	9	7.84e+03	4.03e+04	5.88e+01	2.86e+04
6	11	1.10e+04	7.74e+03	1.15e+01	8.124e+02
7	13	1.08e+04	1.74e+05	6.82e+01	3.38e+04

The temperature variation is seen due to the following reason, when air encounters a fast moving in reentry vehicle the air near the leading edge is a most stopped and its kinetic energy is converted into heat energy. After the air gets deflected and flows over the aerofoil there is a variation in temperature because of the heated air flowing on the vehicle and there is a free stream of air above the hot air which exchanges heat with it and causing a temperature difference along the vehicle. It can be observed from the obtained results the maximum pressure of 2090 Pa is observed at Mach 5 from there it gradually decreases until a minimum pressure of 1420 Pa at Mach number 7 and then gradually increases to 40300 Pa at Mach number 9 from there it gradually decreases of 7740 Pa at Mach number 11 and then increases of 174000 Pa at Mach number 13 during the reentry vehicle moving into atmosphere. It can be observed from the obtained results as the maximum velocity of 58.89 m/s is obtained at Mach number 9 then there a gradually decreases of velocity of 11.5 m/s at Mach number and then increases to 68.2 m/sec at Mach number 13.That the maximum turbulence of 28600 m/s² is obtained at Mach number 9 then there a gradually decreases of turbulence of 814 m/s² at Mach number 11 and then increases of turbulence of 33800 m/s². From the figure3, it is observed that the pressure is decreases with increasing the Mach number so the force will reduce on the vehicle. Hence energy required for moving vehicle is less at high mach number. There is a saving of energy and weight on the vehicle also will reduce. The life of vehicle will increase and the power consumption will reduce.

Mach number is directly proportionally to velocity of flow. Velocity increases with increasing the mach number. The vehicle speed will be increased at safest. Efficiency will be high. Less time is needed to move vehicle in the atmosphere. Figure4 shows variation of velocity with respect to Mach number. Its shows the velocity increased at Mach number 1,3,5,7 and 9 but above 9 up to 11 the velocity decreased due to inherent losses, again it increases from Mach number 13.

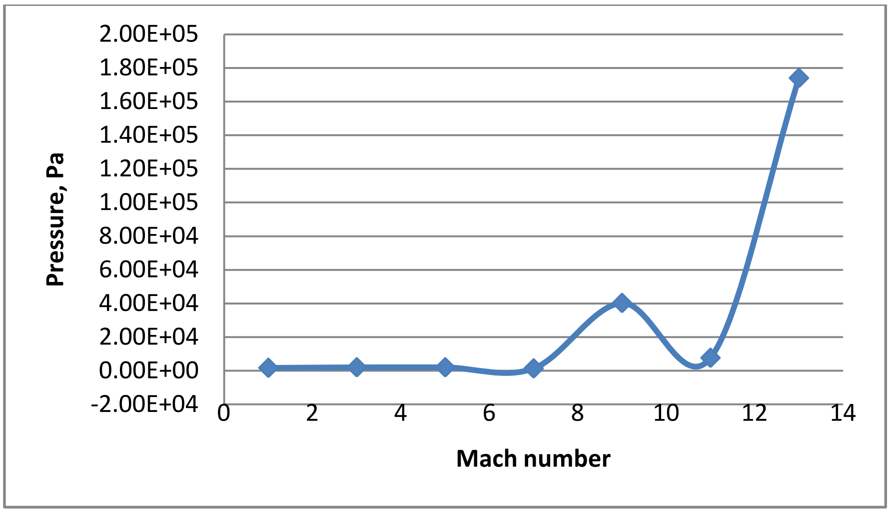


Fig3. Variation of pressure with respect to mach number

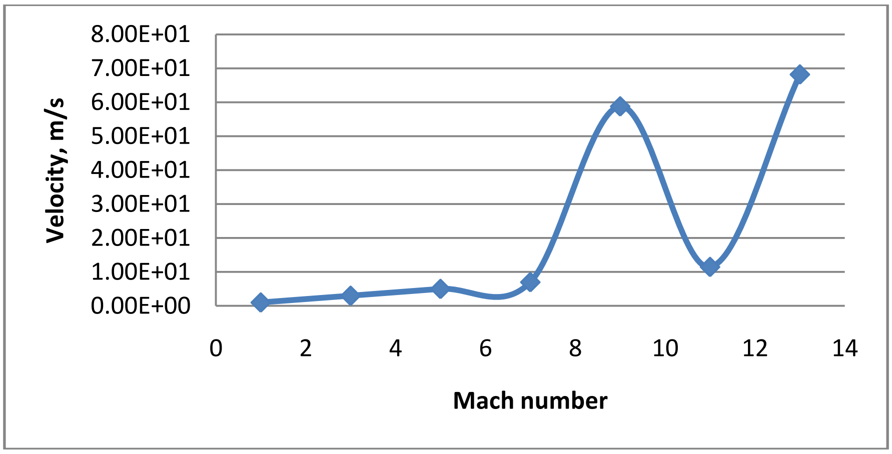


Fig 4. Variation of velocity with respect to mach number

Temperature decreased with increase of Mach number(Figure5). Temp is low at high Mach number hence safe for vehicle. At high Mach number low temp will obtained so corrosion of the vehicle's skin will be reduced. The temperature is reduced from mach numbers 1 to 7 and it's increased from 9 to 13.

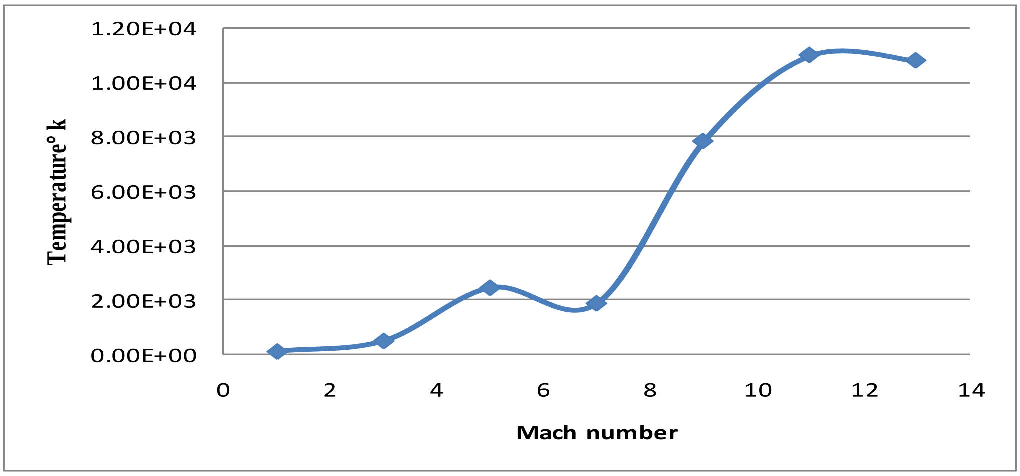


Fig5. Variation of Temperature with respect to mach number

The turbulence is reduced from Mach number 1 to 5 (Figure6) but it raised from 7 to 9 and again falls down at 11 and again it raised at mach number 13 onwards. Turbulence is most effect on the object .There is mainly three effects unstable flow of liquid or gas occurs in the vehicle. Instability in the atmosphere for the vehicle hence its efficiency will affect.

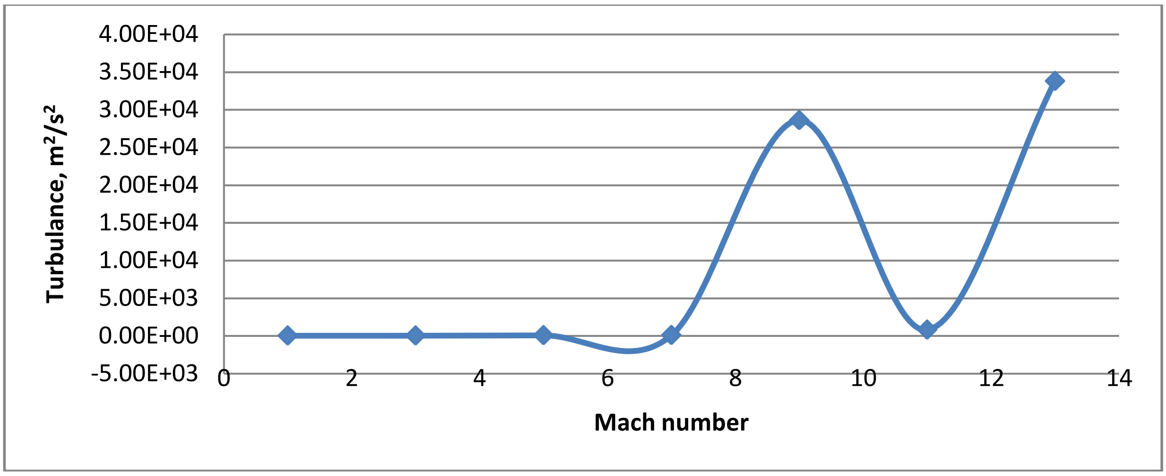


Fig 6. Variation of turbulence with respect to mach number

V. CONCLUSIONS

At high Mach number of vehicle, velocity of flow increased and the pressure of the vehicle decreases drastically. At low Mach number of vehicle, there is an effect on vehicle skin will be corrodes by high temperature due to low Mach number of vehicle used. The reentry vehicle behaves differently at different mach numbers. At Mach numbers 5 and7 the temperatures on the vehicle increases from 2460 ^{0}K and 7840 ^{0}K respectively. At mach numbers 5,9 and 13 the pressure on the vehicle increases from 2090 Pa, 40300 Pa and174000 Pa. At mach number 11 the Velocity of medium is increasing from 58.8 to 68.2 m/s. At Mach numbers 9 and 13 the turbulences of vehicle increasing from 28600 m/s² to 33800 m/s².

REFERENCES

[1]. Baiocco P., Guedron S., Plotard P., Moulin J., “The Pre-X Atmospheric Re-entry Experimental Lifting Body: Program Status and System Synthesis”, 57th IAC Congress, Valencia, Spain, 2-6 October 2006.
[2]. Tumino G., Gerard Y., “Europe Among the World Players in Atmospheric Reentry”, ESA bulletin 128, November 2006.
[3]. Oswald J. et al., “DLR-ONERA accurate CFD support to the Pre-X project”, 6th International Symposium on Launchers Technologies, Munich, Germany, 8-11 November 2005.
[4]. Arini M., “FESTIP Technology Developments in Aerothermodynamics for Reusable Launch Vehicles – Body-Flap Efficiency Studies for a Simplified Two-Dimensional FSSC-15-OAE Concept Vehicle Configuration”, Final Technical Report CIRA-CR-AEP-99-206, December 1999.
[5]. Marini M., “Body-Flap Efficiency Prediction of a FESTIP Concept Vehicle”, Second International Symposium on Atmospheric Reentry Vehicles and Systems, Arcachon, France, March 26-29, 2001.
[6]. Roncioni P., Ranuzzi G., Marini M., “FLPP-IXV Project – Phase B1.2 – Preliminary CFD Activities Synthesis Report”, CIRA Internal Report, CIRA-CF-07-0223, March 2007.
[7]. Ranuzzi G., Borreca S., “CLAE Project. H3NS: Code Development Verification and Validation”, CIRA internal report. CIRA-CF-06-1017, September 2006.

[8]. Dumbacher D., "NASA's Second Generation Reusable Launch Vehicle Program Introduction, Status and Future Plans". 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Indianapolis, Indiana, July 7-10, 2002. AIAA-2002-361 3.

[9]. Whitmore S., Dunbar B., *"Orbital Space Plane, Past, Present, and Future"*. AIAA International Air and Space Symposium: The Next 100 Years, Dayton, Ohio, July 14-17, 2003. AIAA-2003-2718

[10]. Rasky D., "Access from Space: A New Perspective on NASA's Space Transportation Technology Requirements and Opportunities". Space 2004 Conference, San Diego, California, Sep. 28-30, 2004. AIAA-2004-61 03.

[11]. Reuther, J., Kinney, D., Smith, S., Kontinos, D., Gage, P., Saunders, D., *"A Reusable Space Vehicle Design Study Exploring Sharp Leading Edges"*. AIAA Thermophysics Conference, 35th, Anaheim, CA, June 11-14, 2001. AIAA-2001 -2884.

[12]. Saunders, D., Allen, G. Jr., Gage, P., Reuther, J., "Crew Transfer Vehicle Trajectory Optimization".35th AIAA Thermophysics Conference, Anaheim, CA, June 11-14, 2001. AIAA-2001-2885.

[13]. The site for the Mars Reference Mission: http://www_sn.j sc.nasa.gov/marsref/contents.html.

[14]. G. Polishchuka, K. Pichkhadzea, V. Vorontsovb, K. Pavela, "Proposal on application of Russian technical facilities for International Mars Research Program for 2009–2015", Acta Astronautica, Vol. 59, pp.113-118. 2006.

[15]. A. Viviani and G. Pezzella, "Aerodynamic Analysis of a Capsule Vehicle for a Manned Exploration Mission to Mars". 16th AIAA/DLR/DGLR International Space Planes and Hypersonic Systems and Technologies Conference. Bremen Oct. 2009. AIAA-2009-7386.

[16]. G. M., Hanley, F. J. Lyon, "The Feasibility of Spacecraft Deceleration by Aerodynamic Braking at the Planet Mars", Proc. of the 1th AIAA Annual Meeting . Washington, D.C. June 29- July 2, 1964. AIAA-64-479.

[17]. Ivanov M.S., Rogazinskii S.V. Method of direct statistical simulation in rarefied gas dynamics. Novosibirsk: Publ. Comput. Center, Sib. Branch, USSR Acad. Sci., 1988.

[18]. Ivanov M.S., Rogazinskii S.V. Comparative analysis of efficiency of numerical schemes of the DSMC method in rarefied gas dynamics: Preprint No.19-87 / Inst. Theor. Appl. Mech., Sib. Branch, USSR Acad. Sci. Novosibirsk, 1987.

[19]. Borgnakke C., Larsen P.S. Statistical collision model for Monte Carlo simulation of polyatomic gas mixture // J. Comput. Phys. 1975. Vol. 18. P. 405-420.

[20]. Ivanov M.S., Markelov G.N., Gimelshein S.F., Statistical simulation of reactive rarefied flows: numerical approach and applications. AIAA Paper 98-2669.

[21]. A. Schettino, R. Votta, et al. *Proc. of the West-East High Speed Flow Field Conference*, Moscow, 2007

[22]. M. Ivanov, P.Vashchenkov, A. Kashkowsky Numerical Investigation of the EXPERT Reentry Vehicle Aerothermodynamics Along the Descent Trajectory, 39th AIAA thermophysics Conference, Miami, June 2007.