MATHEMATICAL MODELING OF TURBULENT SUPERSONIC FLOWS IN INLETS WITH ROTATING COWL

Bedarev I.A. *, Fedorova N.N. *, Goldfeld M.A. *, Falempin F.**
Institute of Theoretical and Applied Mechanics, Institutskay 4/1, 630090, Novosibirsk, Russia
** *MBDA, Chatillon, France*

Summary Methods of physical experiment and mathematical modeling have been used to study the properties of flows in adjusted inlets designed to operate in a wide range of Mach number. The variation of inlet geometry was performed by cowl rotation**.** The experimental investigations have been carried out in the blow-down wind tunnel at Mach numbers from 2 to 6 and in the hot-shot wind tunnel at Mach numbers from 5 to 8. The computations were performed on the basis of the full Navier-Stokes equations and the two-equation turbulence model by Wilcox. The experimental pressure distributions along the inlet walls were used for the verification of simulation results. Comparison have shown a good agreement. Computations within the wide range of flow and geometric parameters help to carry out the experiments and provide a basis for the choice of optimum configurations and explanation of flow features.

The computer design tools are widely used in developing the optimum geometry of the engines. Automatic designing performed in the Center for Computation of Rutgers University (USA) in cooperation with Aerospatiale Matra Missiles (France) and United Technologies Research Center (USA) permits one to get the hypersonic aircraft inlets shapes which must guarantee inlet start and effective operation within the wide range of Mach numbers [1]. The main parameters used to evaluate the efficiency of geometrical configuration were the coefficient of total pressure losses and air flow rate. The experiments were carried out at different Mach numbers [2]. In the paper, the results of the numerical simulations of the 2-D flows at M=2-8 performed on the basis of Navier-Stokes equation are presented. The computation results are compared to the experiments carried out in ITAM (Novosibirsk).

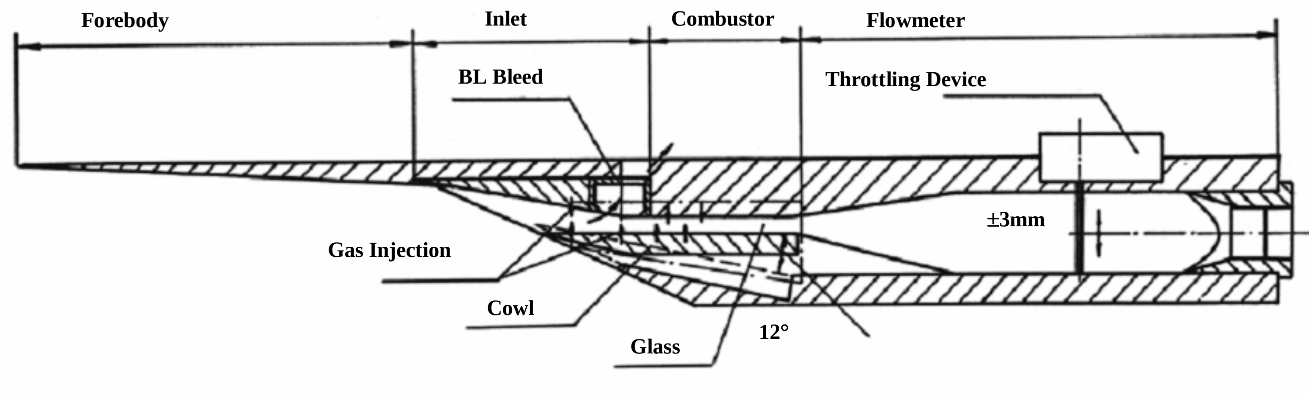

Fig. 1. Model scheme

The experimental model presents a channel with flat walls and 3D forebody (Fig.1) The quasi-two-dimension configuration of the forebody allows one to simulate a boundary layer developed on the aircraft nose part and guarantees preliminary flow deceleration in two shock waves. The variation of inlet geometry was performed by cowl rotation [2]. Several model configurations with different cowl inclination angle have been considered (Fig. 2) for the corresponding Mach numbers to provide the inlet "start". Numbers 1 to 7 in Fig.2 denote the cowl positions relevant to the angles $\theta = 12^{\circ}$, 10°, 5.5°, 2.5°, 2°, 0° and -1° that were chosen as basic configuration for the flows at M=2-8.

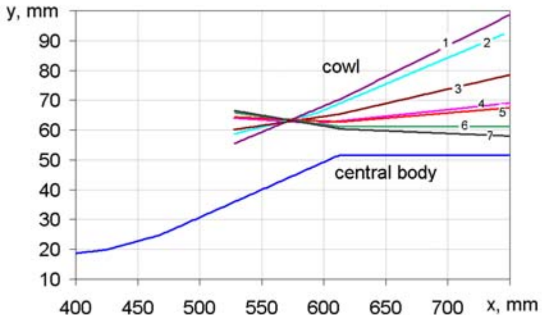

Fig. 2. Inlet configurations

The experimental investigations have been carried out in the ITAM blow-down wind tunnel at Mach numbers from 2 to 6 and in the hot-shot wind tunnel at Mach numbers from 5 to 8. The bottom and top walls of the model were equipped with static pressure taps to measure the distribution of static pressure along the longitudinal axis. Additionally, in three cross-sections the installation of Pitot pressure probes has been provided to determine the local Mach numbers and total pressure losses in the channel. The side walls of model was equipped with transparent windows to provide Schlieren visualization of the internal flow structure.

The calculations have been performed on the basis of the full transient Favre-averaged Navier-Stokes equations and Wilcox two-equation turbulence model. The original numerical algorithm [3] used earlier for the 2-D turbulent supersonic flow was extended to a new class of internal flows.

In inlets, many types of Shock Wave /Boundary Layer Interactions (SWBLI) take place, making the flowfield picture rather complex. To ensure the numerical simulation gives the reliable results for various SWBLI, the verification of the turbulence model and computation methods have been carried out for forward- and backward-facing steps [4], impinging shock wave [5],cylinder-flare configuration [6] etc.

The computation were performed for various inlet configurations. Fig. 3 presents the static pressure and skin friction distributions along the central body obtained in computations carried out under the conditions of blow-down facility. The given data indicate a big lengthwise pressure non-uniformity and high pressure loads that increase together with Mach number. At M≥4 a saw-toothed structure appears in the pressure distribution related to the alternation of shock waves and rarefaction waves inside the channel. The increment of shock wave intensity with Mach number rise results in the formation of an extensive separation region on the channel bottom wall. This region covers the channel section and causes the significant pressure rise. The presence of the boundary layer separation region on the channel bottom wall at M≥4 is testified by the skin friction distribution.

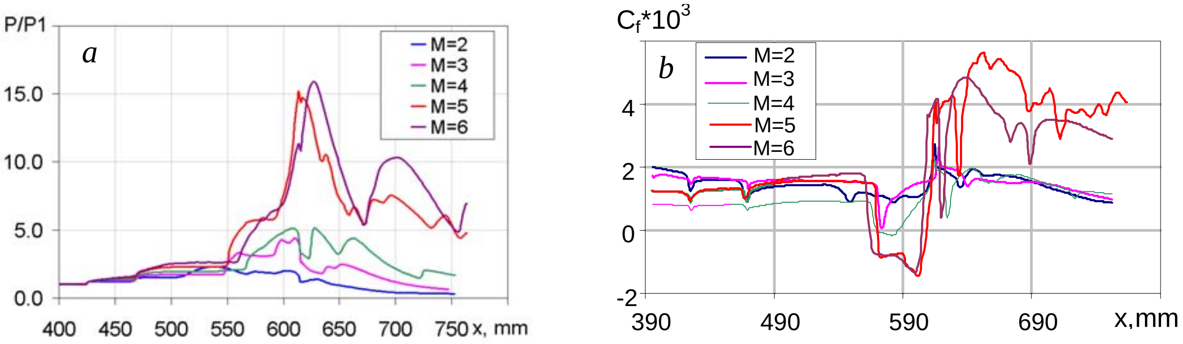

Fig. 3. Computed pressure and skin friction distributions along central body

The experimental and computed pressure recovery coefficient ν for inlets of various geometry are shown in Fig. together with USA standard curve MIL-500. The double and triple points at the same Mach number show the results for different experimental runs and inlet configuration studied This data demonstrates that the experiments underpredict ν values whereas the computations get the values over the standard curve. The possible reason of disagreement between computed and experimental . ν values that are the laminar-turbulence transition and temperature factor influence, 3-D effects etc, are discussed.

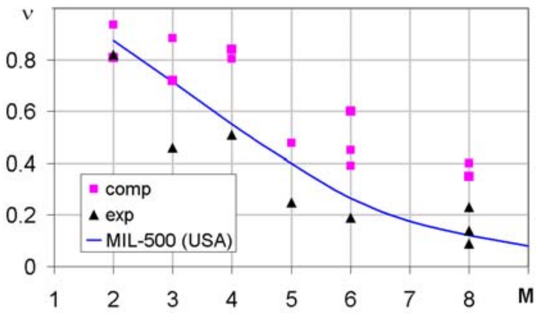

Fig. 4. The experimental and computed pressure recovery coefficients

Thus, the methods of physical experiment and mathematical modeling have been used to study the properties of complex flows in the inlets in a wide range of Mach number. The experimental findings present a basis for the mathematical model and calculation algorithm verification. At the same time, the computations performed within the wide range of flow and geometric parameters help one to carry out the experiments and provide a basis for the choice of optimum configurations and explanation of flow features.

The investigations has been supported by MBDA (France) and European Commission within the frame of ISTC project #887.

References

[1] Bourdeau C., Blaize M. and Knight D. Performance Analysis for an Automated Optimal Design of High Speed Missile Inlets. *AIAA Paper 99-0611*,1999.

[2] Falempin F., Goldfeld M., Nestoulia R., Starov A. Evaluation of a Variable Geometry Inlet for Dual Mode Ramjet *AIAA Paper No 2002-5232*, 8 pp.

[3] Borisov A.V., Fedorova N.N. Numerical simulation of turbulent flows near the forward-facing steps *Thermophisics and Aeromechanics* (1996). Vol. 4, No.1. C. 69-83.

[4] Bedarev I.A., Borisov A.V., Fedorova N.N. Numerical Simulation of the Supersonic Turbulent Separated Flows in Vicinity of the Backward–and Forward– Faced Steps *Computational Fluid Dynamics Journal* (2001) Special Number. P. 194 – 202.

[5] Fedorova N.N., Fedorchenko I.A., Shuelein E. Experimental and Numerical Study of Oblique Shock Wave / Turbulent Boundary Layer Interaction at M=5 *Computational Fluid Dynamics Journal* (2001) Vol.10, No.3. P. 376-381.

[6] Bedarev I.A., Fedorova N.N. Numerical Simulation of Axisymmetric Super- and Hypersonic Separated Flows in Vicinity of Cylinder-Flare Configuration // First International Conference on Computational Fluid Dynamics, July 9-July 14, 2000, Kyoto Research Park, Kyoto, Japan, p. 343-344.