Optimization of the Supersonic engine inlet

Koval Svetlana
Moscow Aviation Institute (National Research University)
Moscow, Russia
(koval-svetlana@ro.ru)
June, 2019

The most important issue in the design of a perspective aircraft is the development of a highly efficient power plant. One of the factors affecting its efficiency is the choice of input device. Thus, increasing the requirements for the characteristics of the power plant of the aircraft leads to the need to develop spatial air intakes or air intakes of complex shape.

According to [1], the choice of input device depends largely on the estimated flight number M of the aircraft, the required range of deviations of the numbers M from the calculated one, the location of the power plant on the aircraft, the type of engines used and a number of other factors.

Depending on the estimated flight speed, the input devices can be divided into three types: subsonic, supersonic.

Keywords: *input device, shock wave, optimization.*

Introduction

The engine inlets are designed to slow down the flow of air entering the engine. The main requirements [2] imposed on input devices are:

- ensuring high values of the total pressure retention coefficient;
- creating a uniform flow at the engine inlet or the desired (permissible) irregularity;
- minimal aerodynamic drag;
- minimum dimensions and weight;
- ensuring stable and efficient operation in the entire required range of flight modes and engine operating modes.

Modern aircraft input devices are characterized by the presence of a large number of controls to ensure optimal operation, as well as strong integration with the body and systems of the aircraft.

Modern software systems make it possible to reliably calculate the three-dimensional flow around an air intake in a stationary setting, considering viscous effects and with a duct through the internal channels of air intakes and engine cavities.

The main objective of this work was to calculate and optimize the shock wave system in a supersonic input device. By optimal we mean a system that has the maximum possible value of the total pressure retention coefficient [3]. The position of the first shock wave due to the shift of the air intake cone in axial movement is considered as an optimization parameter.

As a result of the calculation, it is possible to optimize the flow part of the air intake, which has the optimum position of the jump system, or specify the geometry of the flow part of the inlet device that will best meet the requirements.

I.Air intake device of an engine

1.1 Purpose and classification of air intakes

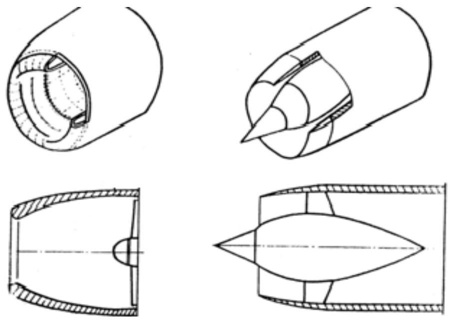
Air intake devices are designed to take air from the surrounding atmosphere, supply it to the engine with the least losses and the process of compressing this air from the high-speed head.

Input devices used on various aircraft differ in a wide variety of types and structural forms. The range of flight speeds of the aircraft and the requirements for its maneuverable properties have the greatest impact on the appearance of the input device. Accordingly, the airplane air intake device is divided into:

Subsonic - M numbers of cruising flight not exceeding 0.8...0.9. Installed on civil aviation aircraft, helicopters. In the power plants of these aircraft, air compression is carried out mainly by the compressor, and the pressure increase from the high-speed head is small.

Transonic - large subsonic cruising and relatively small supersonic maximum flight speeds (M < 1.5...1.7). The pressure increase due to the high-speed head in such input devices is more significant. Air intake input devices of these aircraft are simple in design, as they are usually performed unregulated.

Supersonic - installed on aircraft with high values of the maximum number of M flight (usually at M > 2.0). They are usually performed adjustable [4].



a) b)

Figure 1.1 Scheme of the air intake device: a – subsonic;
b – supersonic

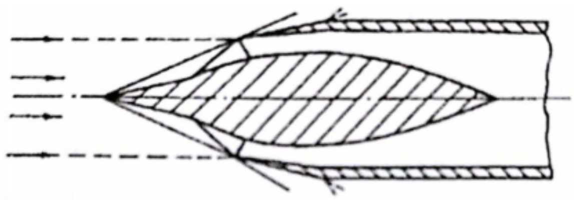
The numerical calculation of the gas-dynamic characteristics of the air intake supersonic air intakes is characterized by a wide variety of types and schemes. They are classified according to the signs:

1. The number of shocks – single- shock multishock intake:

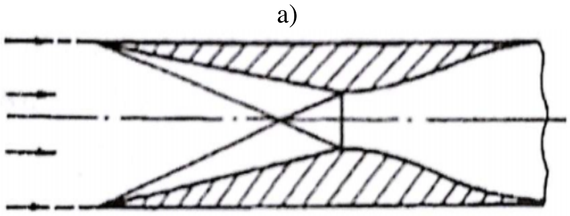
The braking of supersonic flow to subsonic speed occurs in shock waves. The simplest case – a single straight shock wave. But it cannot provide effective air compression due to the increase in its intensity and loss of full pressure at high flight speeds. To reduce the intensity of the direct jump, the airflow in front of it is pre-braked in several oblique shock waves of low intensity.

To create a system of shock waves, a special profiled surface, called the braking surface, is used. Its forming is a broken line with one or another number of fractures. When this surface is flowed around by a supersonic flow, oblique shock waves are formed at its fractures, in which the supersonic flow is pre-compressed before the closing direct jump.

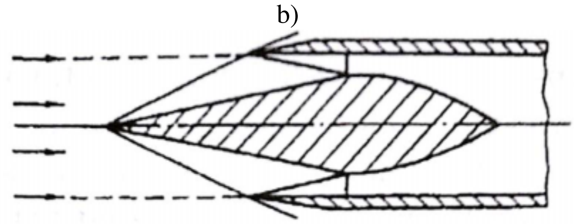
2. By the location of the shock waves relative to the entry plane:



a)



b)



c)

Figure 1.2 Types of supersonic diffusers:
a - external- compression; b – internal – compression; c – external – internal compression

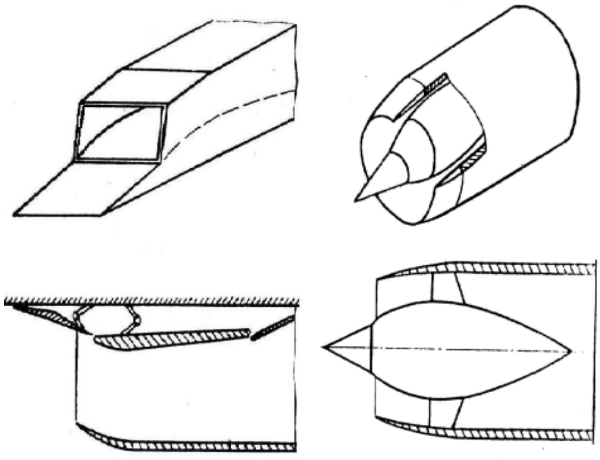
At the supersonic input external compression device, all the shock waves formed during the flow around the braking surface are located in front of the input plane of the input device. The area of the smallest section of the internal channel ("throat") is close to the plane of the entrance;

In the supersonic input internal compression device, all oblique shock waves are located behind the input plane, and compression is carried out inside the channel.

In the supersonic input device of mixed compression, one part of the oblique jumps is placed in front of the input plane, and the other part – in the internal channel. In this case, the inner channel from the input plane to the "throat" has a significant narrowing, and the minimum cross-section of the channel, called "throat", is located at some distance from the input plane;

Currently, supersonic external compression input devices are used in aviation. the supersonic input device of mixed and especially internal compression can in principle provide a more efficient process of compression of the supersonic flow at high m flight numbers, but there are some difficulties in their practical use.

3. According to the shape of the braking surface and the inlet section, there are flat and axisymmetric supersonic air intake devices:



a) b)

Figure 1.3 air intake devices:
a – two – dimensional; b – axisymmetric

4. The arrangement of the aircraft intake devices are divided into frontal, located in the forward fuselage or engine nacelles, and adjacent, installed near any surface area of the aircraft.

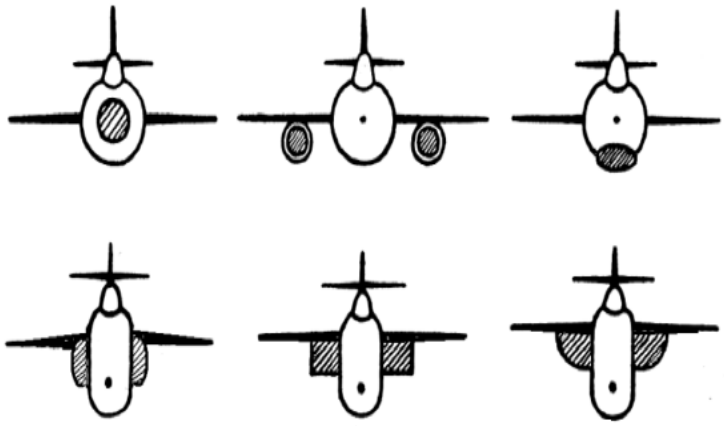


Figure 1.4 The location of the air intakes on the plane:
a – frontal; b – underwing; c – side

The disadvantage of adjacent supersonic air intake devices (as compared with frontal ones) is that they have a higher degree of irregularity and unsteady flow in the output section. This is explained both by the presence of non-uniformity of the flow at the inlet and by the length of the air supply channels, which is small according to the conditions of the layout, which does not provide sufficient leveling of the flow parameters.

Frontal axisymmetric supersonic air intake devices were widely used on second-generation supersonic aircraft. When placed in the nose of the fuselage, these supersonic air intakes at low angles of attack due to uniform flow and axial symmetry of the flow provide good performance in terms of losses, mass, and flow structure at the engine inlet. However, with increasing angles of attack, their characteristics deteriorate sharply, especially at supersonic flight speeds.

Supersonic air intake devices adjacent to the surface of the aircraft, as a rule, are flat and are rarely made semicircular or oval. They began to be used on highly mobile aircraft of the third and fourth generations to improve the performance of power plants at high angles of attack and slip, reduce the length and mass of air supply channels to the engine and use positive interference of supersonic air intakes and aircraft.

The wing, ventral, dorsal and lateral (relative to the fuselage) inlet devices of the adjacent type are known.

The main problem of choosing the location of supersonic air intakes near the surface of the aircraft is to ensure a small change in the local angles of attack and slip in the area of the supersonic air intakes compared to changes in the angle of attack and slip of the aircraft itself. This is achieved by directing the influence of the surfaces to which the supersonic air intake device is adjacent [5].

II.Numerical simulation of the operation of an axisymmetric three - shock air intake of external compression type

To carry out a numerical study, a model of an axisymmetric three-shock air intake device of the external compression type was chosen, the general view of this model is presented in (fig. 2.1).

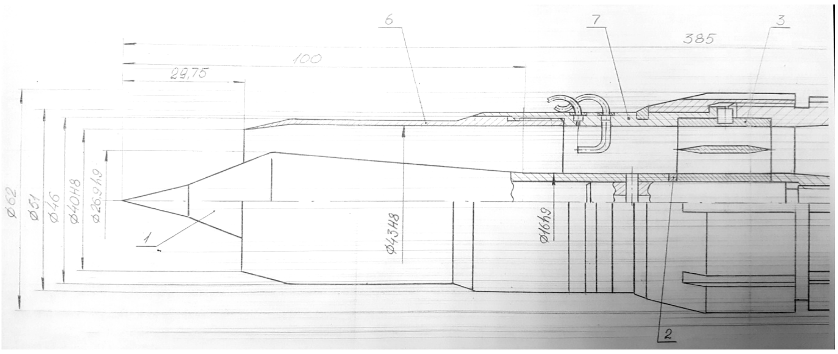


Figure 2.1 General view of the input device

According to the available experimental model drawings, a geometric model of the input device was presented (fig. 2.2). It can be seen that the calculated geometry of the cone air intake device and its channel are fully consistent with the experimental model.

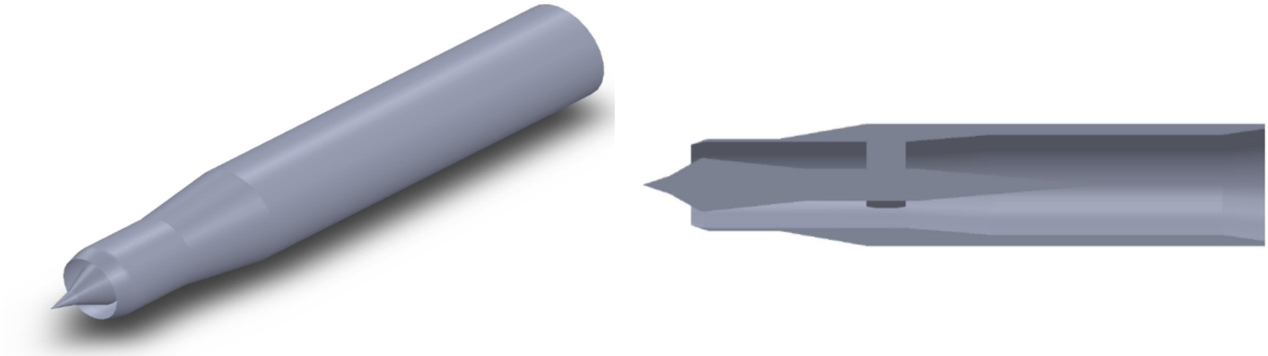


Figure 2.2 Geometric model of the considered intake device

2.1 Mathematical model of the air intake device

The Navier-Stokes Equations are the basic governing equations for a viscous, heat conducting fluid. It is a vector equation obtained by applying Second Newton's Law of Motion to a fluid element and is also called the momentum equation. It is supplemented by the mass conservation equation, also called continuity equation and the energy equation. Usually, the term Navier-Stokes equations is used to refer to all of these equations.

Navier-Stokes Equations are the governing equations of Computational Fluid Dynamics (CFD). Computational Fluid Dynamics is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.).

To solve this problem, we used a numerical method that allows one to calculate spatial viscous three-dimensional turbulent gas flows in the time-based process. The system of Reynolds-averaged Navier-Stokes equations describing the spatial flow of a viscous compressible gas in the cartesian coordinate system is as follows:

$$\partial(\rho u_i)/\partial t + \partial(\rho u_i u_j - \tau_{ij})/\partial x_j + \partial p/\partial x_i = 0 \quad \partial(\rho e)/\partial t + \partial(\rho u_i H - u_j \tau_{ij} - q_i)/\partial x_i = 0,$$
$$\partial\rho/\partial t + \partial(\rho u_j)/\partial x_j = 0$$

где: i, j=1,2,3.

The stress tensor τ and the heat flux vector q are related to the parameters of the averaged flow using the relations:

$$\tau_{ij} = \tau_{lij} - \rho\overline{u_i u_j} = (\nu + \nu_t)[(\partial u_i/\partial x_j + \partial u_j/\partial x_i) - \tfrac{2}{3}\delta_{ij}\,\partial u_k/\partial x_k]$$
$$q_i = -C_p(\nu/\mathrm{Pr} + \nu_t/\mathrm{Pr}_t)\partial T/\partial x_i,$$

где: $\nu = \mu/\rho$ – laminar kinematic viscosity; μ – molecular viscosity determined by Sutherland law; $\nu_t = \mu_t/\rho$ – turbulent kinematic viscosity.

In the above equations p, ρ, T, u_i – pressure, density, temperature, and components of the velocity vector U in the cartesian coordinate system, respectively; $e = \varepsilon + 0.5\, u_i u_i$ – total specific energy and $H = e + p/\rho$ – total specific enthalpy. The specific internal energy ε is defined as $\varepsilon = p/(æ-1)/\rho$, где æ – the ratio of specific heat, C_p – specific heat at constant pressure Pr и Pr_t – ordinary and turbulent Prandtl numbers, equal respectively 0.72 и 0.90.

For viscous flows on the walls, the following conditions should be specified:

$$q_w = 0,\ (\nu_t)_w = 0,$$

where q_w, $(\nu_t)_w$ – the velocity vector and turbulent viscosity on the wall and, in addition to the above conditions, at the boundaries of the computational domain, the normal derivatives of the flow parameters and the set of values (at the boundaries through which gas flows into the computational domain) must vanish.

As initial, for all types of flow, it is necessary to set flow parameters in the entire computational domain. As a basic assumption, it was considered that the calculation was carried out under the assumption of heat-insulated walls for a three-dimensional stationary formulation.

Formulation of the problem. Calculated grid. Boundary conditions

The calculations were carried out using an unstructured computational grid, the construction of which was carried out in ANSYS CFX, with adaptation at the entrance to the device. The general view of the computational grid is presented in (fig. 2.3), in (fig.2.4) the computational grid in the region of the central body is shown. The number of calculated cells - 3764428 pcs.

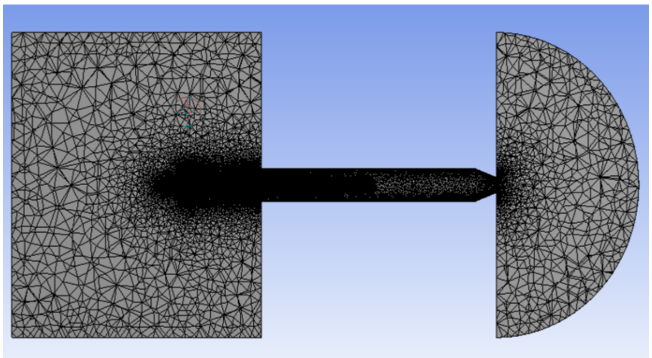


Figure 2.3 Calculated Grid

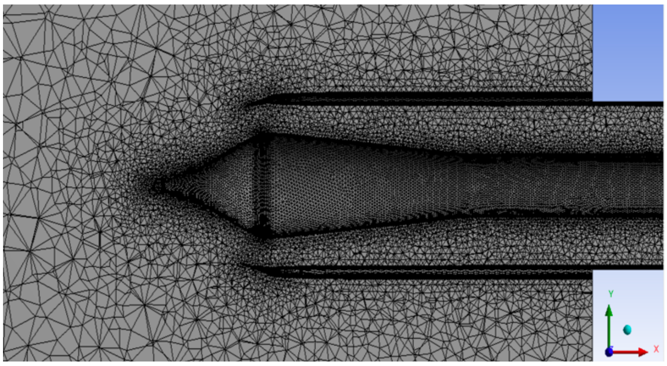


Figure 2.4 Calculated grid in the area of the central body

Selection of turbulence model

The main models of turbulence for calculating the flow in the air intake: k - ε, k ω, and SST.

The equations k - ε of turbulence model fairly accurately describe the flow of liquids and gases in the main part of the flow. But at the same time, the k - ε model encounters difficulties in describing boundary layers with a pressure gradient, strongly swirling flows with the curvature of streamlines, laminar-turbulent transition, three-dimensional boundary layer flows, as well as in modeling compressible and separated flows. Despite these limitations, the propagation of the k - ε model in practice is explained by a stable iteration process, resistance to errors in specifying input data, and reasonable accuracy for a wide class of turbulent flows.

In the k - ω turbulence model, two parameters are used: the turbulent kinetic energy k and the specific energy dissipation rate ω. In this model, to improve the description of the near-wall flows, instead of the equation of the dissipation rate, an equation is used that characterizes the dissipation rate per unit of kinetic energy. In general, this model of turbulence provides adequate results when calculating pre-discontinuous flows, and also finds application in solving problems with boundary heat transfer. But greater sensitivity to boundary conditions can lead to significant errors in the calculation results.

The k - ω and k - ε turbulence models do not accurately predict the parameters of the boundary layer near a curved surface under the action of a pressure gradient. The reason for this is that these models ignore the effect of shear stress transfer.

The SST turbulence model combines the main advantages of the k - ε and k - ω turbulence models: near the wall, the k - ω model is used that resolves small-scale turbulence, and k - ε is used in the free shear stress region and the flow core to describ e large-scale structures. The transition between them is provided with the help of weight functions for model coefficients. The main advantage of the SST turbulence model is that the use of the k - ε model in the main flow and the k - ω models near the wall makes it possible to abandon complex near-wall functions, which makes the model more universal in terms of calculatin g currents of various types and characteristics. On the other hand, the SST model of turbulence has drawbacks caused by the s eparation of the calculation area between the k - ε and k - ω model, leading to a failure in turbulent viscosity at the point of tra nsition from one model to another.

The SST turbulence model was chosen as the main one, since, in general, it is superior in quality to the k - ε and k - ω mo dels [6-7].

Assignment of boundary conditions for calculation

To provide the necessary input data for the calculation of the aerodynamic characteristics of the air intake, it is necessary to assign boundary conditions to all surfaces of the calculated volume, to fully simulate the most accurate gas flow. The following boundary conditions are specified in the calculation:

1. At the entrance to the settlement area (inlet) - flow velocity (with the corresponding value of M), static pressure (P=101325 Pa) and static temperature (T=288 K), corresponding to flying at altitude H=0 км (fig. 2.5);
2. On all other surfaces (excluding the walls of the air intake) (opening) - static pressure (P = 101325 Pa) and temperature (T = 288 K), corresponding to a given flight height H = 0 km (fig. 2.6).

Also, when solving the problem in the calculation model, the following options are set:

– all air intake surfaces are defined as surfaces of no-slip wall, however, the roughness affecting the boundary layer parameters was not taken into account;
– set the value of the parameter *Reference Pressure* = P_h, to reduce rounding errors when the dynamic pressure changes in the computational domain are small compared with the absolute pressure value;
– as the working substance was chosen - air as *Air Ideal Gas*;
– Heat transfer calculation option - total energy;

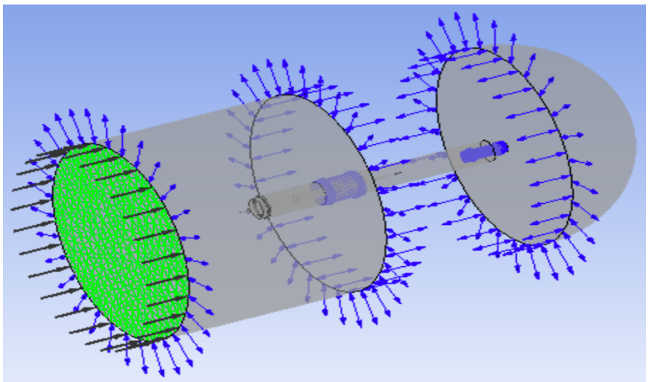


Figure 2.5 Boundary conditions for solving the Inlet problem

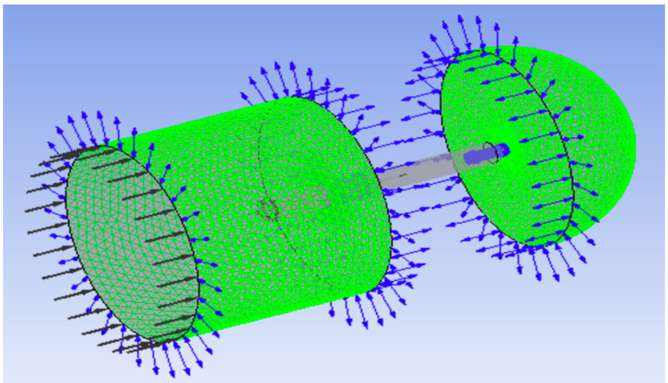


Figure 2.6 Boundary conditions for solving the Opening problem

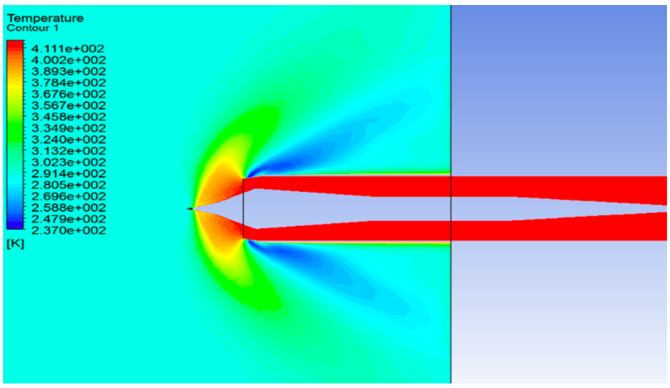
2.2 Results of numerical simulation

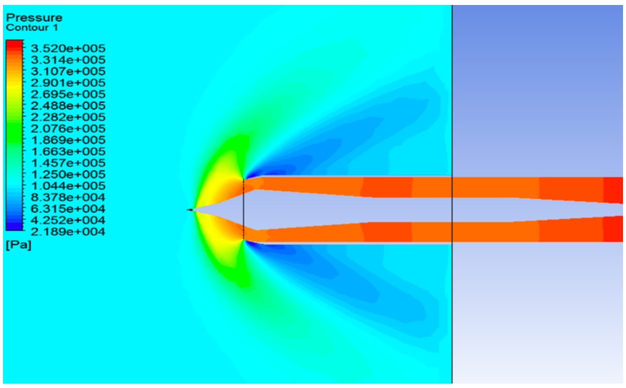
The calculation was performed in the ANSYS CFX environment in automatic mode using the ANSYS WORKBENCH shell. Some numerical simulation results are presented below. The following are the Mach number distribution fields and the pressure fields for various flight speeds from 1.5 M to 2.5 M (fig. 2.7-2.9).

The (fig. 2.7) shows the fields of static, total pressure, static temperature and Mach number of the air intake device at mode M = 1.5. It can be seen that the stabilization of the flow occurs on a knocked-out shock wave.

The (fig.2.8) shows the fields of static, total pressure, static temperature and the Mach number of the air intake device in the M = 2 modes. It is seen that this mode of operation of the air intake device is calculated because the system jumps came to the side. It should also be noted that to the measuring section, the flow in the channel of the intake device is stabilized.

The (fig.2.9) shows the fields of static, total pressure, static temperature and Mach number of the air intake device at mode M = 2.5. It can be seen that the stabilization of the flow takes place not on the sidewall, but a knocked out a shock wave. As can be seen from the obtained results, the static pressure field in the dimensional section will have an uneven profile, which can lead to a difference in the numerical simulation results from the experiment, since in the experiment, local measurement of parameters in the cross-section is used (nozzles of full and static pressures are established), while the averaging over the cross-sectional area is used in the calculation.





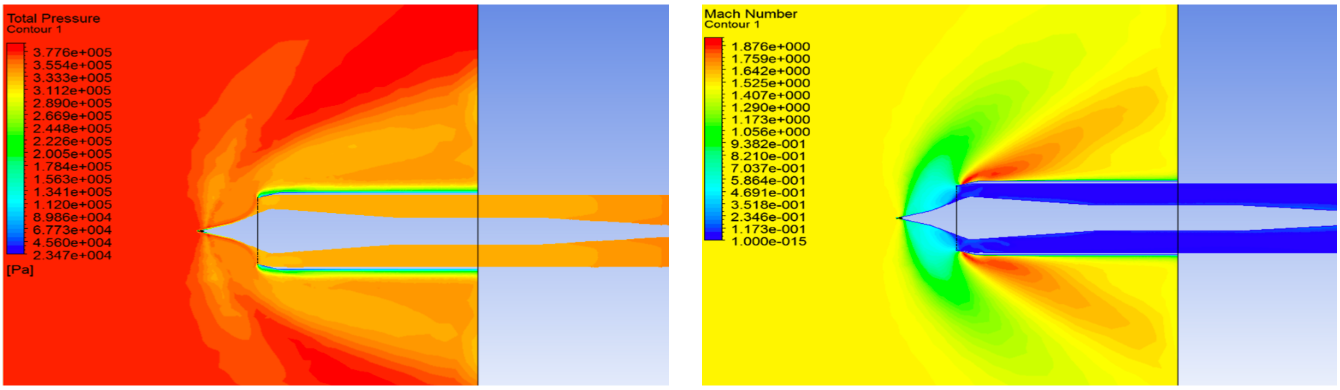


Figure 2.7 fields of static pressure, total pressure, static temperature and the Mach number
of the air intake device at mode M = 1.5

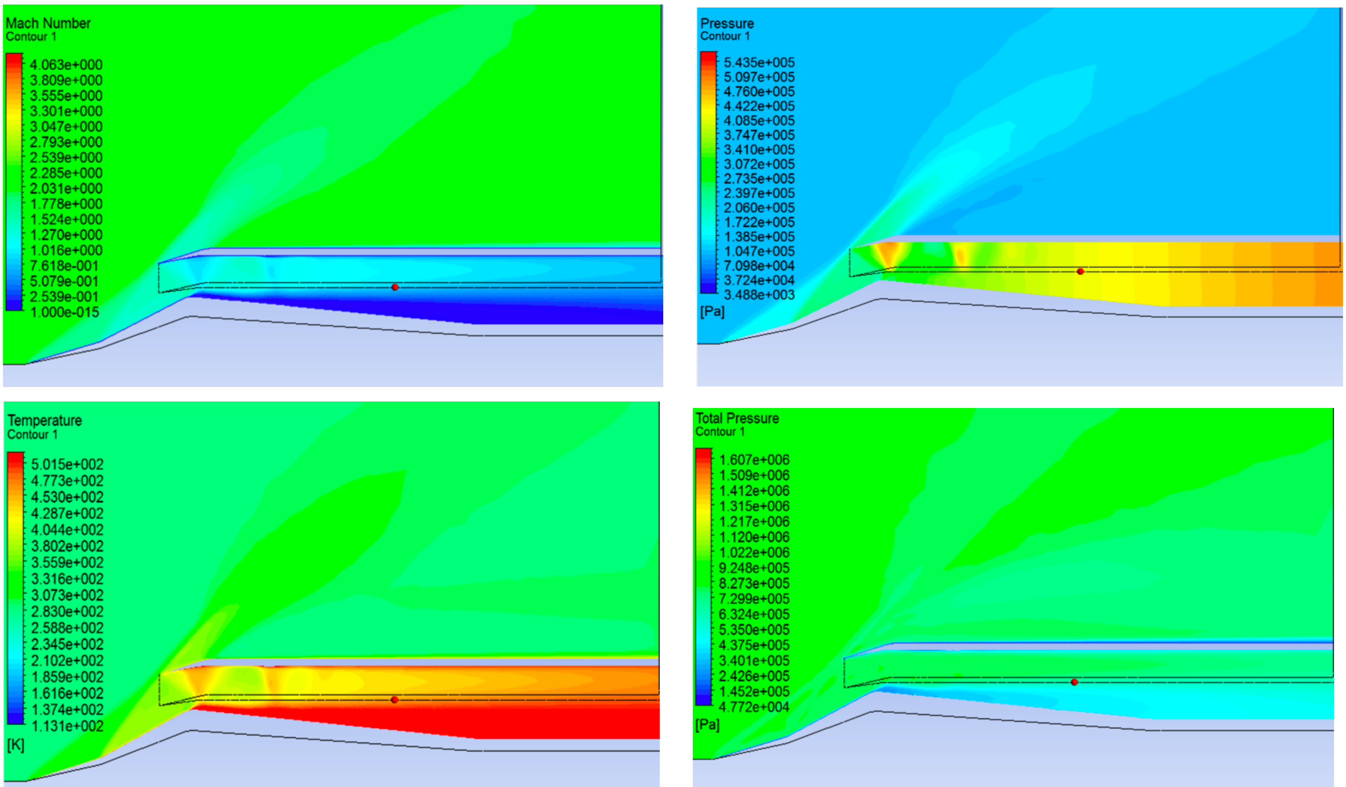
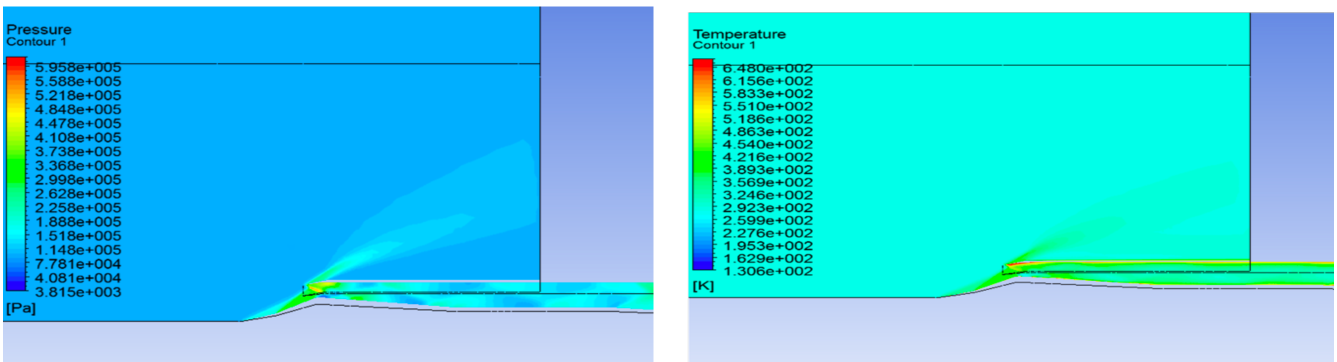


Figure 2.8 fields of static pressure, total pressure, static temperature and the Mach number
of the air intake device at mode M = 2



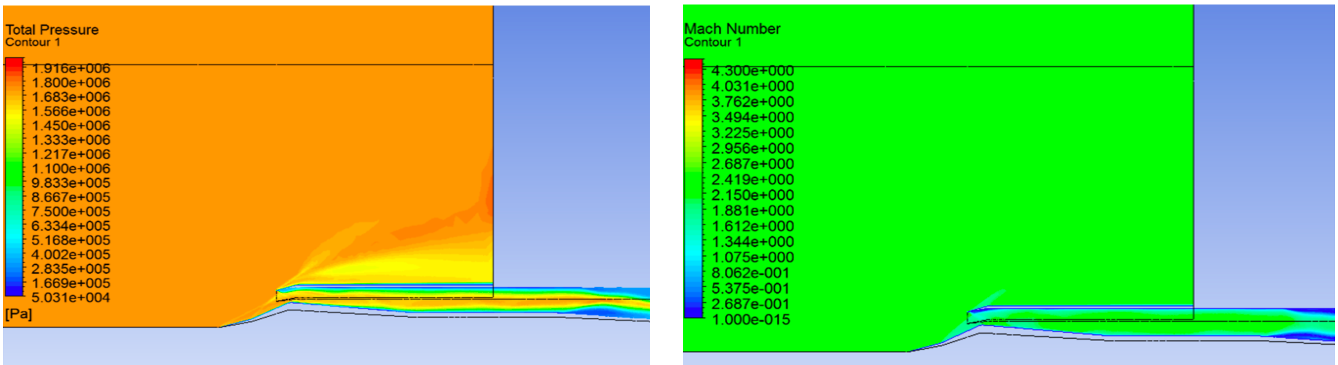


Figure 2.9 fields of static pressure, total pressure, static temperature and the Mach number of the air intake device at mode M = 2.5

2.3 Numerical simulation analysis

Comparison of the flow visualization results in the experiment with the results of numerical simulation (fig. 2.10 - 2.11) shows that the flow pattern outside the air intake device in the calculation corresponds to the picture obtained in the experiment, i.e. in the experimental modes M = 2.23 and M = 2.62, a picture is observed analogous to numerical calculation.

The results of the numerical simulation were processed and the dependencies similar to the experimental ones were obtained. For a more correct comparison, the resulting graphs are presented below.

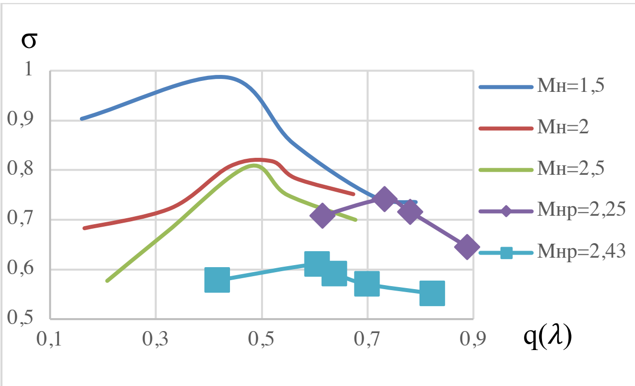


Fig. 2.10 Dependence of the loss coefficient of the total pressure σ on the gas-dynamic function q (λ)

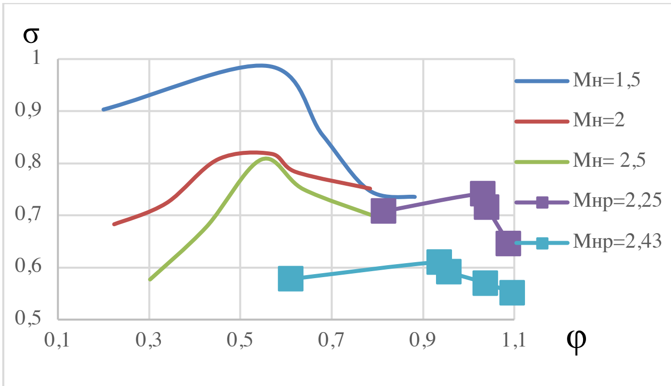


Fig. 2.11 Dependence of the loss coefficient of the total pressure on the flow coefficient φ

Comparison of the results shows that, in general, the nature of the dependencies in the experiment and during the numerical simulation is the same. The difference may be due to the influence of the measurement system in the experiment, the influence of the computational grid, the turbulence model, which was not considered in this work, as well as the influence of the presence of the boundary layer during the numerical simulation and experimental research [8-9].

III. Analysis of the possibility of calculating the air intake device in the ANSYS package when using the IOSO optimization system when choosing σ_{in} as the optimization criterion

During the flight of a supersonic aircraft in a variety of conditions (altitude, flight number M, etc.), it becomes necessary to coordinate the operating modes of the air intake and the engine.

The optimal control program for the power plant of the aircraft is a program for changing the regulatory factors of the power plant in which the selected optimality criterion reaches an extreme value.

The modern level of development of computer technology and numerical methods allows for automated optimization, allowing to determine the optimal position of the regulatory elements of the investigated product to achieve its maximum efficiency.

The section presents the results of the study of the possibility of using modern optimization methods to determine the optimal position of the wedge panels of a supersonic air intake in order to maximize σ_{in}.

3.1 Optimization method

According to the technical task within the framework of this work, the IOSO NM (Indirect Optimization on the Base of Self - Organization) software package was selected for optimization studies. IOSO NM uses the method of indirect statistical optimization based on self-organization (MNSO). This method is based on using the technology of constructing response surfaces for approximation of the objective function and constraints.

At the initial stage of the work of the basic MNSO, an experimental plan is formed by generating random values of the vector components of variable variables with uniform distribution. For all points from the experiment plan, a direct appeal is made to a mathematical model that calculates the values of the optimization criterion and the parameters to be limited. The minimum required number of points in the experiment plan is small and for a problem with a dimension of 100 variables it can be 40 ... 60 points.

In the optimization process, information is accumulated about the system under study in the vicinity of the optimal solution, which leads to a gradual improvement in the quality of the response surface in the optimization process. The operation scheme of the IOSO optimization algorithm is presented in (fig. 3.1).

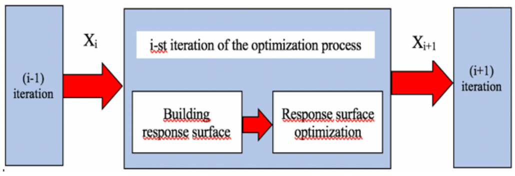


Figure 3.1. IOSO technology iteration scheme

Distinctive features consist in the possibility of constructing response surfaces with high predictive properties for objective functions of large dimension and with complex topology with the minimum possible number of points in the initial plan of the experiment (40 ... 50).

3.2 Stages of optimization of the position of the regulatory elements of the air intake of a supersonic aircraft

In the general case, the structure of the optimization study of the input device of the power plant of a supersonic aircraft can be divided into several main stages. The (fig. 3.2) presents a flowchart illustrating the procedure for conducting an optimization study.

For automation, presented in the figure of technology, a complex was developed that includes:

- the program Ansys CFX, designed for gas dynamic calculations of the flow and determine the value of the optimization criterion – σ_{in}.
- Ansys CFX program designed to create a geometric model of the input device of the power plant, taking into account the specified position of the adjustable elements;

The search for the optimal solution is carried out by the procedure of multicriteria optimization IOSO. This program manages the entire settlement complex: it generates a source data file, runs the programs in the required order, saves the change in the settlement project, analyzes the results obtained.

Besides, the IOSO NM program preserves the history of the study, which allows identifying the relationship between the geometrical parameters of the product and its gas-dynamic characteristics.

Automation of data transfer from one program to another is implemented using developed macros.

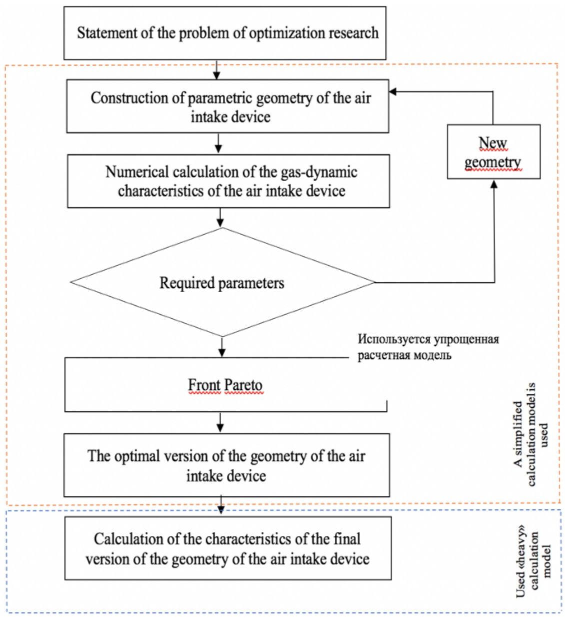


Figure 3.2 A block diagram of the technology developed to optimize the position of the regulating elements of the air intake of a supersonic aircraft

Optimization of the position of the central body is carried out under the following conditions:

- Changing the position of the central body will be done by varying the adjustable parameter;
- The following parameters will be recorded in the optimization study: flight Mach number (M = 2.5), the initial posi-

tion of the input device.

- The target function is the coefficient of full pressure σ_{in}. Recovery averaged over the area in the cross-section of the engine inlet;
- To calculate the gas-dynamic parameters of the flow, a coarse grid is used, which makes it possible to significantly reduce the research time [10-11].

Conclution

The paper presents an overview of classical and promising input devices of a power plant of a supersonic aircraft, as well as an analysis of the results of their research. The analysis led to the following conclusions:

1. The choice of the type of supersonic input device depends largely on the purpose of the aircraft. For supersonic aircraft flying at numbers M <2.5, air intakes of external and internal compression are used. On airplanes flying at M> 2.5, mixed compression air intakes are used, which can significantly reduce air intake losses at high flight speeds;
2. On airplanes of the last generation, they predominantly install the input devices of the ventral and winged layout, which are less sensitive to changes in the angle of attack than the side air intakes;
3. In the program ANSYS Workbench, a geometric model was built for numerical modeling and a geometric model of the computational domain was created on its basis;
4. A numerical simulation of the operation of the air intake device in automatic mode using the ANSYS CFX package;
5. The analysis of the results of numerical simulation and their comparison with the results of an experimental study;
6. Further work is planned to conduct an optimization study of the geometry of the supersonic air intake in the IOSO system using the ANSYS software package to evaluate its gas-dynamic characteristics

The analysis showed that the flow pattern during the numerical simulation and experimental research is the same. At the same time, there are differences in the numerical values - the calculation gave a lower result, this may be due to the influence of the computational grid, the boundary layer, and the turbulence model used.

Reference

1. Borovikov A., Gavriliouk V., Gilevich I., Duganov V., Khokhlov A., Stepanov V. Gasdynamic design of supersonic and hypersonic airframe integrated inlets and nozzles //AIAA Paper. - 96-4549, 1996.
2. Supersonic fighter development / Roy Braybrook. - Somerset; Newbury Park (Calif.): Haynes, 1987. - 208 c.
3. J.R. Wilson, “The New Shape of Supersonics,” *Aerospace America*, June 2002, pp. 26-33.
4. Susan E. Cliff, Scott D. Thomas and Veronica M. Hawke, “Swing-Wing Inline-Fuselage Transport Design Studies at Supersonic Flight Conditions,” AIAA Paper 2009-7073, Sept. 2009.
5. Kumar G. Bhatia and Jin Wertheimer, “Aeroelastic Challenges for a High-Speed Civil Transport,” AIAA Paper 1993-1478.
6. Molchanov, Alexander M. “THERMOPHYSICS AND FLUID DYNAMICS. Special Chapters.” OSF Preprints, 20 Dec. 2018. Web.
7. Molchanov, Alexander M. “NUMERICAL METHODS FOR SOLVING THE NAVIER-STOKES EQUATIONS.” OSF Preprints, 13 Mar. 2019. Web.
8. B. Andersson,R.Andersson,L.Ha ˚kansson,M.Mortensen,R.Sudiyo,B.vanWachem,L.Hellstro ‘Computational fluid dynamics for engineers’, 2012
9. Scott J. Ormiston Jeffrey R. Berg “ANSYS CFX”, 2013
10. Egorov, I.N., Kretinin, G.V., Leshchenko, I.A. and Kuptzov,S.V. “Use of the IOSO NM Software for Complex Optimization Problems”. 11th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference, Portsmouth, Virginia, USA, 6-8 September, 2006. Inverse Problems, Design and Optimization Symposium, Miami, Florida, USA, 16-18 April, 2007.
11. Egorov I.N., Kretinin G.V., Leschenko I.A., Kuptcov S.V. “Multi-objective Optimization Using IOSO Technology”, 7-th ASMO UK / ISSMO Conference on engineering Design optimization, Bath, UK, 7-8 July, 2008.