VERIFICATION AND VALIDATION OF MODELS AND METHODS OF NUMERICAL MODELING OF SPATIAL FLOWS

A.M. Kharitonov

Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, 630090 Novosibirsk, Russia.

Intense development of numerical methods of modeling the flow around various vehicles necessitates establishment of rigorous standards for determining the degree of reliability of results and/or the area of their applicability. This especially refers to spatial flows, with models lacking complete adequacy and various assumptions used in numerical simulations. Reynolds-averaged Navier-Stokes (RANS) equations are often used to solve various problems of spatial flows, with differential models of turbulence being used for closing the system of equations. All these models contain different numbers of empirical constants, which are valid, strictly speaking, only for certain conditions and, hence, can be used to solve a rather limited class of problems. Therefore, available models of turbulence can be reasonably used within the framework of a zonal approach. Long-term experience [1-3] of using various turbulence models for solving versatile problems of spatial flows shows that a particular model usually allows acceptable results to be obtained only for certain classes of flows. Thus, the known (k-ω) model [1] provides a satisfactory description only for near-wall flows with a developed boundary layer, whereas the (k-ε) turbulence model [2, 3] is more suitable for flows in mixing layers. For this reason, hybrid approaches have appeared lately [4]; in these approaches, a smooth transition from one model to another is performed, depending on the distance from the solid walls. Other methods of turbulence modeling are also under development [5]: for example, the method of direct numerical simulation (DNS), large eddy simulation (LES), etc. Application of DNS is limited because it is impossible in practice to resolve all dynamically important scales; hence, this method can be used at the moment only to calculate simple geometries in low-Reynolds-number flows. The LES method is also approximate, because it allows the contribution of large eddies to be calculated fairly precisely, but small-scale vortices are modeled as approximately isotropic ones.

A specific problem of the methods developed is the low accuracy of calculating flows with shock wave-boundary layer interaction leading to formation of separation regions.

Problems of reliability of numerical methods of solving various problems of spatial flows have been discussed at international conferences for the last 15 years [6-9]. The possibility of development of databases containing test cases for comparisons of numerical solutions with experimental data is considered [10]. Such databases stimulate further development of the methodology of numerical and physical research and lead to a higher level of aerothermodynamic data, as applied to in-flight conditions, with the aim of determining uncertainties.

Adequate numerical modeling of complex flows in the vicinity of spatial configuration requires a strategy for verification and validation of software products being developed. As numerical realizations are only approximations of real flows and, hence, only approximations of reality, their reliability has to be estimated via known procedures of verification and validation. To use these procedures successfully, one has to understand the difference between them.

As was stated in [6], verification is the process of determining the degree of adequacy and the level of accuracy of numerical modeling of a particular conceptual model, while validation shows how accurately the chosen conceptual model describes the examined flow through comparisons of numerical and experimental data. These procedures are schematically shown in Fig. 1.

Hence, the process of verification and validation includes both the computational and physical aspects. High reliability of modeling is achieved by estimating the degree of accuracy of a chosen conceptual model in representing the real phenomenon and by comparisons with experimental data.
Section I

The adequacy of numerical modeling is evaluated through the following procedures:
- determination of the order of convergence of numerical solutions by comparisons with exact analytical solutions (if they exist) or with the size of the coordinate grid tending to zero;
- evaluation of the sensitivity of the discretization algorithm to various uncertainties: space and time constraints, adaptation of the grid to the model geometry and boundary conditions, etc.

Validation implies careful comparisons of results of numerical calculations of the phenomenon under study with experimental data to answer the question: Is the numerical solution physically correct?

The process of validation involves the following procedures:
- evaluation of the degree of reliability of the chosen numerical and experimental methods;
- evaluation of systematic errors of experimental data: nonuniformities of the velocity field, effect of the test-section walls and supporting devices, and accuracy of measurement equipment and processing of measured information. It is also necessary to know the level and scale of turbulence at the test-section entrance.

In accordance with [8,11], validation of results of numerical simulations implies a comparative analysis of the physical model and all uncertainties, including uncertainties associated with initial and boundary conditions from the physical and computational viewpoints.

Uncertainties are usually inherent in both numerical and experimental modeling. Numerical modeling involves uncertainties associated with 1) insufficient correspondence between the theoretical and numerical models, 2) insufficient accuracy of computations, 3) inaccurate modeling of the phenomenon, 4) influence of external phenomena, and some others. Wind-tunnel modeling implies overcoming difficulties related to 1) effects of interference of phenomena, 2) insufficient measurement accuracy, 3) insufficient number of tests, 4) choosing and defining a particular test program, etc. The critical difference between numerical and physical modeling is the degree of modeling of the phenomenon and insufficiency of measurements.

Thus, careful verification and validation allows researchers to analyze the sensitivity of models and their implementations to existing uncertainties. As a result, the degree of reliability of models and methods of numerical modeling can be estimated, and the areas of applicability of various approaches can be determined.

Two examples of validation for spatial flows are given below: 1) the flow around a delta wing at an angle of attack and 2) the flow around two separating stages of an aerospace vehicle.

Supersonic flow around a delta wing at an angle of attack

Regimes of the flow around a delta wing are characterized by the number, size, and position of streamwise vortices and internal shock waves. Owing to this variety of features, this class of flows is a good test case for methods of calculating complex spatial flows under development. For this purpose, the Khristianovich Institute of Theoretical and Applied Mechanics of the Siberian Branch of the Russian Academy of Sciences (ITAM SB RAS) and the Keldysh Institute of Applied Mathematics of the Russian Academy of Sciences (KIAM RAS) performed a comprehensive numerical and experimental study of the flow field on the leeward side of the delta wing [11÷19]. The flow around the delta wing was numerically
modeled with the use of the spatial steady Euler equations and unsteady Navier-Stokes equations. An algorithm was developed, which allowed the Euler and Navier-Stokes models to be simultaneously used in different blocks. A grid with six blocks was developed, and the zonal approach was applied with the use of algorithms of parallelization of computations on high-resolution grids containing up to 640 nodes along the wing half-span. Such an algorithm facilitates grid generation in areas of different shape, allows some solution discontinuities to be identified in the form of moving boundaries of areas, and makes it possible to use different physical models in different blocks.

Advanced methods used in experiments include optical visualization of the limiting streamlines on the model surface by the oil-film method, visualization of the spatial pattern of the flow by the laser sheet technique, measurement of pressure distributions over the model surface, and measurements of fields of gas-dynamic parameters with the use of a rake with five-channel pneumometric probes.

Figure 2 shows the delta wing geometry and its basic parameters.

The analysis of the results obtained and the validation of the method of calculations within the framework of the Euler equations through comparisons with experimental data are illustrated in Fig. 3. It is seen that numerical modeling predicts the flow regime with formation of the primary and secondary vortices and also a system of weak compression waves above the primary vortex on the leeward surface. The shock wave under the primary vortex is not identified, which may be caused by insufficient grid resolution.

There are grounds to believe that the use of a larger number of grid nodes will allow this shock wave to be resolved. At the same time, the neglect of the turbulent state of the boundary layer leads to violation of flow conicity, thus, making the flow more sensitive to pressure gradients on the wing surface. Hence, it is necessary to solve this problem with the use of the Navier-Stokes equations and appropriate turbulence models. It was shown [13] that such an approach provides adequate simulation of the flow topology on the wing surface and in its vicinity and predicts reasonable values of gas-dynamic parameters.

Verification and validation of the numerical realization of solving the problem considered stimulated the use of a hybrid method of numerical modeling, combining the Euler and Navier-Stokes models.
The Navier-Stokes model was used in two out of six blocks adjacent to the wing surface, and the Euler model was used in the remaining four blocks. The patterns of the surface streamlines calculated by the Navier-Stokes and Euler equations are compared in Fig. 4. Figure 5 shows the spanwise distributions of the pressure coefficient in two cross sections of the wing in comparison with experimental data.

The calculations of the flow around the delta wing and comparisons of the results obtained with experimental data demonstrate fairly good agreement with the Navier-Stokes calculations.

The differences in pressure distributions and in formation of vortex structures above the wing, calculated by the Navier-Stokes and Euler equations, are illustrated in Fig. 6.
As a whole, the results obtained made it possible to study the specific features of the supersonic flow on the leeward surface of delta wings in detail and to supplement the diagram proposed earlier by Wood and Miller (gray lines), which is illustrated in Fig. 7. The colored lines are new regimes added to this diagram:

1) boundary of the emergence of secondary separation;
2) boundary of the emergence of tertiary separation;
3) boundary of regimes where shock waves are formed above the primary vortex sheet;
4) region of transition from regimes without shock waves above the primary vortex sheet to regimes with shock waves being formed;
5) right boundary of regimes characterizing the formation of a horizontal shock wave between the pair of primary vortices.

Thus, the process of numerical modeling accompanied by careful validation allows the area of applicability of the Euler model for this class of flows to be determined. As a consequence, perfect software products can be developed to study the specific features and to calculate complex spatial flows.

**Flow between separating aerospace stages**

Two-stage-to-orbit systems are among the most realistic aerospace transportation systems. In this case, the second stage is the orbiter, and the first stage equipped by an air-breathing engine ensures separation of the stages in the range of Mach numbers of 6÷12 at altitudes of ~ 30 km and, hence, at high dynamic pressures. Under these conditions, the aerodynamic interference between the stages exerts a significant effect on the separation maneuver safety. In the general case, the flow around separating stages is a complicated three-dimensional unsteady gas-dynamic problem. As is shown in Fig. 8, the flow between the stages is accompanied by interaction of incident and reflected shock waves and expansion waves with each other and with boundary layers.

This class of flows was intensely studied, and results were reported in many publications, including [20-32]. The pressure fields in the flow
around two bodies of revolution and interference between these bodies and a flat plate were analyzed in detail in [20].

Aerodynamic interference of separating generic two-stage winged systems was studied numerically and experimentally in [21-26]. Adequate numerical simulation of such flows is impossible without profound understanding of the physical pattern of interaction of the stages. Therefore, a comprehensive aerophysical experiment was performed in a supersonic wind tunnel based at ITAM SB RAS at a Mach number $M_\infty=3$ and Reynolds number per meter $Re=35 \times 10^6 \text{ m}^{-1}$. The experiment included measurement of integral and distributed characteristics in a wide range of relative positions of the separating stages.

![Diagram of the experimental setup](image)

The models of the first and second stages are combinations of an axisymmetric cone-cylinder body with a flat tapered wing with sharp leading and trailing edges (Fig. 9). The sweep angle of the leading edge of the wing is $\chi=53^\circ$. The wing has a hexagonal profile with a constant spanwise thickness equal to 4% of the wing chord. The second stage model is a halved copy of the first stage. Modeling of separation of the stages within the ranges $\Delta z = D \div 3D$ and $\Delta x = -0.5D \div 1.5D$ was provided by a special device. The angle of attack of the second stage was changed simultaneously with the change in the angle of attack of the first stage within the range $\alpha = 0 \div 10^\circ$.

An analysis of uncertainties and their possible contributions to the integral error allows one to estimate the maximum deviations of aerodynamic coefficients (they stay within the interval $\pm 0.01 \div 0.03$), which allows the differences between numerical and experimental data to be found with high reliability.

Numerical modeling of separation of the stages was performed to determine the area of applicability of gas-dynamic equations for this class of problems. The algorithm of solution of three-dimensional steady Euler equations was implemented at KIAM RAS on multiprocessor computing systems and was described in detail in [24-26].

An example of calculated and experimental values of the pressure coefficient on the surfaces of the stages in the plane of symmetry is shown in Fig. 10. The character of the dependences $C_p(x)$ reflects the above-mentioned features of the flow in the space between the stages. Satisfactory qualitative and quantitative agreement is observed everywhere between the calculations and experiments in terms of the coordinate of arrival of the bow shock wave from the second stage on the first stage and the values of pressure in this region. There are significant differences, however, especially at $\alpha > 0$, in the values of pressure directly behind the inflection of the body (cross section $x=3.5D$) of the first stage.

The calculated pressure distributions at $\Delta z = 1.5D$, where the governing factors of the influence of the second stage on the first stage are the bow shock wave and the expansion fan from the junction of the nose part with the body, are also in good agreement with the experiment (Figs. 10a and 10b).

As the nose part and the major part of the body of the second stage at $\Delta z = 1$ is located in the conical flow field of the first stage, the positive phase of excess pressure is additionally realized on
a significant part of the lower surface of the second stage (Fig. 10c). Under the influence of the expansion fan from the first stage, however, the pressure on the second stage decreases, and a region with the negative phase of excess pressure is formed. The pressure increases again when the shock wave emanating from the wing arrives on the lower part of the body.

As the angle of attack increases, the positive deflections of the conical flow field become smaller, and the effect of the expansion fan becomes more pronounced. Correspondingly, the positive phase of excess pressure decreases, and the negative phase increases (Fig. 10d). At both angles of attack, significant differences between the calculations and experiments are observed in the region of the negative phase of excess pressure. This can be attributed to the arrival of the shock wave from the second stage wing on the lower part of the body and the corresponding increase in pressure, which is predicted by calculations to be much closer to the base region than it follows from the experiment. Apparently, these differences are associated with insignificant density of the computational grid. Clearly identifying the features of the flow around the first stage, the grid is insufficient for adequate calculation of the flow around the second stage because of the smaller size of the latter. The second reason for these differences may be the insufficient adequacy of the mathematical model chosen for this flow, because the model does not take into account the specific features of the boundary-layer flow. As was shown in [25], the angle of shock-wave reflection from the surface in the boundary layer is noticeably greater in the experiment than in the calculation. This fact is largely responsible for the differences in the subsequent flow regions. If the near-wall region is calculated by the Navier-Stokes equations with an appropriate turbulence model, good agreement between the calculation and experiment can be expected.

Thus, with allowance for the differences mentioned above, the calculated and experimental pressure diagrams are in good agreement in the cases considered. Validation of the numerical method allows the latter to be improved. The above-given example gives grounds to believe that the Euler model in the range of parameters considered provides fairly accurate modeling of the flow in the space between the stages of a two-stage aerospace system during separation of the stages.

A detailed analysis of all information obtained allowed us to identify three typical regions of interaction of the separating stages, depending on their relative positions, which are shown in Fig. 11.
Section I

1. Weak interference region, where the bow shock wave from the first stage reaches the surface of the second stage in the vicinity of the rear part, and the positive interference increment $\Delta C_{n_2}$ is mainly caused by the bow shock wave from the first stage.

2. Strong interference region, where a large part of the lifting surface of the second stage is located in the conical flow field between the bow shock wave from the first stage and its expansion fan. Typical features of this region are a significant positive increment $\Delta C_{n_2}$ at a zero angle of attack and a considerable negative gradient of the dependence $\Delta C_{n_2}=f(\alpha)$.

3. Negative interference region, which is formed at small distances between the stages and non-negative angles of attack as a result of the influence of the reflected own shock wave and expansion fans from the first and second stages.

This classification of interference regions is also valid for the longitudinal force of the second stage. Turning of the second stage at an additional angle of attack $\Delta \alpha$ for all values of $\Delta x$ and $\Delta z$ intensifies the interference effect of the first stage on $\Delta C_{n_2}$. As the velocity increases to $M^\infty=6.11$, the general typical features of the flow around the separating stages remain unchanged. The relative changes in the normal force of the second stage owing to interference become more pronounced.

Thus, an analysis of results of three-dimensional inviscid Euler computations with the use of the zonal approach and validation of these results by experimental data confirm the possibility of predicting a complicated structure of supersonic flows in the vicinity of separating stages.

Another example of a two-stage-to-orbit aerospace system is the ELAC-EOS concept developed at the Institute of Aerodynamics at the Aachen Technical University. This configuration is as close to the real one as possible. The geometry of the first and second stages is shown in Fig. 12.
The results of vast numerical and experimental studies of the ELAC-EOS two-stage aerospace system were obtained together with the Aerodynamic Institute at the Aachen Technical University, Munich Technical University, and ITAM SB RAS in Novosibirsk. They are described in [27-32].

The characteristics of the ELAC-EOS models were also studied in the T-313 supersonic wind tunnel of ITAM SB RAS at $M_a = 4.0$ and $Re=4.8\cdot10^6$. The following parameters were varied:

- Distance between the models $h/L=0.225$, 0.325, and 0.450; angles of sideslip of the second stage $\Delta\beta=0^\circ$, 2°, and 4°;
- Angles of fixation of the second stage with respect to the first stage $\Delta\alpha=0^\circ$, 2°, and 5°;
- Angles of rolling $\Delta\phi=0^\circ$, 2°, and 4°.

The angles of attack of both models were simultaneously changed in the range $\alpha=0^\circ+6^\circ$ with a step of 1° for each parameter being fixed.

Separation of the ELAC-EOS stages was calculated by three-dimensional laminar Navier-Stokes equations, which were solved by the finite-volume method in curvilinear coordinates including structured and unstructured blocks. For more careful verification and validation of the numerical method, special experiments were performed to study the flow during separation of the EOS orbital stage from a flat plate.

As an illustration, the Schlieren picture of on interaction case is compared in Fig. 13 with the calculated static pressure distribution on the surface of the second stage.

As in the experiment, the calculation (Fig. 13b) at a distance $h/L=0.150$ and angle of attack $\alpha=0^\circ$ predicts the position of the bow shock wave from the model and the reflected shock wave from the flat plate, which is then incident on the rear part of the model in the vicinity of its base region. Thus, a comparison between the calculation and experiment shows that the calculation correctly reproduces all basic features of the flow.

The calculated pressure distribution on the flat plate is compared with experimental results in Fig. 14. It is clearly seen that small differences are observed only in a local region of incidence of the bow shock wave onto the plate and in the vicinity of arrival of the oblique shock wave from the wing.
Section I

A visual analysis of the Schlieren pictures of interaction between the stages showed that the main factors of influence of the first stage on the second stage are the bow shock wave of the first stage, the expansion fan induced by flow deflection when the flow enters the cavity for the second stage, the barrel shock wave induced by flow turning along this cavity, and (in some cases with the minimum distance between the stages) the expansion fan induced by flow deflection at the maximum thickness cross section of the first stage. There are only two factors of influence of the second stage on the first stage: the bow shock wave of the second stage and the shock waves induced by the wings of the second stage.

The influence of these factors reduces the normal force of the first stage and changes its pitching moment. This is confirmed by the dependences of the coefficients of the longitudinal force $C_{ XB}$, normal force, and pitching moment $C_{ MB}$ on the angle of attack for the limiting case ($h_{ min}$, $\Delta \alpha =5^\circ$). An analysis and reconstruction of the measured aerodynamic characteristics testify that the second stage EOS does not affect the characteristics of the first stage ELAC at distances greater than $h/L=0.450$. This conclusion is valid for all parameters $\alpha$, $\Delta \alpha$, $\Delta \beta$, and $\Delta \phi$.

The effect of the first stage on the second one with changes in intensity of all factors mentioned, depending on $h$, $\Delta \alpha$, and $\alpha$, reveals the governing role on the aerodynamic characteristics of the second stage. A complicated dependence between the interference components of the normal force and pitching moment was found. With changing the distance between the stages, a reverse change in the pitching moment is observed. At small distances between the stages, an additional normal force is formed, and the center of pressure is shifted upstream, which makes the pitching moment decrease with increasing angle of attack. As the distance $h$ increases, the bow shock wave from the first stage is shifted downstream, and a more significant contribution to generation of the additional lift force is made by the lifting surface of the second stage by shifting the center of pressure downstream, which increases the negative pitching moment.

Figure 15 illustrates the behavior of the normal force $C_{ zb}(\alpha)$ and pitching moment $C_{ mb}(\alpha)$ for different distances $h(\alpha)$.
As the angle of attack increases, the disturbed flow field from the first stage becomes attenuated, and this effect degenerates. With a further increase in the distance between the models, the second stage gradually leaves the disturbed flow field, and its normal force and pitching moment approach the nominal values. Changes in the yawing and rolling angles do not produce any noticeable effect on the normal force and pitching moment.

As a result of these studies, a database of aerodynamic characteristics of separating winged vehicles, including their interference, was formed.

The information obtained favors better understanding of aerodynamics of separating stages and can be used for validation of numerical methods, which were developed within the DFG projects SFB 255 at the Munich Technical University and SFB 253 at the Institute of Aerodynamics at the Aachen Technical University [27-29].

Depending on the geometry of the first and second stages and their relative positions, the following events are possible:

● negative interference region, where the second stage completely loses its lifting properties;
● formation of reverse changes of the pitching moment owing to displacement of the center of pressure upstream or downstream during separation of the second stage.

These and other interference effects can substantially influence the separation maneuver safety.

Thus, the examples of validation of numerical methods for calculating complex spatial flows considered above and many others show that this procedure is extremely important. It allows the area of applicability of numerical methods to be determined and their reliability and accuracy to be improved. For these reasons, the validation procedure should undoubtedly become a necessary attribute of numerical methods under development.

Experimental data collected for this purpose should be as complete as possible and should be accompanied by an actual credibility interval.

This work was sponsored by RFFI in frame of the projects № 03-01-00249, 06-01-00774.

References


10. Marini M. An example of CFD codes verification and validation in aeronautics and turbo machinery: The European Flownet database project. Proceeding of the EWHSSF Conference Beijing, China, October 19 22, 2005.


