Experimental and Computational Modeling of Parachutes Fluid Dynamics

I.I. Lipatov
Central Aerohydrodynamic Institute
1 Zhukovsky Street
Zhukovsky, Moscow Region
140180 RUSSIA

lipatov@tsagi.ru

PHYSICAL AND NUMERICAL MODELING

The paper consists of two parts devoted correspondingly to the experimental opportunities and results along with the computational methods and results obtained mainly in the Central Aerohydrodynamic Institute (TsAGI).

1.0 EXPERIMENTAL TOOLS

Experimental tools include two subsonic wind tunnels T-101 and T104 allowing conducting experiments on large-scale models in a large range of parameters. Other facilities will be mentioned below.

The biggest wind tunnel T-101 is a continuous operation return-flow facility which has an open test section designed for testing different objects including parachutes, paragliders etc.

This wind tunnel is equipped with the six-component balance, remote control of articles under tests, computer complex for test data processing. Two fans consuming power of 30 mW generate airflow. Measuring equipment allows to determine main aerodynamic characteristics of aircraft models: Cx, Cy, Cz, Mx, My, Mz.

The facility is characterized by next parameters:

- Flow velocity: 5 ... 55m/sec, Reynolds number: up to $3.3 \cdot 10^6$ (L=1m), total pressure: atmospheric, dynamic pressure: up to 1.8kPa, stagnation temperature: ambient, test section dimensions-length: 24m, elliptical nozzle: 14 x 24m

Another experimental facility T-104 is a continuous return-flow wind tunnel with an open test section. It is designed for testing different models at low subsonic flow velocity in a wide range of angles of attack and sideslip angles.

The wind tunnel T-104 provides conducting more than 20 types of tests, including tests parachutes and rescue means, physical phenomena investigation, etc.

This facility is characterized by the next parameters:

- Flow velocity: 15... 125m/sec, Reynolds number: up to $8 \cdot 10^6$ (L=1m), total pressure: atmospheric, dynamic pressure: up to 8.5kPa, test section dimensions-length: 13m, axisymmetrical nozzle diameter 7m
- Test section dimensions Length: 13m Nozzle diameter: 7m

Lipatov, I.I. (2006) Experimental and Computational Modeling of Parachutes Fluid Dynamics. In *Fluid Dynamics of Personnel and Equipment Precision Delivery from Military Platforms* (pp. 2-1 – 2-10). Meeting Proceedings RTO-MP-AVT-133, Paper 2. Neuilly-sur-Seine, France: RTO. Available from: http://www.rto.nato.int/abstracts.asp.

T-104 is equipped with additional tools allowing investigations of unsteady aerodynamics of parachutes. One of such experiments is presented on the figure 1.

Figure 1: Experiments in the wind tunnel T-104.

Concrete results of experiments belong to designing bureaus and cannot be presented in the paper.

2.0 NUMERICAL MODELS AND METHODS

2.1 Method of Discrete Vortices

The second direction of investigations is associated with the mathematical modeling.

One approach is based on the method of discrete vortices [1]. This method allow to investigate the flow nearby an airplane along with the near and far wake parameters. This method can be evaluated as an approximate or engineering one, but along with experimental verification it provides reliable and rather quick tool for different flow parameters investigation. The method suggests that the flow is described by Euler equations. Viscosity influence is taken into account on the surface of an airplane. The method is especially accurate for the bluff bodies geometries, like open ramp where boundary layer separation is fixed.

As an example the trajectory of one airplane (JAK 40) in the wake of another one (IL 76) [2] is presented on the figure 2.

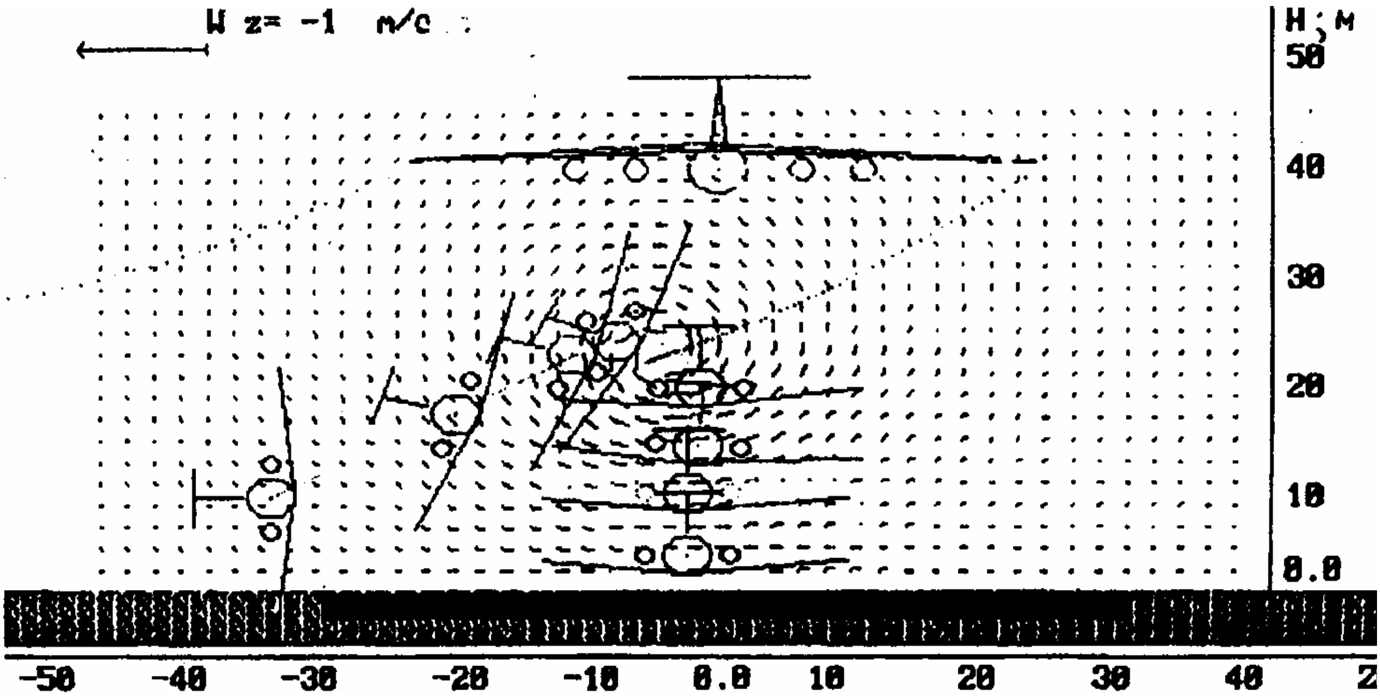

Figure 2: Influence of the vortex wake on the dynamics of an aeroplane.

Method of discrete vortices may be applied as well to describe the flow nearby opening parachute located in the near wake of an airplane.

2.2 Full Euler Models

One of the models used to describe near wake is the model based on the panel method [3].

Another model is based on the full Euler equations approximation.

Key feature of the problem is associated with the complex grids generation describing airplane geometry. The problem is formulated as follows

$$\frac{\partial \vec{q}}{\partial t} + \frac{\partial \vec{E}}{\partial x} + \frac{\partial \vec{F}}{\partial y} + \frac{\partial \vec{G}}{\partial z} = 0$$

where

$$\vec{q} = [\rho, \rho u, \rho v, \rho w, e]^T$$
$$\vec{E} = [\rho u, \rho u^2 + p, \rho uv, \rho uw, u(e+p)]^T$$
$$\vec{F} = [\rho v, \rho uv, \rho v^2 + p, \rho vw, v(e+p)]^T$$
$$\vec{G} = [\rho w, \rho uw, \rho vw, \rho w^2 + p, \rho vw, w(e+p)]^T$$

The pressure is determined from the state equation for perfect gas characterized by the specific heat ratio γ

$$p = (\gamma - 1)[e - \rho(u^2 + v^2 + w^2)/2]$$

On the surface having the normal vector $\vec{n} = (n_x, n_y, n_z)$ the next boundary condition is valid

$$un_x + vn_y + wn_z = 0$$

For the incoming flow $\rho \to \rho_\infty, e \to e_\infty, \vec{V} \to \vec{V}_\infty$. On outer boundary in the external flow $p \to p_\infty$.
On the side boundaries next non-reflection conditions are posed

$$R_{ext/\text{int}} = V_n \pm 2a/(\gamma - 1)$$

where a is the speed of sound, V_n is the velocity normal to the boundary. On incoming parts of external boundary entropy, tangential velocity and R_{ext} are known, second invariant R_{int} is extrapolated on the boundary from internal points. On the out coming parts R_{ext} is known at the same time entropy, tangential velocity and R_{int} are extrapolated from inner points. During the process of finding steady solution the Kutta-Zhukovsky condition needed to find out unique steady solution is not posed. It is supposed that the solution belongs to the family of limited function and may be found automatically using dissipative schemes.

Method of finite volumes is used where integral equations are used instead of differential ones.

$$\frac{\partial}{\partial t} \iiint_v \vec{q}\,dxdydz + \iint_s (\vec{E}n_x + \vec{F}n_y + Gn_z)ds = 0$$

where s is a boundary of grids cell. Due to some reasons explicit scheme was chosen. The computational procedure includes four steps on each time level

$$q^{(1)} = q^{(n)} - \alpha_1 \tau [R(q^{(n)}) + D(q^{(n)})]/\theta$$

$$q^{(2)} = q^{(n)} - \alpha_2 \tau [R(q^{(1)}) + D(q^{(1)})]/\theta$$

$$q^{(3)} = q^{(n)} - \alpha_3 \tau [R(q^{(2)}) + D(q^{(2)})]/\theta$$

$$q^{(n+1)} = q^{(n)} - \tau [R(q^{(3)}) + D(q^{(3)})]/\theta$$

where $q^{(n)}$ and $q^{(n+1)}$ flow parameters on the n-th and (na+1) -th time levels; $R(q^{(n)})$ is the vector consisting of convective and diffusion terms; $D(q^{(n)})$ determines artificial dissipation; θ - is the cell volume; τ - time step; α_1, α_2, α_3 - Runge-Kutt coefficients. Artificial dissipation was introduced to provide stability of numerical algorithm and to avoid oscillations arising nearby shock waves [4].

For computations of isolated wing and wing + fuselage grid C-H type was used. Details are discussed in [5].

2.2.1 Additional Experiments

Auxiliary experiments have been conducted to evaluate different factors and to justify mathematical models. It was used low speed wind tunnel T-105 where model of an airplane with the pylons and engine cells was investigated.

These experiments have been conducted to analyze influence of small-scale elements on the flow field as well as on the wake structure. Such influence is difficult to investigate using computational fluid dynamics. Model investigated is presented on the figure 3.

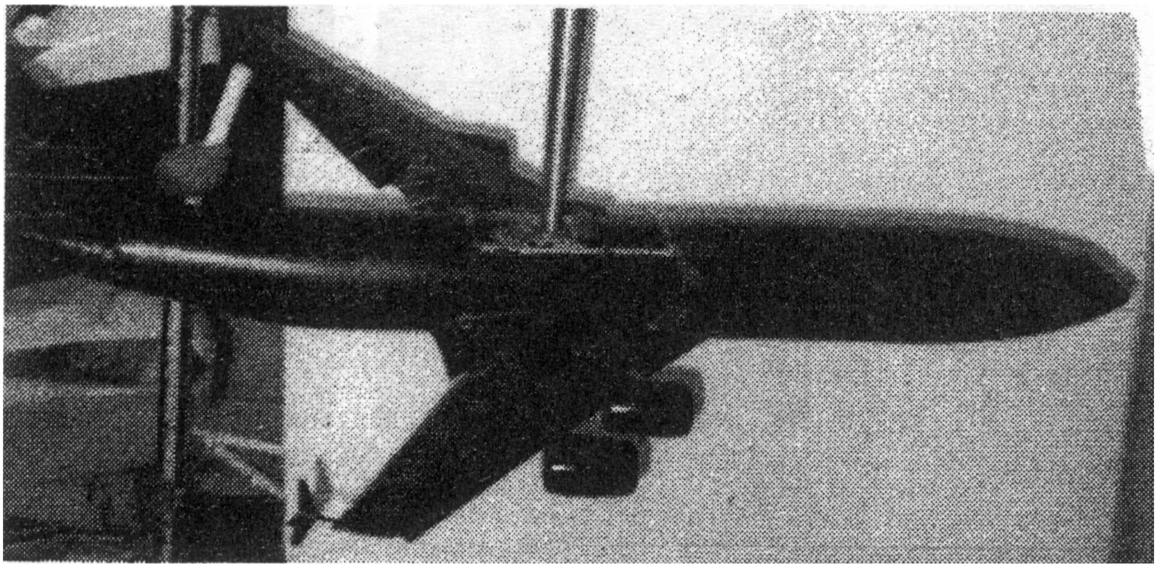

Figure 3: Model of an aeroplane in the wind tunnel.

The flow velocity was equal to 25 m/sec, angle of attack 12°. Wingspan was equal 133.2 cm/. The wind tunnel has an open working section with level of turbulence 0.5%. Two sorts of grids have been used in experiments to control the turbulence level. Both grids provided the same turbulence level and scale of turbulence comparable with grid size.

Another series of experiments have been conducted in the T-124 wind tunnel. These experiments showed that small-scale turbulence has insignificant influence on the structure, size and position of the vortex core and other parameters. The engines jets, turbulent boundary layers and separation on the fuselage and other elements of and airplane generate initial turbulence. Experiments with the An-124 airplane wing allowed choosing constants needed for the algebraic model of turbulence. Experiments allowed as well verifying modified turbulence models.

For the verification of numerical model results of computations have been compared with the experimental results obtained in the aerodynamic wind tunnel T-112 for subsonic and transonic regimes.

Model of swept wing ($\chi = 30^0$) had relative thickness $\overline{c} = 9\,\%$. Results of calculated and measured pressure distributions on the wing surface are presented on the figure 4. It may be seen rather good agreement. For subcritical flow regimes coincidence is quantitative and qualitative. For supercritical flow regimes there is some difference especially in the vicinity of shock waves.

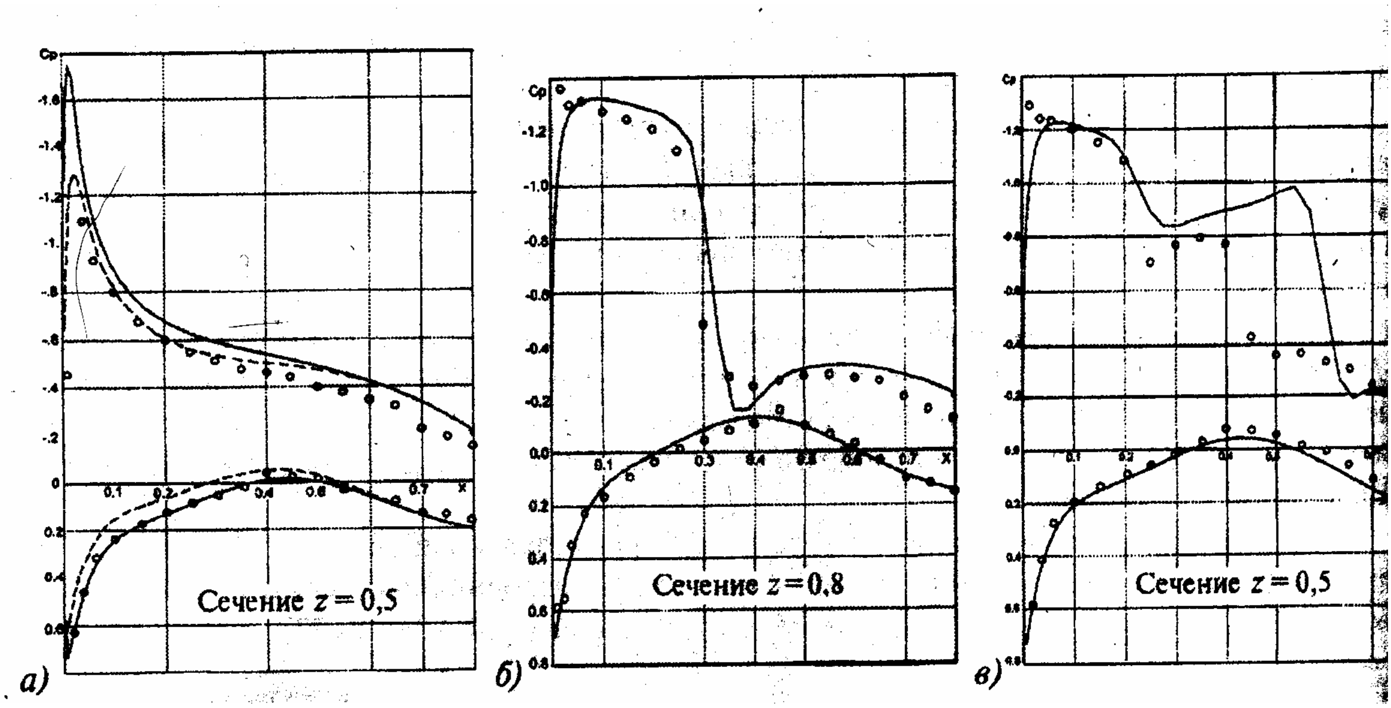

Figure 4: Comparison of numerical and experimental data.
Pressure coefficient distributions in different wing sections.

The main aim of abovementioned computational analysis was to describe mostly vortical system behind an airplane. Nevertheless this approach as applicable as well for the near wake modeling.

Example of the parachute modeling in uniform subsonic flow is described in [6]. Computational modeling of parachute systems needs special computational techniques based on geometrically and dynamically adaptive meshes. Examples of this technology application are depicted on the figures 5-6.

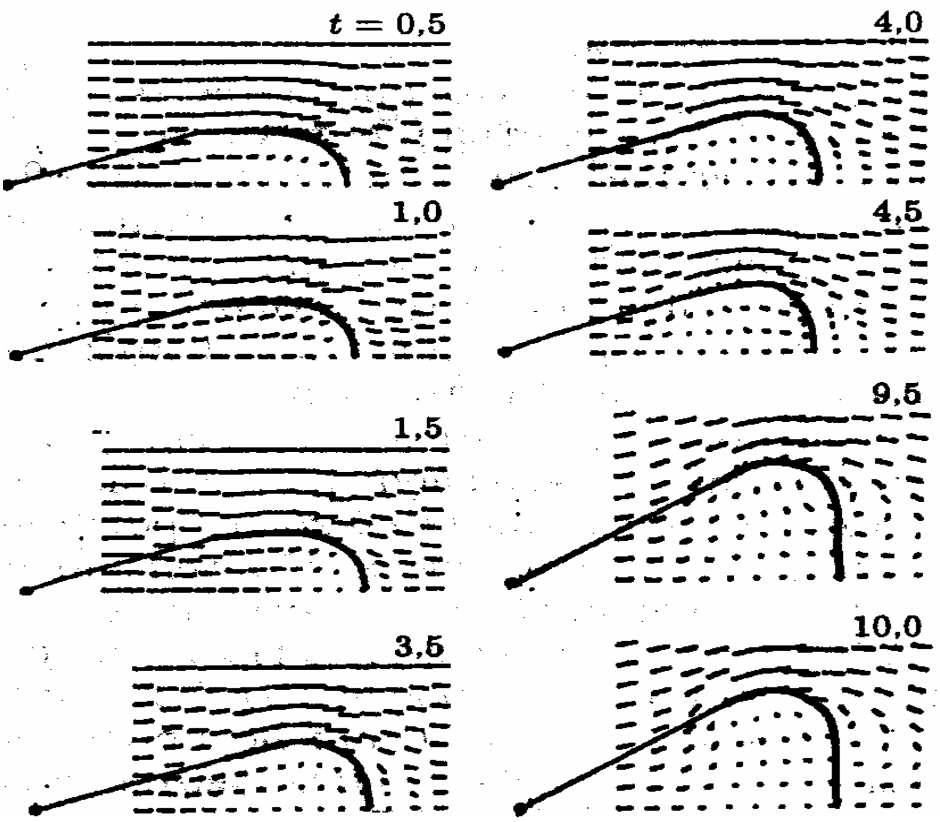

Figure 5: Flowfield near the parachute at different time values.

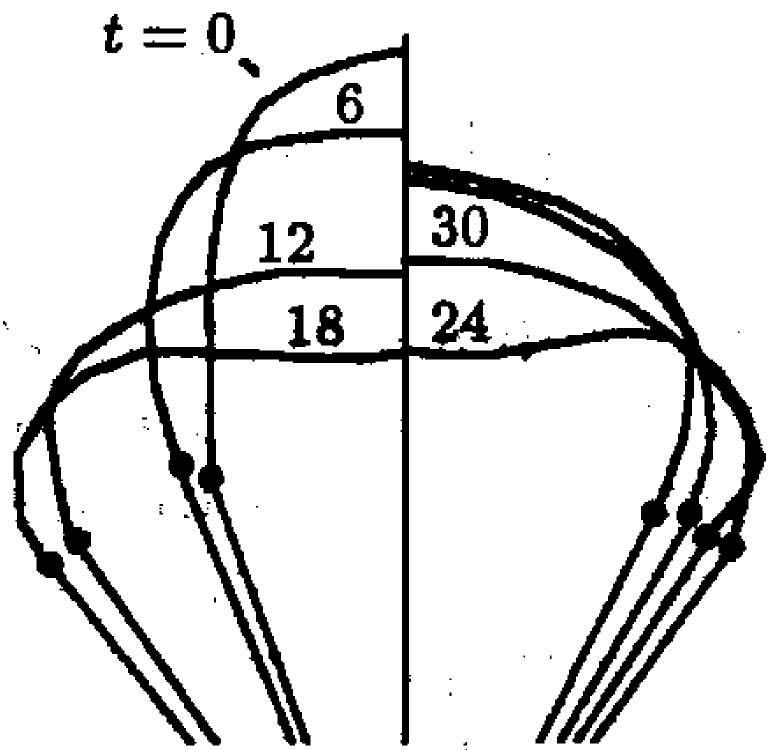

Figure 6: Geometry of parachute at different time values.

REFERENCES

[1] Belocerkovsky S.M., Lifanov I.K. Numerical methods of singular integral equations solving and their application in aerodynamics, theory of elasticity and in electrodynamics. M. Nauka. 1985 (in Russian).

[2] Vyshinsky V.V., Sudakov G.G. Vortical wake of an aeroplane in the turbulent atmosphere. Moscow. TsAGI. 2005.(in Russian).

[3] Sudakov V.V., Voevodin A.V. Numerical method of separated subsonic flow computation near an aeroplane //Uchen. Zap. TsAGI. V.23.N. 3. 1992.(in Russian).

[4] Jameson A. Transonic airfoil computations using the Euler equations // Numer. Meth. in Aeronautical Fluid Dynam., 1982.

[5] Vyshinsky V.V., Kravchenko S.A. Calculation of three-dimensional separated flows within the frame work of nonstationary Euler equations // TsAGI Journal. V.1. N. 2. 1994.

[6] Gilmanov A.N. Methods of adaptive meshes in gas dynamics problems. M. Nauka. 2000. (in Russian).

SYMPOSIA DISCUSSION – PAPER NO: 2

Author's Name: I.I. Lipatov

Discussor's Name: T. Jann

Question:

1) Availability of cited literature?
2) Presented diagrams included in AVT-133 paper 2?

Author's Response:

1) Most in Russian but available and can probably be translated.
2) Yes, apart from one (very recent results).

Discussor's Name: P. Starke

Question:

When using arbitrary Euler-Lagrange method (ALE), did you observe a loss in precision, when the mesh started deforming?

Author's Response:

Yes, that is true, nevertheless this approach looks like the most effective for now. As for accuracy we are trying to use experimental results to verify numerical results.