DOI: 10.13009/EUCASS2019-461

Application of CFD to WIG-craft motion control system design

S. A. Brodsky, A. V. Nebylov and A. I. Panferov
International Institute for Advanced Aerospace Technologies of State University of Aerospace Instrumentation
67, Bolshaya Morskaya, 190000, Saint Petersburg, e-mail: iiaat@aanet.ru

Abstract

The article discusses the possibilities of using the Comsol Multiphysics software package at the stage of preliminary design of a motion control system for a promising WIG-craft. The development of new methods for synthesizing motion control algorithms in the area of the WIG effect is based on the results of a numerical study of the vehicle aerodynamics and the calculation of aerodynamic forces and torques using the CFD computational fluid dynamics module, which expands the capabilities of the COMSOL Multiphysics numerical simulation environment.

1. Introduction

The efficiency of the synthesized control algorithms largely depends on the reliability of the mathematical models of the controlled vehicle. At present, there are no sufficiently reliable results of analytical calculations of aerodynamic forces and torques acting on a winged vehicle as it moves near flat surface (ground-effect), and even more so, near the surface with complex shape. The complexity of piloting an EKP (WIG), determined by the strong dependence of their stability and controllability parameters on the flight speed and altitude above the screen, requires extensive experimental studies using specially equipped wind tunnels. Experimental studies are costly and time consuming.

Significantly reducing the cost of analyzing the inherent stability and controllability of various design schemes, their balancing conditions, at different heights above the screen and flight speeds, will allow the creation of a universal software complex for simulating research in the design of a wind tunnel.

The principal distinctive feature of the ground effect vehicle in comparison with the airplane is the significant non-linear dependence of the aerodynamic forces and torques on the height above the ground. This fact determines the need to use non-linear control laws. Under these conditions, the control system must be sufficiently robust and provide an acceptable quality control with any changes in the parameters of the control object.

The main purpose of our study is to develop the most accurate non-linear computer mathematical model of motion in the area of ground-effect and the development of new methods for synthesis of motion control algorithms. For the development of mathematical model of the vehicle motion, the last modification of the CFD Comsol Multiphysics software package was used.

The ekranoplan is promising in terms of efficiency, but still very difficult to control. Fly at low altitude itself is very dangerous. Rough irregularities of the underlying surface and limitations in height maneuvering require high precision control. It requires accurate and fast sensors, an effective control system and, most importantly, an adequate model of the dynamics of the control object, taking into account all the operating forces and moments. Today, the main problem is still connected with the features of the screen effect and consists in determining the operating aerodynamic forces and moments for all flight modes.

Stationary flight modes are determined by the speed, altitude and angular orientation of the vehicle. Aerodynamic forces and moments in all modes are determined by the distribution of local aerodynamic loads over the entire surface of the vehicle. Aerodynamic controls actuators change the geometry of the aerodynamic surface of the vehicle, propulsion systems with thrust vector control create additional aerodynamic flows of a given speed and direction at the inlet and outlet boundaries, which change the flow parameters in the boundary layer.

The determination of aerodynamic forces and moments for various stationary flight modes is successfully solved by experimental and numerical methods. But each control is another additional dimension in the variable parameter space. The article describes the numerical computational fluid dynamics (CFD) methods we used to find the stationary solutions of the Reynolds-averaged Navier-Stokes equations (RANS) for various models of turbulence and for all admissible combinations of variable parameters.

DOI: 10.13009/EUCASS2019-461

The using the popular software package Comsol Multiphysics allows achieving convergence of the solution for most combinations of parameters. To reduce computation time, methods for organizing batch computations are suggested. The problems of calculating aerodynamic forces and moments and their interpolation on a non-uniform grid for all values of variable parameters are considered.

Calculation of the parameters of aerodynamics in only in a small neighborhood of the balancing modes allows reducing the amount of calculations for non-stationary modes. There is a need to solve the problems of centering and balancing for the complete mathematical model of the WIG dynamics. Issues of creating the model and methods for solving these problems are considered.

2. Preliminary calculation of aerodynamic parameters using the COMSOL Multiphysics software package with CFD module

The development of new methods for synthesizing motion control algorithms in the area of the WIG effect is based on the results of a numerical study of the aerodynamics of an aircraft and the calculation of aerodynamic forces and moments using the CFD computational fluid dynamics module, which expands the capabilities of the COMSOL Multiphysics numerical simulation environment. We use numerical methods of computer fluid dynamics (CFD) to find stationary solutions of the Reynolds-averaged Navier-Stokes equations (RANS) for various models of turbulence and for all admissible combinations of variable parameters.

2.1 Geometry design

The original geometry of the vehicle was created based on available in the public access photo and video materials of the selected prototype, using the method of digitizing individual sections and projections. Small and unprincipled for integral aerodynamics design elements were not taken into account. The design was created directly in the COMSOL graphical editor in order to avoid unpredictable conversion errors.

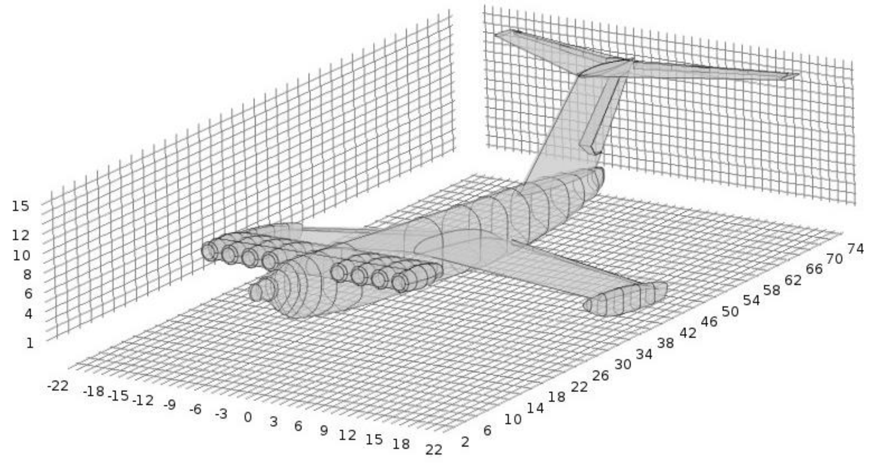

Figure 1: Preliminary WIG-craft geometry

During the experiments, the following parameters were varied: the distance from the lowest point of the vehicle to the ground, horizontal speed, pitch, speed of gas jets flowing from the nozzles of jet engines, the angle of rotation of the thrust vector, the angle of rotation of the aileron flaps and the angle of rotation of the elevator.

Changes in the height and orientation of the aircraft, as well as in the deviation of the aerodynamic surfaces of the controls corresponded to the corresponding computational grids of the flowing airflow. The boundary and initial conditions correspond to the speed of the aircraft, as well as the speed and direction of the gas jets flowing from the nozzles of jet engines.

All possible flight modes and all possible options and methods for controlling the change in the geometry of the structure and the change in the speed and direction of thrust were considered.

The final, optimized geometry of the aerodynamic surface of the aircraft, allows you to create a grid of splitting the surrounding space, a closed gas flow for effective numerical integration of the system of differential equations of gas dynamics by the method of finite volumes. The recursive use of automatic grid generation methods in the CFD module of the COMSOL Multiphysics package is combined with manual partitioning into areas with different types of finite

DOI: 10.13009/EUCASS2019-461

elements and different values of element parameters. Multigrid methods we use to increase accuracy and reduce solution time.

2.2 Computational grid for CFD

The recursive use of automatic grid generation methods in the CFD module of the COMSOL Multiphysics package is combined with manual partitioning into areas with different types of finite elements and different values of element parameters. Multigrid methods were used to increase accuracy and reduce solution time. When building the airflow grid, the volume was divided into zones that differ in the size and shape of the cell.

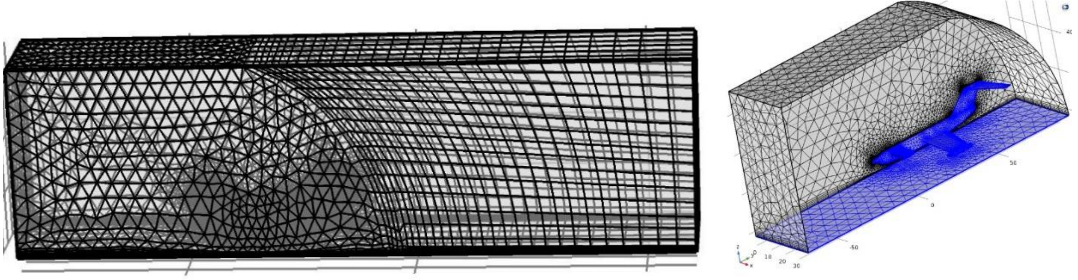

Figure 2: Multigrid with detailed drawing of the boundary layer

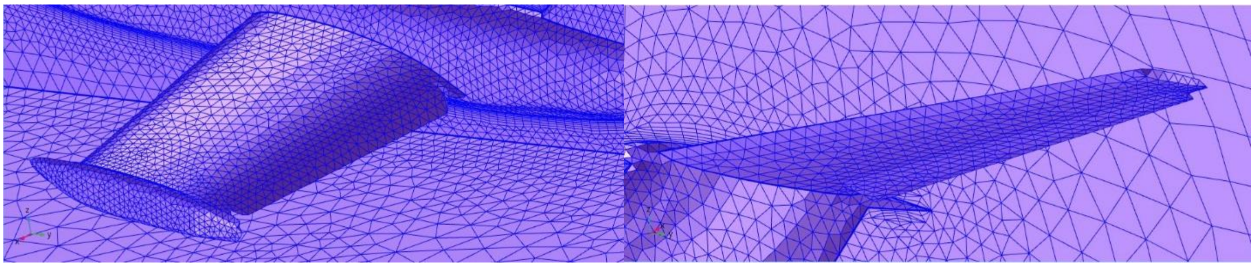

Fig. 3 Features of the grid in the area of articulation of the steering surfaces of the wing and stabilizer

Multigrid methods were used to increase accuracy and reduce solution time. Mainly used free computational grid on tetrahedral finite elements (Free Tetrahedral), border layer grid (Boundary Layers) and swept semi-structured grid (Swept) with different element sizes.

2.3 Models of turbulent aerodynamic flows

The study includes the development of a model of turbulent aerodynamic flows arising from the movement of the aircraft in the area of the screen effect. Calculation of turbulent flow is one of the methods for calculating the Reynolds-averaged Navier-Stokes equations (RANS), where the instantaneous solution parameters in the exact Navier-Stokes equations are divided into averaged and pulsating components.

To simulate turbulent flows in the COMSOL Multiphysics package, several turbulence models are implemented: L-VEL, yPlus, Spalart-Allmaras, *k-ε, k-ω*, low-Reynolds *k-ε* and SST. All these models are available using the Computational Fluid Dynamics module (CFD Module).

The airflow around the WIG-craft is turbulent, which is determined by the Reynolds number for a given body length and input velocity. The Reynolds number for WIG aerodynamics is based on its length *L* and air velocity *u*.

$$Re = \frac{u\,L\,\rho}{\mu} = \frac{100\frac{m}{s}\cdot 73.8\,m\cdot 1.205\frac{kg}{m^3}}{1.82\cdot 10^{-5}\,Pa\cdot s} = 4.886\cdot 10^{8} \qquad \textbf{(1)}$$

For such numbers, the flow is turbulent; when choosing a turbulence model, we limited ourselves to the possibility of using the RANS *k-ε* and *k-w* turbulence models and the Spalart-Allmaras model. For the *k-ε* turbulence model, the averaged Navier-Stokes equations for a compressible perfect gas are the equations of motion for a single-phase fluid are the continuity equation:

DOI: 10.13009/EUCASS2019-461

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

and the momentum equation:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot \left[-p\mathbf{I} + (\mu + \mu_T) \cdot (\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) - \frac{2}{3}(\mu + \mu_T) \cdot (\nabla \cdot \mathbf{u})\mathbf{I} \right] + \mathbf{F} \tag{2}$$

where
- ∇ is the Nabla operator
- ρ is the density (SI unit: kg/m3)
- **u** is the velocity vector (SI unit: m/s)
- p is pressure (SI unit: Pa)
- μ is dynamic viscosity (SI unit: Pa·s),
- τ is the viscous stress tensor (SI unit: Pa)
- **F** is the volume force vector (SI unit: N/m3)

These equations are applicable for incompressible as well as for compressible flow with density and viscosity variations. The turbulence model interface, *k-ε* (spf), is used to simulate single-phase flows at large Reynolds numbers. The physics interface is suitable for incompressible flows, weakly compressible flows, and compressible flows at low Mach numbers. The model introduces two additional transport equations and two dependent variables: the turbulent kinetic energy, *k*, and the turbulent dissipation rate, *ε*. The turbulent viscosity is modeled as

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

where C_μ is a model constant. The transport equation for *k* reads:

$$\rho \frac{\partial k}{\partial t} + \rho(\mathbf{u} \cdot \nabla)k = \nabla \cdot \left[(\mu + \frac{\mu_T}{\sigma_k})\nabla k \right] + P_k - \rho\varepsilon \tag{4}$$

where the production term is

$$P_k = \mu_T \left[\nabla \mathbf{u} : (\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) - \frac{2}{3}(\nabla \cdot \mathbf{u})^2 \right] - \frac{2}{3}\rho k \nabla \cdot \mathbf{u} \tag{5}$$

The operation **":"** denotes a contraction between tensors defined by

$$\mathbf{a} : \mathbf{b} = \sum_n \sum_m a_{nm} b_{nm} \tag{6}$$

The transport equation for ε reads:

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho(\mathbf{u} \cdot \nabla)\varepsilon = \nabla \cdot \left[(\mu + \frac{\mu_T}{\sigma_\varepsilon})\nabla \varepsilon \right] + C_{\varepsilon 1} \frac{\varepsilon}{k} P_k - C_{\varepsilon 2}\rho \frac{\varepsilon^2}{k} \tag{7}$$

The model constants in these equations are determined from experimental data and the values are:

$$\begin{gathered} C_\mu = 0.09 \; C_{\varepsilon 1} = 1.44 \; C_{\varepsilon 2} = 1.92 \\ \sigma_k = 1.0 \; \sigma_\varepsilon = 1.3 \end{gathered} \tag{8}$$

The parameters of the turbulence model are optimized for most of the possible types of flows. In our task, the default parameters of the turbulence model are used in the calculation.

The equations solved using the turbulence model, the *k-ε* interface are the Reynolds-averaged Navier-Stokes equations (RANS) for momentum conservation and the continuity equation for mass conservation. Turbulence effects are modeled using two standard equations for the *k-ε* model with realizable constraints. The flow near the walls is modeled using the boundary function. The turbulent *k-ε* interface can be used for stationary and non-stationary analyzes. The main feature is the addition to the properties of the current material of additional RANS equations and the transport equation for k and ε. When modeling, in addition to the velocity field u and pressures p, the kinetic energy of turbulence k and the energy dissipation rate ε are calculated. Similar systems of partial differential equations for other RANS models are known to specialists and are represented in the interfaces of the CFD module.

DOI: 10.13009/EUCASS2019-461

When choosing a turbulence model, the results convinced us of the feasibility of using the Spalart-Allmaras model as a more algorithmically stable and leading to the same results of a static solution in a shorter calculation time.

The Spalart-Allmaras turbulence model is a one-equation turbulence model designed mainly for aerodynamic applications. It is a low Reynolds number model, that is, it does not utilize wall functions. “Low Reynolds number” refers to the region close to the wall where viscous effects dominate. Compared to the low Reynolds number k-ε model, the Spalart-Allmaras model is generally considered more robust and is often used as a way to obtain an initial solution for more advanced models. It can give reasonable results on relatively coarse meshes for which the low Reynolds number k-ε model does not converge or even diverges.

This module includes the standard version of the Spalart-Allmaras model without the trip term. The model solves for the undamped turbulent kinematic viscosity:

$$\frac{\partial \tilde{\nu}}{\partial t}+(\mathbf{u}\cdot\nabla)\tilde{\nu}=c_{b1}\tilde{S}\,\tilde{\nu}-c_{w1}f_w\left(\frac{\tilde{\nu}}{l_w}\right)^2+\frac{1}{2}\nabla\cdot\left[(\nu+\tilde{\nu})\nabla\tilde{\nu}\right]+\frac{c_{b2}}{\sigma}\nabla\tilde{\nu}\cdot\nabla\tilde{\nu} \qquad \textbf{(9)}$$

The Wall Distance interface calculates the reciprocal distance to selected walls. The value will be small when the object is far away from the respective walls and larger when closer. The exact distance D to the closest wall can be found by solving the Eikonal equation:

$$\left|\nabla D\right|=1 \qquad \textbf{(10)}$$

where $D=0$ on selected walls and $\nabla D\cdot n=0$ on other boundaries. COMSOL Multiphysics solves for a modified version of the Eikonal equation, where the dependent variable is $G=1/D$ and an additional smoothing parameter σ_w is used. This results in the following equation:

$$\nabla G\cdot\nabla G+\sigma_w G(\nabla\cdot\nabla G)=(1+2\sigma_w)G^4,\ l_w=\frac{1}{G}-\frac{l_{ref}}{2} \qquad \textbf{(11)}$$

The resulting wall distance and the direction to the nearest wall are available in COMSOL Multiphysics as predefined variables. Once we add the Wall Distance interface to a model, we simply have to add a Wall boundary condition and select the walls from which we want to calculate the distance.

The model also includes the following auxiliary variables:

$$c_{w1}=\frac{c_{b1}}{\kappa_v^2}+\frac{1+c_{b2}}{\sigma},\ \chi=\frac{\tilde{\nu}}{\nu},\ f_{v1}=\frac{\chi^3}{\chi^3+c_{v1}^3} \qquad \textbf{(12)}$$

$$f_{v2}=1-\frac{\chi}{1+\chi f_{v1}},\ f_w=g\left(\frac{1+c_{w3}^6}{g^6+c_{w3}^6}\right)^{1/6},\ g=r+c_{w2}(r^6-r) \qquad \textbf{(13)}$$

$$r=\min\left(\frac{\tilde{\nu}}{\tilde{S}\kappa_v^2 l_w^2},10\right),\ \tilde{S}=\max\left(\Omega+C_{Rot}\,\min(0,S-\Omega)+\frac{\tilde{\nu}}{\kappa_v^2 l_w^2},0.3\Omega\right) \qquad \textbf{(14)}$$

$$S=\sqrt{2\mathbf{S}:\mathbf{S}},\ \Omega=\sqrt{2\mathbf{\Omega}:\mathbf{\Omega}} \qquad \textbf{(15)}$$

where

$$\mathbf{S}=0.5(\nabla\mathbf{u}+\left(\nabla\mathbf{u}\right)^T),\ \mathbf{\Omega}=0.5(\nabla\mathbf{u}-\left(\nabla\mathbf{u}\right)^T) \qquad \textbf{(16)}$$

are the mean strain rate and mean rotation rate tensors, l_w, is the distance to the closest wall and $\nu=\mu/\rho$ is the kinematic viscosity. The turbulent viscosity is calculated by

$$\mu_T=\rho\tilde{\nu}f_{v1} \qquad \textbf{(17)}$$

The default values for the modeling parameters are:

$$\begin{aligned}&c_{b1}=0.1355\ c_{b2}=0.622\ c_{v1}=7.1\ \sigma=2/3\\&c_{w2}=0.3\ c_{w3}=2\ \kappa_v=0.41\ \ C_{Rot}=2.0\end{aligned} \qquad \textbf{(18)}$$

DOI: 10.13009/EUCASS2019-461

2.4 Wall Boundary Condition

The model of turbulent aerodynamic flows arising from the movement of the aircraft in the area of the screen effect has the following sets of boundary conditions corresponding to the boundary conditions on various surfaces streamlined by air flow.

Boundary conditions "Wall 1". Mathematically, the conditions can be formulated as:

$$\begin{aligned}
&\mathbf{u}\cdot\mathbf{n}=0\\
&\left[(\mu+\mu_T)(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}}-\frac{2}{3}(\mu+\mu_T)\cdot(\nabla\cdot\mathbf{u})\mathbf{I}-\frac{2}{3}\rho k\mathbf{I}\right]\mathbf{n}=-\rho\frac{u_\tau}{u^+}\mathbf{u}_t\\
&\mathbf{u}_t=\mathbf{u}-(\mathbf{u}\cdot\mathbf{n})\,\mathbf{n}\\
&for\ k-e\ model:\ \nabla k\cdot\mathbf{n}=0,\ \varepsilon=\rho\frac{C_\mu k^2}{k_v\,\delta_w^+\mu}\\
&for\ Spalart-Allmaras\ model:\ \tilde{v}=\kappa_v\delta_w u_\tau
\end{aligned} \qquad \mathbf{(19)}$$

Boundaries "Wall 2" and "Symmetry 1" conditions can be formulated as:

$$\begin{aligned}
&\mathbf{u}\cdot\mathbf{n}=0\\
&\mathbf{K}-(\mathbf{K}\cdot\mathbf{n})\mathbf{n}=\mathbf{0},\ \mathbf{K}=\left[(\mu+\mu_T)(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})-\frac{2}{3}(\mu+\mu_T)\cdot(\nabla\cdot\mathbf{u})\mathbf{I}-\frac{2}{3}\rho k\mathbf{I}\right]\mathbf{n}\\
&for\ k-e\ model:\ \nabla k\cdot\mathbf{n}=0,\nabla\varepsilon\cdot\mathbf{n}=0\\
&for\ Spalart-Allmaras\ model:\nabla\tilde{v}\cdot\mathbf{n}=0,\nabla G\cdot\mathbf{n}=0
\end{aligned} \qquad \mathbf{(20)}$$

Boundaries "Inlet" and "Nozzles" conditions can be formulated as:

$$\begin{aligned}
&\mathbf{u}=\mathbf{u_0}\\
&U_{ref}=\|\mathbf{u_0}\|\\
&for\ k-e\ model:\ k=\frac{3}{2}(U_{ref}\,I_T)^2,\varepsilon=C_\mu^{3/4}\frac{k^{3/2}}{L_T}\\
&for\ Spalart-Allmaras\ model:\ \tilde{v}=v_0,\nabla G\cdot\mathbf{n}=0
\end{aligned} \qquad \mathbf{(21)}$$

Boundaries "Outlet 1" and "Intakes" conditions can be formulated as:

$$\begin{aligned}
&\left[-p\mathbf{I}+(\mu+\mu_{\mathbf{T}})\,(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})-\frac{2}{3}(\mu+\mu_T)\cdot(\nabla\cdot\mathbf{u})\mathbf{I}-\frac{2}{3}\rho k\mathbf{I}\right]\mathbf{n}=-p_0\cdot\mathbf{n}\\
&for\ k-e\ model:\ \nabla k\cdot\mathbf{n}=0,\nabla\varepsilon\cdot\mathbf{n}=0\\
&for\ Spalart-Allmaras\ model:\nabla\tilde{v}\cdot\mathbf{n}=0,\nabla G\cdot\mathbf{n}=0
\end{aligned} \qquad \mathbf{(22)}$$

For this module, the turbulent intensity I_T, turbulence length scale L_T, and reference velocity scale U_{ref} values are related to the turbulence variables via

$$k=\frac{3}{2}(U_{ref}\,I_T)^2,\ \varepsilon=\frac{C_\mu^{3/4}}{L_T}\left(\frac{3(U_{ref}\,I_T)^2}{2}\right)^{3/2} \qquad \mathbf{(23)}$$

DOI: 10.13009/EUCASS2019-461

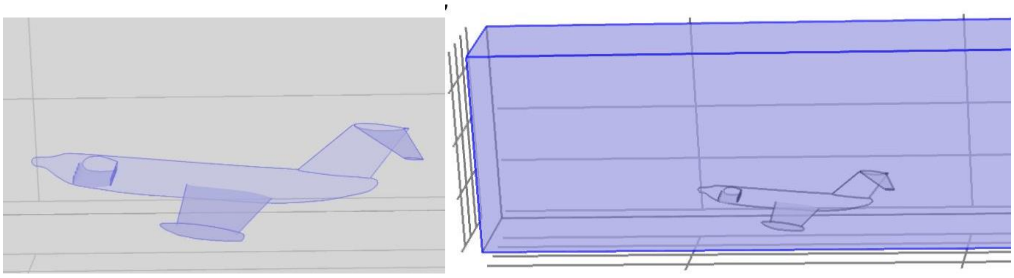

Figure 4: Vehicle Surfaces (Wall 1) and Free Flow Surfaces (Wall 2)

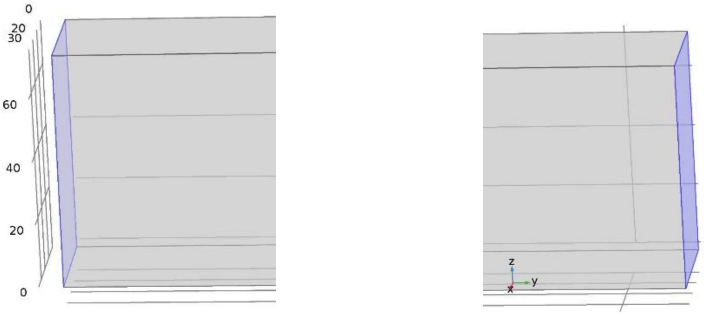

Figure 5: Inlet surface (Inlet) and airflow outlet (Outlet)

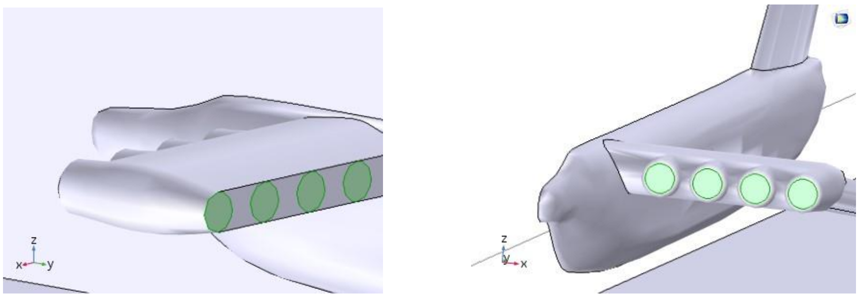

Figure 6: The output surface (Intakes) and the input surface of the air flow at the outlet of the engine (Nozzles)

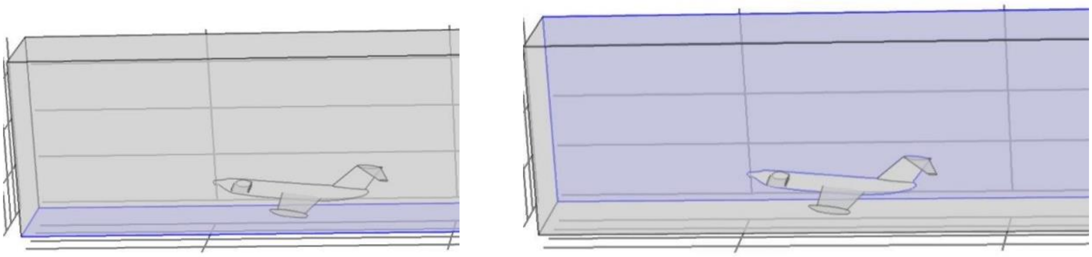

Figure 7: Underlying surface (Wall 2) and the plane of symmetry (Symmetry)

When choosing a turbulence model, the results convinced us of the feasibility of using the Spalart-Allmaras model as a more algorithmically stable and leading to the same results of a static solution in a shorter computation time.

The resulting model of aerodynamics is used in a series of computational experiments designed to calculate aerodynamic forces and torques. Changes in the altitude and orientation of the vehicle, the deviation of the aerodynamic surfaces of controls, with the calculation of the corresponding grid for flowing air flow with boundary and initial conditions determined by the speed of the vehicle, as well as the speed and direction of gas jets emanating from the

DOI: 10.13009/EUCASS2019-461

nozzles of jet engines. In order to reduce the computation time, an artificial increase in the viscosity parameter of the medium is carried out to preliminary search for a stationary solution.

2.5 Numerical experiments

Using the CFD Comsol Multiphysics program, a numerical simulation of the aerodynamics of a hypothetical aircraft with a geometry similar to the selected prototype in the range of the screen effect with the following variable parameters was carried out:

- ground clearance (it turned out to be more convenient for forming the base and subsequent interpolation to use not the height of the center of mass, but the distance from the lowest point of the aircraft to the screen);
- horizontal speed;
- pitch (at certain points for calculating lateral movement we use angles of roll and yaw);
- thrust speed (gas jets flowing from the nozzles of jet engines);
- thrust angle;
- flap angle;
- angle of the elevator.

Changes in the height and orientation of the aircraft, as well as in the deviation of the aerodynamic surfaces of the controls corresponded to the corresponding computational grids of the flowing airflow. The boundary and initial conditions correspond to the speed of the aircraft, as well as the speed and direction of the gas jets flowing from the nozzles of jet engines.

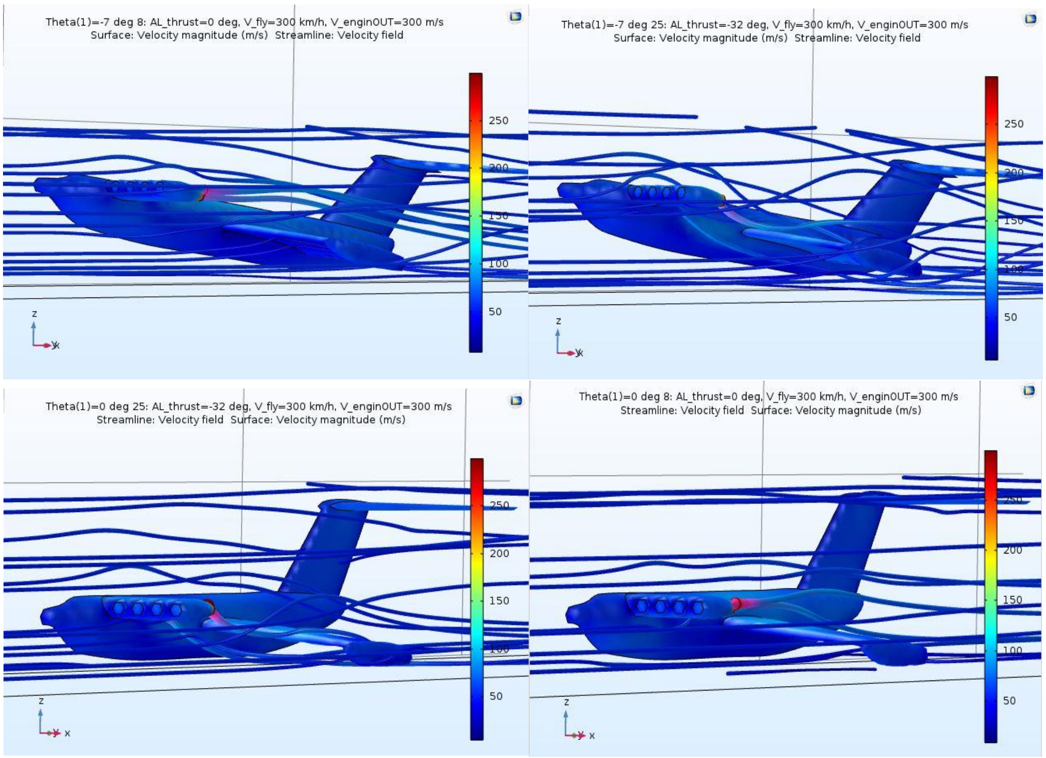

Figure 8: The results of virtual wind tunnel blowing, taking into account the deviation of controls and changes in the force and direction of thrust engines

New original results were obtained by CFD modeling, depending on the angle of rotation and the speed of the outflow of jet jets of the propulsion system. Engine thrust is taken into account when calculating reactive, as well as aerodynamic forces and torques.

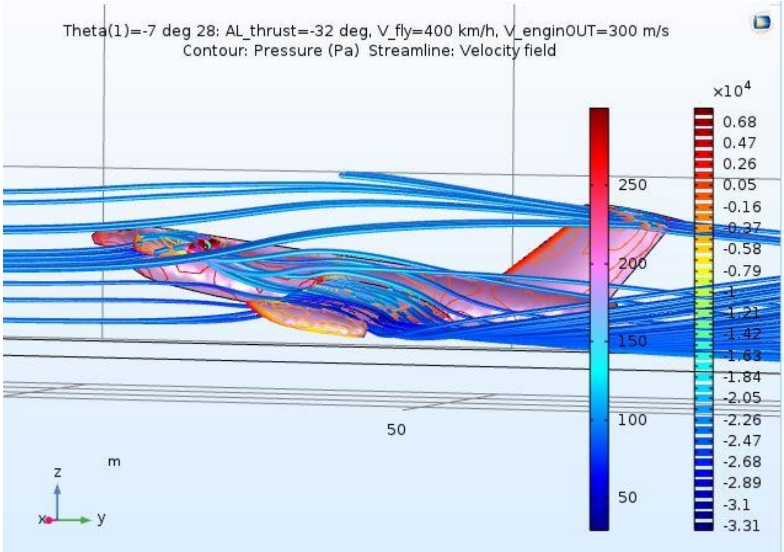

Figure 9: Reactive streams in the boundary layer and pressure distribution on the surface

In the course of numerous computational experiments, a database was created of the results of calculating the integral aerodynamic forces and torques for all admissible flight and control modes. The calculation was carried out by the method of numerical integration of distributed aerodynamic forces over the surfaces of the vehicle. The calculation of aerodynamic forces and moments (integral over the surface) and the coordinates of the center of pressure was carried out according to the following table.

Table 1: Calculation of aerodynamic forces and moments

Parameter name	**Formula**
Drag force Fy only Pressure (N)	intop1(ny*p)
Lift force Fz only Pressure (N)	intop1(nz*p)
Moment of Fz + Fy for (x,y)=0 (J):	intop1((nz*y-ny*z)*p)
Moment of Fz + Fy for (x,y)=Rc.g. (J)	intop1((nz*(y-Y_CG)-ny*(z-Z_CG))*p)
Lift force Fz Total, (N)	intop1(spf.T_stressz)
Drag force Fy Total (N)	intop1(spf.T_stressy)
Lift force Fz only Viscosity (N)	intop1(spf.K_stressz)
Drag force Fy Viscosity (N)	intop1(spf.K_stressy)
The Y coordinate of the pressure center (m)	intop1(nz*y*p)/intop1(nz*p)
The X coordinate of the pressure center (m)	intop1(ny*z*p)/intop1(ny*p)

[a] The designations in the table correspond to the accepted designations in the interface of the CFD COMSOL Multiphysics software module.

The batch mode of calculations with the optimal sequence of the choice of the calculated points was used, allowing the most efficient use of the results of each experiment as initial conditions for the subsequent one. A new original approach to organizing packet computing reduced the total computation time and provided the required number of irregularly distributed computation points for subsequent interpolation.

DOI: 10.13009/EUCASS2019-461

2.6 Features of the organization of batch computing

The settings of the parameters of the algorithm of packet calculations made by us when calculating a series of stationary solutions for turbulent flow (Spalart – Allmaras) for points corresponding to the values of seven variable model parameters turned out to be extremely significant.

We organized batch processing in Batch Sweep mode for four geometry parameters. Deviations were automatically determined by one of the parameters from the central point:

1. Pitch
2. Altitude
3. Flap angle
4. Angle of rotation of the elevator

Nested batch processing in Auxiliary Sweep mode was also organized for three physics parameters:

5. Flight speed (boundary condition of the front wall of the domain).
6. The speed of the jet stream (the boundary condition on the rear section of the nozzles).
7. Angle of rotation of thrust.

We made the following decisions:

1. Batch sweep only for geometrical parameters, to exclude the transfer of initial conditions from the previous step for the modified geometry and to be able to work with individual files to correct the model for errors.
2. Nested Auxiliary sweep for the parameters of physics, the organization of the sequence of iteration of combinations of parameters for all points with a minimum distance between adjacent points.

The transfer of the initial conditions from the previous step contributes to the successful solution of auxiliary problems of computing automation in order to improve the convergence, accuracy and reduce the computation time. However, in the case of a variable geometry error in the determination of the parameters of turbulence, on the contrary, the “segregated solver” is non-reducible according to the parameters of turbulence.

The question arises, how can the initial conditions be transferred more accurately to the boundary layer by transferring them from the surface of the deformed object. That is, to establish the correspondence of points on the surface of the body before and after the deformation or rotation of the element.

We organized a chain of changes of parameters that are independent of the geometry on the basis of the transition with the least changes, without jumps. For example, the first parameter changes from minimum to maximum (“up”). In the first step, the second parameter goes “up” in the second “down”, etc. Then the first parameter goes down, and the second continues to swing. We organized such a “swing” in all four parameters. Unfortunately, these features are not incorporated in COMSOL.

3. Aerodynamics data processing with MATLAB

3.1 Integral aerodynamic parameters interpolation

Using the Matlab program, the obtained experimental data of the virtual "wind tunnel test" was analyzed. A multidimensional interpolation of the results was carried out using an iterative algorithm that calculates penalized least square regression based on a multidimensional discrete cosine transform for automatic smoothing and interpolation of multidimensional incomplete data on a non-uniform grid resulting from the lack of convergence of the aerodynamics equations for the steady state at individual points. The results obtained are stored in the form of multidimensional arrays corresponding to forces and torques. The dimensions of the arrays correspond to the number of variable parameters.

DOI: 10.13009/EUCASS2019-461

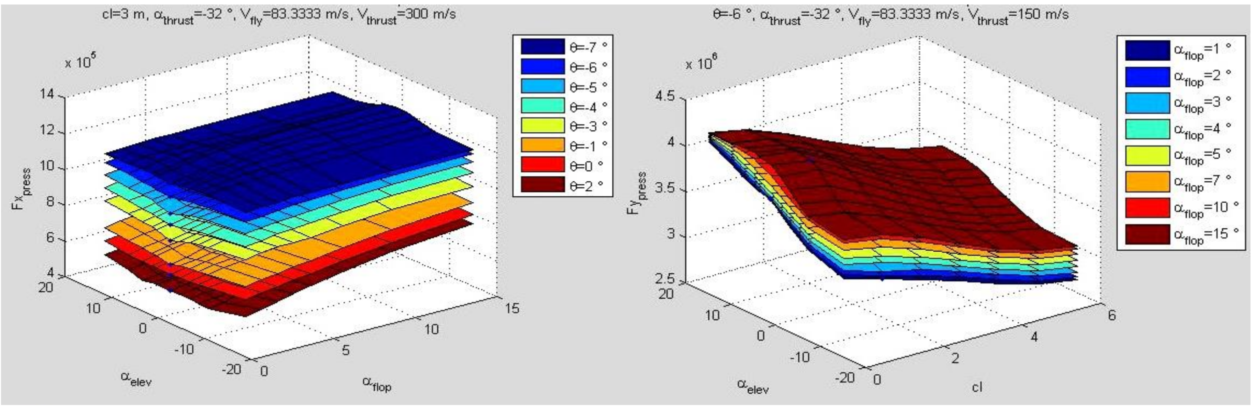

Figure 10: Results of multidimensional interpolation incomplete aerodynamic data on a non-uniform grid (drag and lift forces)

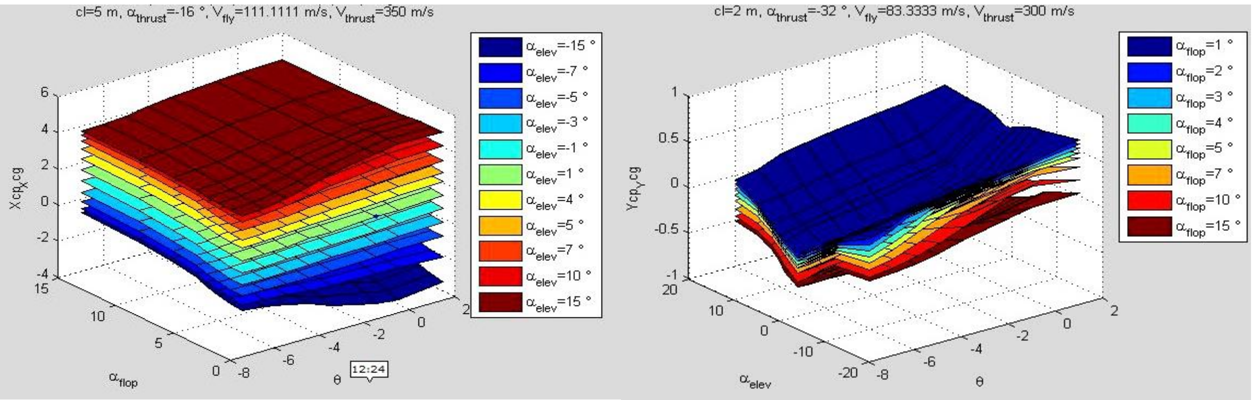

Figure 11: Results of multidimensional interpolation incomplete aerodynamic data on a non-uniform grid (center of pressure coordinates)

A mathematical model of the vehicle dynamics in the “virtual blowing” coordinate system is used, taking into account the action of jet thrust and aerodynamic forces and torques for the following purposes:
-determination of the optimal position of the center of mass of the vehicle to ensure balancing on the maximum range of motion parameters (altitude and speed), taking into account the restrictions on the ranges of variation of parameters of control actions;
-determination of the optimal values of the parameters of speed and direction of thrust, providing balancing with a minimum deviation of the flaps and the neutral position of the elevator on the entire range of motion parameters taking into account the optimum centering of the vehicle;
-determination of the optimal values of the parameters of the speed and direction of thrust, balancing the entire range of flap deviations.

3.2 Mathematical model of dynamics and synthesis of control system

A mathematical model of the dynamics of the aircraft in the area of the ground effect includes the classical model of the dynamics of a solid body, the model of aerodynamics, the model of thrust. The nonlinear model of aerodynamics, presented in the form of multi-dimensional tables of forces and moments, was obtained using the CFD software package Comsol Multiphysics. The model of jet thrust of the engines is determined by the thrust speed and the angle of inclination of the thrust, these parameters determine the acting reactive force and torque and affect the aerodynamic forces and torques that are taken into account in the aerodynamics model.
For all permissible combinations of altitudes above the ground and horizontal speeds, the solution to the problem of balancing was found for values of angle of attack, speed of thrust, angle of thrust and angle of rotation of the flaps, with the elevator in a neutral position, corresponding to the compensation of all forces and torques acting on a solid body.
The resulting mathematical model of aircraft dynamics in the area of the WIG effect includes a model of solid body dynamics, a model of the propulsion system (shown in Fig.12), a model of aerodynamic forces and torques in a tabular form (shown in Fig.13). The solution of the problems of optimization of determining the position of the center of

gravity of the plant model and balancing are used by us in the synthesis of the nonlinear laws of motion vector control and stabilization of ground-effect vehicle.

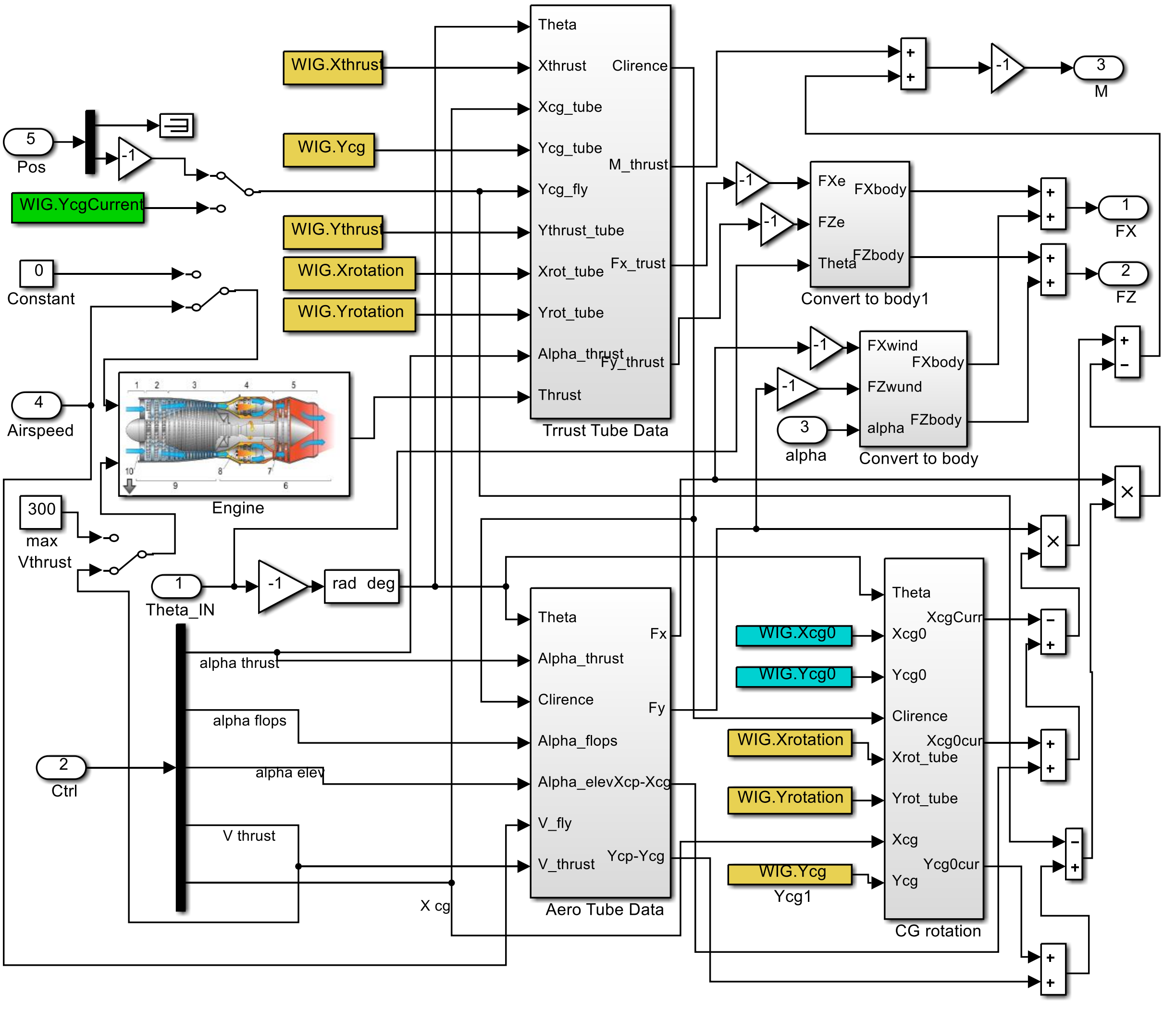

Figure 12: The model of the aerodynamic and reactive forces and moments

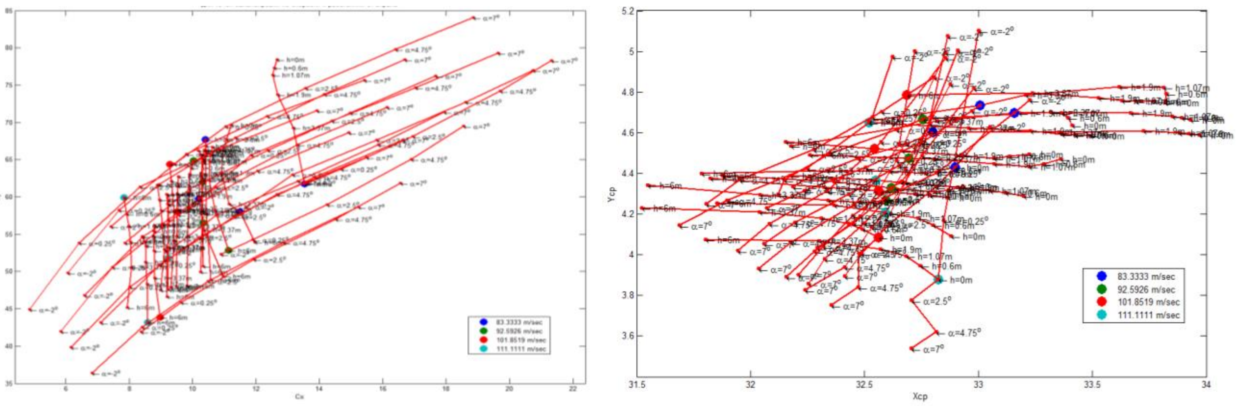

Figure 13: Polar diagram F_L/F_D of the entire wing-in-ground effect vehicle and the position of the point of application of aerodynamic forces Fa for the selected balancing points (h, V) depending on the incidence angle

Solving the problems of balancing and control is sought by mathematical programming methods taking into account restrictions on the ranges of variation of parameters of control actions, such as determining the optimal values of

DOI: 10.13009/EUCASS2019-461

parameters of speed and direction of thrust with a minimum deviation of flaps and elevator on the entire range of variation of motion parameters ensuring the stationary state of the control plant or its stabilization relative to the pilot-specified transition mode.

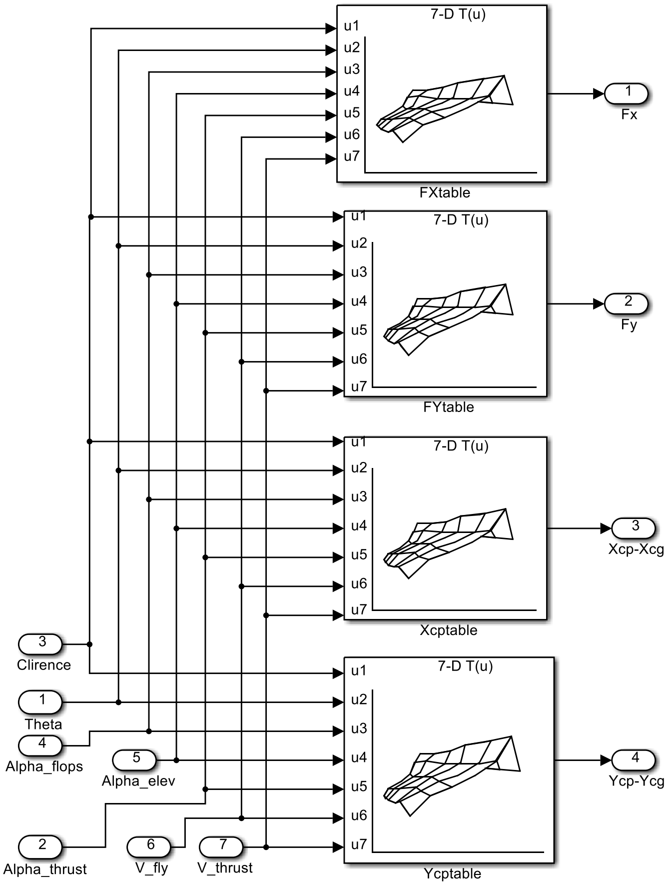

Figure 14: Entering aerodynamic coefficients in the Simulink model

Thus, it becomes possible to synthesize optimal control of a solid along the balancing curve in the parameters of speed and altitude of flight by small increments of forces, providing the required angle of attack with small increments of torques. Oscillations arising in the system are suppressed by a linear stabilization system using the elevator.

The implementation of such a control system is complicated by the discrete representation of the parameters of the model of aerodynamics and the corresponding balancing parameters. The use of cubic splines for n-dimensional interpolation solves this problem at the design stage.

Synthesis taking into account the models of sensors, drives, taking into account harmonic disturbances from the swelling of the underlying surface and the influence of elastic vibrations of the structure is carried out using known techniques.

For such a model of longitudinal motion of the ground-effect vehicle, vector control laws are obtained, which allow one to transfer it from one stationary mode to another with a small vertical speed.

Redundancy in the organization of hierarchical control allows us to reserve control channels for solving a separate problem of stabilizing unstable movement caused by the uncertainty of the dependence of forces and moments on the vertical and angular pitch rates, which are still represented by empirical dependencies based on an incomplete experimental base.

The proposed setting of goals and objectives for determining a nonlinear optimal control of a moving vehicle, transferring the system from one stable state of balancing to any other stable state over the entire allowable range of motion parameters in the area of the ground effect allows to:

- to solve the problem of deterministic control and stabilization;
- to calculate the optimal parameters of control actions, providing a grid of increments of forces and torques in the vicinity of specified balancing points;

DOI: 10.13009/EUCASS2019-461

- conduct a numerical study of the flight trajectories taking into account external disturbances, uncertainties of the parameters of the vehicle model and the control system caused by interpolation of the calculated balancing parameters, as well as taking into account inaccuracies and delay in the execution of commands by the control units.

The methods of synthesis developed by us for automatic motion control system affect all modes of take-off, landing and flight of the vehicle. The resulting models and control laws allow to create a simulator of controlled motion of an ekranoplane, with flight simulator functions.

On the basis of the simulator, it is planned to create a system of short-term predictive interactive control in accelerated time. The hardware implementation of this system as part of the automatic motion control system includes information display system implies the permissible reduction of the system model and control laws.

The system analyzes the actions of the pilot on its impact on the classic, aircraft manual controls, determines the parameters of the probable planned trajectory for the forecast period, calculates the parameters of the nearest allowable trajectory, converting the ground effect vehicle from the balancing safe mode corresponding to the current state to the planned one, and displays it and the limits series of permissible trajectories on the display.

3. Conclusions

Solving the problems of centering and balancing allows us to proceed to the synthesis of nonlinear control laws and stabilization systems that provide the necessary quality of transient processes, within the optimization of automatic motion control systems using a dynamic airbag (ekranoplan and aeroboat), synthesis of control algorithms for seaplane during takeoff and landing in flight mode by terrain, etc.

Preliminary calculations for individual flight modes have already shown that for the WIG-craft that does not have its own stability, a new and original approach to the synthesis of a control system is required for creating a multidimensional control system with restrictions on control parameters.

In the course of the research the following results were obtained.

– Implemented the mathematical aerodynamics model of a generalized (hypothetical) vehicle of a given geometry in the software package CFD “Comsol Multiphysics”.

Obtained a series of numerical stationary solutions of aerodynamic flows for predetermined ranges of change of flight parameters, and predetermined ranges of controlled variables of the aerodynamics model of the object, allowing to form a multidimensional array of aerodynamic parameters of the model.

– Obtained and implemented the mathematical model of dynamics of the controlled object was, which allows solving the problems of balancing and optimal alignment of the controlled object.

– Resolved the problem of determining the optimal position of the centre of gravity of the object model, providing the possibility of balancing on the maximum ranges of change of the parameters of the motion of an object in the zone of ground effect action with a minimum deviation of controls, to the corresponding maximum aerodynamic quality.

– Resolved the problems of conditional optimization of control parameters values, corresponding to the balancing points as a solution to the problem of mathematical programming with restrictions on controlled variables corresponding to the ranges of change in parameters of controls.

– Obtained the optimal physically realizable laws of control parameter changes, corresponding to safe manoeuvres of the vehicle in the zone of the ground effect action.

– Resolved the problems of stabilization of the motion of the vehicle at stationary points, as well as the problems of the finite control and stabilization of the motion of the wing object, as a problem of optimal transition from one steady state to another with optimal stabilization of deviations from the chosen trajectory in the state space.

The solutions obtained are saved in form of multidimensional arrays of parameters corresponding to the developed structure of the control system.

Methods and algorithms developed to resolve the tasks, formed the basis of the software package being created, designed to resolve practical problems of designing an vehicle control system of arbitrary design in the zone of ground effect action taking into account the uncertainties of the parameters of the mathematical model, as well as taking into account the inaccuracies and delay in the execution of commands by the control elements.

It should be noted that the approach of using the dependence of aerodynamic forces and moments on the pitch angle to be erroneous when analyzing the stability of a ground-effect vehicle by applying them to the angle of attack. This approach does not reflect the specifics of the screen effect and leads to erroneous results reducing the real values of the compensating forces and moments.

3. Acknowledgments

The study was supported mainly by the Russian Foundation for Basic Research under the project No. 18-08-00234. The model of WIG-craft was designed with the support of the Russian Science Foundation, the project 19-19-00103.

DOI: 10.13009/EUCASS2019-461

References

[1] Nebylov Alexander, Joseph Watson, Editors. Aerospace Navigation Systems.// John Wiley & Sons, Inc. UK/ - 2016. - 371 pp.

[2] Nebylov A.V, Wilson Ph.. Ekranoplane - Controlled Flight close to Sea. Monograph. WIT-Press/Computational Mechanics Publications, Southampton, UK, 2001. 300 p.

[3] Nebylov A.V. Structural optimization of motion control system close to the rough sea. 13th IFAC World Congress, San Francisco, USA. Elsevier Ltd, Oxford, 1996, Vol.Q p. 385-394.

[4] Nebylov, A.V. Ensuring control accuracy. Springer-Verlag, Heidelberg, Germany. 2004.

[5] Nebylov, A.V. Editor. Aerospace Sensors. Momentum Press, NY, USA. 2013, 350 p.

[6] Hahn T. et al. Analysis of Wing-in-Ground-Effect Vehicle with regard to Safety Ensuring Control. Proceedings of the 19th IFAC World Congress. Cape Town, South Africa. 2014.

[7] Nebylov A.V., Nebylov. V.A. Controlled WIG flight concept. 19th IFAC World Congress. Capetown, 2014.

[8] Knyazhsky, A.Y., Nebylov, A.V., Nebylov, V.A. Increase in the aerodynamic quality of ground effect vehicle due to the big waves turning around, CYBERNETICS AND PHYSICS, Vol. 6, No. 2, 2017, p.71-75.

[9] Benzerrouk Hamza , Nebylov Alexander, Nebylov Vladimir Interactive Multiple Model Target Tracking Based on Seventh-Degree Spherical Simplex-Radial Cubature Information Filter / IFAC Aerospace Controls TC Workshop Networked & Autonomous Air & Space Systems (NAASS 2018) 13 – 15 June 2018, Santa Fe, NM USA.

[10] Nebylov A.V., Nebylov V.A. Metrology problems of WIG-craft motion control /5th International Workshop of Metrology in Aerospace (MetroAeroSpace), 2018, Rome, Italy.