Advances in Systems Science and Application (2015) Vol.15 No.4 338-350

Simulation Modelling for Computer Aided Design of Secondary Aerodynamic Wing Surfaces

Aleksandr A. Gorbunov, Aleksej D. Pripadchev, Irina S. Bykova and Valerij V. Elagin

Federal State Educational Government-financed Institution of Higher Professional Education, Orenburg State University, Orenburg region, Russia

Abstract

The simulation modelling technique of secondary aerodynamic wing surface of the long-haul aircraft using a high-precision mathematical simulation in the aerodynamics and hydrodynamics computer programme has been formulated in the present article. The method proposed is based on the modern methods in the field of computer-aided design of aircraft using CATIA three-dimensional simulation and simulation modelling in SALOME environment. The use of high-precision mathematical simulation in computer-aided design of secondary aerodynamic wing surfaces allows us to determine the aerodynamic characteristics of the secondary aerodynamic surface of the model developed and to verify the previous results of the project definition.

Keywords Secondary aerodynamic surfaces; High-precision computer mathematical simulation; Simulation model; A long-haul aircraft; Synthesis; Design automation; Project procedures; Three-dimensional modelling

1 Introduction

Building the new long-haul aircraft (A/C) with improved performance characteristics and enhancing the currently used aircraft is effected in various ways. One of them is improving its aerodynamics depending largely on the A/C look. In our view, the most rational way to improve the aerodynamic characteristics of the A/C is to install the secondary aerodynamic surfaces (SAS) at the wingtip. The use of SAS reduces the induced drag of the aircraft, enhances the effective wing aspect-ratio and ascensional power at the wingtip, improves A/C longitudinal/transverse stability, reduces specific fuel consumption, cuts takeoff run and landing roll of the aircraft. Currently, there are many wing SAS designs installed at the long-haul A/C differing in geometrical and aerodynamic characteristics [1-3]. However, despite years of experience based on the empirical approach, there is no uniform method and engineering tools of computer-aided design of SAS both individually and as a part of the wing in the available scientific matter. In this regard, development of the scientific foundations and the method of computer-aided designing of SAS wing as a component of the simulation modelling technique is a priority task.

SAS designing involves numerous aerodynamic, energetic, structural and geometrical, technological and operating characteristics. Thus, modern computer technologies are to be used for the analysis, synthesis and appropriate design solutions. In this connection, the design and construction of SAS is reasonable to do using a high-precision mathematical simulation in the computer programs of computational aero- and fluid dynamics (CFD).

2 Methods

The purpose of this research is to develop methods of simulation modelling of the designed SAS of the wing of the long-haul aircraft using the latest methods of high-precision mathematical simulation and engineering analysis. The object of the research is simulation modelling of aircraft SAS of the wing within the proposed method of simulation. The subject of the research is the methodology of high-precision mathematical simulation of aircraft SAS wing. The methodological provision of the study is in using the latest methods of mathematical simulation in the computer programs of computational aero- and fluid dynamics.

Due to the fact that designing and construction of the SAS wing is a complex and modifying with time process producing a large number of iterations, there is a necessity to describe rationally the design process with simulation and physical models. We shall consider the process of simulation modelling of the secondary aerodynamic wing surface of the long-haul aircraft. Simulation modelling means a method of a high-precision mathematical simulation in the computational aero- and hydrodynamics programme [4-6]. Currently, to solve CFD problems, several software products can be used, for example:

- COSMOSFloWorks;
- NAFEMS EFD. Lab;
- EFD.V5 forCATIAV5;
- SALOME;
- ANSYS.

We propose SALOME product to be used. It is intended for solving problems of computational aero- and hydrodynamics. This is an open integrated software platform for numerical computations and simulation modelling. The SALOME user interface of the computational aero- and hydrodynamics provides the following options:

- it sets the master data and displays the results immediately in the graphic design window;
- it has a user friendly interface;
- it takes minimal time to prepare the data and view the results of the experiment.

CAD used is a finite-element pre-post processor, being the computational environment kernel surrounded by the united multitude of CAE solvers. With

SALOME, one can also develop proprietory software solutions, which has been done by us, program for investigation and calculation of aircraft aerodynamic characteristics being the resulting product [7]. This programme based on CAE solver Open FOAM (liquids and gases flow analysis) helped us to carry out simulation of SAS of the long-haul aircraft. The revised Open Foam package uses Parasolid geometric simulation kernel which is a platform for the latest CAD such as: ANSYS, Icem-CFD, Femap, Solid Works having full technical support.

2.1 Discussion

Development of the wing SAS 3D model was performed in CATIA high-level CAD system. Use of CATIA CAD is conditioned by the fact that the system possesses a hybrid design function, combining both surface and solid elements in a single model. Another important thing is the fact that the system has a function of free parameterization and constructs models with the help of the earlier created drawings, Fig.1.

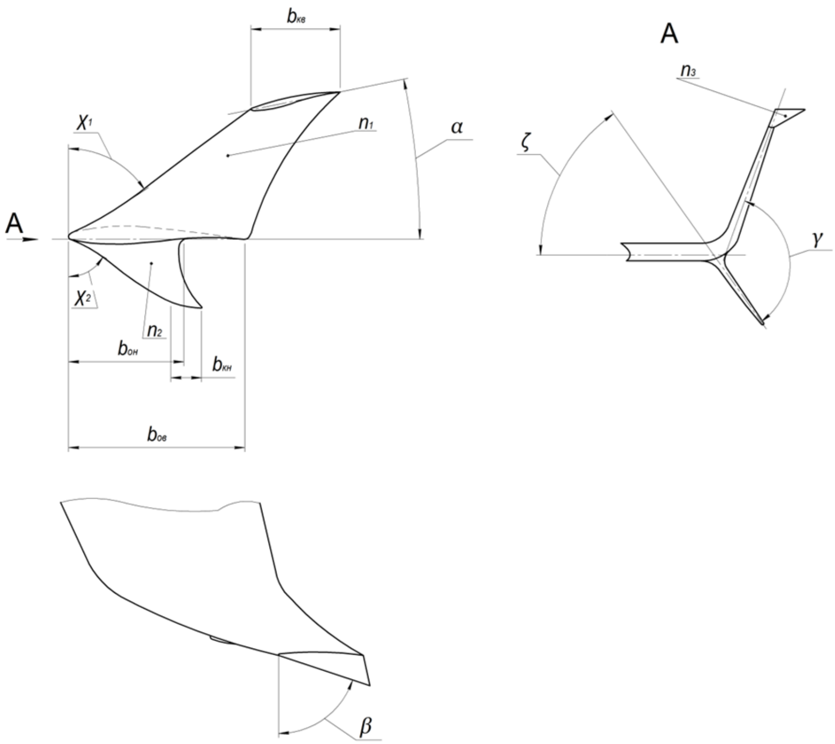


Fig.1 General View of SAS, Patent No.2481242

n_i - the number of working surfaces; χ_1 - leading edge sweep angle of the upper surface; χ_2 - leading edge sweep angle of the lower surface; b_k - tip chord of the upper surface; b_0 - root chord of the upper surface; b_k - tip chord of the lower surface; b_0 - root chord of the lower surface; γ - angle between the working surfaces; α - installation angle end plate relative to the end rib axis; β- leading edge sweep angle of the end plate.

Based on the previously developed SAS drawing, its 3D model was created, Fig.2. The 3D model developed was converted into STEP format to support to carry out simulation modelling in CAD SALOME programme. STEP format makes it possible to describe the aerodynamic surfaces obtained using mathematical model of CAD programme. STEP data format selection was done according to the requirements of the software used for simulation modelling.

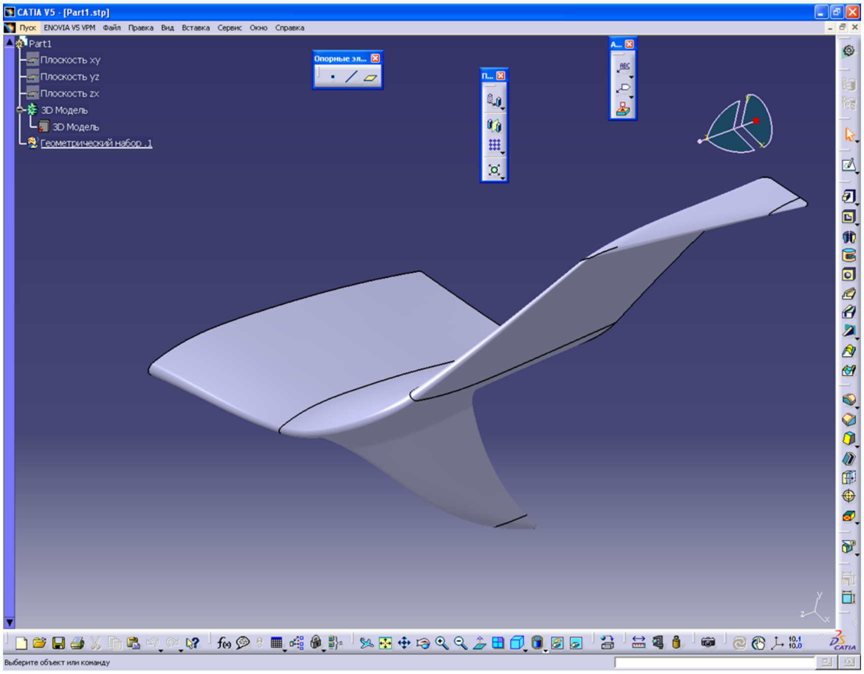
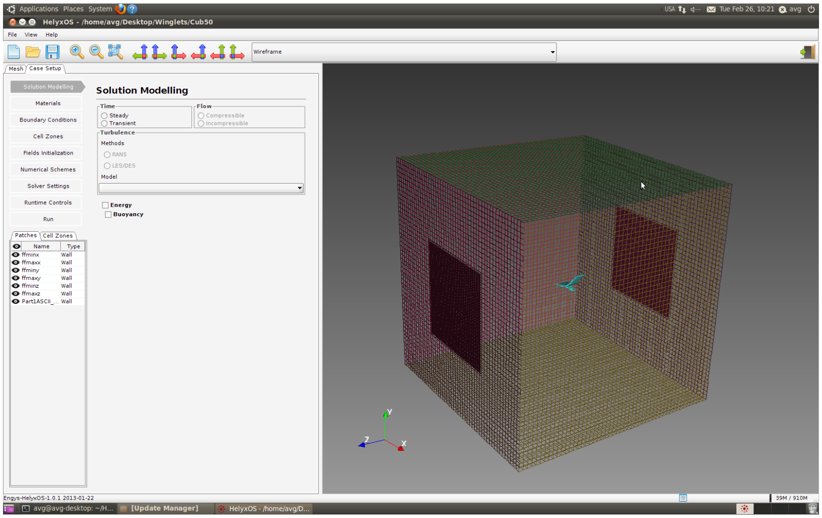


Fig.2 3D SAS model of the wing in CATIA system

The figure shows that all the coupling parts of SAS, namely the interface between the working surfaces have smooth fairings with the given radii. Obtaining accurate aerodynamic shapes has become possible owing to the high-level CATIA CAD system. In the dialogue box, a SAS 3D model is mounted on the main wing of the long-haul aircraft on the left side in the flight direction [8].

The problem of computational aero- and hydrodynamics being solved leads to the analysis of the impact of air on the body. To make calculations it is necessary to set the initial data in the form of the initial and boundary conditions. The body is restrained by the surfaces which limit the range of the fluid. Such conditions are called a Wall. Due to the fact that for SAS the case of external flow is used, which is characterized by such parameters as speed, pressure, temperature, etc., this type is called Ambient Conditions (Environmental Conditions). In accordance with the specified conditions, the External type of task will be solved. As a result, the design area is limited on one side by the field of the set boundaries in the form of the walls forming a rectangular hollow parallelepiped and solid SAS model, placed in a hollow parallelepiped. The size of the design area is set either automatically or manually by the user. In this case, the design grid had been done by the user, which allowed us to obtain a more accurate mathematical problem solution by means of a smaller calculation grid in the design area, Fig.3.

The design area is divided by the grid with a different step: a smaller one immediately in the design area for more accurate solutions, further with a larger step as the high accuracy of the calculation is not required for the remaining area. The design area construction method used results in the efficient use of the computation resources leading to the significant reduction of the computation time [9, 10]. Fig.3 shows the view of a 3D SAS model on the right located in the design area.



○ Steady
○ Transient
○ Compressible
○ Incompressible
○ RANS
○ LES/DES
☐ Energy
☐ Buoyancy

Name	Type
ffminx	Wall
ffmaxx	Wall
ffminy	Wall
ffmaxy	Wall
ffminz	Wall
ffmaxz	Wall
Part1ASCII_...	Wall

Fig.3 Design area

Thus, the design area shown in Fig.3 is a spatial cube, which contains a 3D SAS model of the wing of the long-haul aircraft, in what connection, boundary conditions Wall in the form of four faces on axes OY and OX are set around the model, and the initial conditions graphically displayed in the form of two faces on axis OZ are set at the front and at the back of the 3D model. The boundary and initial conditions are set in accordance with the conditions, which SAS in natural scale will be found under. The initial condition in the form of the face, located in front of the SAS in the program is referred to as (Inlet), and a face located behind the SAS is called (Outlet) i.e. an output parameter. Setting Inlet and Outlet conditions, we thereby define the mode of the fluid flow in the design area. To eliminate the influence of the boundary conditions Wall on the model SAS to neutralize vortex formation around Wall faces, the movement of the flow rate in the direction of the fluid has been set. The parameters of the fluid are set by the following initial data given in Table 1.

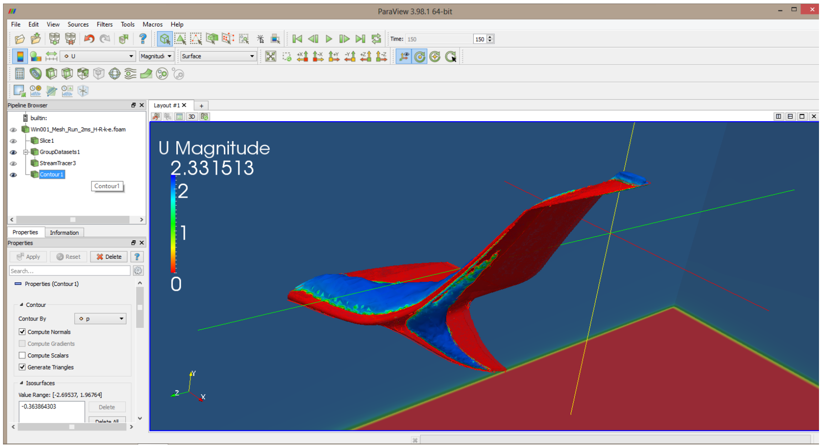
In addition to the parameters listed in Table 1 and initial conditions, we set the turbulence model which suits in the best way to the current design case and select the model of turbulence (k-e), where the calculation of two additional equations for kinetic energy turbulence transport and turbulence dissipation transport. Other turbulence models implemented in the program can be used, but in this design case it is advisable to use the very (k-e) model [11]. All further calculations will be performed for the case when the body is at rest while the medium is moving. The selected design case is applicable for finding ascensional power, drag force, induced drag and fluid force acting on the body located therein.

Table 1 Initial data for the software experiment implementation

Item No	Name	Unit of Measurement	Value
1	Name	-	air
2	Density	kg/v^3	1.205
3	Dynamic Viscosity	$Pa{\cdot}s$	0.000019137
4	Specific Heat Capacity	$l/kg\cdot$	1.006
5	Kinematic Viscosity	m^2/s	0.0000158813
6	Laminar Number	-	0.9
7	Turbulent Prandtl Number	-	0.85
8	Thermal Conductivity	$W/m\cdot$	0.024
9	Reference (absolute) Pressure	Pa	101.325
10	Thermal Expansion Coefficient	k^1	0.00333
11	Reference Temperature	K	300

The problem of computational aero- and hydrodynamics for SAS wing being solved is made to study the movement of air around the body. Solving the problem using CAD gives us the parameters describing the fluid flow, namely: speed, pressure, temperature, etc. Simulation modelling of SAS of the A/C wing in CFD in SALOME programme has been performed within the time equal to 200 calculation steps. The number of the calculated steps is set by the user and selected in the view of the following consideration: to obtain the sufficient experimental data under the steady flow process.

Eventually, repeatedly conducted purges in CFD programme allowed us to establish that the steady flow process starts with the 65th design step when the interference processes become steady, and the number of iterations of 200 design steps selected from the obtained information volume of 250 GB. A larger volume of information is not required, since the available data is sufficient for making design decisions.



Properties (Contour1)

Contour

Contour By: p

☑ Compute Normals

☐ Compute Gradients

☐ Compute Scalars

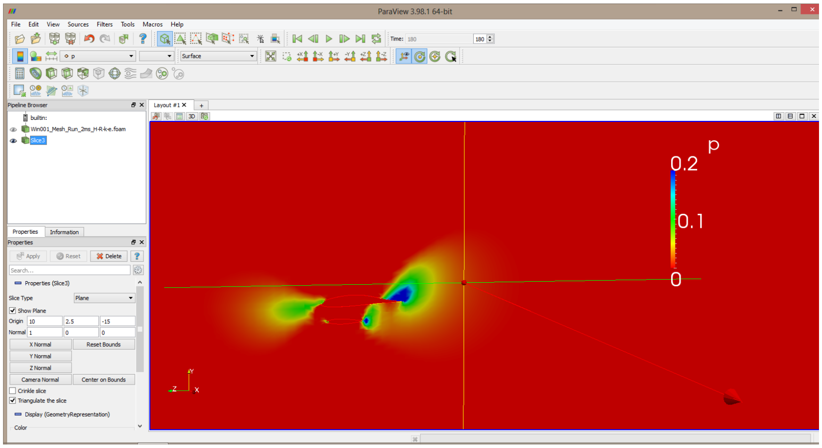
☑ Generate Triangles

Isosurfaces

Value Range: [-2.69537, 1.96764]

-0.363864303

Fig.4 Fluid distribution rate change over SAS



Properties (Slice3)

Slice Type: Plane

☑ Show Plane

Origin: 10, 2.5, -15

Normal: 1, 0, 0

☐ Crinkle slice

☑ Triangulate the slice

Fig.5 Pressure distribution in SAS section

The results of simulation modelling presented further feature the same moment of time corresponding to design step No.180 in various design sections. Fig.4 shows SAS of the wing with the air distribution rate change in the form of relief growths that change their colour with the distribution rate over SAS. Flow distribution value is found using the graduated scale changing its colour from red to blue and graph curves shown below. The field of low rates is marked with the red colour and the field of high rates is marked with the blue colour.

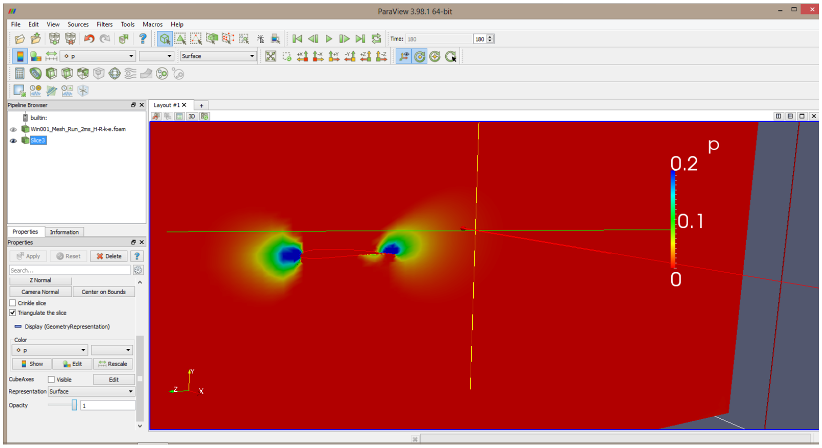


Fig.6 Pressure distribution in the wingtip section.

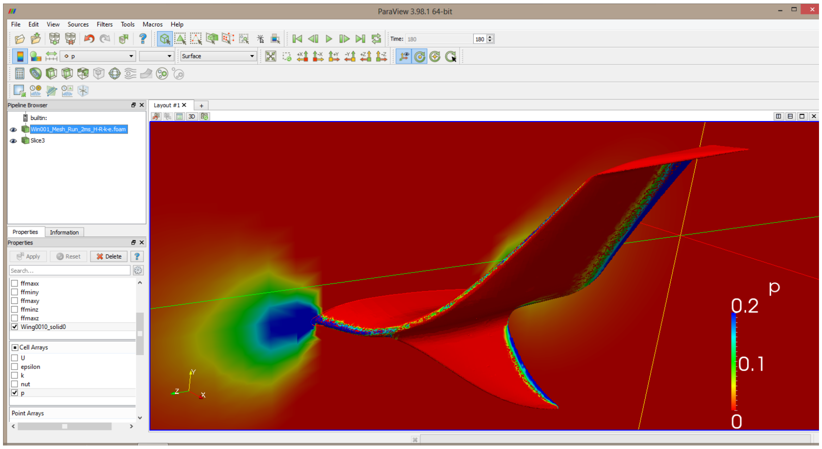


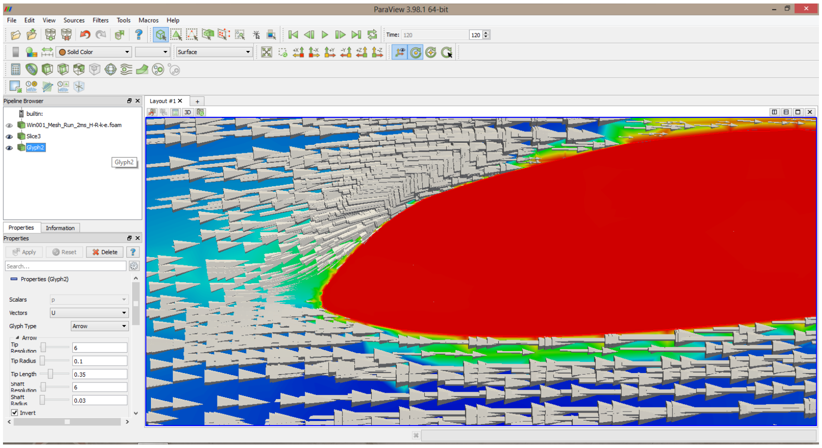
Fig.7 Pressure distribution over SAS of the wing

Fig.5 shows SAS section using which we can find the change in pressure over the SAS. This section clearly shows pressure rise at the leading edge of SAS and also a marked increase at the trailing edge, and further decrease due to the distance from the model owing to air circulation.

Fig.6 presents a section of the wingtip, which shows pressure distribution over the surface of the wing airfoil located close to SAS.

Fig.7 is a 3D SAS model showing the pressure distribution fields and the set cutting plane at the wingtip shown by the section in Fig.6. The pressure distribution over SAS makes it possible to trace the character of pressure change over time.

Fig.8 shows the enlarged section of the wingtip close to SAS displaying air particles vector movement and rate distribution fields.



Time: 120

builtin:
Win001_Mesh_Run_2ms_H-R-k-e.foam
Slice3
Glyph2

Properties (Glyph2)

Scalars p
Vectors U
Glyph Type Arrow
Arrow
Tip Resolution 6
Tip Radius 0.1
Tip Length 0.35
Shaft Resolution 6
Shaft Radius 0.03
☑ Invert

Fig.8 Pressure distribution over SAS of the wing

A graphic curve in the form of the bar graph in Fig.9 shows the change in the Reynolds Number (Re) and kinematic speed over SAS conforming to Fig.4.

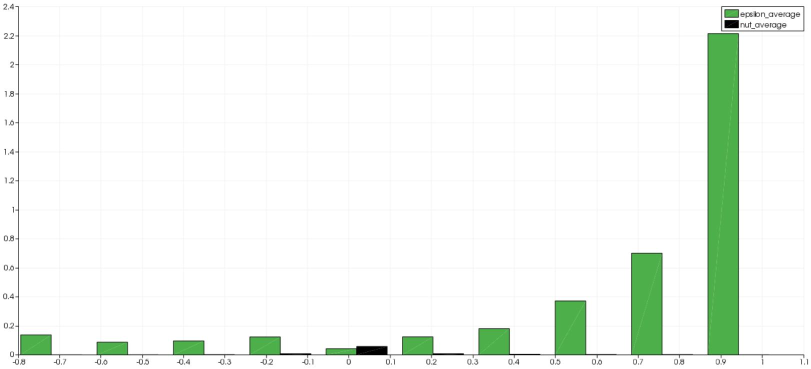


Fig.9 Graphic curve of Re number change and kinematic speed over SAS

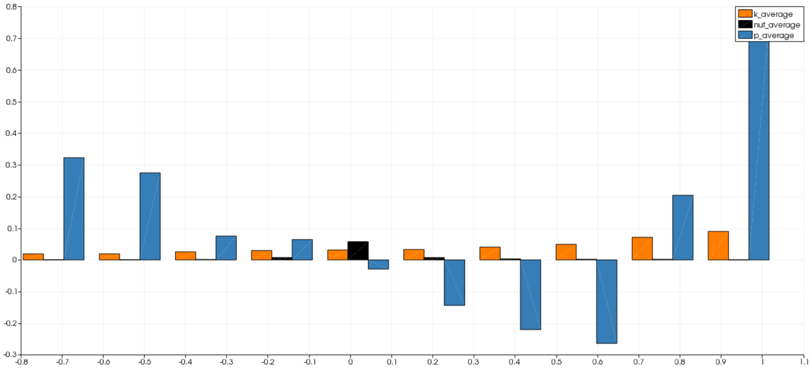
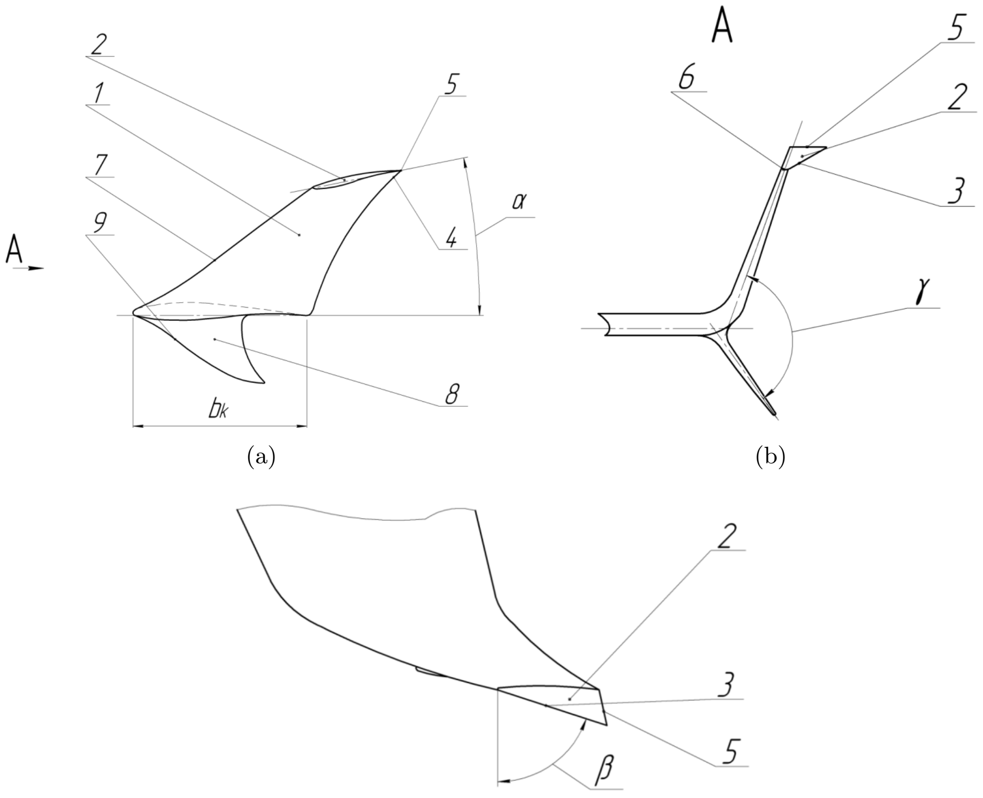


Fig.10 A curve of a temperature change, kinematic speed and pressure change over SAS



(a) (b)

(c)

Fig.11 Secondary aerodynamic surface, Patent No. 2481242

Fig.10 shows a curve describing air parameters, namely, a temperature change, kinematic speed and pressure change over SAS conforming to Fig.4.

Simulation modelling for SAS wing made it possible to find the aerodynamic characteristics of the designed SAS and to confirm flow processes around SAS. Let us consider the flow for SAS wing, Patent No.2481242. While the air is flowing the wing, the air overflows from the lower plane of the wing to the upper one, at the same time, at the endplate 1 supplied with the additional wind swept surface 2 of low aspect with a sharp front edge 3, mounted on the outer side of the endplate 1, there formed a vertical chamfering field transforming into a stable vortex flow with a conical vortex formation on the leading edge 3 of the secondary aerodynamic surface 2 mounted on the endplate 1, Fig 11. At lower vertical aerodynamic surface 8, the vertical chamfering field due to the low aspect the lower surface, does not result in the early formation of the vortex at the leading edge 9 but it is transformed at the surface end into an end conical vortex.

3 Results

The SAS proposed improves aerodynamic efficiency of the long-haul A/C and provides a maximum effect from air overflowing throughout the entire field of the effective values [12, 13]. In this way, it makes it possible to enlarge the effective wingspan reducing the induced drag generated by the vortex flying from the sweptwing tip and, consequently, increasing the ascensional power at the wingtip; to increase the effective aspect of the wing, without changing its span; to improve fuel efficiency of the A/C and the flying range.

4 Conclusions

1. High-precision computer mathematical simulation and simulation modelling techniques in the analysis of the flow process around the designed SAS model provides a means to find out the aerodynamic characteristics for given initial and final conditions affecting the induced drag of the aircraft and ascensional power generated by the system of lifting surface areas of the A/C.

2. Using a simulation modelling technique can be used for complex multivariate iterative calculations of the new SAS designs, high quality design solutions being secured due to the accumulation of output data followed by statistical analysis with the time from the previous to the next point in time set with the predefined step.

3. The behaviour of the flow processes around the SAS model designed identified as a result of the high-precision mathematical simulation modeling enables us to confirm or deny (in the case of unsound designing) aerodynamic characteristics obtained at the preceding stages and to develop physical SAS models.

Acknowledgement

The project has been performed under Contract No.14.Z56.15.5527-MK of February 16, 2015, Grant of the President of the Russian Federation for the state support of young Russian scientists to carry out research on the subject of "Computer-aided design of secondary aerodynamic wing surfaces as an element of the aircraft life cycle".

The outcomes of the scientific research have been accepted for implementation in JSC. KAPO TUPOLEV (Kazan).

References

[1] V.A. Komarov et al (2013), Conceptual design of the aircraft, House of Samara State Aerospace University, Samara. pp.120.

[2] LLC Novaya Tekhnika Enterprise(2013), Ontology of design, Novaya Tekhnika Publishing,Samara. No.1.

[3] Daniel P. Raymer.,(1992), Aircraft Design: A Conceptual Approach, American Institute of Aeronautics and Astronautics, Washington. pp.391.

[4] Shannon, R. (1978),"Systems simulation modelling", *Art and Science*, pp.268.

[5] Pavlovsky, Yu.N. (2000), Simulation Models and Systems, FASIS: Exhibition Centre of Russian Academy of Sciences, pp.134.

[6] Clayton Bargsten Winglets. (2011), Striving for Wingtip Efficiency, National Aeronautics and Space Administration, Washington DC: UNCL, pp.68.

[7] A.V. Gordiyenko, A.A. Gorbunov,A.D. Pripadche. (2013), "Program for research and calculation of the aircraft aerodynamic characteristics: a certificate of the official registration of the software", Certificate No. 2013616240 the Russian Federation.Applicant and Patent holder Orenburg State University. No. 2013616240.

[8] A.A. Gorbunov,A.D. Pripadchev (RF).(2013), "Aircraft wingtip", Patent RU No. 2481242, ECC 643/10.No. 2011148436.

[9] Jeppe Johansen and Niels N. Sorensen, (2006), "Aerodynamic Investigation of Winglets on Wind Turbine Blades Using CFD", Roskilde, Denmark: Riso R-1543.

[10] Hepperle, M.(2005),"Aerodynamic Optimisation of a Flying Wing Transport Aircraft", *New Results in Numerical and Experimental Fluid Mechanics V*

(Notes on Numerical Fluid Mechanics and Multidisciplinary Design), Vol.92, pp.69-76

[11] G. J. Kennedy.(2012), “Aerostructural analysis and design optimization of composite aircraft”, *PhD thesis*, University of Toronto.

[12] SCHIKTANZ, Daniel.(2011), “Conceptual Design of a Medium Range Box Wing Aircraft”. Department Fahrzeug technik und Flugzeugbau, Master Thesis, Hamburg, HAW Hamburg.

[13] Joseph Katz, Allen Plotkin.(1991), “From Wing Theory to Panel Methods”, *Low Speed Aerodynamics*, Singapore, pp.632.

Corresponding author
Tatiana can be contacted at:yal05@mail.ru