

The Hydrodynamic Characteristics Calculus for unmanned underwater vehicle wing using Solid works Flow Simulation Module

BALANDIN KIRILL

Abstract: - The objective of the application is to determine, through Solid Works Flow Simulation module, the drag coefficient and the lift coefficient of the unmanned underwater vehicle wing profile, for various values of the incidence attack angle α and various length of wing. The wing profile is immersed in a uniform air stream, oriented perpendicular to the stream. The velocity of the water stream is $V=2$ m/s and water density is $\rho=998.2$ kg/m³. Finally, the results predicted by 3D simulation will be compared with each other in order to determine more suitable length of wing.

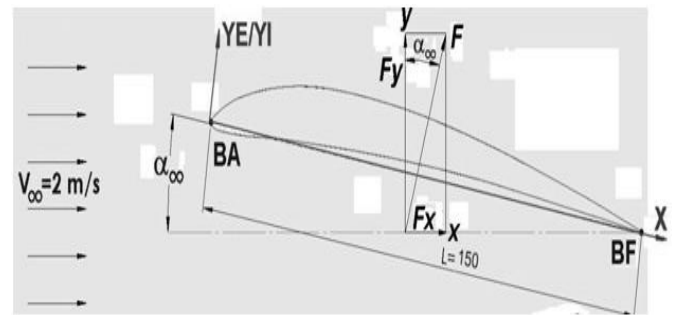


Fig. 1 Coordinate system of wing profile

I. INTRODUCTION

The application's goal is to calculate the hydrodynamic characteristics (drag coefficient and the lift coefficient) resulting from the underwater vehicle wing and fluid interaction, fig. 1, for different values of the incidence attack angle α and different length of wing. Other goal is to detect optimal wing ratio under two conditions: max lift coefficient and min drag coefficient. The water velocity is $V=2$ m/s. The profile chord is $L=150$ mm and the width of the wing is $B=90$ mm. The coordinate systems of the wing profile are presented in fig. 1, where: X is the abscissa and YE/YI are Y values for suction/pressure side. The origin of the coordinate system is placed in the leading edge point BA with X axis positive oriented to the trailing edge point BF and Y axis perpendicular in the BA point.

II. PROBLEM FORMULATION

The force \vec{F} , which is perpendicular on the profile chord, is the resultant force between the drag force \vec{F}_x and lift force \vec{F}_y , calculated by the following relations:

$$F_x = C_x \rho \frac{V^2}{2} L \cdot B \quad (1)$$

$$F_y = C_y \rho \frac{V^2}{2} L \cdot B \quad (2)$$

The C_x and C_y coefficients of (1) and (2) relations are determined in this work for various values of the incidence attack angle α and the different ratios $\lambda=B/L$. The C_y lift coefficient is negative for negative incidence angles, became zero for null lift angle α_0 , than grow almost linear until the critical incidence angle, with values between 1 rad and 1.5 rad then, the lift coefficient decrease and the drag coefficient grow.

III. THE STAGES OF APPLICATION

- Activation of the Flow Simulation module;
- Create Flow Simulation project;
- Define Computational Domain and goals;
- Running flow study;
- View the results;
- Cloning the project;
- Modify the α incidence attack angle, change wing length and rerun the study;
- Simulation and experimental results comparison.

A. Activation of the Solid Works Simulation module

Flow Simulation is based on advanced Computational Fluid Dynamics (CFD) techniques and is created to analyze a wide range of complex problems including [1]:

- External and Internal flows;
- Steady state and Transient flows;
- Incompressible liquid and Compressible gas flows including subsonic, transonic, and Supersonic regimes;
- Water vapor (steam) condensation;
- Calculation of relative humidity in gas flows
- Non-Newtonian liquids (laminar only);
- Compressible liquids (liquid density is dependent on pressure);
- Real gases;
- Laminar, turbulent, and transitional flows;
- Swirling flows and Fans;
- Multi-species flows;
- Flows with heat transfer within and between fluids and solids;
- Cavitation.

Once installed, Solid Works Flow Simulation module can

be activated inside Solid Works using Tools→ Add-Ins menu; as a consequence, the Flow Simulation menu bar will be added to the main menu.

B. Create Flow Simulation project

A Flow Simulation project contains all the settings and results of a problem. Each project is associated with a SolidWorks configuration. By modifying a Flow Simulation project it is possible to analyze flows under various conditions and for modified SolidWorks models. When a basic project has been created, a new Flow Simulation Design Tree tab appears on the side of the SolidWorks Configuration Manager tab. For this application, the main project characteristics are: SI unit system, External flow, Water fluid, 2 m/s velocity in X direction.

C. Define Computational Domain

The flow and heat transfer calculations are performed inside the computational domain. Flow Simulation analyzes the model geometry and automatically generates a Computational Domain in the shape of a rectangular prism enclosing the model. The computational domain's boundary planes are orthogonal to the model's Global Coordinate System axes. For External flows, the computational domain's boundary planes are automatically distanced from the model. In this application, to reduce the required CPU time and computer memory, will perform a two-dimensional (2D) analysis. For most cases, to study the flow field around an external body and to investigate the effects of design changes it is recommended to use the default Computational Domain size as determined by Flow Simulation. The accuracy will be increased at the expense of required CPU time and memory due to the larger size of Computational Domain [2].

D. Define goals

Flow Simulation initially considers any steady state flow problem as a time dependent problem. The solver module iterates on an internally determined time step to seek a steady state flow field, so it is necessary to have a criterion of determining that a steady state flow field is obtained, in order to stop the calculations. Flow Simulation contains built' in criteria to stop the solution process, but it is best to use specific criteria, which are named Goals. It is possible to set the following type of goals: global, point, surface, volume and equation [2]. For this application both the X - Component of Force and Y - Component of Force were specified as a Global Goal. This ensures that the calculation will not be finished until both components, in the entire computational domain, are fully converged. Also, the following two equation goals were imposed (in this case wing length = 230 mm and wing width = 90 mm):

$$\{GG \text{ Force (X) } 1\} / (0.5 * 998.2 * 2^2 * 0.23 * 0.09) \quad (3)$$

$$\{GG \text{ Force (Y) } 1\} / (0.5 * 998.2 * 2^2 * 0.23 * 0.09) \quad (4)$$

to obtain the numerical values of the drag and lift

coefficient at the end calculation. The expressions were obtained by extraction of the C_x and C_y coefficients from relations (1) and (2), with the following values:

- 998.2 kg/m³ – the water density;
- 2 m/s – the fluid velocity in X direction;
- 0.09 m – the wing width;
- 0.23 m – the length chord of the profile.

E. Running flow study

The Flow Simulation → Solve → Run command start the calculation. Flow Simulation automatically generates a computational mesh, by dividing the computational domain into slices, which are further subdivided into cells. The cells are refined as necessary to properly resolve the model geometry. After the calculation starts, the Solver Monitor dialog provides information about the current status of the solution, by monitoring the goal changes and view preliminary results at selected planes. In the bottom pane of the Info window Flow Simulation notifies with messages if inappropriate results may occur [2].

F. View the results

When the calculation is finished, the flow parameters distribution can be seen and analyzed the results with various results processing features and tools available in Flow Simulation: Cut Plot, Surface Plot, Iso-surfaces, Flow Trajectories, Particle Study, Surface Parameters, Volume Parameters, Point Parameters, XY Plot, Goal Plot, Report, and Animation. The Goal Plot offers the possibility to study how the goal value changed in the course of calculation. Flow Simulation uses Microsoft Excel to display goal plot data.

The converged values of all project goals are displayed in the Summary sheet and numerical values are placed in Plot Data sheet of an automatically created Excel workbook. This summary contain the numerical results of all imposed goals: X - Component of Force, Y - Component of Force, LIFT and DRAG coefficients, calculated with rel. (3) and (4) [3].

G. Cloning the project

In the first project, the profile was placed at 0.1 rad incidence angle and wing length/width ratio 0.8. This project will multiplied by cloning, to place the profile at incidence angles: 0.1 rad, 0.5 rad, 1 rad, 1.5 rad, 2 rad for 3 ratios: 0.8, 1.6 and 2.5. The two components: the drag and lift coefficients, will be calculated for every angle and ratio case. After the α incidence attack angle changes the study must be rerun.

IV. SIMULATION AND EXPERIMENTAL RESULTS COMPARISON

The Excel values from Goal Plot of max C_y and min C_x values for each wing ratio was centralize in the Table 1 and Table 2. The diagram from fig. 2 shows values comparison between min C_x of the each wing ratio case in the same angle of attack conditions.

ratio	0.1 rad	0.5 rad	1 rad	1.5 rad	2 rad
2.5	0.085	0.09	-0.236	-0.357	-0.19
1.6	0.07	-0.06	-0.316	-0.394	-0.322
0.8	0.066	-0.07	-0.306	-0.379	-0.35

Table 1. Min drag force coefficients

ratio	0.1 rad	0.5 rad	1 rad	1.5 rad	2 rad
2.5	0.227	0.921	1.771	1.863	1.55
1.6	0.371	1.347	2.379	2.35	2.14
0.8	0.423	1.603	2.574	2.655	2.328

Table 2. Max lift force coefficients

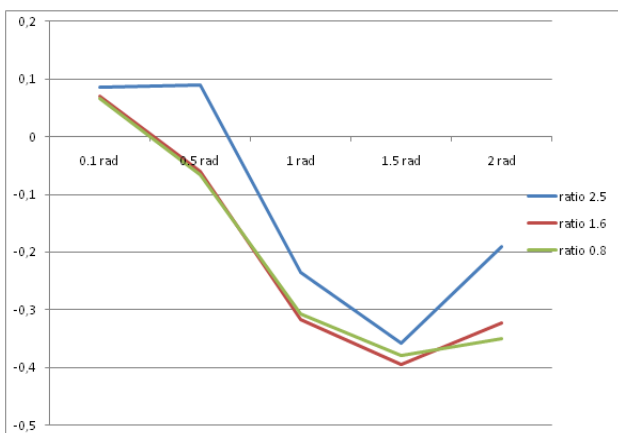


Fig. 2 Min Drag coefficient values

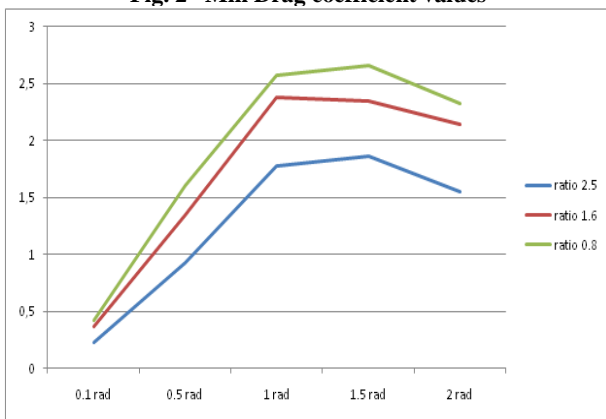


Fig. 3 Max Lift coefficient values

The diagram from fig. 3 shows values comparison between max C_y of the each wing ratio case in the same angle of attack conditions.

V. CONCLUSION

The curves from fig. 2 and values from Table 1 shows that optimal ratio of underwater vehicle wing for minimum drag force will be 1.6 with angle of attack 1.5 rad. Curves from fig. 3 and values from Table 2 shows that maximum lift force coefficient will be at ratio 0.8 with angle of attack 1.5 rad. The results were obtained without great computational efforts, since the number of the generated finite elements were situated between

19000 and 32000 and the CPU time was about 3...6 minutes per angle, for a computer with 16 GB RAM memory and 4.5 GHz Intel Core i5 processor.

REFERENCES

- [1] Dassault Systems, SolidWorks Flow Simulation Technical Reference, 2010, pp. 1-2.
- [2] Dassault Systems, SolidWorks Flow Simulation Tutorial 2010, 2010, pp. 5-6.
- [3] Dassault Systems, Flow Simulation 2010 Online User's Guide, 2010.