

The Aerodynamic Force Calculus for a Plate Immersed in a Uniform Air Stream Using SolidWorks Flow Simulation Module

DORIAN NEDELCU

DRAGHITA IANICI

Department of Materials Engineering, Mechanical and Industrial

„Eftimie Murgu” University of Resita

P-ta Traian Vuia 1-4, 320085 Resita

ROMANIA

d.nedelcu@uem.ro

d.ianici@uem.ro

MARIAN-DUMITRU NEDELONI

DANIEL DAIA

FLORENTIN MIREL POP

RAOUL CRISTIAN AVASILOAIE

Department of Engineering Faculty

„Eftimie Murgu” University of Resita

P-ta Traian Vuia 1-4, 320085 Resita

ROMANIA

m.nedeloni@uem.ro

d.daia@uem.ro

f.pop@uem.ro

r.avasiloaie@uem.ro

Abstract: - The objective of the application is to determine, through SolidWorks Flow Simulation module, the aerodynamic force and its components: the drag force and the lift force, for a 1500x1500x10 plate immersed in a uniform air stream; the plate is oriented perpendicular to the stream. The velocity of the air stream is 10 m/s and air density is 1.2 kg/m³. Finally, the results predicted by 2D simulation will be compared with experimental data.

Key-Words: - aerodynamic force, drag force, lift force, SolidWorks, Flow Simulation

1 Introduction

The application's goal is to calculate the aerodynamic force resulting from the L=1500 x B=1500 x h=10 plate and fluid interaction, fig. 1, for different values of θ angle [1]. The air velocity is V=10 m/s.

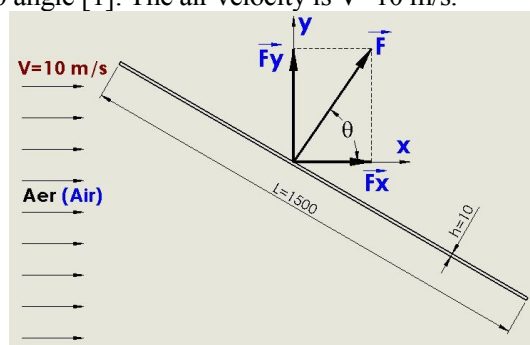


Fig. 1

2 Problem Formulation

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

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

$$F_y = \frac{1}{2} C_y \rho B L V^2 \quad (2)$$

Aerodynamic coefficients C_x and C_y of (1) and (2) relations are chosen according to the ratio L/B and the θ angle, from fig. 2, which were obtained experimentally [1], [2].

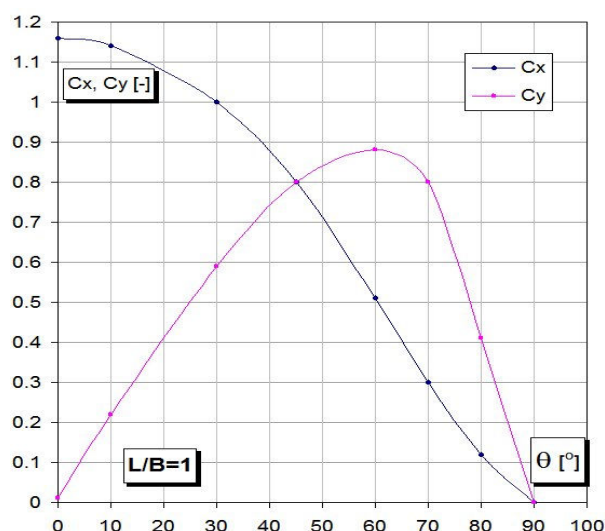


Fig. 2

3 The stages of application

- The plate design;
- Activation of the SolidWorks Flow Simulation module;
- Create Flow Simulation project;
- Define Computational Domain;
- Define goals;
- Running flow study;
- View the results;
- Cloning the project;
- Modify the θ angle layout and rerun the study;
- Simulation and experimental results comparison.

3.1 The plate design

- Create a new part document and save it as **Placa**.
- Create the sketch $L=1500 \times h=10 \times 60^\circ$, fig. 3, drawn in **Front plane**.

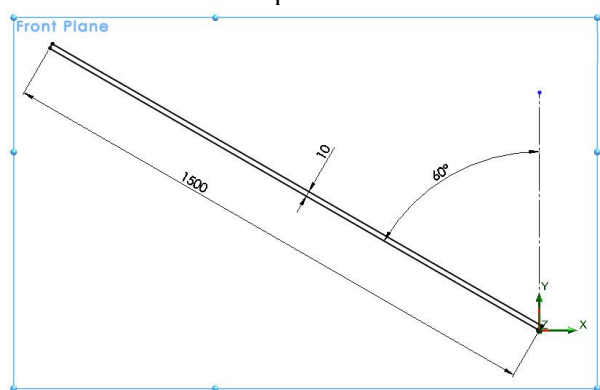


Fig. 3

- Through **Boss-Extrude** command, the sketch will be symmetric extruded on total distance 1500 mm.

3.2 Activation of the Flow Simulation module

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

3.3 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. To create a project, the following informations must defined:

- a project name;
- a system of units;
- an analysis type (external or internal);
- the type of fluid (gas, incompressible liquid, Non-Newtonian laminar liquid, or compressible liquid);
- the substances (fluids and solids);
- initial or ambient conditions;
- the geometry resolution and the results resolution;
- a wall roughness value;
- physical features include heat transfer in solids, high Mach number gas flow effects, gravitational effects, time dependent effects, surface-to-surface radiation and laminar only flow.

For this application, the main project characteristics are: SI unit system, External flow, Air fluid, 10 m/s velocity in X direction.

3.4 Define Computational Domain

In this application we are interested in calculate the aerodynamic force of the plate only, without the accompanying 3D effects. Thus, to reduce the required CPU time and computer memory, a two-dimensional (2D) analysis is performed. Selecting **Flow Simulation** → **Computational Domain**, the dialog box from fig. 4 appears:

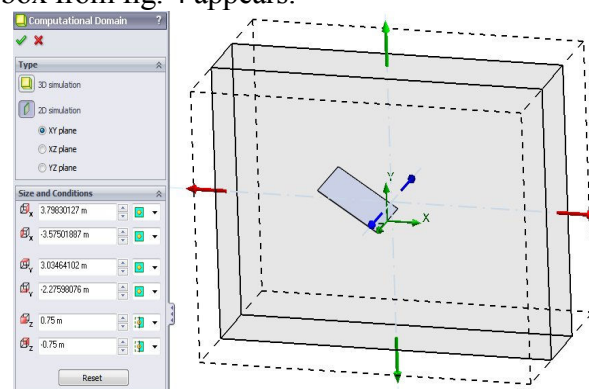


Fig. 4.

3.5 Define goals

The aerodynamic force can be determined by specifying the appropriate Flow Simulation goal. For this application, both the **X - Component of Force 1** and **Y - Component of Force 1** were imposed as a Global Goal. This ensures that the calculation will not be finished until both components, in the entire computational domain, are fully converged.

3.6 Running flow study:

The **Flow Simulation** → **Solve** → **Run** command start the calculation. Flow Simulation automatically generates a computational mesh. The mesh is created 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. During the mesh generation procedure, the current step and the mesh information can be viewed in the **Mesh Generation** dialog box. The initial mesh is constructed in several stages [3], [4]:

- constructing the basic mesh for a specified number of cells and stretching or contracting the basic mesh locally to better resolve the model and flow features by the use of Control Planes;
- splitting the basic mesh cells either to capture the small solid features, or resolve the substance interface (fluid/solid, fluid/porous, porous/solid interfaces or boundary between different solids) curvature (e.g., small-radius circle surfaces, etc.) - i.e. small solid features refinement, curvature refinement and tolerance refinement;
- refining the mesh cells of a specific type (refinement of either all cells, or fluid, and/or solid cells, and/or partial cells);
- refining the obtained mesh to better resolve narrow channels- i.e. narrow channel refinement.

The **Run** window include the following controls.

fig. 5:

- **Mesh** - to create the new computational mesh for a project that is already meshed or calculated;
- **Solve** - to calculate the project; for a project that was already calculated before the following options can be selected:

- **New calculation** - to recalculate a previously calculated project from start, using the initial conditions specified in the **Wizard** or **General Settings**. To start the new calculation using previous calculation results as an initial condition, select the **Mesh**, **Solve**, **New calculation**, and **Take previous results** options, which will ignore the initial conditions specified in the **Wizard** or **General Settings**.
- **Continue calculation** - to continue calculation from the point where the solver was stopped automatically or manually.

- **Run At** - list to select the way to run the calculation:

- **This computer (CAD session)** - the solver runs on the current computer as a part of the SolidWorks application.
- **This computer (standalone)** - the solver runs as a separate process (i.e. as a separate application).
- **Add computer** - to select a network computer to be added to the **Run At** list; this computer can be selected to calculate the study.

- **Use CPU(s)** - if the local or the selected network computer has multiprocessing capabilities, in the **Use CPU(s)** list select the number of processors or processor cores to use for calculation.
- **Load results** - to automatically load results when the calculation is finished or stopped manually.
- **Batch results** - to define the plots, parameter tables and reports to be created.

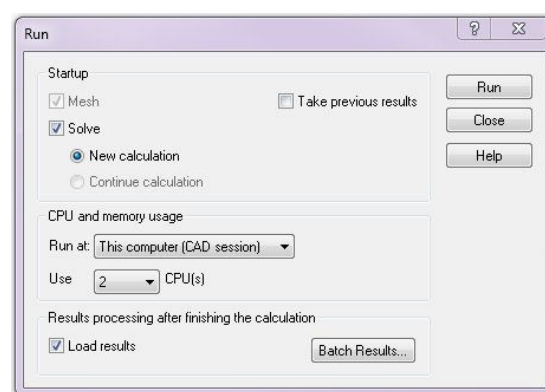


Fig. 5

After the calculation starts, the **Solver Monitor** dialog, fig. 6, provides informations 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.

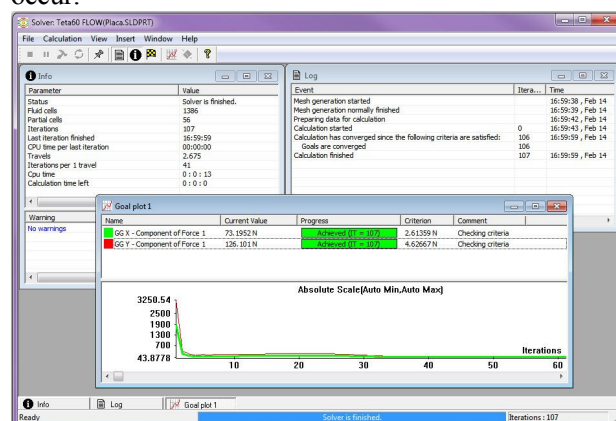


Fig. 6

3.7 View the results

The **Goal Plot** offer the possibility to study how the goal value changed in the course of calculation. Flow Simulation uses Microsoft Excel to display goal plot data. Each goal plot is displayed in a separate sheet, fig. 7.

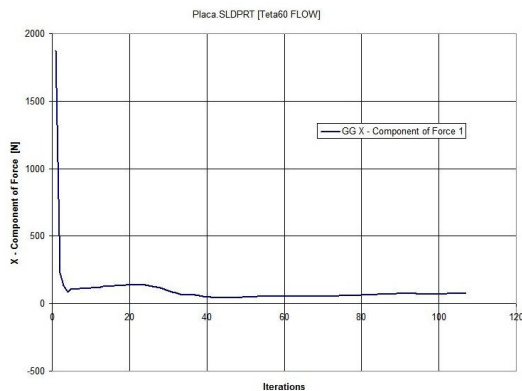


Fig. 7

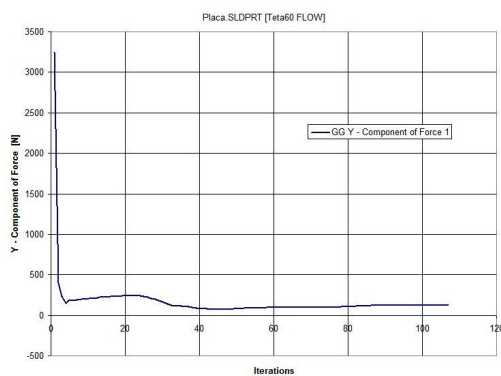


Fig. 8

The converged values of the two project goals are displayed in the **Summary** sheet and numerical values are placed in **Plot Data** sheet of an automatically created Excel workbook, fig. 9.

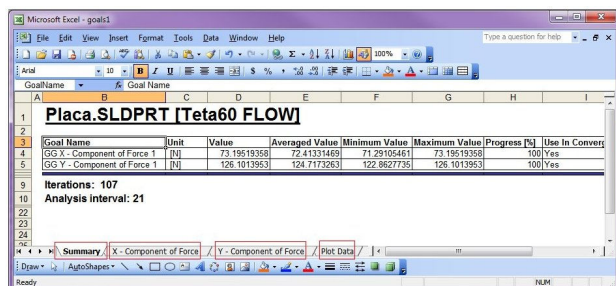


Fig. 9

The **Cut Plot** displays results of a selected parameter in a selected view section. To define the view section, can be used SolidWorks planes or model planar faces (with the additional shift if necessary). The parameter values can be represented as a contour plot, as isolines, as vectors, or in a combination (e.g.

contours with overlaid vectors). Right-click the **Cut Plots** icon and select **Insert**. The **Cut Plot** dialog box appears like in fig. 10.

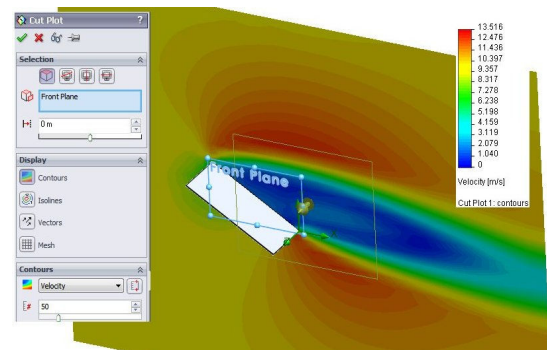


Fig. 10

Using **Flow trajectories** it is possible to view the flow streamlines. **Flow trajectories** provide a very good image of the 3D fluid flow, show how parameters change along each trajectory by exporting data into Microsoft Excel and save trajectories as SolidWorks reference curves, fig. 11.

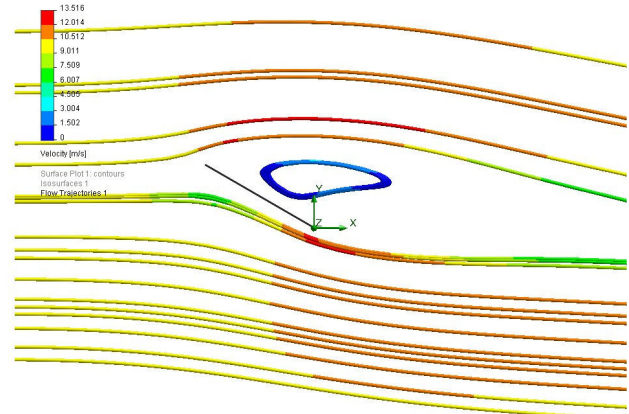


Fig. 11

3.8 Cloning the project

In the first study, the plate was placed at 60° angle. The **Teta60 FLOW** project will multiplied by be cloning, to place the plate at the following angles: 10°, 30°, 45°, 70°, 80°. The two components: the drag force \vec{F}_x and lift force \vec{F}_y , will be calculated for every angle case. After the θ angle layout change, the study must be rerun.

4 Simulation and experimental results comparison

The experimental and Excel values from **Goal Plot** option was centralized in the tab. 1, for all six values of the θ angle.

Tab. 1

θ [°]	Experimental						Flow Simulation	
	Cx [-]	Cy [-]	Fx Rel. (1)	Fx Err [%]	Fy Rel. (2)	Fy Err [%]	Fx [N]	Fy [N]
10	1.14	0.22	153.9	-19.4	29.7	-7.2	183.9	31.8
30	1.00	0.59	135	0.33	79.65	2.49	134.5	77.6
45	0.80	0.80	108	5.88	108	6.36	101.6	101.1
60	0.51	0.88	68.85	-6.31	118.8	-6.1	73.20	126.1
70	0.30	0.80	40.5	2.42	108	1.40	39.52	106.4
80	0.12	0.41	16.2	37.87	55.35	5.88	10.07	55.35

The diagram from fig. 12 show the graphical comparison between the Flow Simulation and calculated values of the drag force \vec{F}_x and lift force \vec{F}_y .

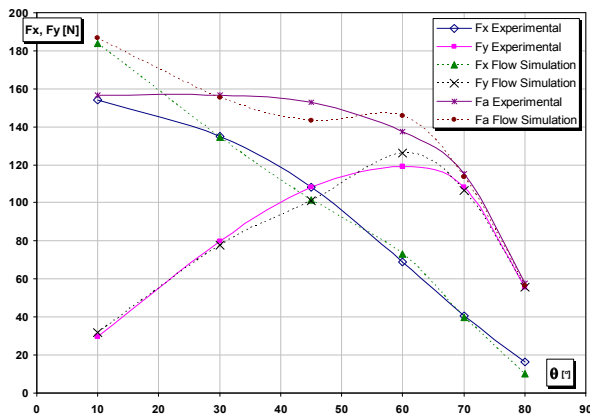


Fig. 12

The values were calculated on the base of the relations (1) and (2) and with the coefficients values resulted from fig. 2. Tab. 2 show the aerodynamic force values calculated through rel. (3).

$$F_a = \sqrt{F_x^2 + F_y^2} \quad (3)$$

Tab. 2

θ [°]	Fa [N]		Err [%]
	Exp.	Flow Sim.	
10	156.7	186.6	-19.1
30	156.7	155.4	0.9
45	152.7	143.4	6.1
60	137.3	145.8	-6.2
70	115.3	113.6	1.5
80	57.7	56.3	2.5

Fig. 13 show the pressure and velocity distribution for the six angle of the plate.

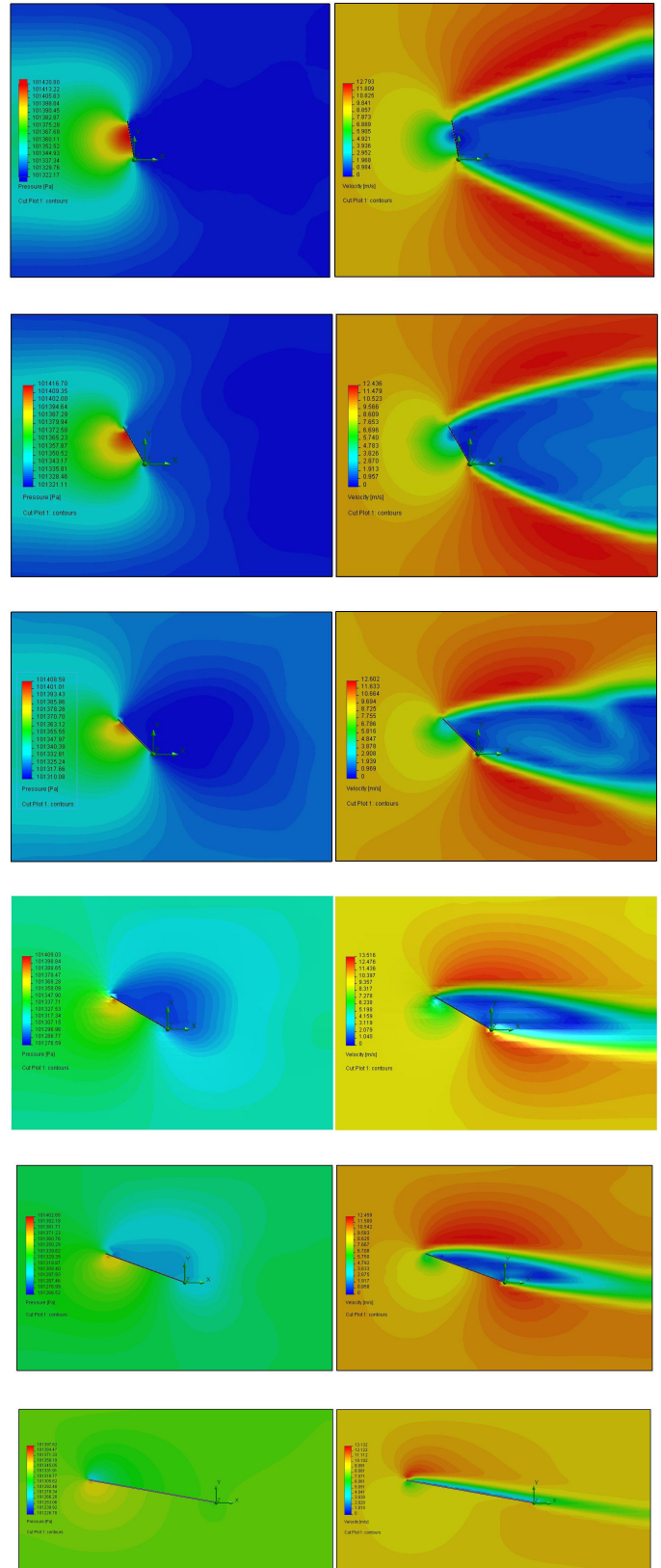


Fig. 13

Fig. 14 show the relative pressure distribution as a function of model X values for the six angle of the plate.

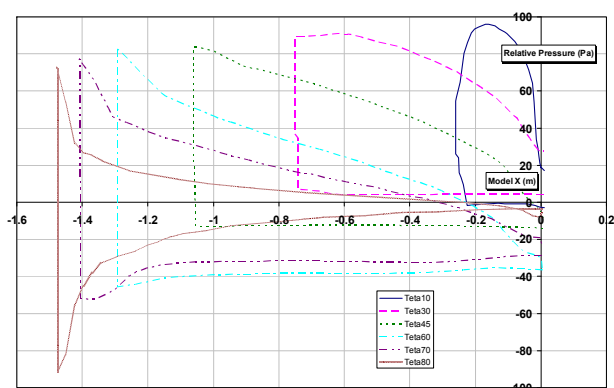


Fig. 14

5 Conclusion

The following conclusions can be obtained from this analyse:

- The curves from fig. 12 confirms a good coincidence of the two components of aerodynamic force, except the extreme limits, where the differences are greater for the component F_x .
- For the 30 ... 70 degrees values of the θ angle, the differences between the two components are less than 6.5%, but for the extreme limits (10 and 80 degrees) the differences increased up to 38% for F_x component.
- F_y component differences are smaller than the F_x component.
- The values in tab. 2 confirms the good overlap of aerodynamic force under 6.5% for 30 ... 80 degree angle, the maximal error of -20% recorded only 10 degrees angle.

Acknowledgments

The authors gratefully acknowledge the support of the Managing Authority for Sectoral Operational Programme for Human Resources Development (MASOPHRD), within the Romanian Ministry of Labour, Family and Equal Opportunities by co-financing the project "Excellence in research through postdoctoral programmes in priority domains of the knowledge-based society (EXCEL)" ID 62557 and "Investment in Research-innovation-development for the future (DocInvest)" ID 76813.

References:

- [1] Iamandi, C., Petrescu, V., Sandu, L., Damian, R., Anton, A., Degeratu, M., *Hidraulica instalatiilor. Elemente de calcul si aplicatii*, Editura Tehnica, București, 1985, pp. 230-231.

- [2] G. Zidaru, *Miscari potentiale si hidrodinamica retelelor de profile*, Editura Didactica si Pedagogica, București, 1981, pp. 68-71.
- [3] Dassault Systems, *Flow Simulation 2010 Online User's Guide*, 2010.
- [4] Dassault Systems, *SolidWorks Flow Simulation 2010 Solving Engineering Problems*, 2010, pp. 2-3.