PRACE INSTYTUTU LOTNICTWA
215, s. 3-16, Warszawa 2011

NUMERICAL ANALYSIS OF WIND RESISTANCE FOR FREE-STANDING ROOFTOP DEVICE

ADAM DZIUBIŃSKI
JERZY ŻÓŁTAK
Instytut Lotnictwa

Abstract

The paper describes research to improve the wind resistance of free standing device. Analysis of the wind influence on object with specified shape, has been simplified to the analysis of two-dimensional case. This helped to formulate the basic criteria for evaluating the examined cases. The study was conducted using numerical methods for fluid mechanics. Flow field has been obtained using the Fluent package. Several configurations of object has been studied. The results of calculations and analysis of the forces has been presented. Some useful hints and tips used during the mesh generation has been shown.

1. Introduction

Commonly used devices are often exposed to many threats leading to their destruction, which could be hazardous for persons and objects appearing in proximity. To reduce the destructive nature of similar phenomena in the process of placing a product on the market, it must meet not only the user requirements but, often, very strict safety requirements. For many products, they are normative.

The analyzed device is free-standing (ie. not fixed to the ground with additional fasteners) base for the flat panels. It is required that such device:

- does not move under the impact of wind, of course to the extent of its allowable velocity. In this case maximum wind velocities of 125 km/h [3] are considered.
- working surface of the device (panel collecting sunlight energy) should not be covered nor changed by the aerodynamic surfaces.
- the suggested solution should be relatively easy and cheap to manufacture (low cost of manufacturing)

Below the following will be presented: model assumptions allowing to reduce task to the two-dimensional case, the criteria for evaluating the analyzed cases, brief discussion of the computational model. At the end the results of analysis will be presented.

During description of results some useful computational methods and tricks will be presented, for both preparation of model and computation.

2. Problem description

The purpose of the analysis is to determine the geometrical parameters of the basis for a flat panel, Figure 1, so to ensure its resistance to wind.

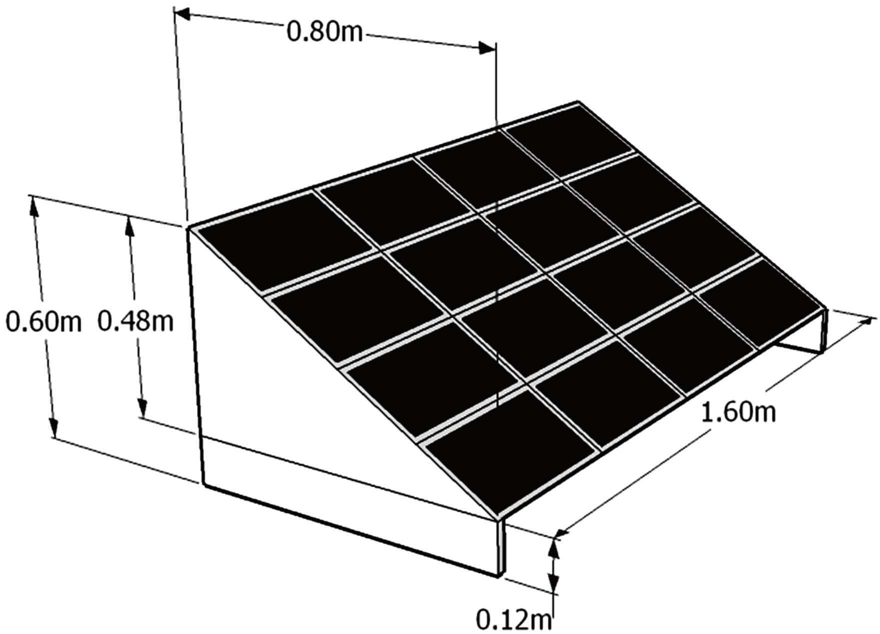

Figure 1. Scheme of an object geometry

Simplified static analysis of the device shows a following forces acting on the object, using commonly defined names :

- ✓ lift and drag: components of aerodynamic force
- ✓ weight
- ✓ normal to surface component of reaction force
- ✓ friction, tangential to surface component of reaction force.

Directions of those forces has been shown in Figure 2.

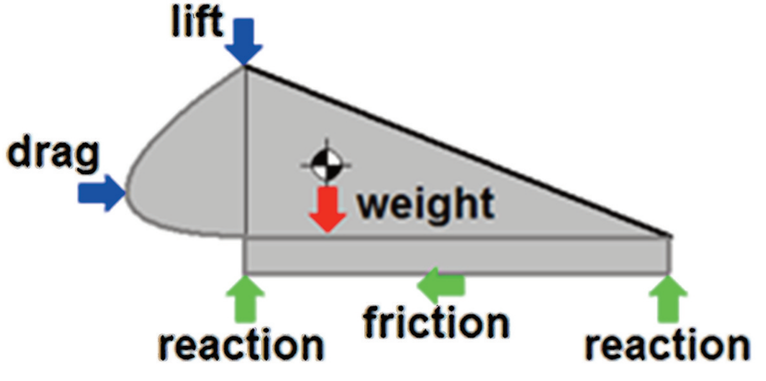

Figure 2. Forces acting on device

It is worth to underline that sum of weight and lift, multiplied by static friction coefficient gives maximum friction that could be used against aerodynamic drag. So it must be assumed sufficient contact area and surface finish (in order to maximize the friction coefficient) to fulfill the requirement of maximum wind resistance. From aerodynamic point of view it is not enough to keep the lift force lower than the weight, if the object will be able to slide because of drag. The goal is rather to keep small as it possible (negative value is prefered) lift force and drag force as low as possible. The conclusion above states, that the search for solutions can start from a simplified geometric model of the object. In this model additional supports on the sides of object have been omitted. This assumption made it possible to reduce the computational model to the analysis of two-dimensional case (2D)

3. Computational model

The widely recognized in industry FLUENT software has been used to obtain the aerodynamic characteristics of tested shapes. This software utilizes finite volume method of solving Reynolds Averaged Navier Stokes equations to obtain the flowfield around complex geometry. Several turbulent models are implemented in the software.

Presented specific problem has been solved using finite volume method on 2D mesh. For purposes of following computations there were some assumptions done: all simulations were done for atmospheric pressure (101325 Pa) and air density of 1.225 g/m^3. For all tests velocity of 125 km/h has been assumed. This velocity has been defined as maximum velocity that the device should withstand. The Spalart-Allmaras [1, 2] model of turbulence was used in all calculation cases.

Common feature of all meshes was assumption of external boundary condition of computational domain. With all meshes the front and upper wall of such domain were defined as pressure farfield and the back wall has been defined as pressure outlet. The bottom wall, which represented the roof surface, has been defined as the wall.

4. Results

Several concepts have been tested. The analysis has been intuitionally divided into the following sections:

- ✓ Analysis of influence of bottom channel on simplified shape
- ✓ Analysis of possible decreasing drag by different shapes of “nose”
- ✓ Analysis of front wall inclination influence
- ✓ Influence of spoiler at different angles of inclination

Each of this phases leaded to some conclusions and caused some choices in the following design.

4.1 Influence of bottom channel on simplified shape.

In order to know the importance of all features of examined shape, first the bottom channel has been tested. Three configuration were considered, see Figure 3:

- ✓ open channel (Case 1)
- ✓ channel closed at forewind wall (Case 2)
- ✓ channel closed at backwind wall (Case 3)

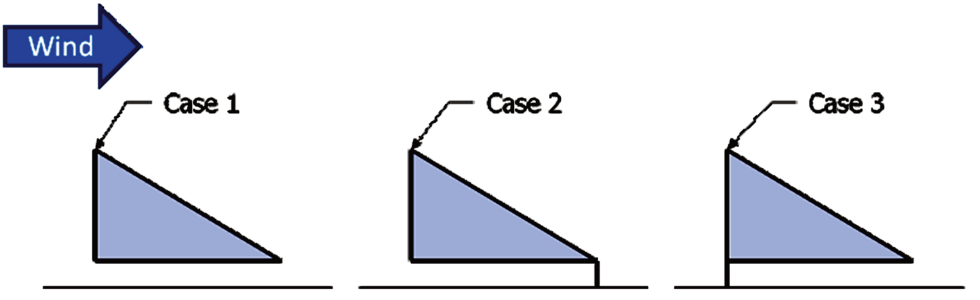

Figure 3. Cases of bottom channel analysis

Tests had to prove, how important is opening the bottom channel and which solution of closing the channel is worse from safety point of view (no one should be hit with the device falling from the roof) . For this numerical test only one mesh, Figure 4, has been created, with assumption of switchable walls [4].

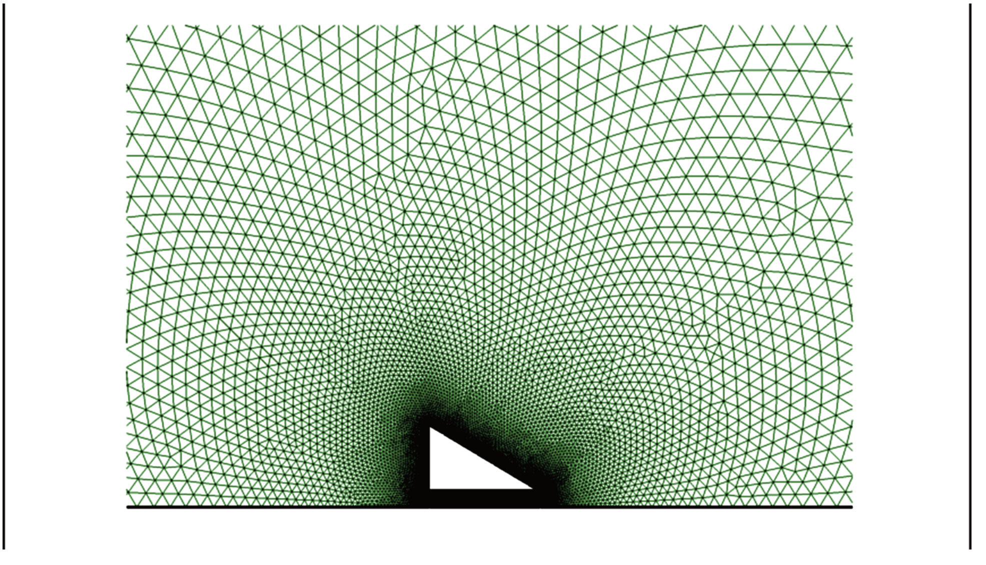

Figure 4. Computational mesh near the object

It means that user can define on specified surfaces, if there is wall or it is a part of interior where flow is able to go through so specified, "transparent to the flow" surface. Such solution was caused by a comparability of results: they were obtained using the same mesh. Similarity of meshes is crucial when results of calculations on non-structural meshes are compared. As we consider 2D geometry, the switchable walls were defined as lines.

The results proved, that most efficient configuration is with the open channel at the bottom of device (Case 1). Its drag is slightly lower than other two configurations (Figure 5a), but lift force is negative as it is demanded (Figure 5b).

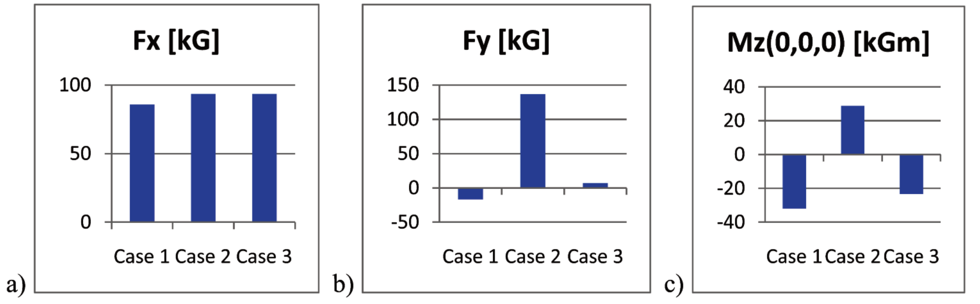

Figure 5. Comparison of (a) lift, (b) drag and (c) moment between Cases 1, 2, 3

Worst configuration is closed backwind (Case 2), which generates high lift force, above weight of the device. The main part of lift is caused by the stagnation region, from which the air with high pressure flows into bottom region and pushes up the bottom wall of the device (Figure 6). The designer has been warned about that phenomena and assured that nothing should cause such blockage of the channel.

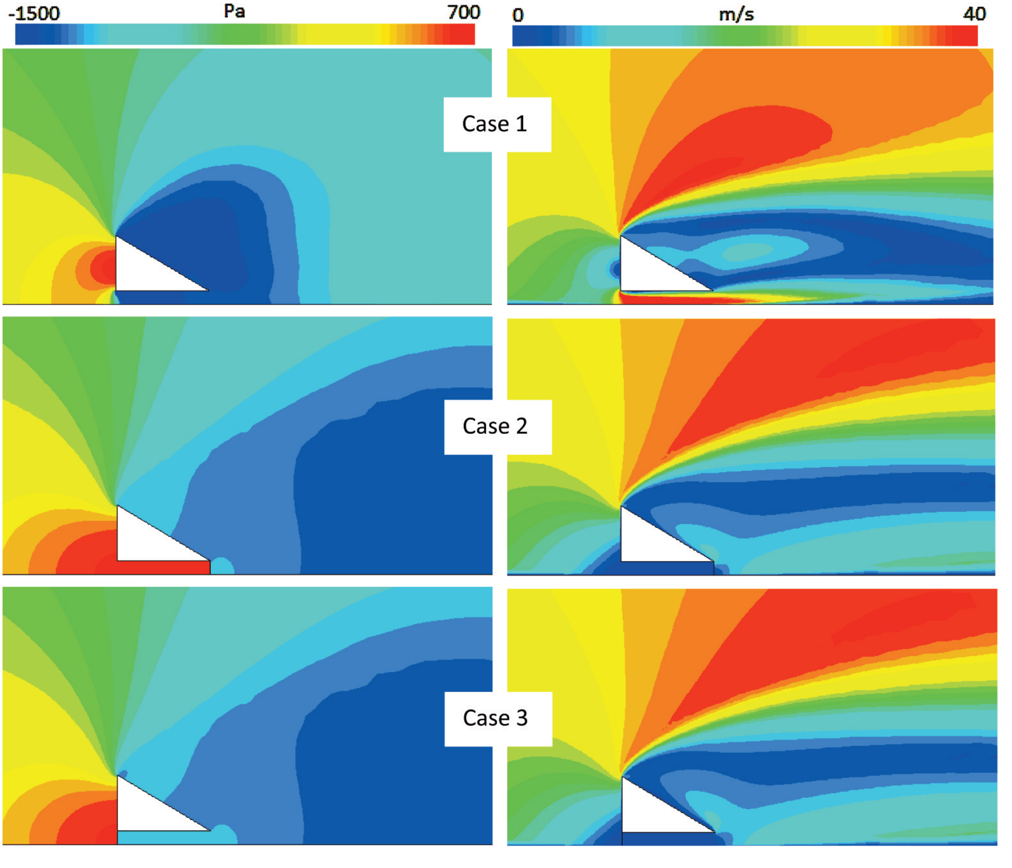

Figure 6. Static pressure (left) and velocity magnitude (right) colour maps for Case 1, 2 and 3 (bottom canal)

Case 3 is an example of how a low pressure in separation region can interact with object generating lift on the upper side of the device and negative lift on the bottom wall. A part of lift has been also generated by friction of air moving up by the front wall.

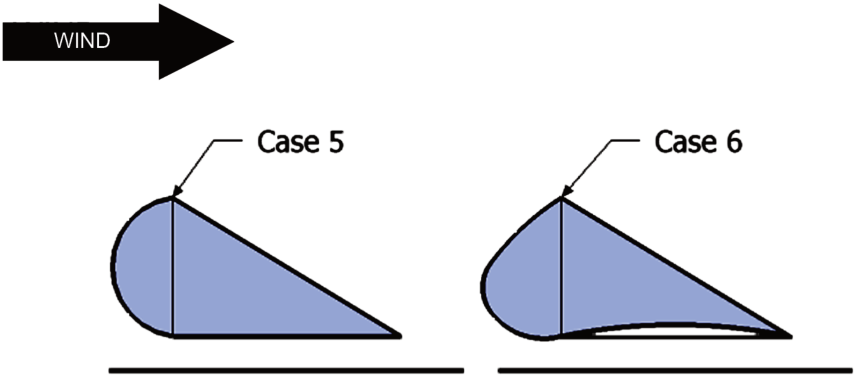

Figure 7. Filleted nose shapes

This set of calculations leaded to the conclusion that keeping the channel under the device opened is necessary. This created an option to use a flat diffuser shape in this canal, but that idea has been abandoned due to other requirements.

4.2 Attempt to decrease the drag using shielded nose

After obtaining some knowledge about advantages of simplified shape, first thing that has been done, was an attempt to decrease the drag. Two configurations has been generated – Figure 8.

First geometry (Case 5) in which to the front part of object (Case 1) half-circular nose was added.

Second geometry (Case 6) in which the nose shape has been deflected down to decrease area where stagnation pressure pushes the device up. Generally idea was to not allow too much air to go through the bottom channel. Additionally for this case the top wall of channel was modified. Two different meshes for each case has been generated (Figure 8).

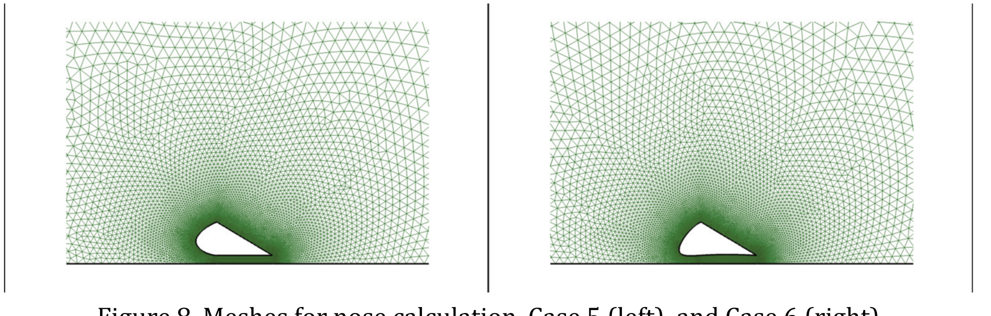

Figure 8. Meshes for nose calculation, Case 5 (left) and Case 6 (right)

The static pressure and velocity magnitude of flowfield distribution for Cases 1, 4 and 5 are presented in Figure 9. The modification of front part of object leaded to reduce value of relatively high static pressure on it and decreased size of domain with low pressure at the top, flat part of the object. In bottom channel decrease of velocity magnitude can be observed.

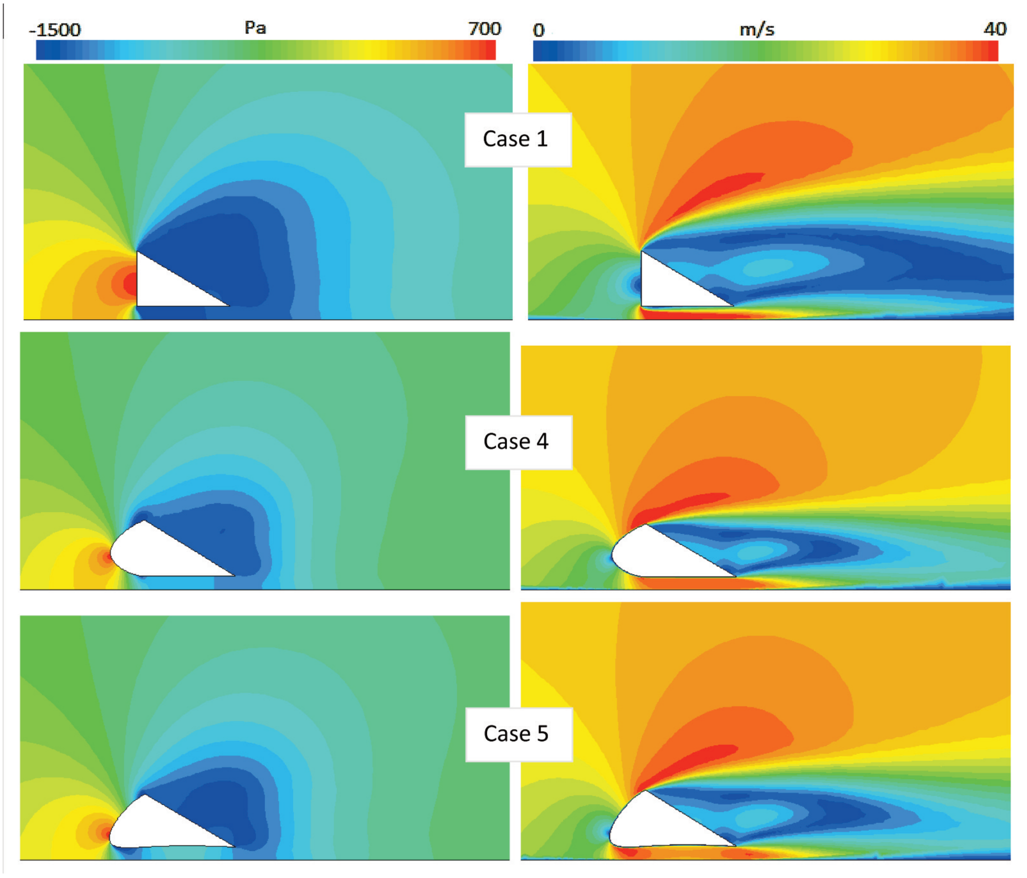

Figure 9. Static pressure (left) and velocity magnitude (right) colour maps for Case 1, 4 and 5 (nose shape modification)

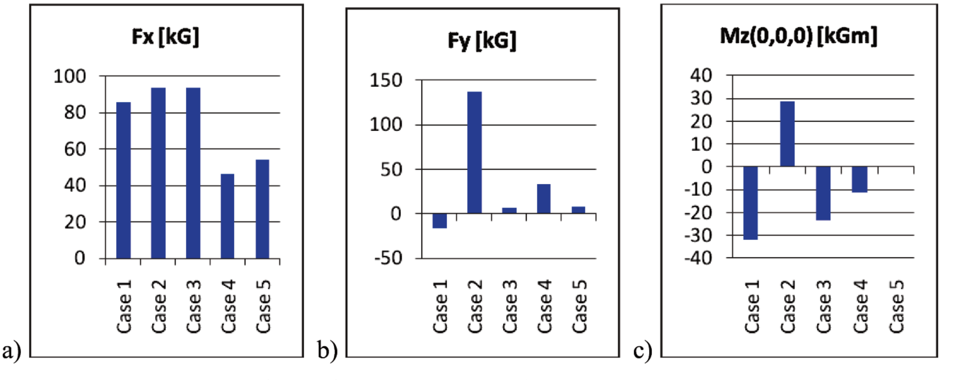

Figure 10. Comparison of (a) lift, (b) drag and (c) moment between cases: 1, 2, 3 and 5 (flat nose with and without canal)

The forces calculated for Case 5 and Case 6 are compared with results for Case 1 in Figure 10. Both solutions caused almost 50% decrease of drag on the front area. The nose down (Case 6) configuration has slightly greater drag but the lift has been decreased almost to the value for flat case with opened bottom channel (Case 1).

4.3 Front wall inclination analysis

After so promising effect of nose down configuration an angle of attack of front wall has been taken into consideration. A set of calculations for different walls angle has been done examining four angles of inclination (alpha) – Table 1 and Figure 11.

	Alpha[deg]
Case 6	67.38
Case 7	50.19
Case 8	38.66
Case 9	30.96

Table 1. Front wall inclination angle cases

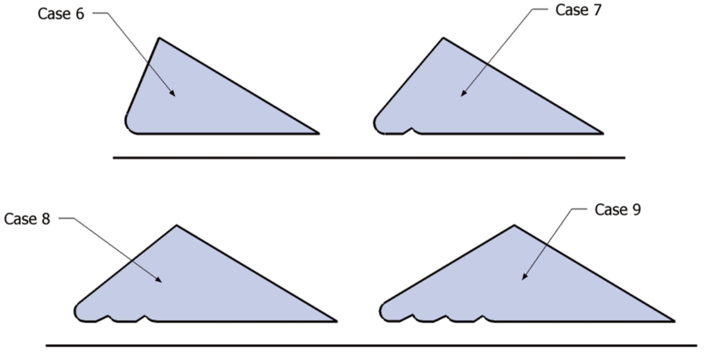

Figure 11. Geometry of front wall inclination cases

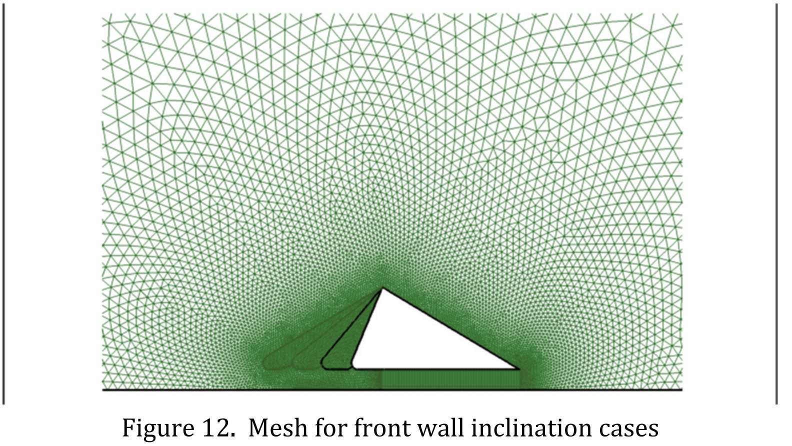

Figure 12. Mesh for front wall inclination cases

A goal of this rough research was to find only tendency of drag and lift changes in function of angle of inclination. This part of research had to be rather an argument in discussion with the designer than a systematic research. Given the assumption, that the shape will be further optimized, this was a reasonable approach.

Mesh for calculation was generated in order to easily change from one configuration to another putting on and off a proper set of walls –Figure 12. Bottom surface of every switchable wall is not flat because of meshing issues. In case of meshing with flat bottom wall the mesher would generate very bad cells (with unacceptable deformation). This could cause errors in calculation or, in worst case, inability to converge the case. It was assumed that those grooves have a negligible effect on flow.

The Figure 13 shows the static pressure and velocity magnitude flowfield distribution for tested configuration. Analysis of those distributions leads to following conclusion: decrease of inclination angle leads to decrease of stagnation area size, and decrease of stagnation pressure.

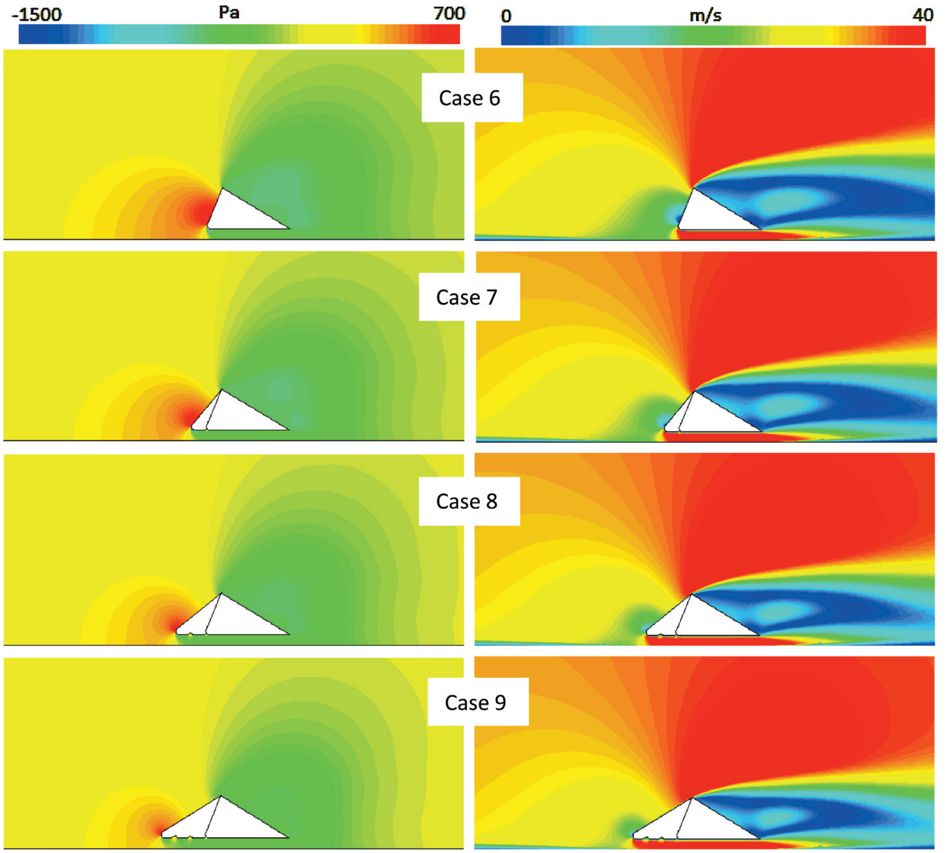

Figure 13. Static pressure (left) and velocity magnitude (right) colour maps for Case 6, 7, 8 and 9 (front wall inclination cases)

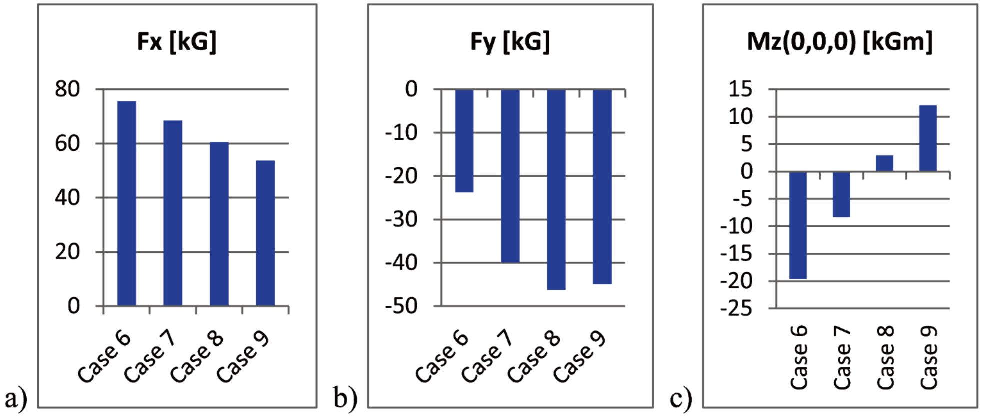

Figure 14. Comparison of (a) lift, (b) drag and (c) moment between cases: 6, 7, 8, and 9

The analysis of forces proves, that with decrease of inclination angle the values of most important parameters are changing as follows – Figure 14 and 15:

- ✓ Drag force deceases in nonlinear way
- ✓ Lift force after achieving the local minimum starts to increase.

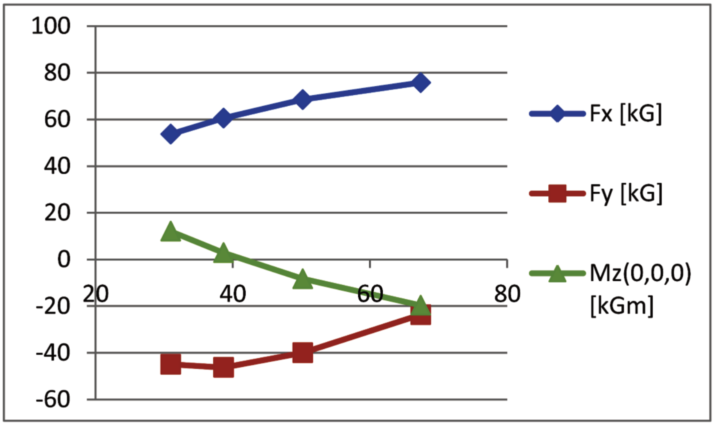

Figure 15. Value of forces and moments dependence versus angle of incidence

Moment changes from negative (Case 6) to positive (Case 9) values because of translation of force generating surface's center of lift (front wall) against flow, in negative direction of local coordinate system x axis.

4.4 Spoiler analysis.

At the end an influence of spoiler mounted on top of the device on the forces generated on the test object. The study aimed to determine if spoiler could be used as an additional lift control surface when the pull force will be not enough to keep the device in its place. Five configurations of spoiler has been tested: 30, 45, 60, 75 and 90 deg – Table 2 and Figure 16. The above mentioned angle was an angle between the spoiler surface and the roof surface. For baseline geometry the best of the front wall cases (Case 8) has been chosen.

	Alpha[deg]
Case 8	0.0
Case 8/30	30.0
Case 8/45	45.0
Case 8/60	60.0
Case 8/75	75.0
Case 8/90	90.0

Table 2. List of spoiler test cases.

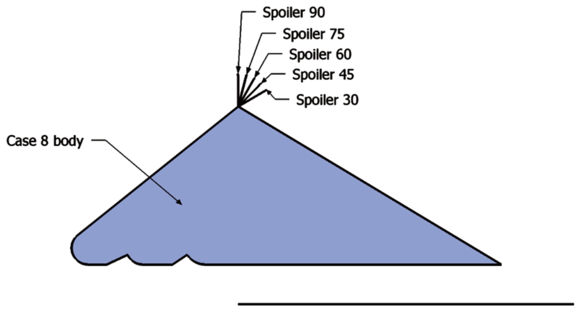

Figure 16. Geometry of the spoiler test cases

Mesh for the spoiler configuration has been generated in similar way as in previous cases, basing on idea of switching on and off the "transparence to the flow" of set of walls inside the domain – Figure 17. Assuming that no boundary layer mesh modeling will be present, design of such mesh (in terms of setting the mesh densities) has been rather easy.

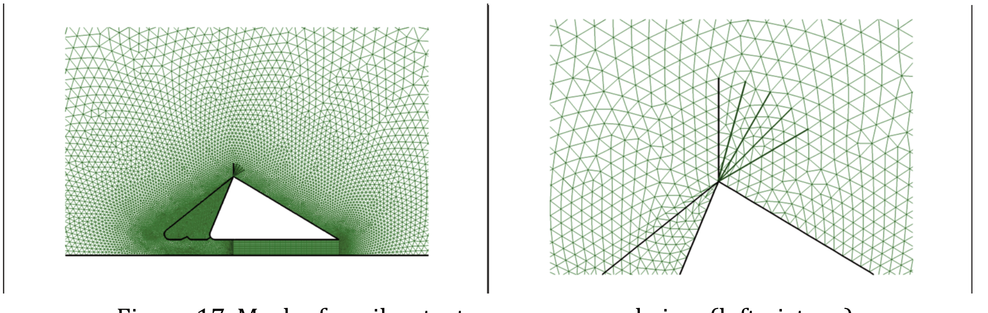

Figure 17. Mesh of spoiler test cases – general view (left picture), zoom of spoiler domain (right picture)

Analysis of forces shows that above angle of 75 deg. efficiency of spoiler decreases, with constant increase of drag – Figure 18 and 19. Using spoiler one can obtain up to 35 % of lift force decrease, what gives similar increase of drag efficiency. The drag itself increases till about 20%, so the device gives in summary only about 15% of effectiveness.

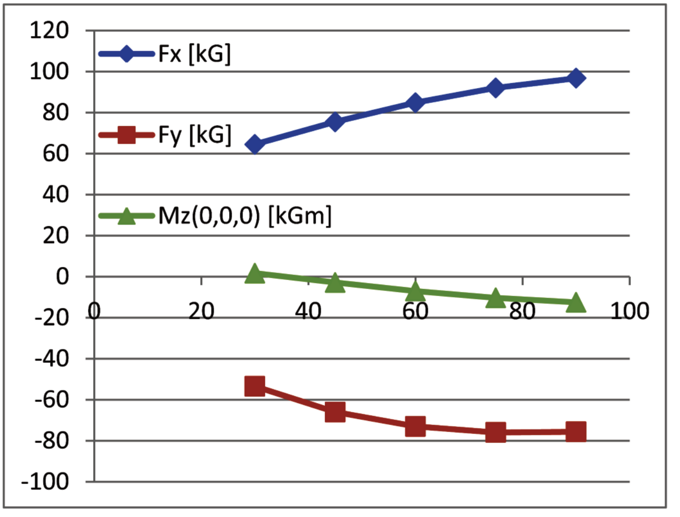

Figure 18. Forces and moments dependence on spoiler angle of incidence

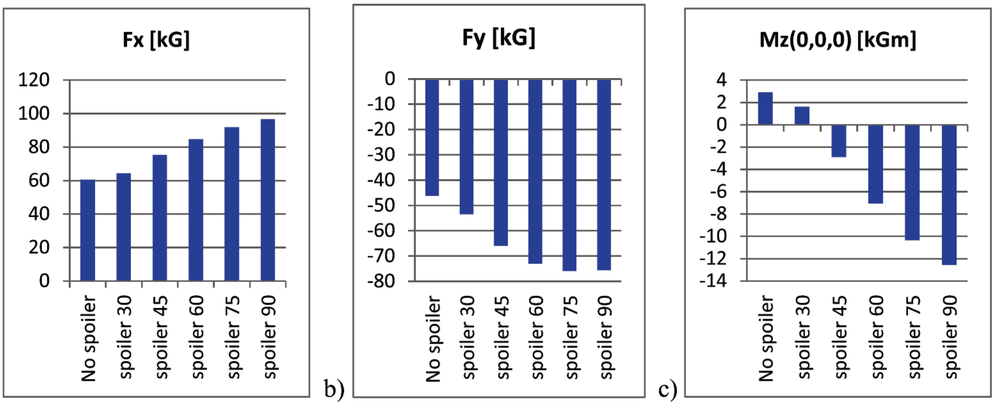

Figure 19. Comparison of (a) lift, (b) drag and (c) moment between spoiler cases

What is worth to underline, spoiler has a reasonable influence on moment, so the device gives some ability to control the moment on the device.

Based on flowfield qualitative analysis – Figure 20 - one can conclude, that negative lift is caused rather by generating larger stagnation area with increased pressure, than by force caused by air on the device. And this is true, but unfortunately increasing the angle the device starts to generate increased area of separation, which gives higher drag. To decide if the device is useful or not, an optimization of lift – to – drag ratio becomes necessary.

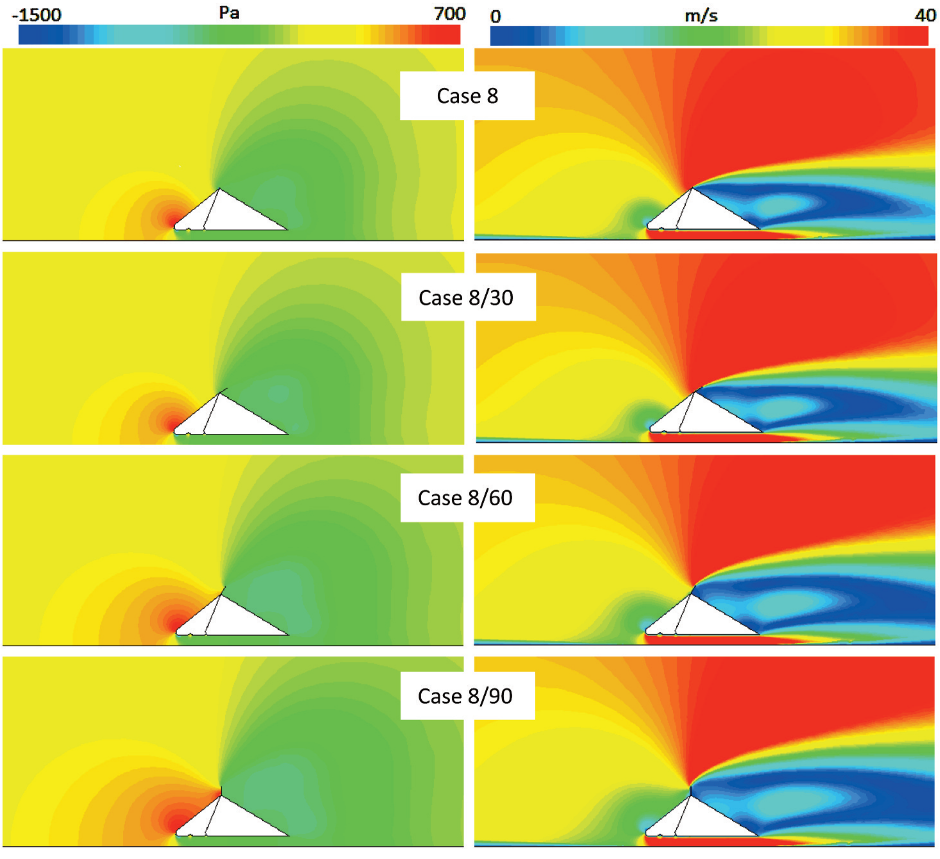

Figure 20. Static pressure (left) and velocity magnitude (right) colour maps for chosen spoiler cases.

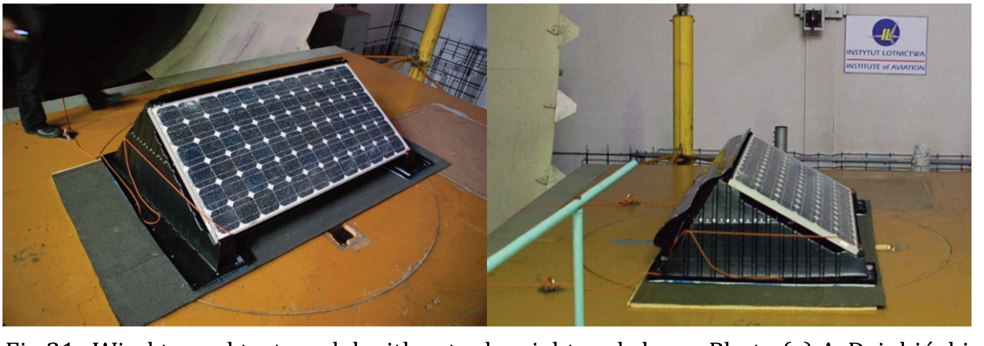

Fig 21. Wind tunnel test model with actual weight and shape. Photo (c) A. Dziubiński

5 Conclusions

Computational method was useful used to analyze the wind resistance. The 2D simple model of the free standing object was tested. Set of configurations were researched. Results of works shows that:

- ✓ opening of bottom canal gives the best result in terms of lift coefficient (less value of lift)
- ✓ filleted nose decreases the drag of the device by half with respect to basic configuration,
- ✓ nose moved down decreases the moment and lift force,
- ✓ front wall inclination has its optimal influence close to 38.66 deg
- ✓ with front wall inclination one can control also the moment of the device, which could be crucial to obtain maximum friction,
- ✓ spoiler inclination increases the drag and decreases the lift coefficient, and with this device combined with a front wall inclination a proper moment coefficient could be obtained.

From tested configurations as the compromise between presented results and technological constrains defined by customer, the Case 7 was down selected to wind tunnel tests – Figure 19.

6 Bibliography

[1] K. Szumański, „Określenie granicznych warunków użytkowania śmigłowców w systemie operacji z wysokich budynków", Projekt rozwojowy Nr R 00 033 02

[2] A. Dziubiński, W. Stalewski, J. Żółtak „Przykłady zastosowania pakietu FLUENT™ w analizach bezpieczeństwa lotu śmigłowców." Prace Instytutu Lotnictwa 2008, Nr 3-4 (194-195), Warszawa 2008.

[3] NORMA EN1991-1-4:2005 (E)

[4] R. Garcia „GAMBIT 2.1 & FLUENT 6.1 Tips & Tricks", IV FLUENT Users Meeting, Barcelona 2003

Acknowledgments

This research was supported by BudroTech GmbH represented by Mr. Josef Drozynski. The authors wants to express that they are very grateful to him for so effective collaboration.

A. Dziubiński, J. Żółtak,
Instytut Lotnictwa

ANALIZA NUMERYCZNA ODPORNOŚCI NA WIATR WOLNOSTOJĄCEGO URZĄDZENIA MONTOWANEGO NA DACHACH

Streszczenie

W pracy opisano badania poprawy odporności na wiatr obiektu wolnostojącego. Analizując obiekt oraz działające na niego pod wpływem wiatru siły, rozważania sprowadzono do analiz dwuwymiarowych. Analiza ta pozwoliła również sformułować podstawowe kryteria oceny badanych przypadków. Badania przeprowadzono z wykorzystaniem metod numerycznej mechaniki płynów. Pole przepływu analizowano używając pakietu Fluent. Przebadano kilka konfiguracji obiektu. Przedstawiono wyniki obliczeń oraz analizy sił. Zaprezentowano również kilka użytecznych procedur i sposobów wykorzystywanych podczas generacji siatek