International conference on Graphics Engineering
June 19th – 21st, 2013 Madrid,Spain
INGEGRAF – ADM – AIP PRIMECA

A new methodology to evaluate the flying shape of a sail

T. Ingrassia [(a)], V. Nigrelli [(a)], G. Gelsomino [(a)]

[(a)] Università degli Studi di Palermo – Dipartimento di Ingegneria Chimica, Gestionale, Informatica, Meccanica

Article Information

Keywords:
Sails,
Flying Shape,
CFD analysis,
FEM method.

Corresponding author:
Tommaso Ingrassia
Tel.:+39 091 23897263
Fax.: +39 091 23860672
e-mail: tommaso.ingrassia@unipa.it
Address: Viale delle Scienze – Ed. 8, 90128 Palermo - Italy

Abstract

In the last years, many improvements have been made in sailing yacht design with particular regard to shapes, materials and building technologies. As racing yacht competitiveness is more and more growing, there is an increasing need for improvements of the sails and the hulls performances. In this context, the Computational Fluid Dynamics (CFD) method is increasingly used in the design process of the sails. Numerical simulations, nevertheless, are usually performed only on the design shape of the sail, without considering that the real flying shape of a sail greatly varies when subject to the normal working conditions. For these reasons, often, the real performances of a sail are very different from the ones estimated during the design phase. To overcome this problem, in the presented paper, an innovative methodology has been developed to evaluate the real (flying) shape of a sail during the design process. In particular, an iterative procedure has been setup to analyse, by means of a coupled CFD/FEM analysis, the fluid-structure interaction in a simplified model of a sail. The method allows to predict the real flying shape of a sail and, consequently, to optimize its performances.

1 Introduction

The study of the optimal shape of a sail has become, over last years, one of the most interesting activities of research, particularly in the field of racing yachts.

Nowadays, to design the best shape of a sail and, consequently, to optimize the performances of a yacht, designers use software thanks to which they can define the “design shape” of a sail, characterized by a particular size, curvature and lengths depending on the imposed boundary conditions (for example wind intensity and direction).

These kinds of software, usually, are based on Computational Fluid Dynamics (CFD) methods and allow to evaluate the aerodynamics behaviour of the designed sails. Unfortunately, during a CFD simulation, the sail is considered as rigid and, consequently, it is impossible to predict its performances because, when subject to aerodynamic forces, it considerably changes its shape. Under these conditions a sail modifies its design shape into the “flying shape” [1-2]. That because sails are very deformable and under usual working loads they are subject to very large displacements. The flying shape, usually, is known when the sail is mounted on the rig and trimmed, after the prototype has been made.

The evaluation of the flying shape could be made through experimental techniques, like the photogrammetric method [2-3], but these approaches only allow to evaluate a posteriori the three-dimensional shape of a sail and cannot be used during the design phase to estimate the real sail performances.

Because of the remarkable differences between the design and flying shapes of a sail, usually, the real performances of a sail noticeably differ from the ones calculated during the design phase.

To reduce this performances gap it needs to calculate and to minimize the differences between the flying and the design shape.

This work aims to develop a new methodology able to evaluate, during the design phase, the flying shape of a generic transverse section of a sail. By means of an iterative procedure, basing on coupled numerical Fluid-Structure Interaction (FSI) analyses [4-5], the flying shape of a sail is calculated starting from its design shape.

To this goal numerical approaches, like the CFD and the Finite Element Method (FEM) analyses, have been used to setup 2D fluid-dynamic and structural simulations of the sail.

The developed procedure can be easily improved to study the three-dimensional flying shape of the sails and also integrated in an optimization loop to find the best design shape before manufacturing the prototype.

2 Evaluation of the flying shape: study of the fluid-structure interaction

The procedure here presented allows to evaluate the flying shape of the cross section of a sail through the numerical study of the fluid-structure interaction. The developed methodology is structured in the following way. In the first step the aerodynamic load, that is the pressure distribution on both sides of the sail, is calculated by means of a CFD code. After, the displacements of the sail, subject to the calculated pressure distribution, are evaluated by a FEM structural analysis. The CAD model of the deformed shape of the sail is then exported and used for a new CFD analysis in which the (previous) boundary conditions remain unchanged. The new calculated pressure distribution is used to setup another FEM simulation and to evaluate the up-to-date deformed shape of the sail section. The iterative process is continued as soon as the difference between the maximum elastic strain values, $\Delta\varepsilon_{max}$, of two following iterations is lower than a fixed threshold value $\Delta\varepsilon_t$. The (deformed) shape of the sail obtained during the last run of the iterative procedure represents the flying shape.

The implemented methodology can be summarized (fig. 1) with the following steps:

1) Setup of the CAD model (at the first loop equal to the design shape);
2) CFD analysis and evaluation of the pressure distribution;
3) FEM analysis and evaluation of $\Delta\varepsilon_{max}$;
   - if $\Delta\varepsilon_{max} > \Delta\varepsilon_t$, the CAD model is up to date with the current deformed shape and the procedure restarts from point 1;
   - if $\Delta\varepsilon_{max} < \Delta\varepsilon_t$, the procedure stops and the current deformed shape represents the flying shape.

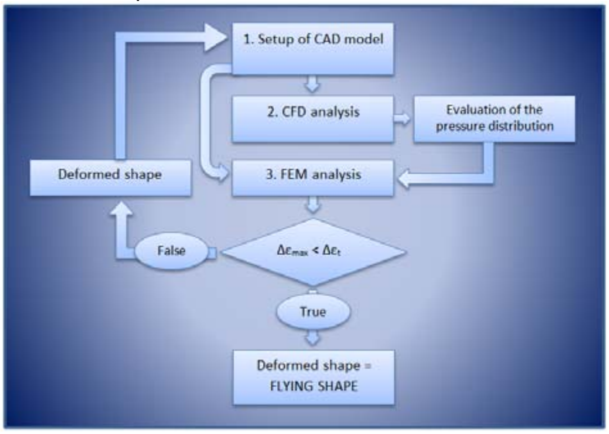


Figure 1 – Flow diagram of the developed procedure

3 Setup of the CFD simulation

The 3D model of the sail used in this work has been supplied by Doyle SailMaker [6], one the most important company in the field of yacht sails design and production. The CAD model of the analysed sail is shown in figure 2.

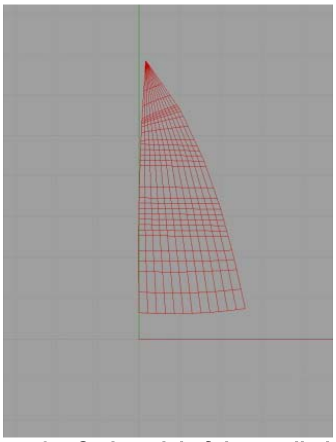


Figure 2 – Cad model of the studied sail

In the first phase of this research activity, to reduce the analysis computational time, it has been chosen to study only a 2D section of the sail, postponing the improvement of the procedure to analyse the full (3D) flying shape of the sail in a later stage. The evaluation of the flying shape has been made on the bottom section of the sail (fig. 3).

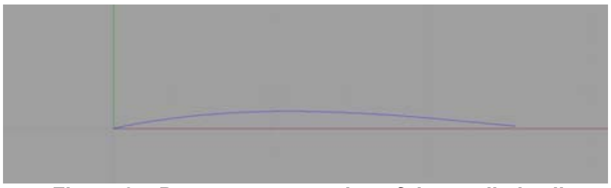


Figure 3 – Bottom cross section of the studied sail

Main dimensions of the sail section are the following:

- chord length L = 2,250m;
- maximum camber C = 0,185 m;
- thickness t=1,e – 003m.

An important factor in a fluid dynamics simulation is the choice of the studied domain. In this kind of problems, in fact, only a restricted region (where the analysed fluid dynamics parameters are really influenced) is analysed.

For this reason, to reduce the hardware resources requirement and to effectively reproduce the real physical conditions, additional fictitious boundaries must be conveniently created. The definition of these additional boundaries represents one of the most crucial steps in the setup of a CFD analysis [7] and, usually, are related to the delimitation of the fluid mass around the mast and the sail through an entry (inlet) and an exit (outlet) boundary of the fluid (fig. 4).

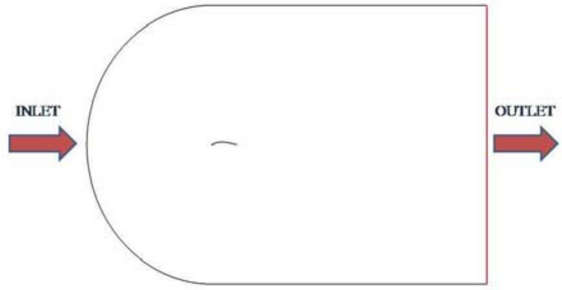


Figure 4 – Inlet and outlet boundaries of a CFD domain

If the inlet is positioned enough far from the must, the aerostatic pressure distribution can be assumed as uniform and, consequently, it can be hypothesized the condition of undisturbed fluid, so setting the turbulence parameters equal to zero.

With regard to the outlet boundary, instead, it is not simple to formulate any hypothesis and all the variables are, in general, unknown. For this reason, to simplify the calculation, avoiding for example the reflection and propagation of noises [8], all the derivatives related to the fluid longitudinal motion are imposed equal to zero and the fluid mass leaving the domain is regulated by means of the continuity equations.

For the presented case study, the fluid domain has been limited by imposing a distance from the must varying from 10 L (10 times the sail chord length) for the inlet boundary to 20 L for the outlet.

Another important aspect in a fluid dynamics analysis is the choice of an adequate quality of the mesh able to give reliable results. In some cases, a poor quality of the mesh can affect the real fluid flow. On the other hand, it is necessary to limit the nodes number because a too high quality mesh could require excessive computational resources and long calculation times. The mesh of the studied domain should be suitably improved only where high pressure and velocity gradients are expected. Often, when a simulation is very complex, the high gradient zones are not known a priori; in these cases, preliminary analyses should be carried out to roughly identify the fluid flow and the pressure distribution. In the analysed case, a poor mesh has been created in the undisturbed fluid flow zones, which are around the domain boundaries, and very high quality mesh has been imposed around the sail section (fig. 5). The used mesh is made of 18847 nodes and 18780 elements.

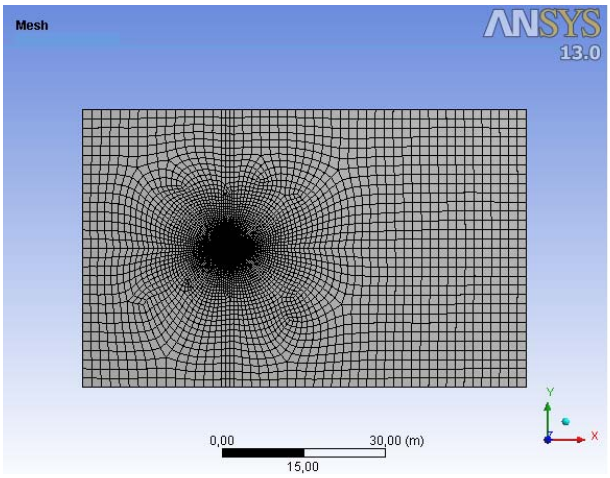


Figure 5 – Mesh of the CFD domain

3.1 CFD boundary conditions

Before running the numerical simulations, some analysis parameters have been set. In particular, the single-precision 2D approach [9] has been chosen to meet the requirements for good accuracy of the results and available hardware resources. The turbulence has been simulated by means of the K – ε model [10] because, in the studied problem, the turbulent flow is mainly composed of the shape resistance and the induced aerodynamic one [11-12]. Moreover, in the presented case study, it is not interesting to calculate the viscous forces into the boundary layer. For all these reasons, the K-ε model and the standard wall functions [7, 13] have been effectively used in simulating the analysed turbulent flow.

The fluid considered in the analysed domain is the air; it has been characterized with the following values:

- density ρ equal to 1,225 kg/m3;
- viscosity ν equal to 1,7894e-05 kg/m s.

Considering the physical characteristic of the case study, the following boundary conditions have been defined (fig. 6):

- velocity inlet for edges 1 and 2;
- pressure outlet for edges 3 and 4;
- wall for the sail section (edge 5).

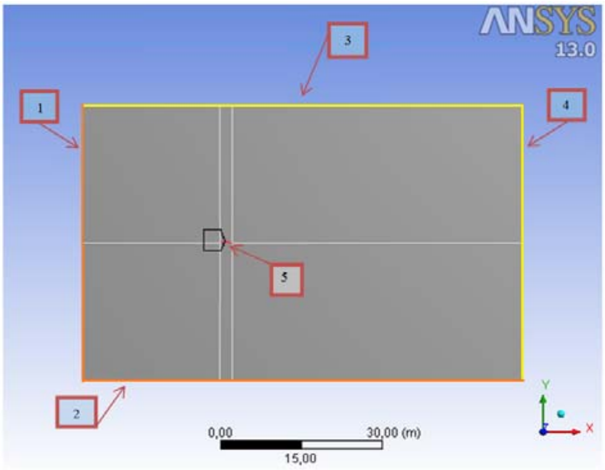


Figure 6 – Edges of the CFD domain

The velocity inlet condition [9] has been used to define the properties (velocity, direction, etc...) of the undisturbed fluid flow at the entrance of the domain. The fluid velocity has been considered as constant and equal to 5 m/s, the apparent wind angle (fig. 7), instead, has been imposed equal to 18°.

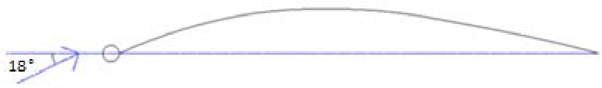


Figure 7 - Apparent wind angle

The pressure outlet condition has been used at the exit boundary of the domain, where the pressure and velocity values of the fluid flow are not well known. This condition has been also used to evaluate any possible contrail due to the sail. The "wall" option has been used for the sail by hypothesizing it fixed in a domain with a moving air flow.

The problem convergence has been determined by monitoring the residuals of both the equations and the most interesting quantities, like the drag and lift coefficients, in order to verify the achievement of the stationary conditions. Figure 8 shows the graphs of the main residuals over the iterations number during the first run.

viscosity ν equal to 1,7894e-05 kg/m s.

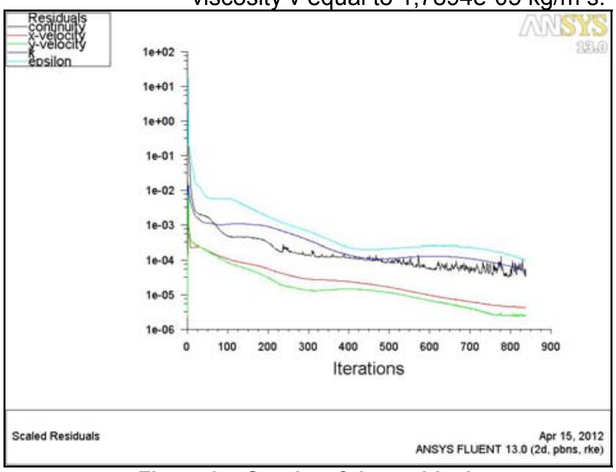


Figure 8 – Graphs of the residuals

4 Setup of the structural FEM analysis

To evaluate the deformation of the sail section due to the aerodynamic loads, a non-linear FEM analysis has been carried out. The material used for the sail is the polyester, having the following characteristics:

- Young modulus - 3400 MPa;
- Poisson coefficient - 0.38;
- Ultimate strength - 55 MPa.

During the meshing of the sail model, the Edge Sizing [9] option has been used and an element size equal to 1, e-003m has been imposed.

The aerodynamic loads applied to the structural FEM model have been deduced from the CFD analysis; they are equal, in fact, to the pressure distributions on both (internal and external) sides of the sail. The imported pressure distribution over the internal side of the sail section, obtained during the first run of the iterative procedure, is shown in figure 9.

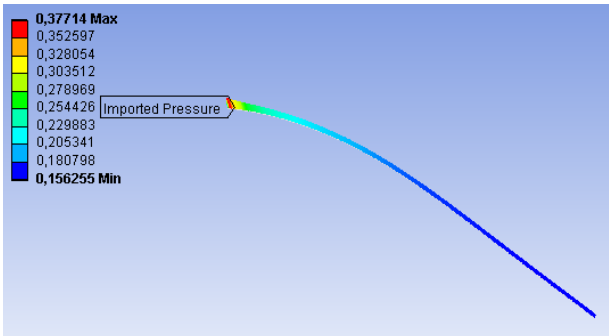


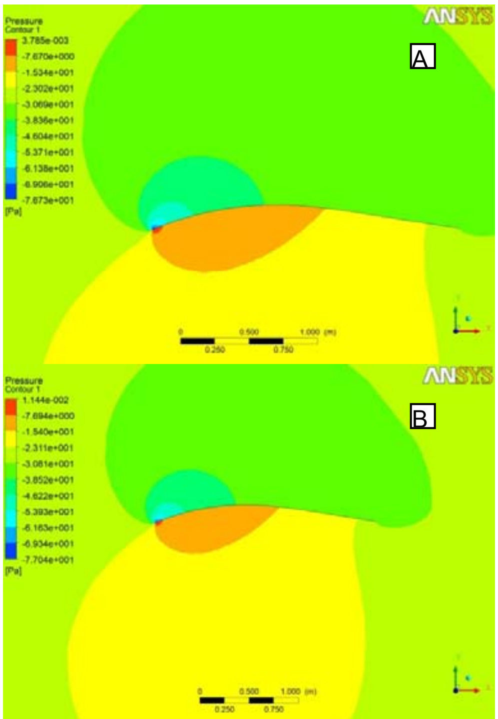
Figure 9 – Imported pressure distribution map on the internal side of the sail section

To simplify the setup of the boundary conditions, in a preliminary stage, the bottom section of the sail has been considered. For this reason, on the leading and the end edges of the sail, the x, y and z displacements have been constrained to simulate the real working conditions.

5 Results

The flying shape of the sail section has been found after three iterations.

The pressure distributions during each run are shown in figure 10.



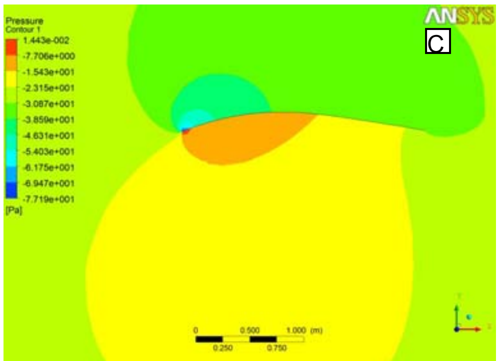


Figure 10 – Pressure distribution maps around the sail in the first (a), second (b) and third (c) run

It can be noticed the maximum pressure value on the leading edge (on the internal side of the sail) decreases more and more from the starting configuration to the final (flying) shape of the sail. To this end, it is useful also to analyse how the drag and lift coefficients, summarized in table 1, vary when the sail changes from the initial shape (run 1) to the flying one (run 3).

Lift coefficient	***Drag coefficient***
Cl_1 = 2,301	***Cd_1 = 0,08627***
Cl_2 = 2,286	***Cd_2 = 0,08628***
Cl_3 = 2,269	***Cd_3 = 0,08630***

Table 1 – Lift and drag coefficients

The lift coefficient decreases when the design shape of the sail changes in its flying shape, inversely the drag coefficient slightly increases. That means the sail, in real working conditions, reduces its performances.

With regard to the results of the structural analyses, in table 2 the maximum values of the equivalent (Von Mises) elastic strain are presented.

Maximum equivalent strain [m/m]
ε_{max1} = 1,7237E-05
ε_{max2} = 1,4646E-05
ε_{max3} = 1,3263E-05

Table 2 – Von Mises strain values

As said, the convergence of the iterative process has been obtained after three loops. During the third run, in fact, the calculated difference $\Delta\varepsilon_{max} = \varepsilon_{max3} - \varepsilon_{max2}$ = 0,1383E-05 is lower than $\Delta\varepsilon_t$ (set equal to 0,15E-05).

Figure 11 shows the maps of the elastic strain on the three deformed shapes of the sail.

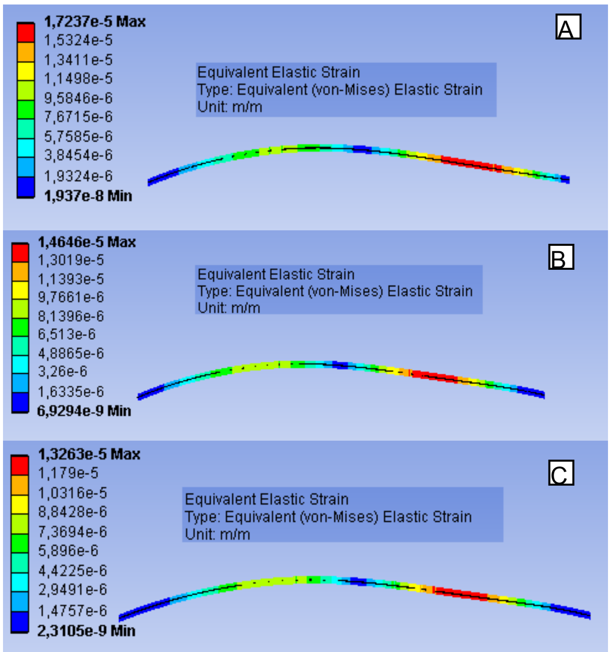


Figure 11 – Von Mises strain maps of the sail in the first (a), second (b) and third (c) run

6 Conclusions

One of the key aspects characterising the performance of a sail, in addition to its trim, is its flying shape. The behaviour of a sail, in fact, is remarkably affected by the real (deformed) shape and, usually, the efficiency of the flying shape is lower than the design one. For this reason, it is very important to predict the real shape of the sail during the design phase, so to reduce at most the gap of performance between the ideal and the real shapes.

The latest trends in the field of the racing yachts, like the America's Cup, related to the use of rigid sails, demonstrate the higher efficiency of these last ones due to a simpler control and prediction of their deformations in use.

In this work a new methodology to predict the real flying shape, during the design phase of a sail, has been studied. The developed (iterative) procedure is based on the numerical fluid-structure interaction analysis of the transverse section of a sail. In particular, in every single loop, the aerodynamic loads are evaluated and transferred to a nonlinear FEM model to evaluate the deformed shape of the sail that can be used for a following run. The process stops as soon as the maximum deformations of two following configurations differ less than a fixed value. In the analysed case study, the flying shape has been obtained after three runs. The obtained results demonstrate the flying shape has a lower efficiency if compared with the design one, so disappointing any prediction of the performances made during the design phase. Promising improvements of the presented procedure are related to the study of the whole three-dimensional shape of a sail and its integration in an optimization process in order to find, during the design phase, the best shape of a sail able to minimize the difference between the ideal and the real performances.

References

[1] Ranzenbach, R., and J. Kleene. 2002. Utility of Flying Shapes in the Development of Offwind Sail Design Databases. High Performance Yacht Design Conference

[2] Clauss G., Heisen W., CFD Analysis On The Flying Shape of Modern Yacht Sails, proceedings of IMAM 200,512 th International Congress of the International Maritime Association of the Mediterranean - Lisbon, 26-30 September 2005

[3] Graf K., Müller O. Photogrammetric Investigation of the Flying Shape of Spinnakers in a Twisted Flow Wind Tunnel, Proceedings 19th Chesapeake Sailing Yacht Symposium. 2009.

[4] Augier B., Bot P., Hauville F., Durand M. Experimental validation of unsteady models for fluid structure interaction: Application to yacht sails and rigs. Journal of wind engineering and industrial aerodynamics, 101, 53-66, 2012.

[5] Trimarchi D., Vidrascu M., Taunton D., Turnock S. R., Chapelle D., Wrinkle development analysis in thin sail-like structures using MITC shell finite elements. Finite Elements in Analysis and Design, 64, 48-64. 2013

[6] www.doylesails.com/

[7] Versteeg H.K., Malalasekera W., An introduction to Computational Fluid Dynamics, Prentice Hall, 1995.

[8] White F. M., Viscous Fluid Flow, Second Edition, McGraw-Hill International Editions, Mechanical Engineering Series, 1991

[9] Ansys Fluent Documentation - https://www.sharcnet.ca/Software/Fluent12/index.htm

[10] Speziale C. G., On nonlinear kl and k-ε models of turbulence. Journal of Fluid Mechanics, 178(1), 459-475, 1987

[11] Wilcox D.C., Turbulence modeling for CFD. Vol. 2. La Canada: DCW industries, 1998.

[12] Milgram J. H., Fluid mechanics for sailing vessel design. Annual review of fluid mechanics, 30(1), 613-653, 1998

[13] Blazek J., Computational fluid dynamics: Principles and applications, Elsevier, 2001