DESIGN OPTIMIZATION OF INTERACTING SAILS THROUGH VISCOUS CFD

V.G. Chapin, R. Neyhousser[+], G. Dulliand and **P. Chassaing**

DAEP/ISAE, Institut Supérieur de l'Aéronautique et de l'Espace, Toulouse, France
[+]Aquitain Design Team, Arcachon, France

SUMMARY

In this paper, we present *ADONF*, a new numerical simulation platform for viscous flows. This platform is developed and validated to compute analysis, design and optimization of real flows around complex sails configurations through a large number of automated Reynolds Averaged Navier-Stokes (RANS) simulations. We illustrate the ability of *ADONF* to search for robust, high performance, optimized fluid flows, in the field of the aerodynamic of sailing boats. Results presented focus on the optimization of the design and trimming of interacting sails because it is a question of major interest for many competitive sailors. Because *ADONF* is based on Navier-Stokes equations in RANS or URANS formulations, sail shape and trimming optimization in upwind and downwind sailing conditions may be addressed as will be shown. For future applications, it will be possible to extend and apply *ADONF* to hydrodynamic of sailing boats or any other fields in fluid mechanics governed by Navier-Stokes Equations.¶

NOMENCLATURE

β - Apparent wind angle
c - Sail chord
C_d, C_l - Drag and lift force coefficients
C_r - Driving force coefficient
C_h - Heeling force coefficient
C_p - Pressure coefficient
δ - Sail trim angle
δ_{GV1} - Windward mainsail trim angle
δ_{GV2} - Leeward mainsail trim angle
δ_f , δ_{jib} - Jib trim angle
f - Sail camber
f_{GV1} - Windward mainsail camber
f_{GV2} - Leeward mainsail camber
F_h - Aerodynamic heeling force
F_r - Aerodynamic driving force
i - Aerodynamic angle of attack
M_c - Aerodynamic heeling moment
S - Sail surface
x_f - Location of the sail maximum camber

1. INTRODUCTION

The work presented in this paper is a tentative of response to the following question: how to better design and better trim interacting sails to maximize sailing boats performance? Sail designers and competitive sailors know that this is a complex question. Existing tools are frequently based on crude modelization of the real flow. Moreover, VPP are necessary to predict sailing boat speed for a given measured or predicted aerodynamic force. Today, VPP are based on simplified aerodynamic and hydrodynamic models to predict aero-hydrodynamic forces and there dependences to design parameters. These models are crude representation of the real forces acting on sailing yachts [7, 9, 12, 20, 22]. Some of their drawbacks are known to result in some misleading predictions but increase their performance is not an easy task [7, 9, 12, 20, 22]. As said before by Korpus [15], experiments are probably the best method to predict theses forces by taking into account real world effects like viscous separation, unsteadiness, etc… But it is difficult to discard scaling effects during the transposition to real yachts. It is always difficult to take into account aero-structural coupling which may be important in sail design. Another difficulty specific to experiment is the ability to access all physical variables needed to better understand flow around bodies which may be helpful to guide future design.

The best validated modelization we have today is viscous Computational Fluid Dynamics (CFD) through RANS simulations [2, 3, 4, 14, 15]. The question of the usefulness of these advanced numerical tools for sail design is open. RANS codes have a critical drawback when used to predict forces acting on a yacht. They are time-consuming. But, the highest time consuming task in the process is the engineer time needed to generate meshes with a high quality standard. Moreover, design and optimization are complex multi parameters processes which need a large number of configuration variations.

These facts drive three questions to make RANS methods useful for yacht and sail designer:

- Is it possible to automate mesh generation and integrate RANS simulations into a user-friendly environment?
- Is it possible to qualify RANS predictions with experiments representative to real separated, unsteady flow conditions?
- Is it possible to decrease the time needed to compute hundred or more RANS simulations to search for optimums and robust designs?

In this paper we propose *ADONF*, a new numerical simulation platform which may be a response to these questions. The first novelty of *ADONF* is that it proposes a solution to the first question with an automated mesh generation process. The second one is we use a solver based on RANS equations and not inviscid ones. And the necessity of a viscous modelization for flow around mast and sails has been shown previously [4, 5, 13]. Also, with *ADONF* it is now easy to simulate hundred or thousand two-dimensional sails configurations to compute optimal design of multi parametric problems on a laptop [2, 3]. Extension to three-dimensional configurations is in progress and should not pose new specific problems. One difference should be true for few years: simulation time should not be compatible with laptop but should require a multiprocessor workstation or a supercomputer for computing hundred or more three-dimensional configurations.

In this paper, we will first present the computational model used by going through the solver, mesh issues, transition and turbulence modelling issues. Then, we will briefly describe the integrated, user friendly environment developed in *ADONF*. Then, in the next part, *ADONF* potential in sail design and optimization will be illustrated though examples. Typical and original configurations with one, two or three sails in upwind or downwind sailing conditions have been chosen.

2. COMPUTATIONAL MODEL

In this section, main elements of the computational model are described. First the fluid dynamics equations used to simulate the flow around interacting sails are presented then the solver and physical models and limitations are described. We are using viscous Navier-Stokes equations on hybrid meshes with structured and unstructured mesh part in the computational domain with conformal or non-conformal interfaces between domains. This is a powerful technology with high flexibility for mesh generation of interacting sails for two and three-dimensional flows.

2.1 GOVERNING EQUATIONS

The flow simulations around interacting sails presented in this paper are based on the numerical resolution of the following Reynolds Averaged Navier-Stokes equations:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial t}(\rho u_i) = 0$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[\tau_{ij} + R_{ij}]$$

With the viscous stress tensor

$$\tau_{ij} = 2\mu\,[S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}]$$

the deformation tensor

$$S_{ij} = \frac{1}{2}(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})$$

and the turbulent Reynolds stress tensor R_{ij} which should be modelled (see turbulence modelling part). Following Boussinesq hypothesis this tensor may be approximated by:

$$R_{ij} \cong \mu_T\,[S_{ij} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}] - \frac{2}{3}(\rho k)\,\delta_{ij}$$

2.2 SOLVER

The software package used to resolve the Navier-Stokes equations is Fluent 6. It is a steady or unsteady, compressible or incompressible, three-dimensional solver which resolve the previously given Reynolds Averaged Navier-Stokes (RANS) equations. In the simulations presented, we have used the segregated solver and the Spalart-Allmaras turbulence model in its vorticity-based or strain-vorticity-based production term. When not explicitly specified, second-order spatial and temporal schemes were used in the steady version.

To solve the Navier-Stokes equations proper boundary conditions are required on all calculation domain frontiers. At wall boundary, the no-slip condition is applied. A pressure outlet boundary condition is applied at the outlet. A velocity inlet boundary condition is applied on other frontiers (inlet, up and down).

2.3 MESH ISSUES

The mesh generation is a crucial step in the process of RANS simulation. First of all, it is a time consuming activity which need engineer experience and long practice to rigorously clean the CAD geometry and do the best choice for the mesh topology. Secondly, the mesh influence on results on typical sails configurations is may be important and should be carefully evaluated and bounded by relevant choices in mesh size distribution over main flow regions. Boundary layers have to be well resolved on all bodies (mast and sails) and this impose some critical criteria on mesh size in the normal and tangential direction to walls. But these criteria are not enough to obtain a good flow description and results independent to mesh. Other flow gradients may have to be well resolved. This is not a simple task on typical sails because of the zero thickness and the subsequent leading-edge pressure gradient when angle of attack is not ideal as frequently supposed in inviscid methods. From that point of view, hybrid mesh technology may be a critical issue for high-fidelity RANS simulations [4].

In fact, it is good to know that results are never totally independent to the chosen mesh as opposed to what is frequently argued. The relevant question when interpreting RANS results on sails is: how bounded is the mesh influence on physical quantities of interest.

To illustrate the mesh convergence, figure 1, the lift-to-drag ratio convergence with mesh number of points on a typical sail (f/c = 12.5%, Re = 1.4 x 10^6) calculated on four meshes have been shown. A good convergence of this critical performance parameter with the mesh number of points may be seen.

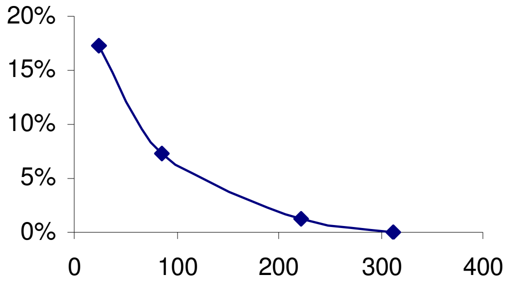

Figure 1: lift-to-drag ratio convergence with mesh refinement (number of points divided by 1000

Another important feature of mesh is their adaptability or flexibility to be used with different kind of sail geometries and trim angles. A critical point for yacht rig aerodynamic studies is the necessity to generate meshes on multiple bodies (mast, mainsail, jib, etc…) which interact and may be displaced relative to others. The challenge is to generate good quality meshes in the boundary layers regions of each body without using too high aspect ratio cells. To respect these topologic constraints and obtain good mesh control, a good candidate is hybrid meshes (as may be seen on figure 2) with eventually non conformal interface between the inner structured region around masts and sails and the outer unstructured region around all interacting structured domains (figure 2) as was done with Gambit 2.2 [10]. The mast trailing-edge with link to the zero-thickness sail is a region of difficulty for the structured mesh part and need much more attention and some tricks.

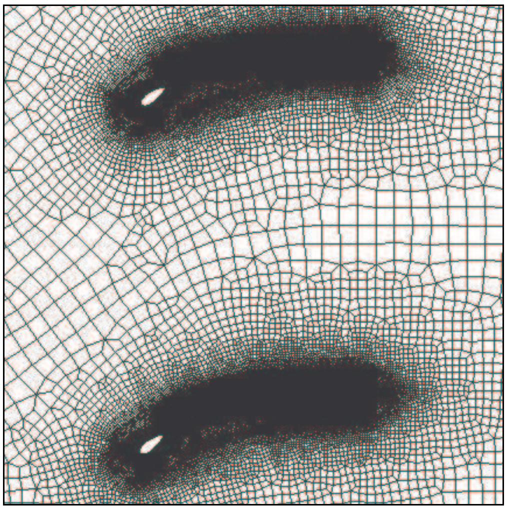

Figure 2: hybrid mesh example

2.4 TRANSITION & TURBULENCE MODELING

A reliable prediction of the boundary layer transition through computer simulation is always a challenge today. The transition of a boundary layer is a highly complex physical phenomenon. It is a problem of stability of the Navier-Stokes equations which are highly sensitive to background turbulence level, pressure gradient, surface roughness, etc… The range of existing transition prediction methods extends from simplified empirical relationships through those based on linear stability to direct numerical simulations. All of these methods have critical limitations. No transition models are implemented in RANS simulations. Eventually transition may be tripped when transition location is known.

In the same time, mast and sail aerodynamic is highly concerned with separation bubble, turbulent transition and turbulent reattachment process and it is well known that these phenomenon and their associated pressure losses may have critical influence on pressure and friction distribution on sails. Also an accurate representation of laminar and turbulent separated flow regions is critical when we are concerned with drag prediction.

Despite this, we will see in this paper that a simple low cost turbulence model like the Spalart-Allmaras model may have coherent qualitative behaviour on mast-sail geometries and may reveal to be better than more sophisticated ones.

The Spalart-Allmaras turbulence model used is a one equation model with standard coefficients values. The equation is a transport equation for the turbulent viscosity as follow:

$$\frac{\partial}{\partial t}(\rho\tilde{\nu})+\frac{\partial}{\partial x_i}(\rho\tilde{\nu}u_i)=P+Diff-Diss$$

$$P=C_{b1}\rho\tilde{S}\tilde{\nu}\ ,\ \tilde{S}\equiv S+\frac{\tilde{\nu}}{\kappa^2 d^2}f_{v2}\ ,\ f_{v2}=1-\frac{\chi}{1+\chi f_{v1}}$$

$$Diff=\frac{1}{\sigma_{\nu}}[\frac{\partial}{\partial x_j}[(\mu+\rho\tilde{\nu})\frac{\partial\tilde{\nu}}{\partial x_j}+C_{b2}\rho(\frac{\partial\tilde{\nu}}{\partial x_j})^2)]$$

$$Diss=C_{w1}\rho f_w(\frac{\tilde{\nu}}{d})^2$$

$$\mu_T=\rho\tilde{\nu}f_{v1}\ ,\ f_{v1}=\frac{\chi^3}{\chi^3+C_{v1}^3}\ ,\ \chi=\frac{\tilde{\nu}}{\nu}$$

$$S_{vorticity-based}\equiv\sqrt{2\Omega_{ij}\Omega_{ij}}\ ,\Omega_{ij}=\frac{1}{2}(\frac{\partial u_i}{\partial x_j}-\frac{\partial u_j}{\partial x_i})$$

$$S_{strain-vorticity-based}\equiv\left|\Omega_{ij}\right|+C_{prod}\min(0,\left|S_{ij}\right|-\left|\Omega_{ij}\right|)$$

$$C_{prod}=2\,,\left|\Omega_{ij}\right|\equiv\sqrt{2\Omega_{ij}\Omega_{ij}}\ ,\left|S_{ij}\right|\equiv\sqrt{2S_{ij}S_{ij}}$$

$$f_w=g[\frac{1+C_{w3}^6}{g^6+C_{w3}^6}]^{1/6}\ ,\ g=r+C_{w2}(r^6-r)\ ,\ r=\frac{\tilde{\nu}}{\tilde{S}\kappa^2 d^2}$$

$$C_{b1}=0.1355, C_{b2}=0.622, \sigma_{\nu}=2/3, C_{v1}=7.1$$

$$C_{w1}=\frac{C_{b1}}{\kappa^2}+\frac{1+C_{b2}}{\sigma_{\nu}}, C_{w2}=0.3, C_{w3}=2.0, \kappa=0.4187$$

3. INTEGRATED & AUTOMATED COMPUTATIONAL ENVIRONMENT

Fluid motion around interacting sails in their real environment is complex with separated flow regions and unsteady phenomena. Because there are a lot of parameters that define a sail design or a complete rig design, there is a crucial need to integrated and automate the entire simulation process. Turnaround time of the simulation process is another constraint. *ADONF* is a response which gives us the ability to analyse a large number of configurations and design. It opens a new way to design and optimize fluid motion around interacting sails through viscous methods.

Because, we think that this is a central question for the usefulness of RANS simulations in yacht design, we have developed this new numerical tool which integrate and automate the entire computational environment of RANS flow simulation from CAD definition, to mesh generation, flow simulation, flow analysis and design modifications through optimization following the diagram of figure 3. The main difficulty is the mesh generation process automation. When the problem has been resolved, we have obtained a tool which generates meshes of high reliability. This particular property of automated meshes increases our ability to compare and rank different design or sail trim. It also increases ease to study the mesh density influence on results.

As it will be shown through examples, with *ADONF*, it becomes possible to investigate and resolve new questions about fluid motion around designed bodies and their related performances.

The first level of new questions that can be investigated is the "what-if" questions. What will be the performance of this rig design if I change the mast section? What will be the performance of this rig if I change the sail recovery factor preserving a constant sail surface? Etc… It is only the imagination of the sail designer which limits the process. We don't need many engineers to generate all the meshes needed to respond to these questions.

In a second phase, we have implemented optimization algorithms in *ADONF*. With optimization algorithms, a second, higher level, set of questions becomes open to the designer. How to change the rig design or the deck plan to increase the performance of that particular sailing boat in given wind conditions? How to change rig trimming to increase boat speed in given wind conditions? What is the best camber and trim of these two interacting sails to maximize driving force or driving to heeling force ratio? Etc… Let us illustrate this through examples now.

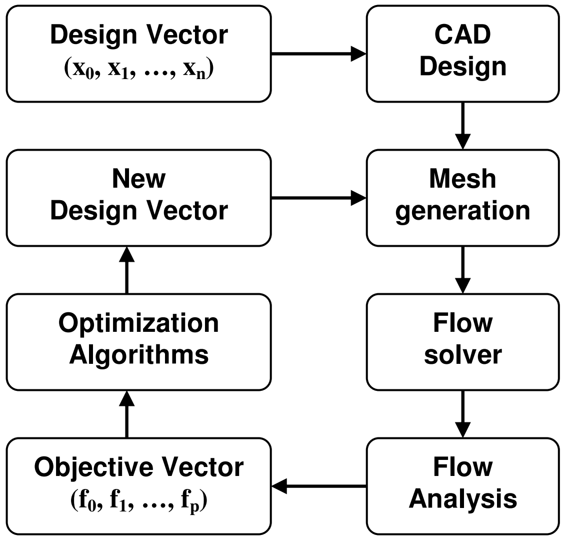

Figure 3: *ADONF* software package diagram

4. RESULTS

4.1 OPTIMUM CAMBER OF A SINGLE SAIL

From the excellent work of F. Bethwaite [1], we know that in real viscous flow, there exists an optimum sail camber. This observed fact has been chosen as an interesting question to test the ability of *ADONF* and implemented optimization algorithms to resolve sail design questions.

For a single sail the optimization problem may be formulated as follow: for a given apparent wind angle, what is the optimal camber and related trim angle which maximize the driving force Fr? The same question is also posed for maximizing the lift-to-drag ratio Fr/Fh. The apparent wind angle chosen was β = 30°. Other sail parameters are listed in the following table:

β	x_f	C	f_0	δ_0
30°	30%	6500	7%	13°

The optimization algorithm first used was a gradient based algorithm known as the Simplex method. After about twenty RANS simulations, a good convergence is obtained (figure 4, 5, 6) and *ADONF* found the following optimal solutions for maximum driving force and maximum lift-to-drag ratio:

Objective	f_{GV}	i_{GV}
Max(Fr)	18%	7°
Max(Fr/Fh)	8%	3°

The number of needed RANS simulations to determine the optimal solution was dependent to the initial condition but the optimal solution was independent to the initial condition. An example of algorithm convergence is given on figures 2 & 3 for camber, trim angle and driving force.

It is interesting to note that the optimal solution, maximizing the driving force, present a separation near the trailing-edge on the suction surface (figure 7). This point clearly illustrates the ability of viscous CFD to make a trade-off, between high camber and massive separation, through RANS simulations.

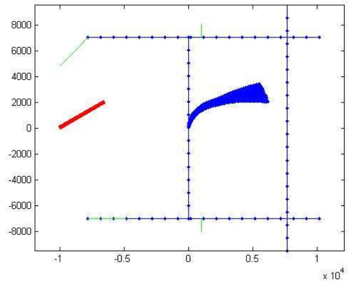

Figure 4: configurations of the optimization problem

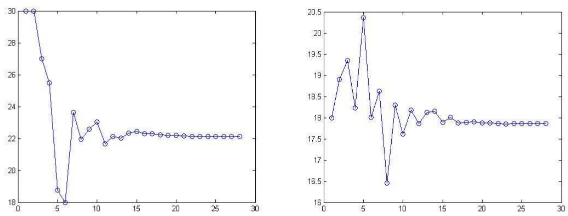

Figure 5: trim and camber convergence

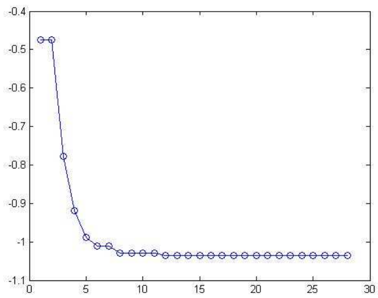

Figure 6: driving force convergence

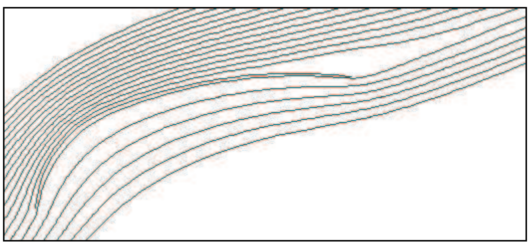

Figure 7: stream function around the sail for maximum driving force at β = 30°

4.2 TWO INTERACTING SAILS

A more challenging optimization problem is the two interacting sails problem which is typical of mainsail-jib interaction on a sailing boat and is well known as a problem of long debates and controversies [11, 16]. The question is to know if *ADONF* may be used to clarify this problem without passion.

The optimization problem may be formulated as follow: for a given apparent wind angle, what are the optimal cambers and related trim angles which maximize the driving force Fr? The same question is also posed for maximizing the lift-to-drag ratio Fr/Fh. The apparent wind angle chosen was β = 30°.

As in the previous case, we have used the Simplex algorithm for optimization process. The results found are listed in the following table:

Table 4: optimum design & trimming results

Objective	f_{GV}	i_{GV}	f_{JIB}	i_{JIB}
Max(Fr)	27%	27°	30%	-2°
Max(Fr/Fh)	4%	20°	19%	-1°

The solution that maximizes the driving force is visualized on figure 8. As in the previous case, small separation regions are found near the trailing-edge on the suction surfaces.

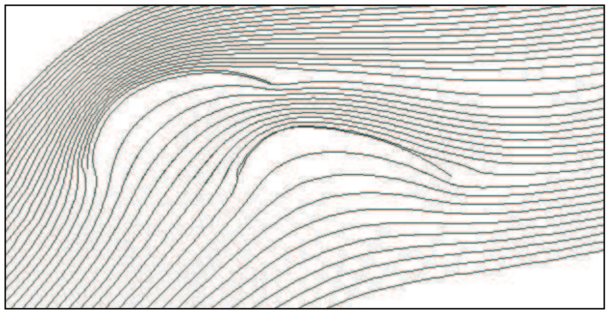

Figure 8: stream function around two interacting sails for maximum driving force at β = 30°

For further investigation on this problem, it will be possible to take into account a constraint on the heeling moment to obtain more realists designs for a given boat. This constraint may be easily added through a penalty method or another efficient constraint handling method [8, 19].

4.3 THREE INTERACTING SAILS

Another interesting and complex problem of interacting sails on which we have had the chance to work is the double rig of the Hydraplaneur of Yves PARLIER, an ocean racing multihull design by the Aquitaine Design Team [6]. This new rig, as shown in figure 9 opens new questions about sail design. One question is related to the differential trimming and differential loading between the windward and the leeward rig. Is there an optimum? How this optimum change with apparent wind direction?

Here, the question of the optimum driving force of this double rig with three interacting sails in downwind sailing conditions is open.

A design of experiment on a three sails configuration with five variables has been done with *ADONF* (figure 10). A set of points have been obtained in the driving force, heeling moment plane (figure 11). The mesh influence on results has been qualified to choose the mesh number of points. From the three sails design of experiment, a complementary set has been defined and another one with 4 sails and nine variables (figure 11). It is shown that the 3 sails complementary set and the 4 sails set increase the aerodynamic performance of the initial set. It may be interesting to evaluate multi-objective optimization methods to determine the Pareto frontier of this rig with more precision.

Another recurrent question is to know if the addressed optimization problem presents a complex response surface with local optimums and a global optimum. If the response to this question is positive, optimum solutions found by gradient based algorithm like the Simplex method may be dependent to the initial condition of the optimization process. It may be useful, in this case, to use evolutionary algorithms. This argument is frequently used to justify the usefulness of genetic algorithms but problems with this property are difficult to find [21]! Also we have tried to visualize a sub-problem response surface of the three sails optimization problem by taking into account only the two major variables of the five variables problem. These variables are trim angles of the two mainsails (δ_{GV1}, δ_{GV2}). The response surface based on these two variables is shown on figure 12. It is interesting to note that this response surface exhibits three local optimum of the driving coefficient and a global optimum. This fact clearly illustrates the usefulness of evolutionary algorithms for aerodynamic optimization of viscous flows based on RANS or URANS simulations.

Figure 9: Hydraplaneur double rig

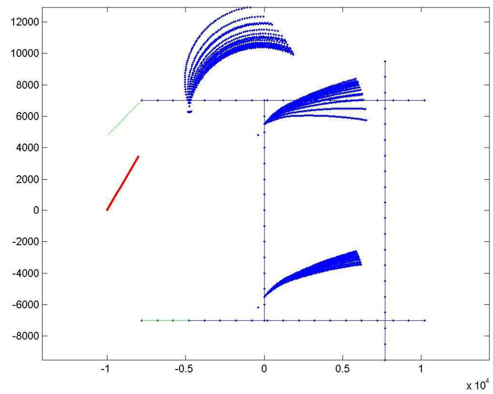

Figure 10: Optimization of the three sails rig of the Hydraplaneur in downwind sailing condition

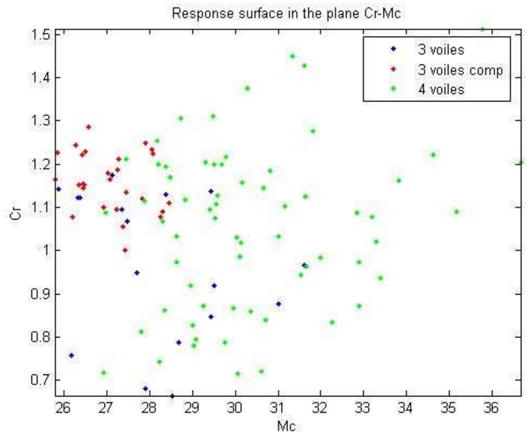

Figure 11: driving force versus heeling moment of the various Hydraplaneur configurations

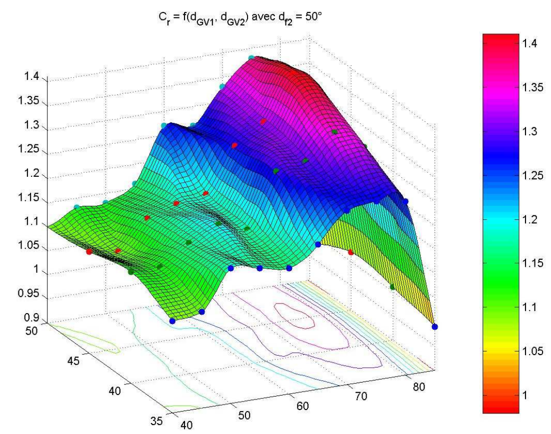

Figure 12: response surface of the driving force versus the two mainsails trim angles (δ_{gv1}, δ_{gv2}).

5. CONCLUSIONS

Results obtained with *ADONF* software package and presented in this paper demonstrate that today, it is possible to do analysis, design and optimization of complex interacting sails configurations with viscous CFD through automated RANS simulations.

Needed conditions to do a good job with this kind of viscous CFD are the following:

- To have a user-friendly environment to run hundred or more RANS simulations with ease on a laptop.
- To use a high-fidelity RANS solver with appropriate hybrid meshes for separated flows on mast and sail configurations [4].
- To implement optimization algorithms and probably evolutionary methods for complex rig configurations with multi-modal optimal solutions.
- To have a background in sail design and an expertise in viscous CFD to resolve conceptual design questions through optimization.

Today, we are working on the extension of *ADONF* to three dimensional flows with a parameterization of sail shape relevant for sail designers. For future applications, it will be interesting to couple *ADONF* with a hydrodynamic solver. Then design optimization might be done directly on boat speed.

6. REFERENCES

1. Bethwaite F., "High performance sailing", Waterline Books, p209, 1996.
2. Chapin V. G., Neyhousser R., Dulliand G., Chassaing P., "Analysis, Design and Optimization of Navier-Stokes Flows around Interacting Sails", *MDY06 International Symposium on Yacht Design and Production*, Madrid, Spain, March 2006.
3. Chapin V. G., Neyhousser R., Jamme S. Dulliand G., Chassaing P., "Sailing Yacht Rig Improvements through Viscous CFD", *SNAME 17th Chesapeake Sailing Yacht Symposium*, Annapolis, Maryland, USA, March 2005.
4. Chapin V. G., "Viscous CFD as a relevant decision-making tool for mast-sail aerodynamics", *Marine Technology*, **42**(1), p1-10, 2005.
5. Chapin V.G., Neyhousser R., Jamme S., "Aérodynamique grand voile - mât", *2nd Workshop Science & Voile*, Ecole Navale, Lanvéoc-Poulmic, May 2004.
6. Chapin V.G., "Gréement Gerris", Contract report n°2002-2, 2002.
7. Claughton A., "Development in the IMS VPP Formulations", *SNAME 14th Chesapeake Sailing Yacht Symposium*, Annapolis, USA, 1999.
8. Deb, K., "An efficient constraint handling method for genetic algorithms", *Computer Methods in Applied Mechanics and Engineering, 186*, 311-338, 2000.
9. Fossati F. & al., "Wind Tunnel Techniques for Investigation and Optimization of Sailing Yachts Aerodynamics", *Proceeding of the 2nd High Performance Yacht Design Conference*, Auckland, February 2006.
10. Gambit 2.3 & Fluent 6.2 users guide and tutorials, http://www.fluent.fr/
11. Gentry Arvel, "The Aerodynamics of Sail Interaction", *3th AIAA Symposium on the Aero/Hydronautics of Sailing*, November 1971.
12. Jackson P.S., "An improved upwind sail model for VPPs", SNAME 15th *Chesapeake Sailing Yacht Symposium,* Annapolis, USA, 2001.
13. Jones P., & Korpus R., "International America's Cup class yacht design using viscous flow CFD", *SNAME 15th Chesapeake Sailing Yacht Symposium*, Annapolis, Maryland, USA, January 2001.
14. Korpus R., "Reynolds Averaged Navier-Stokes in an Integrated Design Environment", *MDY04 International Symposium on Yacht Design and Production*, Madrid, Spain, 2004.
15. Korpus R., "Performance Prediction without Empiricism: A RANS-Based VPP and Design Optimization Capability", *SNAME 18th Chesapeake Sailing Yacht Symposium*, Annapolis, Maryland, USA, March 2007.
16. Marchaj C. A., *Sail Performance*, McGraw Hill, Great Britain, 1996.
17. Milgram J.H., "Sail Force Coefficients for Systematic Rig Variations", September 1971.
18. Milgram J.H., "Effects of Masts on the Aerodynamics of Sail Sections", Marine Technology, vol. 15(1), 35-42, 1978.
19. Müller S.D., "Bio-Inspired Optimization Algorithms for Engineering Applications", PhD Thesis, 2002.
20. Oossanen P. van, "Predicting the Speed of Sailing Yachts", SNAME Transactions, Vol 101, 337-397, 1993.
21. Pulliam T.H., Nemec M., Holst T., Zingg D.W., "Comparison of Evolutionary (Genetic) Algorithm and Adjoint Methods for Multi-Objective Viscous Airfoil Optimizations, AIAA paper 2003-0298, Reno, USA, 2003.

22. Teeters J. & al., “Changes to Sail Aerodynamics in the IMS Rule”, *SNAME 16th Chesapeake Sailing Yacht Symposium*, Annapolis, Maryland, USA, March 2003.

8. AUTHORS’ BIOGRAPHIES

Vincent G. Chapin, Associate Professor at ISAE. He is responsible of the team “Flow Control & Optimization” in the Fluid Mechanics Department of ISAE. His research is related to flow control, optimization and innovating aerodynamic.

Romaric Neyhousser, Aquitain Design Team leader and crew member during the Hydraplaneur project.

Guillaume Dulliand, Mesh specialist at the Fluid Mechanics Department of ISAE during this work. Now he is in formation at the von Karman Institute.

Patrick Chassaing, Professor INPT.