High Performance Yacht Design Conference
Auckland, 4-6 December,2002

IMPROVING THE DESIGN OF SAILS USING CFD AND OPTIMIZATION ALGORITHMS

Sriram Shankaran[1], ssriram@stanford.edu
Tyler Doyle[2], tyler@stanford.edu
Margot Gerritsen[3], margot.gerritsen@stanford.edu
Gianluca Iaccarino[4], jops@ctr.stanford.edu
Anthony Jameson[5], jameson@baboon.stanford.edu

Abstract. Two current Stanford Yacht Research projects are presented, both related to the analysis and design of upwind sails, and part of a larger research effort to develop efficient and robust sail and hull shape optimisation methods.
First, we discuss the in-house development of a methodology to predict and improve the flying shape and forces on America's Cup sails. We adapted and parallelized a 3D unstructured incompressible Euler solver developed by Jameson and colleagues. The computed pressure loading is transferred to the structural package NASTRAN, which computes the deflected shape of the sail. The mesh is displaced accordingly, and a new pressure loading is computed. This process is repeated until convergence. The computed lift and induced drag for a main sail with elliptic planform compares well with results from lifting surface theories. The flow solver is more efficient than commercial solvers and highly suitable for the computationally intense viscous sail flow calculations.
Second, we present a sail shape optimisation method, which combines the commercial CFD package FLUENT with gradient-based cost function minimisation. Results are presented for the optimisation of sheeting angles for the rig of a three masted clipper yacht. We investigate two cost functions, both characteristic of a sail's aerodynamic performance: the reciprocals of the driving force coefficient and the ratio of driving to heeling force coefficients. Comparing results for upwind and close reaching apparent wind angles shows that the latter leads to well trimmed sails, whereas the former causes the sails to be over trimmed, which is as expected.

1. INTRODUCTION

What CFD and optimisation methods are (most) suitable for efficient and accurate performance analysis and shape optimisation of yachts? At the newly founded Stanford Yacht Research group (SYR) we are looking to answer this question for a variety of yacht designs and sailing configurations. Our ultimate goal is to develop efficient and reliable shape optimisation codes for both sails and hulls. We investigate the use of commercially available codes, but also develop methods and software in-house specifically tailored to yachting.

1.1 Sail and hull shape optimisation

Several strategies can be used to optimise the shape of sails or hulls, varying in degrees of robustness and computational intensity. Parametric studies are easiest to implement, as they only require an efficient flow solver. Here, various control parameters, such as camber, draft and sheeting angle, are manually adjusted in a trial-and-error fashion. More robust and automated procedures can be built using gradient-based cost function minimisation, adjoint optimisation methods, or evolutionary algorithms.

In the first approach, a cost function characteristic of the performance of the sail or hull is minimised with respect to one or more control parameters. We describe this optimisation method in more detail in section 3, and discuss results for the optimisation of sheeting angles of a three-masted clipper yacht. The iterative procedure requires the (usually expensive) calculation of the sensitivity derivatives of the cost function with respect to each of the control parameters in each iteration step.
An alternative is the adjoint or control theory approach. Here, the necessary gradients are obtained via the solution of the adjoint of the equations describing the fluid flow. The adjoint method is much more efficient because the expense incurred in the calculation of the gradient is effectively independent of the number of design variables [1]. The cost of the adjoint solve is comparable to the cost of the flow solution. This method has become a popular choice for design problems involving fluid flow, and has been successfully used for the aerodynamic design of aircraft configurations [2]. We are currently extending our in-house flow solver described in section 2 to include adjoint optimisation.
A third optimisation approach is to use evolutionary algorithms (EAs) [3]. EAs are biologically inspired optimisation algorithms, imitating the process of natural evolution. EAs do not require gradient information, but use a set of solutions (population) to find the optimal

1 Doctoral Candidate, Department of Aeronautics and Astronautics, Stanford University
2 Doctoral Candidate, Department of Mechanical Engineering, Stanford University
3 Assistant Professor, Stanford Yacht Research (http://syr.stanford.edu), Stanford University
4 Research Associate, Center for Turbulence Research, Stanford University
5 Thomas V. Jones Professor of Engineering, Department of Aeronautics and Astronautics, Stanford University

designs. The population based search allows parallelization, and may avoid premature convergence to local minima. However, the population normally must be large, thus requiring many flow calculations. Also, the procedure is further removed from the physics than the adjoint method, and is therefore perhaps less appealing to engineers. We will be exploring this approach in the near future as extension of the design project described in section 3.

1.2 Current studies

In sections 2 and 3 of this paper, we present preliminary results of two of our main projects.

The first project is the in-house development of efficient solvers for the incompressible Euler and Navier-Stokes equations. These solvers will be extended with an adjoint method for sail and hull optimisation. Here, we present the application of the incompressible Euler code to predict the flying shape and forces on upwind America's Cup sails. The Euler code is an adapted and parallelized version of the 3D unstructured Aircraft Euler code developed by Jameson and colleagues to compute flows past aircraft [4]. This code is much more efficient than commercial solvers such as FLUENT or CFX, and therefore highly suitable for the computationally intense 3D sail flow calculations. The code is iteratively coupled to the structural package NASTRAN to compute the deflected shape of the sail.

In the second project we investigate gradient-based cost function minimisation methods and evolutionary algorithms for sail shape optimisation. The commercial CFD package FLUENT is used to solve for the fluid flow. In section 3, we present results for the optimisation of the sheeting angles for a three masted yacht, based on gradient-based optimisation. Here, we compute flow past 2D sail cross sections half way up the masts, and limit ourselves to upwind and close reaching apparent wind angles. We are extending the technique to full 3D calculations, and downwind calculations.

2. PROJECT I: PERFORMANCE ANALYSIS FOR UPWIND AMERICA'S CUP SAILS

In this project, we adapted and parallelized a in-house 3D unstructured incompressible Euler solver for the performance analysis of upwind AC sails. The solver is iteratively coupled to the structural package NASTRAN to determine the flying shape of the sails. The use of parallel computing combined with multigrid and residual averaging techniques allows for fast turn-around times which becomes critical when the simulations are embedded in a design environment. The resulting flow solver is much more efficient than commercial solvers and highly suitable for the computationally intense viscous sail flow calculations.

2.1 The mathematical model

Jameson and co-workers developed very efficient numerical solvers for compressible fluid flow with embedded supersonic regions [4]. These solvers can not be used directly for the incompressible sail flow calculations. There are two complications. First, an extra time dependent constraint must be imposed on the momentum equations to ensure a divergence-free velocity field. Second, the use of compressible flow solvers in the incompressible flow limit results in extremely stiff equations because the speed of sound goes to infinity. To overcome this difficulty, we adopt the artificial compressibility method, an approach first proposed by Chorin in 1967 [5]. The basic idea behind artificial compressibility is to introduce a pseudo-temporal equation for the pressure through the continuity equation. This approach removes the troublesome sound waves. When combined with multigrid acceleration procedures, artificial compressibility proves to be particularly effective [6]. Converged solutions of incompressible flows over a main sail can be obtained in about 75-100 multigrid cycles.

Using the idea of artificial compressibility, the equations of motion of an incompressible, inviscid fluid can be cast in the form

$$\frac{\partial W}{\partial t} + P\left\{\frac{\partial f}{\partial x} + \frac{\partial g}{\partial y} + \frac{\partial h}{\partial z}\right\} = 0 \qquad (1)$$

The dependent variables are stored in the state vector *W*, equal to $W = [p, u, v, w]^T$. The inviscid flux vectors *f*, *g* and *h* and the preconditioning matrix *P* are described by

$$f = \begin{bmatrix} u \\ u^2 + p \\ uv \\ uw \end{bmatrix}, g = \begin{bmatrix} v \\ vu \\ v^2 + p \\ vw \end{bmatrix}, h = \begin{bmatrix} w \\ wu \\ wv \\ w^2 + p \end{bmatrix}$$
$$P = \begin{bmatrix} \Gamma^2 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \qquad (2)$$

In integral form, the equations are

Conservation of Mass

$$\frac{\partial}{\partial t}\int_V p\, dV + \int_S \Gamma^2 (\mathbf{u}\cdot\mathbf{n}) dS = 0 \qquad (3)$$

Conservation of Momentum

$$\frac{\partial}{\partial t}\int_V \mathbf{u}\, dV + \int_S \mathbf{u}(\mathbf{u}\cdot\mathbf{n})dS = -\int_S p\mathbf{n}\, dS \qquad (4)$$

Here, p is the pressure, $\mathbf{u}$ is the velocity vector, $\mathbf{n}$ is the surface unit normal, and V and S are the volume and surface of the control volume respectively. This integral form is used in a finite volume method on an unstructured tetrahedral grid. The unstructured grid allows for flexibility in handling multiple sail or hull geometries. More details on the finite volume method can be found in [7].

The parameter Γ is called the artificial compressibility parameter and may be interpreted as an artificial speed of sound, or a relaxation parameter for the pressure iteration. When the temporal derivatives tend to zero, the set of equations satisfy precisely the incompressible Euler equations, so that the correct pressure may be established.

The choice of Γ is crucial in determining the convergence and stability properties of the numerical scheme. Typically, the convergence rate of the scheme is dictated by the smallest wave speeds present and the stability of the scheme by the largest. In the limit of large Γ, the difference in wave speeds can be large. Although this situation would presumably lead to a more accurate solution through the penalty effect in the pressure equation, very small time steps would be required to ensure stability. Conversely, for small Γ the difference in the maximum and minimum wave speeds may be significantly reduced, but at the expense of accuracy. A compromise between the extremes is achieved by choosing Γ to be $\Gamma^2 = C(u^2 + v^2 + w^2)$, with C a constant of order unity. In regions of high velocity and low pressure where suction occurs, Γ is chosen large to improve accuracy, and in regions of low velocity, Γ is correspondingly reduced.

We note that to prevent odd-even decoupling at adjacent nodes, which may lead to oscillatory solutions, a dissipation term is added to the flux calculations in the finite volume scheme. For further details see [8].

The resulting system of equations is integrated in time using an explicit multistage scheme combined with local time-stepping and residual averaging techniques [8].

Further acceleration of convergence is provided by the use multigrid techniques, which are described in [9].

The flow solver was parallelized using efficient load-balancing and inter-processor communication routines. Linear speed-up has been observed on variety of platforms, which include the SGI Origin 2000, and linux clusters using Athlon processors.

2.3. Structural sail model and aero-elastic coupling

The structural model used for the present calculations assumes the sail to be a flexible isotropic 1 *mm* thick membrane. Quadrilateral membrane elements are used to model the sail geometry thereby forcing the sail to resist all external loads through tension. The battens on the main sail and the tension stays are not included in the structural model. Also, the translation degrees of freedom at the boom and along the mast of the main sail are suppressed. For the head sail, the translational degrees of freedom along the front stay and the clew are suppressed while allowing the boom to deflect.

The loads from the fluid flow computation are transferred in a conservative manner to the sail geometry. The deflected shape of the sail is obtained by using a non-linear model which increments the loads in stages. Usually, about 5 load increments are used to obtain the deflected shape for a given pressure loading.

The deflected shape obtained from the structural model is used to deform the computational mesh for the fluid to obtain a new pressure loading. The above process is repeated until a converged deflected shape of the sail is obtained. The computational mesh is deformed using a spring analogy method, whereby the edges of the mesh are substituted with springs whose stiffness constants are inversely proportional to their lengths. Alternate methodologies to deform the mesh are under investigation. The use of elasticity equations with different structural properties in different directions seems to be an attractive alternative.

2.4. Results and discussion

2.4(a) Main sail computations

We first analyse the main sail alone, with and without the mast. The sail planform is roughly elliptic in shape with a height of 24 *m* and a boom length of 10.5 *m*. The boom is off-set by 2 *m* from the symmetry plane to include the presence of the foot vortex in the simulations. The sail camber is roughly constant at around 15 % through the length of the mast. The incoming air-stream has a boundary layer profile with the maximum airspeed of 10 m/s at around the head of the sail. The flow is also twisted with the change in the inlet angle varying from 0 to 19 degrees. The sail geometry is twisted to account for the twist in the incoming air-stream so that individual sections operate at maximum lift while not stalling.

The computational mesh for this geometry has around 2.3 million cells. The structural model used 8000 quadrilateral membrane elements distributed evenly along the height of the sail. The structural properties that used for this computation are Young's Modulus $E = 11114803\ N/m^2$ and density $\rho = 1.653\ kg/m^2$.

The flow solver was run on 8 processors on a SGI-Origin 2000 with three levels of multigrid to obtain a solution in under 10 minutes. Per solve, 50-75 multigrid W-cycles were used. Five aero-elastic iterations were required to obtain the steady deflected shape of the sail.

The pressure distributions at mid-height and three-quarters height are given in figure 1. The computed lift and drag at and angle of 19 degrees were 1.6 and 0.19

respectively. The non-dimensional lift matches well with that from a finely tuned panel method [10].

No appreciable difference in the computed lift and drag was observed between simulations run with and without the mast. To better investigate the mast-sail interaction, viscous flow calculations are needed.
The twisted nature of the onset flow produces a change in the lift and drag coefficient of around 10-15 %. In the absence of twist to the onset flow the lower sections are operating at a higher angle of attack than designed for thereby producing more lift and drag due to the stronger foot vortex.
The sail deflection is shown in figure 2. The computed deflection is larger than what is observed from photographic images. The incomplete nature of the structural model could be a possible source of this error. The absence of battens and tension stays from the structural model allows the structure to deflect to a flying shape that has more twist and less camber along the sections. Inclusion of these structural models has not yet been attempted. Also, the absence of models to predict wrinkling in the sail cloth, may also limit the accuracy of the aero-elastic calculations. However, the rather rigid nature of the sail cloth leads to reasonable estimates of the lift, drag and flying shape.

2.4(b) Combined head and main sail computation

The structural properties of each sail are the same as for the main in the previous section. The total number of cells in the fluid mesh is around 3.8 million. The head sail has 480 membrane elements and the main sail 1240. The pressure distributions on the head and main sail at mid-height are shown in figures 3 and 4. The flying shape of the head sail is shown in figure 5. The computed lift and drag coefficients at an angle of 19 degrees are computed to be, respectively, 0.9794 (head) and 0.9556 (main), and 0.1476 (head) and 0.2240 (main). The lift results are in close agreement with those obtained by a traditional well-tuned panel method [10]. The pressure distribution on the head sail shows that the twist along the sail matches well with the twist in the inlet flow profile. Hence, the flow enters smoothly without separating off the leading edge. The pressure distribution along the suction side of the main sail highlights the slot effect produced by the head sail. The flying shape of the individual sails is along expected lines. The leading edge of the head sail is `deflated' while the main sail is essentially inflated due to the loading.

We note that grid convergence studies were performed for all computations to ensure solution quality.

2.5 Future work

Future work will include

- Enhancement of the structural model
- Extension to viscous computations
- Addition of adjoint optimisation

The solver has already been extended to include adjoint optimisation and viscous computations on structured meshes. We are currently in the process of validating the solver on unstructured meshes.

3. PROJECT II: OPTIMIZATION OF SHEETING ANGLES FOR THE MALTESE FALCON

We investigate the use of a gradient-based cost function minimisation algorithm combined with a commercial CFD package to optimise the sheeting angles for the rig of the *Maltese Falcon*. Three square-rigged masts will rig this future megayacht, shown in figure 6. The yards are 12% chord circular arc cross sections. The rig is based on an original design by W. Proells, which was further developed at Hamburg University in the early 1960’s. Because the main purpose of the work presented here is to assess the usefulness of the optimisation approach, we limit ourselves to 2D analysis for upwind and close reaching apparent wind angles. We note that because of the type of rigging, 2D analysis likely gives very reasonable optimisation results. We are currently extending the procedure to full 3D calculations and a larger range of incident angles.

3.1 Gradient-based cost function minimisation

In the gradient-based optimisation approach, a cost function characteristic of the performance of the sail or hull is minimised with respect to one or more control parameters. This iterative procedure requires the sensitivity derivatives of the cost function with respect to the control parameters in each iteration step. These derivatives determine the direction of improvement. A step is taken into this direction and the procedure is repeated until convergence [11].

In this work we optimise the sheeting angles only. Therefore, we have only three control parameters (the three sheeting angles), and the gradient information can be computed relatively efficiently using a finite difference method.

We investigate two cost functions J_1 and J_2, with

$$J_1 = \frac{1}{C_x} \quad (5)$$

equal to the reciprocal of the driving force coefficient C_x, and

$$J_2 = \frac{C_y}{C_x} \quad (6)$$

equal to the ratio of heeling force coefficient C_y and C_x. The cost function J_1 is generally used in downwind situations, whereas J_2 is generally preferred in upwind situations. Given an apparent wind direction the optimisation procedure starts by assuming initial sheeting angles for the three masts and a base flow solution is calculated giving the base cost function J_{old}. One by one each mast is then perturbed by a small amount and the flow solution recalculated giving the new cost function J_{new}. The gradient of the cost function with respect to each sheeting angle θ_i is evaluated as

$$\frac{dJ}{d\theta_i} \approx \frac{J_{new} - J_{old}}{\Delta\theta_i}, \quad i = 1,2,3 \tag{7}$$

Once the gradients have been calculated the new sheeting angles are calculated as

$$\theta_{i,new} = \theta_{i,old} - g\frac{dJ}{d\theta_i}, \quad i = 1,2,3 \tag{8}$$

The weighting parameter g is used to weight the gradient information and changes with cost function definition.

The procedure is repeated until the cost function is minimised. Initial results show that, if starting from reasonable initial sheeting angles, the cost function is minimised in 12-15 iterations. Figure 7 shows the typical behaviour of the cost function during an optimisation run.

3.2 Implementation details

We developed a central program that automatically creates the grids, runs the flow solutions and evaluates the flow field variables needed by the optimisation algorithm. We use unstructured non-conforming grids that are robust and computationally efficient. An example is shown in figure 8. The grids are generated by GAMBIT, which is part of the commercial CFD package FLUENT. We note that when the sheeting angles change, only the grids that are directly around the sails, which are highlighted in the figure, need to be adjusted. The grids are coarsened away from the sails to reduce computations.

The flow solution is calculated using FLUENT 6.0 for the incompressible Reynolds Averaged Navier-Stokes equations. The turbulence model used is the Spallart-Allmaras turbulence model, which is sufficiently accurate for upwind and close reaching conditions, and computationally efficient [12]. More sophisticated models must be used for larger angles of incidence because of flow separation.

The code was run on an Athlon computer with a Pentium IV 1.2 GHz processor. A single flow solution usually took just over a minute and an entire iteration consisting of 4 solutions and data processing about 5 minutes.

3.3 Results and discussion

Optimisation runs were performed for both cost functions for apparent wind angles ranging from 30 to 90 degrees. We present data results for apparent wind angles of 30, 60 and 80 degrees in table 1 and figures 9, 10 and 11. Table 1 displays the driving force and heeling force coefficients for each apparent wind angle for both cost functions. The optimal sheeting angles are also given. Figures 9, 10 and 11 show the pressure distributions on the sails in the optimal arrangement for both cost functions, as well as the corresponding pressure distributions on the sails.

30 degrees apparent wind angle					
Cost function	*Force coefficient* *Driving*	*Heeling*	*Sheeting angle* *Aft*	*Mid*	*Fore*
J_1	0.9304	1.9038	-6.5	-19.3	-31.6
J_2	0.7988	1.4754	-16.0	-26.6	-36.1
60 degrees apparent wind angle					
J_1	1.4943	1.0116	-42.6	-49.9	-55.8
J_2	1.1327	0.6888	-56.0	-57.3	-60.9
80 degrees apparent wind angle					
J_1	1.6193	0.3893	-68.45	-70.90	-73.12
J_2	1.2571	0.2499	-77.17	-78.34	-79.60

Table 1. Force coefficients and sheeting angles for optimal sail configurations

For all apparent wind angles tested, cost function J_2 results in more open (larger angle) sheeting arrangements with a more even pressure coefficient distribution along the length of the sail cross sections.

The velocity fields are compared in figure 12. Imagining telltales placed on the leading edge of the sail reveals that J_1 results in the sails being over trimmed (leeward telltale lifting) while J_2 leads to well trimmed sails (telltales both flying).

Figure 13 shows plots of sheeting angle versus angle of attack for both cost function definitions. The most notable feature of these plots is that the sheeting angles of the three masts are more uniform when using the J_2 cost function. We conclude that, as expected, J_2 leads to better optimisation results for the tested upwind and close reaching apparent wind angles.

Again, all numerical results were validated with grid convergence studies.

3.4 Future work

Future work will include

- Extension of the optimisation procedure to 3D
- Extension of the optimisation procedure to broad reaching and downwind conditions
- Integration of the optimisation procedure with a Velocity Prediction Program (VPP)
- Investigation into evolutionary algorithms

We note that the procedure presented here is immediately applicable to a 3D model. The only part to be adjusted is the grid generation. It will be interesting to compare 2D optimisation results to 3D optimisation results to assess the value of 2D optimisation.

Extension to larger apparent angles is challenging. At broad reaching and downwind conditions, the sail flows become separated leading to unsteadiness. Optimisation must therefore be based on a time-averaged flow field, which requires longer solve times. Also, more sophisticated turbulence models will have to be used, such as the SST model developed by Menter, leading to extra equations to be solved, and a much denser computational grid [13].

Acknowledgements

We would like to thank Mr. Steve Collie of the Auckland Yacht Research Unit, Mr. Burns Fallow of Team New Zealand, and Professor Parviz Moin of the Center for Turbulence Research at Stanford University, for their valuable contributions to this research, and Mr. Tom Perkins for allowing us access to the Maltese Falcon rig geometries.

References

1. Kim, S., Alonso, J. & Jameson, A. (2002), "Design optimization of High-Lift Configurations Using a Viscous Continuous Adjoint Method", *AIAA Paper 2002-0844*, 40th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 2002.
2. Reuther, J., Jameson, A., Farmer, J., Martinelli, L. & Saunders, D. (1996), "Aerodynamic Shape Optimization of Complex Aircraft Configurations via an Adjoint Formulation", *AIAA Paper 96-0094*, 34th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 1996.
3. Bueche, D., Stoll, P. & Koumoutsakos, P. (2001), "An Evolutionary Algorithm for Multi-Objective Optimization of Combustion Processes", *Center for Turbulence Research Annual Research Briefs 2001*, Stanford University.
4. Jameson, A. & Baker, T.J. (1987), "Improvements to the Aircraft Euler Method", *AIAA Paper 87-0353*, 25th Aerospace Sciences Meeting, Reno, NV, January 1987.
5. Chorin, A. (1967), "A Numerical Method for Solving the Incompressible Viscous Flow Problem", *Journal of Computational Physics*, **2**, 12-26.
6. Farmer, J., Martinelli, L. & Jameson, A. (1993), "A Fast Multigrid Method for Solving Incompressible Hydrodynamic Problems with Free Surfaces", *AIAA Paper 93-0767*, 31st Aerospace Sciences Meeting, Reno, January 1993.
7. Jameson, A., Schmidt, W. & Turkel, E. (1981), "Numerical Solution of the Euler Equations by Finite Volume Methods using Runge-Kutta Time-stepping Schemes", *AIAA Paper 81-1259*.
8. Jameson, A. (1995), "Analysis and Design of Numerical Schemes for Gas Dynamics 2, Artificial Diffusion and Discrete Shock Structure", *International Journal of Computational Fluid Dynamics*, **5**, 1-38.
9. Jameson, A. (1985), "Multigrid Algorithms for Compressible Flow Calculations", *Proceedings of the 2nd European Conference on Multigrid Methods*, Cologne, 1985.
10. Fallow, B. (2002), *Private Communication*.
11. Mohammai, B. & Pironneau, O. (2001), *Applied Shape Optimization For Fluids*, Oxford University Press.
12. Collie, S., Gerritsen, M. & Jackson, P. (2001), "A Review of Turbulence Modelling for Use in Sail Flow Analysis", *School of Engineering Report No. 603*, University of Auckland, New Zealand.
13. Collie, S., Gerritsen, M. & O'Sullivan, M. (2002), "Numerical Simulation of the Turbulent Flow Past Upwind Yacht Sails", *Journal of Wind Engineering and Industrial Aerodynamics*, under review.

Figure 1. Pressure distribution on the main sail at mid-height (top) and ¾ height (bottom).

Figure 2. Deflection of main sail. Initial geometry (dark) and flying shape (light).

Figure 3. Pressure distribution on head sail at mid-height at 19 degrees apparent wind angle.

Figure 4. Pressure distribution on main sail at mid-height at 19 degrees apparent wind angle.

Figure 5. Deflection of head sail. Initial geometry (dark) and flying shape (light)

Figure 6. Schematic of the three-masted, square-rigged megayacht Maltese Falcon.

Figure 7. Typical behaviour of the cost function during an optimization run.

Figure 8. An example grid used in the simulations. Only the grid blocks directly around the sails must be adjusted in the optimisation procedure.

Figure 9. Results for 30 degrees: Final arrangements for cost function J1 (top) and cost function J2 (mid); Pressure profiles for J1 (solid) and J2 (dashed).

Figure 10. Results for 60 degrees: Final arrangements for cost function J1 (top) and cost function J2 (mid); Pressure profile for J1 (solid) and J2 (dashed).

Figure 11. Results for 30 degrees: Final arrangements for cost function J1 (top) and cost function J2 (mid); Pressure profiles for J1 (solid) and J2 (dashed).

Figure 12. Velocity field near leading edge for J_1 (top) and J_2 (bottom) for degrees apparent wind angle.

Figure 13. Sheeting angle versus apparent wind angle for J1 (top) and J2 (bottom).