APPENDAGE DESIGN FOR THE AMERICA'S CUP USING CFD

Stephen R. Turnock*[1], Nicholas J. Holroyd[2], and James C. Date[1]

[1]School of Engineering Sciences, Ship Science,
University of Southampton, Southampton S017 1BJ. U.K.
email: s.r.turnock@soton.ac.uk, web page: http://www.ses.soton.ac.uk/

[2] Team New Zealand
Email: NHolroyd@compuserve.com

Key words: Appendage Design, Practical CFD, America's Cup.

Abstract. A review is made of the requirements for the design of appendages for International America's Cup Class yachts. The ability of computational fluid dynamics (CFD) to aid effectively this process is considered. In particular, the experience of Team New Zealand's successful defence of the 2000 challenge is discussed. Their approach is based on synthesis of model tests, full scale trials and use of appropriate CFD methods, tuned to capture accurately the actual flow physics. The current ability of Reynolds Averaged Navier Stokes (RANS) flow solver for use in appendage section design is evaluated. It was found that a high quality grid and a systematic approach are required to obtain accurate prediction of lift and drag. It is estimated that similar levels of accuracy can only be achieved in three dimensions for a single appendage using grids with at least 5 to 10 million cells. In addition, severe limitations are experienced as stall is approached. Likewise, the inability of the RANS method to capture transitional flows restricts their accuracy when used in the typical flow regime experienced by yacht appendages. These drawbacks explain the widespread use of viscous-inviscid interaction methods for appendage section design.

1 INTRODUCTION

The winning margins in the last America's Cup were of the order of one or two seconds per mile. Such margins require intense design effort, in order to maximise the performance of the yacht in the expected race conditions. Boats have to be designed to perform well in a variety of seastates, wind speeds and for both upwind and downwind conditions. The ability to predict numerically, using Computational Fluid Dynamics (CFD), the aero/hydrodynamic performance of a chosen design, therefore, is seen to offer benefits in the form of:

- Rapid evaluation of design changes.
- More cost effective than extensive wind tunnel and towing tank testing.
- Greater detail and understanding of the actual flow regime.

The application of numerical techniques in predicting the performance of marine craft has been undertaken since the early 60's, when sufficient amounts of computational power first became available. However, it was not until the mid 1980's that the first serious use of such techniques were applied as a major part of the America' cup challenges. Boppe et. al.[1]'s main application was a three-dimensional surface panel method, which was used to examine the upwind performance of the underwater hull and appendages. Limited use was also made of vortex-lattice methods, for predicting the upwind performance of different sail planforms. Since then, for each campaign, greater effort has been expended and more advanced techniques applied. Recently, fully viscous flow calculations in the presence of a calm free surface have been attempted, due to the vast reductions in the cost of computational power. Throughout these developments the question still has to be asked as to whether actual performance gains have been achieved solely through the power of the available numerical analysis techniques?

This paper considers the current actual capabilities of CFD applied to appendage design. It should be noted that just having access to massive computational resources is in itself no guarantee that the design information so obtained will be the best. This was amply illustrated at a recent workshop on ship hydrodynamics[2]. For similar types of flow solvers on an identical ship hull the total number of cells ranged from 200,000 to 7,500,000 with no definitive correlation with accuracy of solution and applied computational power. In main, the work will be applied to underwater appendages; principally the keel, bulb, winglets and rudder. Mention should also be made of mast-sail flow prediction, but the challenge of sail performance, especially downwind, requires both a fully viscous unsteady flow solution to model the large areas of separation and a coupled fluid-structural calculation, thereby imposing further unknowns. Figs. 1 and 2 illustrate the types of two dimensional calculations applied to multiple sail arrangements and to mast separation respectively. It is hoped that the paper illustrates that the usefulness of the results, is not simply a measure of the sophistication of the numerical analysis tool and computational power available, but is more strongly correlated with the confidence associated with the results, and how well they capture the actual flow physics.

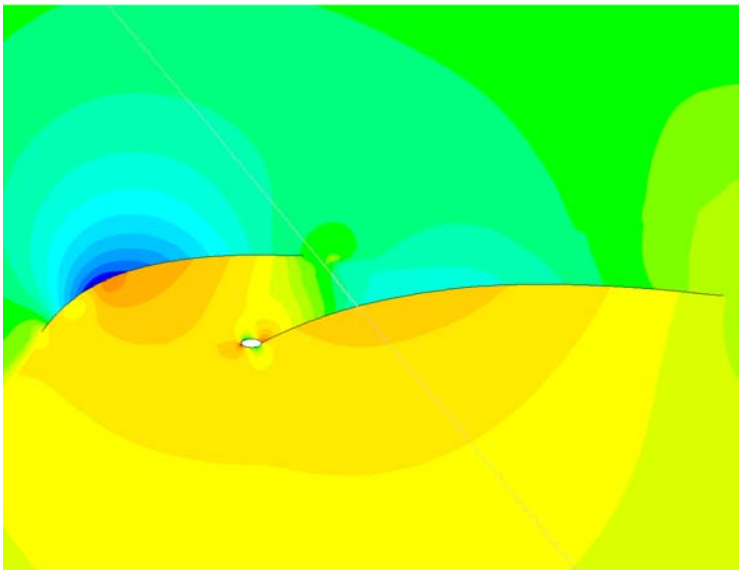

Figure 1: Flow interaction between main and jib

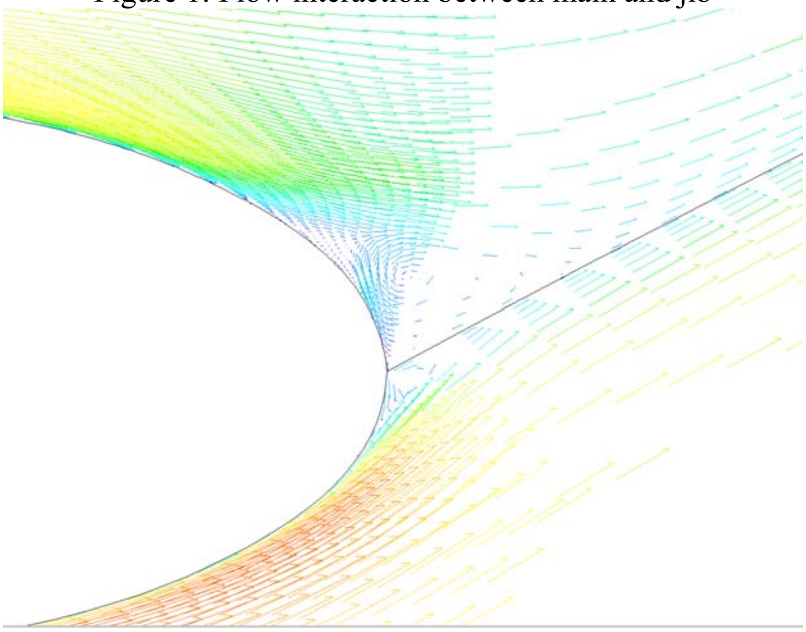

Figure 2: Mast separation

2 AMERICA'S CUP

The America's Cup has been likened to the Holy Grail of yacht racing[3]. The Cup, in competition for a period of 150 years, is the oldest and most distinguished trophy in yachting. The first race was around the Isle of Wight in 1851, where America beat the Royal Yacht squadron and so the trophy was renamed the America's Cup. This began the 132 year American reign. The first proper class, the J boats were established following the introduction of the Universal Rule in 1920. The 12 metre class was instituted in 1957 and remained for thirty years. Australia II finally managed to beat the Americans in 1983, largely because of her innovative and controversial winged keel. A period of dispute occurred in the late 1980's. This encouraged the development of a new class of yacht. The International America's Cup Class (IACC) was introduced for the 1992 competition. The new class is a modern lightweight fast monohull sloop somewhere between an IOR Maxi and an Ultra Light Displacment Boat. The 1992 competition was raced in San Diego, where the Americans retained the Cup until 1995 when Peter Blake in New Zealand beat Stars and Stripes to take the Cup to Auckland for 2000. Team New Zealand successfully defended the Cup against the Italian Prada syndicate in February 2000 with the next Challenge set for early 2003, again in Auckland.

3 APPENDAGE DESIGN

The difficulty of applying modern viscous computational methods to underwater appendage design, stems from the flow regime in which they operate. Whereas most full scale ships are operating in flow regimes, which can be considered exclusively turbulent (Reynolds number based on ship length is greater than $1x10^7$) yacht appendages operate in the transition regime where significant areas are still laminar, before transition to fully turbulent flow occurs. At present, no *a priori* method exists which can accurately capture such behaviour in the three-dimensional, unsteady flow regime present on a yacht.

The problem of the operating regime also exists in the conflicting requirements on the appendage imposed by the necessity for upwind and downwind performance. For the sail rig this problem is overcome through the use of different sail types and sets. However, below water the appendages have to perform both in the downwind condition, where the principal objective is minimising total drag, and upwind when the set of appendages have to resist the forces and moments generated by the sails, in such a way that boat speed is maximised.

By its nature, design for an America' s Cup yacht is constrained by the rules of class, currently IACC. These impose restrictions in a number of areas such as depth, number of control surfaces and so forth. A typical underwater arrangement is shown in Figure 3.

The main difference from conventional aircraft wing design, which has similar altered states between cruise and landing/take-off configurations, is the necessity to situate a large percentage of the yacht's mass (typically >80%) as low as possible below the hull. This mass is located in the bulb. The keel structure connecting the bulb to the hull has to cope with the dynamic and static loads imposed by the bulb, with any deflections resulting in a loss of stability, as well as the hydrodynamic loads generated by itself. Underwater control of the boat's direction and attitude is typically obtained through use of a main rudder, located near or at the stern, and a trim tab on the trailing edge of the keel. Other more radical arrangements are possible, for instance a tandem keel which has two flapped keel supports for the bulb (see Figure 4).

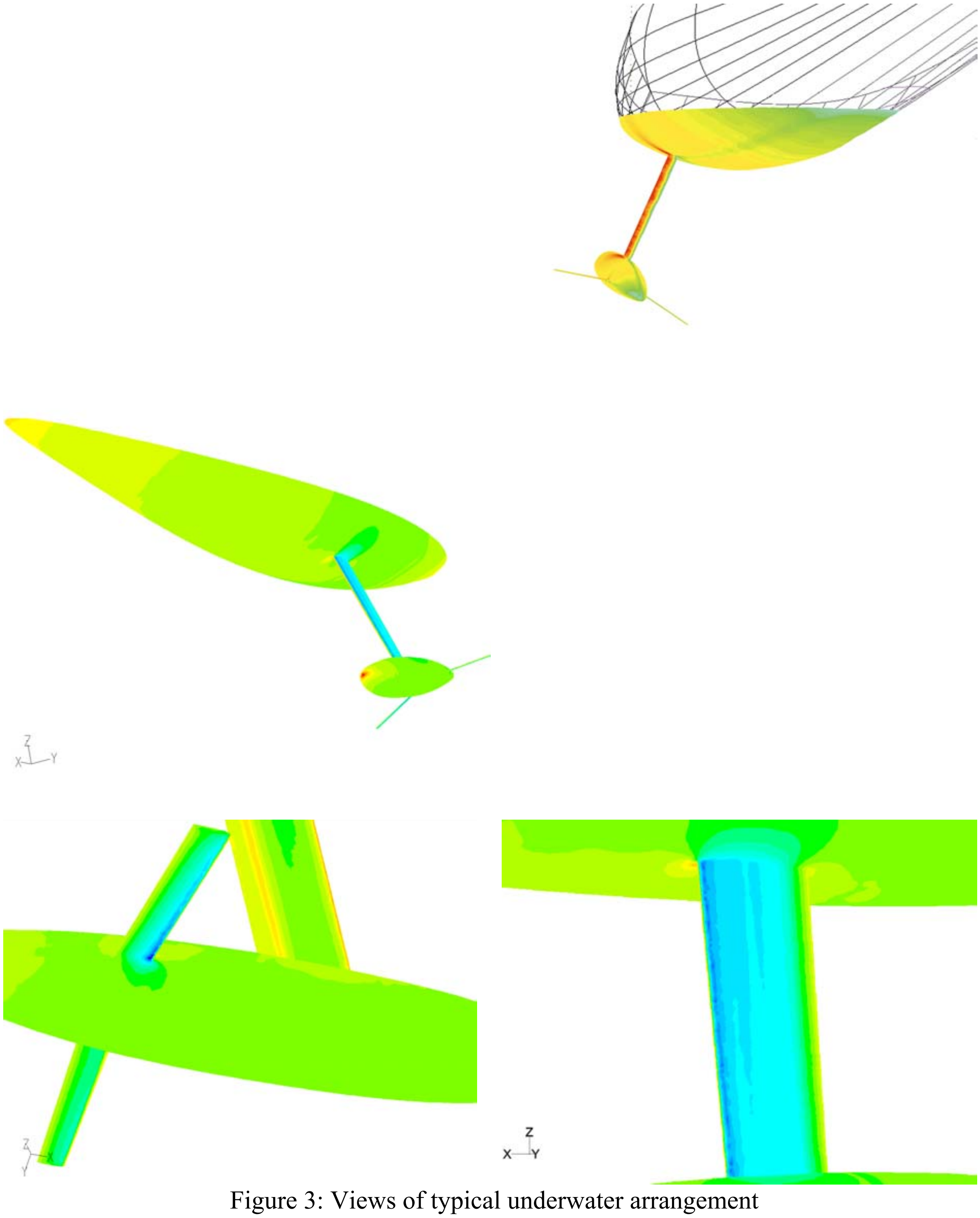

Figure 3: Views of typical underwater arrangement

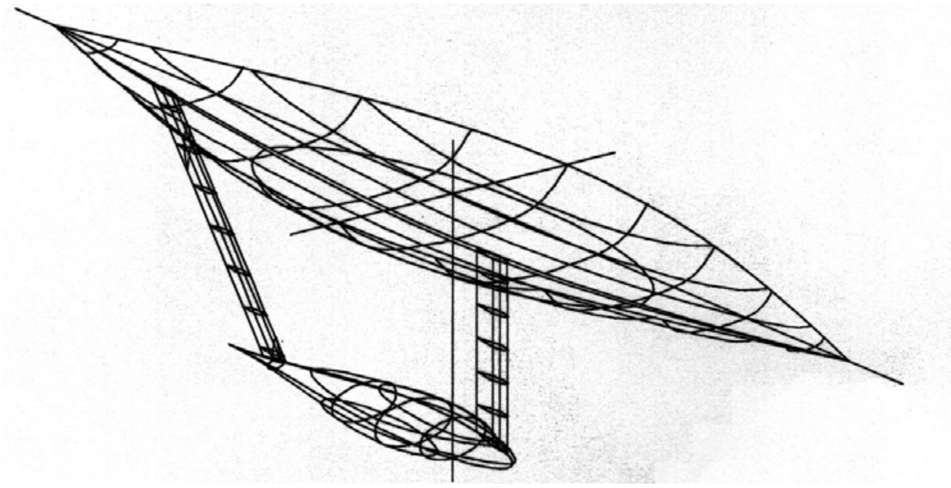

Figure 4: Tandem keel arrangement

However, these more unusual arrangements (i.e. The New Zealand 1992 entry) have yet to show any real promise. Often, too many other design changes would be required and would take the overall design too far away from the known and understood. The final element often present is the two winglets located either side of the bulb. These act both to control tip vortex production and thereby help reduce induced drag, and to generate propulsion as the yacht heaves and pitches, in response to the local seastate. Much was made of the performance gains achieved with the winged keel arrangement used on the 12m class Australia II when the cup was finally wrested from the Americans in 1983. The 12m class yacht was severely restricted in the keel span achievable between the deep canoe body and the rule imposed draft limit, thus making the winglets contribution to keel aspect ratio proportionately large. The shallower canoe forms and greater keel draft allowed by the IACC rule based yachts has resulted in the return of the much more conventional keel bulb arrangements.

Although there are rule based and structural constraints the appendage designer still has considerable flexibility in the following:

- Longitudinal position of keel
- Longitudinal position of rudder
- Keel section shape and planform
- Rudder section shape and planform
- Bulb shape and position relative to the keel
- Winglet section, planform and position on the bulb.

None of these can be treated in complete isolation. Flow behaviour at the intersection between keel-hull, hull-rudder, keel-bulb, and winglet-bulb may be important and can influence the behaviour of the complete system. Not mentioned so far, and of by no means negligible importance, especially in the upwind condition is the influence of the free surface.

The traditional towing tank testing approach allows the complete behaviour, including free surface effects, to be captured but at model scales of between 1:3 and 1:5. However, detailed knowledge of the actual flow interaction between the various elements is difficult to deduce except through extensive parametric testing. Application of the latest in flow measurement technology such as laser doppler anemometry, particle image velocimetry and unsteady pressure transducers requires yet further use of expensive test facilities. More detailed flow behaviour can often be obtained for those parts of the system more deeply submerged through use of wind tunnels. Again, use of such facilities is often expensive.

It is the above economic drivers, which make the application of CFD techniques seem so appealing. It is easy to claim that parametric studies, automated optimisation and detailed flow behaviour can all be captured with a minimum of human intervention. For instance a 50 processor system capable of solving several two million cell problems a day using a viscous RANS code can now be purchased for less than the price of hiring a typical test facility for a week. However, such an approach presumes that the results of such calculations can be used with the same degree of confidence.

4 AVAILABLE METHODS

The full unsteady Navier-Stokes equations, coupled with the conservation of mass and energy are well known. It is currently possible to carry out direct solutions of this system only for very small Reynolds number, typically $R_n<500$ depending on the size of computer applied. For practical calculations Reynolds averaging, hence Reynolds Averged Navier Stokes (RANS), is applied to give the unsteady equations which assume that the time scale of body motion and boundary condition changes are large in comparison to that of turbulent fluctuations within the flow. The result of this assumption, is the need to provide additional models of the relationship between the stress and strain within the fluid. This turbulence closure problem is still the key difficulty with application of the RANS equations to turbulent flow problems. The complexity of approach to solving this problem varies from simple algebraic relationships based on empirical fits, to more complex relationships based on phenomenological models for the behaviour on further flow properties such as turbulent kinetic energy and eddy viscosity(k-ε). Even the more complex models still require various parameters to be set, based on empirical behaviour. Poor performance of the whole gamut of possible models is well known for particular flow regimes. However, the ability to fine-tune the model through modification of the empirical constants does allow the user to achieve good agreement with known experimental data. Caution always then has to be exercised if significant deviations occur from this known condition. The difficulty arising as to how much of a deviation is significant and do the changes in performance occur because of the non-physical behaviour of the turbulence model or do they reflect the actual behaviour of the fluid. In addition the process of flow separation and transition of flow state from laminar to turbulent are difficult to capture within the turbulence closure.

Not withstanding the large limitations inherent in the use of RANS methods their use is widespread. A number of commercial codes are available along with significant numbers of research codes. Each method has its adherents, but the most common impediments to their accurate use is determined by the following:

- Size of the external computational domain.
- The quality of the mesh used to discretise the domain and how that mesh is related to the flow present within the domain.
- The number of unknowns used to discretise the domain, usually limited by the available computational power and computational memory.
- The accuracy with which the geometrical shapes are captured.
- The applicability of the turbulence model to the various flow regimes present.
- How well the free surface is captured is still difficult and provides a further limitation on achievable accuracy.

The alternative approach to estimating appendage performance derives from the infinite Reynolds number approximation of the incompressible Navier-Stokes equations i.e. potential flow. Solutions of Laplace's equations can be found using fundamental Green's function solutions, that can be located on the bounding surface of the domain. These boundary element approaches are perhaps the most powerful tool for capturing the behaviour of the complete system. Usually implemented as surface panel codes, they can include the influence of unsteady behaviour and the free surface for a fully appended hull. Again, a number of commercial and research codes are in widespread use.

A further refinement is to modify the potential flow solution to include the influence of viscosity. At its crudest, this uses empirical expressions for skin friction on each panel to estimate the viscous resistance. A more satisfactory approach is to calculate streamlines on the body surface and to then use solutions of the thin two dimensional boundary layer equations, for example based on the integral momentum equations, to modify the boundary element strengths. This modification ensures that there is a stream surface, which lies on the displacement thickness of the body, and thus represents the potential flow outside the viscous boundary layer. Such an approach, provided a reasonable thin boundary layer calculation procedure exists, allows the viscous friction and form drag to be accurately captured. Effects such as transition and flow separation can be incorporated, provided empirical evidence can be used to tune the model.

5 IMPORTANCE OF VALIDATION

The process of simplifying the complex unsteady flow regime around a full scale yacht can be considered to be one of progressive abstraction of simpler models from the complete problem, see Fig. 5[4]. Each level of abstraction corresponds to the neglect of a particular non-dimensional parameter. Removal of these parameters can be considered to occur in three distinct phases: those which relate to physical parameters, those which relate to the assumptions made when deriving a continuous mathematical representation; and finally those used in constructing a numerical (or discrete) representation of the mathematical model.

The validation process as applied to ship design has been investigated in depth by the Resistance and Flow committee of the International Towing Tank Conference[5,6,7]. As has been already stated, a mathematical model of a physical process generally involves a degree of approximation. In using such a model, it is necessary to appreciate the confidence with which the model can be used. In the same way in which there is always error (or more correctly a degree of uncertainty) in the acquisition of experimental data, numerical modelling gives rise to uncertainty in the answer obtained. The process of validation can be seen as an attempt to eliminate or at least quantify these uncertainties and can be seen as a series of stages.

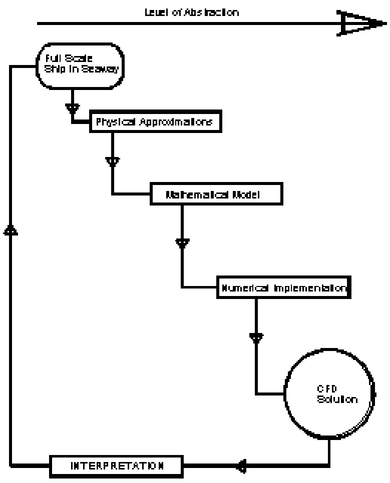

Figure 5: Process of Abstraction and Interpretation

Firstly, verification of the code implementation against the underlying mathematical. Secondly, investigation of the independence of the solution from <u>all</u> numerical parameters. Finally, by comparing numerical and experimental data. As the majority of fluid dynamic codes are an approximation to the actual physics of the flow, there will always be differences between the experimental and numerical results. A comparison will only be valid if both experiment and computation are at the same level of abstraction i.e. all assumptions and values of non-dimensional parameters are the same.

Experimental data has often been used to improve the correspondence between theory and experiment to provide a design tool. Such an approach will usually restrict the range of geometries/conditions for which the model can be used. This can be dangerous if these restrictions are not appreciated/understood by an end user who could use the code for a completely different geometry and base design decisions on what is a fundamentally flawed analysis.

6 APPROACH OF TNZ

Team New Zealand's approach to appendage design for its successful 2000 defence involved careful use of, and corroboration between; full scale on the water testing, ¼ scale towing tank experiments, parametric Velocity Prediction Program (VPP) studies, wind tunnel modelling, and both viscous and inviscid CFD approaches. Although this paper is about CFD, the successful application of this technology requires the practitioner to have a full understanding of the other design methods and how they may be combined synergistically. A brief outline of these methods follows.

- Full Scale Testing: This involves sailing two boats together and measuring their relative performance, changing some parameter on one of the yachts and determining the impact of this change on their relative performance. This has the obvious advantage of correctly modelling all the physical conditions without the need for experimental abstraction. There are drawbacks to this approach, not least of which are; the environmental 'noise' (differences in wind speed and direction, seastate etc. between the two yachts), the human element of how the sails are trimmed and the boat steered, and the very considerable cost of sailing boats of this size and complexity (replacement sails, crew wages, chaseboat fuel, repair and maintenance etc.). The value of this type of testing can be seen in the close correlation between the amount of time each team spent training and testing on the water and the results they obtained in the 2000 event. A fact that has not escaped many of the 2003 challengers and their preparation plans.
- Towing Tank Experiments: The America's Cup protocols allow model testing at up to 1/3 scale. Without modifying either the density/viscosity ratio of the fluid or gravity it is impossible to get both Reynolds (viscous) and Froude (gravity wave) number similarity. Models in tank tests are, therefore, towed at a speed that gives Froude similarity. To obtain full-scale results the viscous contribution to the forces are 'stripped' and scaled empirically (usually via the ITTC formulation), the wave forces are multiplied up to full scale and the two data sources are combined to give the total full scale drag. It can be seen then that the tank can be used to assess essentially inviscid, effects such as wave and lift induced drag, but not Reynolds dependent viscous effects such as section performance.
- Velocity Prediction Program: The VPP is a global force model that determines the yacht's speed for a given wind speed and direction. It is the central design tool and allows the incorporation of experimental and computational results. It allows the designer to assess the overall impact of various trade offs, i.e. a bulb shape with less viscous drag but lower stability.
- Wind Tunnel Testing: Allows both the measurement and visualisation of viscous and inviscid phenomenon. Appendage work is usually conducted in the absence of the hull and free surface shapes. Particularly useful for quickly assessing details such as the impact of strakes, fillets and the like on junction drag, the viscous drag of bulb shapes etc.

As mentioned, the absence of a suitable 3D laminar/turbulent transition model has limited the application of the full RANS methods. The CFD work undertaken then involved an iterative approach based around the decomposition of the problem into the 3D inviscid (both panel and Euler code) assessment of the induced drag and sectional loading along the span of the fin and wings, and then the application of a suitably calibrated 2D coupled boundary layer code to optimise each section for the expected lift coefficient.

The design cases that need to be considered are: upwind at a range of side forces/heel angles, downwind (zero side force) and various manoeuvring cases. The lift produced by a fin is proportional to the C_L, area, and the velocity squared, so in those situations when the boat is slowed (i.e. coming out of a tack) the velocity squared term diminishes rapidly and the inability of the foils to generate the required side force will adversely effect the boats manoeuvrability and competitiveness in a match race situation.

A parametric series of inviscid analyses leads to optimisations of the keel planform, the proportion of total side force carried by the keel and rudder, and the position and spanwise loading of the wings. It should be noted that many of these attributes are tradeoffs between upwind and downwind, and straight line versus manoeuvring performance. At Team New Zealand a customer type relationship exists where the design group aims to provide the best yacht that meets the sailing team's requirements (they are our customer), and whilst we are able to provide data as to the nature of these tradeoffs it is essentially up to the sailors to analyse the tactical implications of such decisions.

The spanwise loading data is then extracted and used to optimise the Lift-to-Drag ratio at the expected load at each section using a 2D coupled boundary layer code. These types of code generally use a Tollmien-Schlichting wave amplification criterion to predict turbulent transition, are carefully calibrated against available wind tunnel data and can be considered amongst the most accurate of the numerical techniques available.

As noted, a typical keel section operates at a Reynolds number of $\sim 3x10^6$ and considerable laminar flow can be expected. In the laminar regime the velocity gradient as you approach the wall is considerably lower and hence the skin friction is less than it would be for turbulent flow. Also, after transition to turbulent flow the boundary layer thickens rapidly and energy is lost to the fluid in terms of the momentum deficit. In order to optimise the sectional performance it is necessary to design the pressure distribution so as to maximise the area of laminar flow. If this approach is pushed too far then the resulting sections will have the maximum thickness a long way aft and the pressure recovery becomes very steep with an inherent risk of flow separation and a massive increase in drag.

Any design changes that significantly change the volume distribution, such as the move to longer bulbs seen in 2000, needs to be assessed for its wave making effect in the towing tank and overall manoeuvrability and tacking performance assessed at full scale. The performance difference between the best and worst boats in the fleet is about one percent and so the performance increments that one is looking for are frequently smaller than the accuracy of the individual test methods available. This is where corroboration of results between different methods is useful and decisions made are not always black and white. In general, the performance of the design on the water is the final arbiter of whether all the proceeding work was successful.

7 SECTION PERFORMANCE

As an illustration of the capabilities of a typical RANS flow solver, the following details the process of validation for the performance of a NACA0012 section and for further details see Date[8]. A systematic approach was used to generate a high quality mesh which captured lift and viscous and form drag over a range of incidence angles approaching stall.

A four-stage investigation was carried out; a boundary location study, a grid independence study, a convergence criteria study together with a validation investigation against experimental data. A number of experimental tests have been carried out on the NAC0012[9,10,11]. The computational model was run at R_n = 2.88x10^6 and 6.0x10^6. The standard and RNG k-ε turbulence models were both tested using the standard constants given in Wilcox[12]. The inlet turbulence parameters were set according to free stream conditions ensuring that k and ε remain positive throughout the domain. QUICK differencing was used for the spatial u and v terms and hybrid for the turbulence quantities k and ε. Pressure correction was carried out using the SIMPLE algorithm. The mass source residual was set at 1.0x10^{-6} kg/s in all computations.

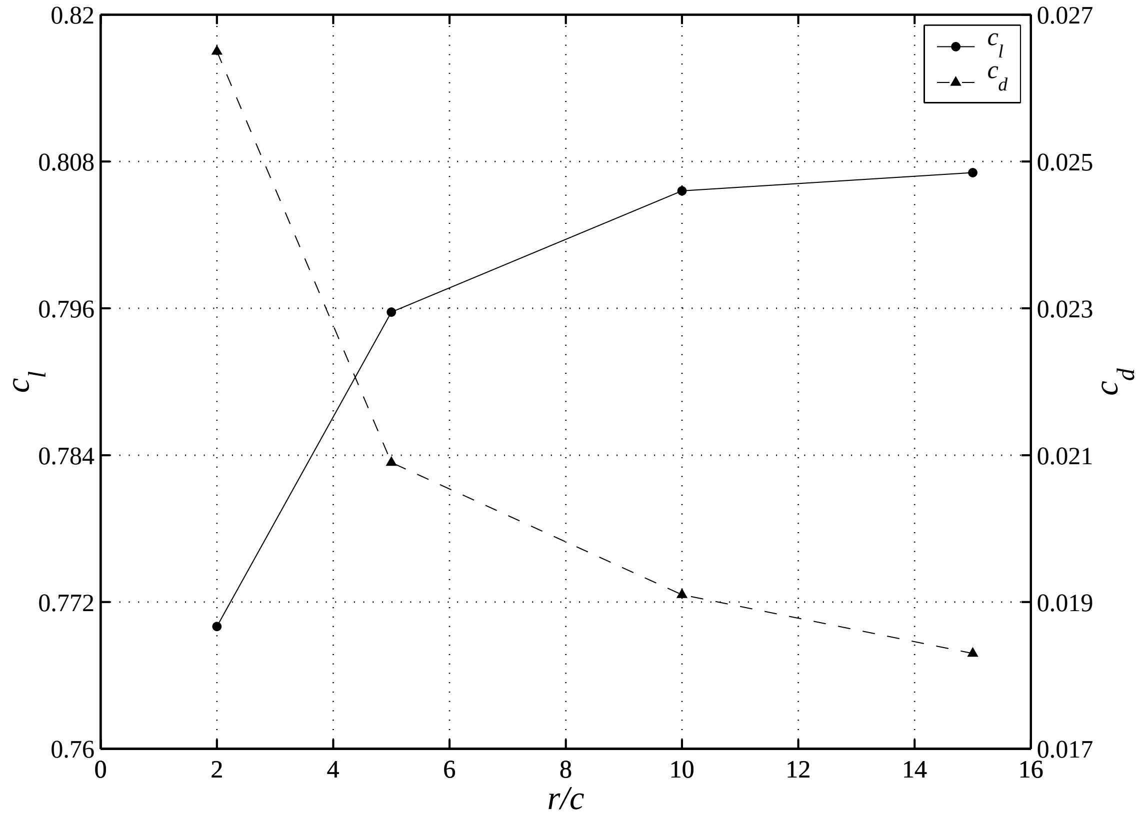

Figure 6: NACA 0012 boundary location study, $\alpha = 8°$, $R_n = 6.0\times10^6$

Figure 6 shows, for an angle of incidence of 8^o the asymptotic convergence in both lift and drag for the outer domain distances up to 15 chords. This chord length outer domain location distance was deemed acceptable and is similar to the criterion of others[13,14,15].

A fully decoupled independence study was conducted to investigate the effect of changes in near-wall grid node location, number of grid nodes in the chordwise, radial and wake directions. The variation in lift and drag, along with pressure distribution were all studied.

Table 1 shows the effect of varying first cell size with a fixed outer cell size. It can be seen that C_f varies little for non-dimensional wall distance y^+ values between 20 and 400, with the number of iterations and CPU time increasing sharply below the lower wall function limit of 30. More importantly, looking at the viscous pressure, it is obvious that use of too large a first cell size, results in an under prediction of the C_{pv}. It is not advisable to use the upper y^+ criterion (~500) as this will result in poor prediction of C_{pv}.

Table 1: Effect of near-wall cell size on computed NACA 0012 drag, $\alpha = 0°$, $R_n = 6.0\times10^6$

y_p (m)	y^+	$C_f \times10^{-3}$	$C_{pv} \times10^{-3}$	$C_d \times10^{-3}$	N_{it}	CPU Time (Min)	No. of Cells
0.016	270-765	7.145	3.478	10.623	1697	26	14400
0.008	118-398	7.090	2.960	10.051	1693	29	15552
0.004	65-200	7.076	2.711	9.788	1686	31	16704
0.002	39-99	7.070	2.508	9.578	1672	34	18432
0.001	20-49	7.080	2.485	9.565	2704	55	19584

Figure 7 shows the effect of increasing the number of cells in the chord-wise direction. A value of 244 chord-wise cells was deemed acceptable. Another variation was achieved by holding the inner cell size fixed at $y^+ = 30$ and by modifying the number of cells in the radial outward direction.

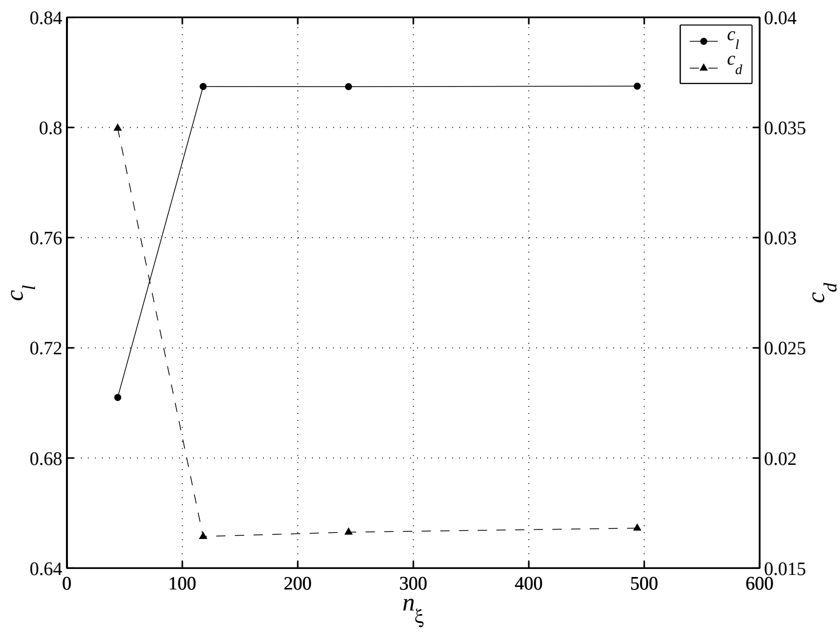

Figure 7: NACA 0012 chord-wise grid study, $\alpha = 8°$, $R_n = 6.0\times10^6$

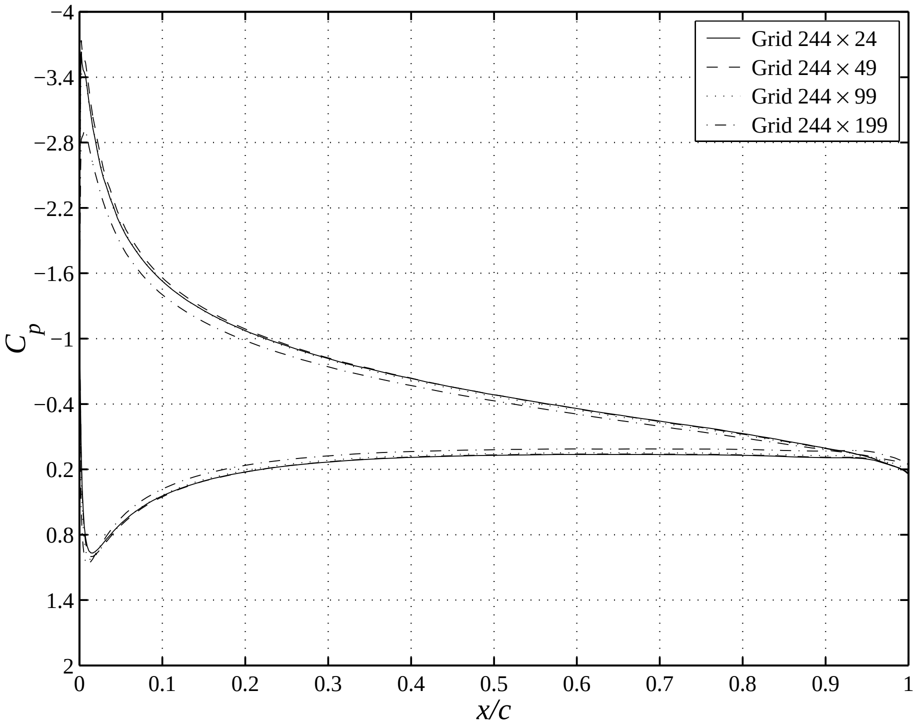

Figure 8: NACA 0012 pressure capture grid study C_p distribution, $\alpha = 8°$, $R_n = 6.0\times10^6$

Figure 8 shows that 49 radial cells gave acceptable results. Similarly it was found that at least 99 cells in the wake were required to give converged results. This gives a minimum total of 21,658 cells to give an acceptable result for both lift and drag. If similar surface cell densities are used then an extrapolation can be made as to the total number of cells TNC required as a function of appendage aspect ratio AR as follows:

$$TNC = 1.061\text{x}10^6 + AR \,.\, 2.642\text{x}10^6 \qquad (1)$$

It is immediately apparent that accurate capture of lifting performance of appendages requires significantly more cells than can normally be applied[16].

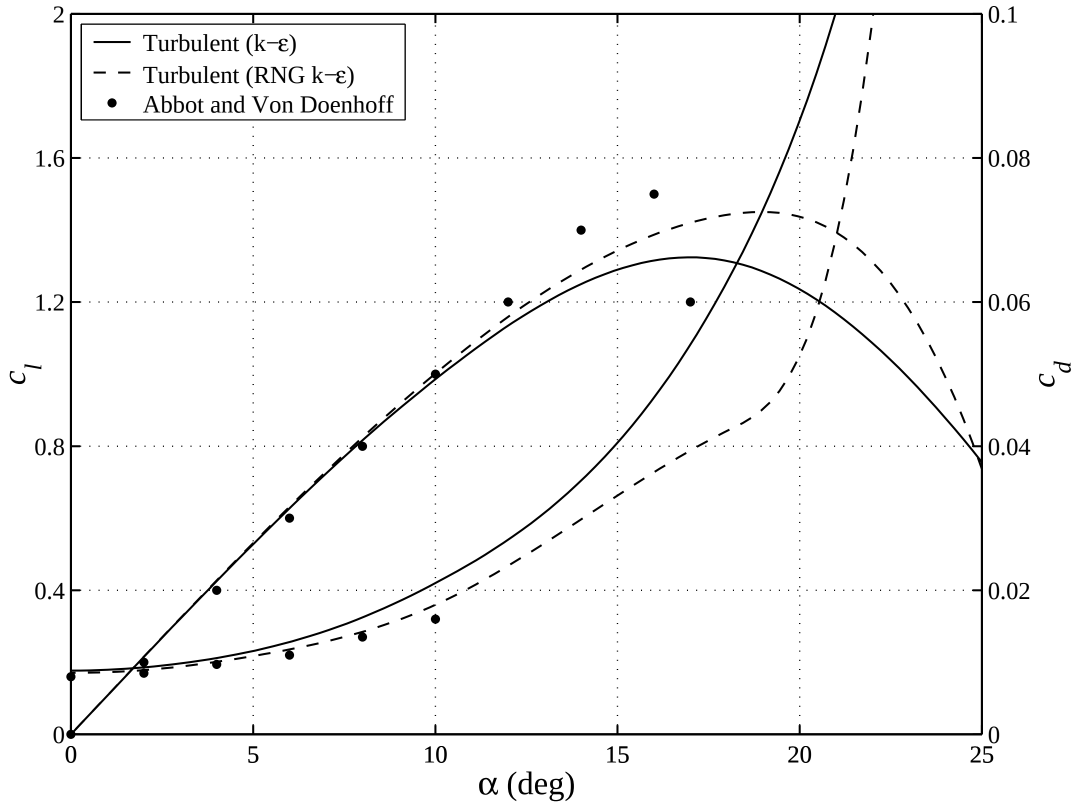

Figure 9: NACA 0012 performance, $R_n = 6.0\times10^6$

The final comparison against experimental data for lift and drag is shown in Figure 9. It can be seen that very close comparison occurs up to 6^o. Significant differences in lift are observed as stall is approached. This is a well known limitation of the standard and RNG k-ε models[13,17,18] and arises from the inherent assumptions within the turbulence model. At zero incidence there is a slight overprediction in drag which arises from the fully turbulent computational model as opposed to the laminar and transition zones actually present.

Figure 10 compares the surface pressure with experimental results[19]. It is evident that for condition that both turbulence models give good agreement on both pressure and suction surfaces. Not withstanding the good performance achieved with the RANS approach, it should be noted that a considerable effort was required to ensure that these results could be used with confidence. Indeed there still remains the problem that the inability to capture laminar-turbulent transition severely restricts use and illustrates the reason why viscous-inviscid interaction methods are still so widely applied.

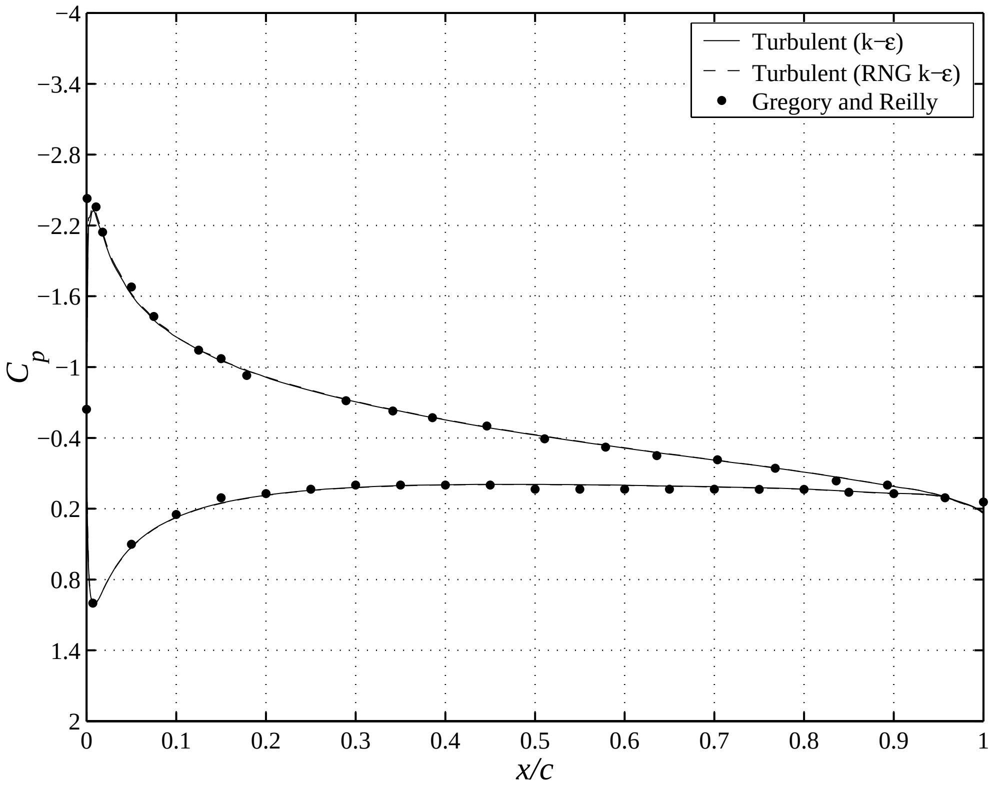

Figure 10: NACA 0012 C_p distribution, , $\alpha = 8°$, $R_n = 2.88\times10^6$

8 CONCLUSIONS

There is no question that with each subsequent America's Cup campaign Computational Fluid Dynamics plays an increasing part in the Appendage design process. As available computational power increases, there is greater flexibility in the computational model and grid generation and most importantly actual experience of the use of CFD for design is accumulated the contribution will become more quantifiable. However, it is equally true that without model scale experimental tests and full scale trail studies use of CFD is to a large extent futile. Intelligent use of the range of numerical tools as discussed, from coupled surface-boundary layer methods to fully viscous RANS calculations with a free surface, is required with detailed knowledge of the strengths and weaknesses of each method.

REFERENCES

[1] Boppe, C.W., Rosen, B.S., Laiosa, J.P, *Stars & Stripes '87:computational flow*

simulations for hydrodynamic design, Proceedings of 8th Chesapeake Sailing Yacht symposium, pp. 123-146. 1987.
[2] Ed. L.Larrson, F.Stern, V.Bertram, Proc. of Gothenburg 2000, A Workshop on Numerical Ship Hydrodynamics, 14-16 September, Chalmers University of Technology, Sweden, 2000.
[3] D.J. Le Pelley, P. Mancebo, R.P.Smith, *A technical proposal for the design of an IACC yacht for the year 2000*, Group Maritime Design Project Report, Ship Science, University of Southampton, 1998.
[4] S.R.Turnock, *Interpretation of CFD results for use in Ship Hydrodynamic Design*, Proc. Of 2nd International CFD Conference, Ulsteinvik, Norway, 1999.
[5] *Resistance Committee Report*, Proceedings of the 21st ITTC, Volume 1, Trondheim, Norway, 1996.
[6] *Resistance Committee Report*, Proceedings of the 20th ITTC, Volume 1, Madrid, Spain 1993.
[7] *Resistance Committee Report*, Proceedings of the 19th ITTC, Volume 1, San Francisco, USA, 1990.
[8] J.C.Date, *Performance prediction of high lift rudders operating under steady and periodic flow conditions*, submitted PhD thesis, University of Southampton, 2001.
[9] I.H.Abbott, A.E. von Doenhoff, *Theory of wing sections*, Dover Publications, 1959
[10] J.J.Thibert, M.Grandjacques, L.H.Ohman, *NACA0012*, AGARD Advisory Report, 1979.
[11] N.Gregory, O.L.O'Reilly*, Low speed aerodynamic characteristics of NACA0012 airfoil section, including the effects of upper surface roughness simulation hoarfrost*, NPL aero Report 1308, National Physical Laboratory, 1970.
[12] D.C. Wilcox, Turbulence modelling for CFD, D.C.W. Industries, 2nd Edition, 1998.
[13] S.W.Chau, Numerical investigation of free-stream rudder characteristics using a multi-block finite volume method, Report 580, Institut Fur Schiffbau der Universitat Hamburg, July 1997.
[14] C.D.Simonsen, *Rudder, propeller and hull interaction by RANS*, PhD Thesis, Dept. of Naval Architecture and Offshore Engineering, Technical University of Denmark,May 2000.
[15] E.Guilmineau, J.Piquet, P.Queutey, *Two-dimensional turbulent viscous flow simulation past airfoils at fixed incidence*, Computers and Fluids, Vol. 26, No. 2, pp.135-162,1997.
[16] M.Caponnetto, A. Castelli, P.Dupont,B.Bonjour, P-L Mathey, S. Sanchi, M.L.Sawey, *Sailing yacht design using advanced numerical flow techniques*, Proc. Of 14th Chesapeake Sailing Yacht Symposium,pp. 97-104, 1999.
[17] C.M.Rhie, W.L.Chow, *Numerical study of the turbulent flow past an airfoil with trailing edge separation*, AIAA Journal, Vol. 21, No. 11, pp.1525-1532, 1983.
[18] D.P.Rizzetta, M.R.Visbal, *Comparative numerical study of two turbulence models for airfoil static and dynamic stall*, AIAA Journal, Vol. 31, No. 4, pp. 784-786, 1993.
[19] E.V.Lewis, *Principles of Naval Architecture*, SNAME, 1988.