Analysis of aerodynamic indices for racing sailing yachts: a computational study and benchmark on up to 128 CPUs

Citation for published version:
Viola, IM, Ponzini, R, Rocchi, D & Fossati, F 2010, Analysis of aerodynamic indices for racing sailing yachts: a computational study and benchmark on up to 128 CPUs. in *Parallel Computational Fluid Dynamics 2008.* vol. 74 LNCSE, Lecture Notes in Computational Science and Engineering, vol. 74, Springer-Verlag GmbH, pp. 61-70. https://doi.org/10.1007/978-3-642-14438-7_6

Digital Object Identifier (DOI):
10.1007/978-3-642-14438-7_6

Published In:
Parallel Computational Fluid Dynamics 2008

Analysis of Aerodynamic Indices for Racing Sailing Yachts: a Computational Study and Benchmark on up to 128 CPUs.

Ignazio Maria Viola[a], Raffaele Ponzini[b], Daniele Rocchi[c], Fabio Fossati[c]

[a] *Yacht Research Unit, the University of Auckland, New Zealand (im.viola@auckland.ac.nz)*

[b] *High-Performance Computing Group, CILEA Consortium, Milan, Italy*

[c] *Mechanical Department, Politecnico di Milano, Milan, Italy*

Keywords: CFD, Parallel Computing, Benchmarks, Yacht Sail Plans, Downwind Sails, Wind Tunnel Tests.

1. SHORT ABSTRACT

This work presents a feasibility study for trustable and affordable CFD analysis of aerodynamic indices of racing sailing yachts. A detailed reconstructed model of a recent America's Cup class mainsail and asymmetrical spinnaker under light wind conditions has been studied using massive parallel RANS modeling on 128 CPUs. A detailed comparison between computational and experimental data has been performed and discussed, thanks to wind tunnel tests performed with the same geometry under the same wind conditions.
The computational grid used was of about 37 millions of tetrahedra and the parallel job has been performed on up to 128 CPUs of a distributed memory Linux cluster using a commercial CFD code. An in deep analysis of the CPU usage has been performed during the computation by means of Ganglia and a complete benchmark of the studied case has been done for 64, 48, 32, 16, 8 and 4 CPUs analyzing the advantages offered by two kind of available interconnection technologies: Ethernet and Infiniband.
Besides to this computational benchmark, a sensitivity analysis of the global aerodynamic force components, the lift and the drag, to different grid resolution size has been performed. In particular, mesh size across three orders of magnitude have been investigated: from 0.06 million up to 37 million cells.
The computational results obtained here are in great agreement with the experimental data. In particular, the fully tetrahedral meshes allow appreciating the beneficial effect of the increasing of the grid resolution without changing grid topology: a converging trend to the experimental value is observed.
In conclusion, the present results confirm the validity of RANS modeling as a design tool and show the advantages and costs of a large tetrahedral mesh for downwind sail design purposes.

2. INTRODUCTION

RANS analysis is playing a central role in the recent America's Cup (AC) races for both hydrodynamic and aerodynamic design aspects. In the last 30 years computational analysis capabilities and affordability have grown so much that in the last AC (2007, Valencia, Spain) all the twelve syndicates had invested a comparable amount of money in experimental tests and in computational resources. It is only in last few years that RANS has become a trustable design tools, in particular in the sail design field. In fact, in some sailing condition the flow around the sails are largely separated and a large computational effort is required to accurately compute the resultant aerodynamic forces.
The aerodynamics of sails can be divided in three branches: the aerodynamic of upwind sails, reaching sails and running sails.
Upwind sails are adopted when sailing at small apparent wind angle (AWA), typically smaller than 35°, where AWA is generally defined as the angle between the yacht course and the undisturbed wind direction at the 10m

reference height above the sea surface. Single mast yachts, namely sloop, adopt a mainsail and a jib or a genoa, which are light cambered airfoils designed to work close to the optimum efficiency, i.e. to maximize the lift/drag ratio. The flow is mainly attached and consequently un-viscous code has been adopted with success since sixties to predict aerodynamic global coefficients [1], [2], and in the last decades several RANS applications have shown a good agreement with wind tunnel tests [3], [4].
Reaching sails are adopted when sailing at larger AWA, typically from 45° to 160°. Sloop modern racing yachts often adopt the mainsail and the asymmetrical spinnaker, which are more cambered airfoils designed to produce the maximum lift [5], [6]; in fact sailing at 90° AWA the lift force component is aligned with the course direction. The flow is attached for more of the half chord of the sail and separation occurs on the trailing edge of the asymmetrical spinnaker. In particular, the flow field is strongly three dimensional because of the increasing of the vertical velocity component, the tip and root vortexes are strongly connected to the trailing edge vortex. Reaching sail aerodynamics requires the capability to correctly compute the separation edge on the leeward spinnaker surface, hence un-viscous code are not applicable and Navier-Stokes code might be adopted. The first RANS analysis has been performed by Hedges in 1993 [7], [8] with limited computational resources. More recently, in 2007 [9] and 2008 [10] two works performed with less than 1 million of tetrahedral cells show good agreement with wind tunnel data: differences between computed and measured force components are between 11% and 7% in lift coefficient and between 12% and 5% in drag coefficient, where lift and drag coefficients are defined as follow in equation (2).
Running sails are adopted at larger AWA and sloop yachts generally adopt a mainsail and a symmetrical spinnaker. The flow is mostly separated and sails work as bluff bodies. Separation occurs on the sail perimeters and the drag has to be maximized [5], [6].
In the AC races, the racing curse is around two marks positioned along the wind direction, in such a way that half of the race has to be sailed upwind and half downwind. In the leeward leg, yacht sails at closer AWA to increase the apparent wind component (due to their own speed) in light air, and sails al larger AWA to reduce the sailed course in stronger breeze. In the recent AC races a wind speed limitation lead to sail mainly reaching than running and for this reason particular focus has been placed on asymmetrical spinnakers.

In the present work, an America's Cup Class, version 5 [11], are studied in a downwind reaching configuration sailing at 45° AWA with mainsail and asymmetrical spinnaker and a RANS analysis has been performed to investigate the benefits in the global force computation accuracy with a very large mesh. A 37 millions of cells mesh has been performed with the commercial codes Gambit and Tgrid by Ansys Inc., which adopt a bottom-down approach: meshes are generated from lower to higher topology, hence from edges to surfaces and than to volumes. Only tetrahedral cells have been adopted. The computation has been performed with Fluent 6.3.26 (Ansys Inc.) solving the uncompressible Navier-Stokes-equations. In Figure 1 a visualization of the mathematical model is showed.
The herein obtained computational results on the 37 million-cell mesh have been compared with both computational (previously obtained on smaller meshes and under the same fluid dynamics conditions) and experimental data acquired in the Politecnico di Milano Twisted Flow Wind Tunnel.
In the following of the paper the experimental set-up is described, then the computational aspects are highlighted together with the hardware and the interconnection technologies used in the parallel run of the numerical simulations, finally numerical results are discussed and compared with experimental measurements in terms of aerodynamics indices such as lift and drag global coefficients.

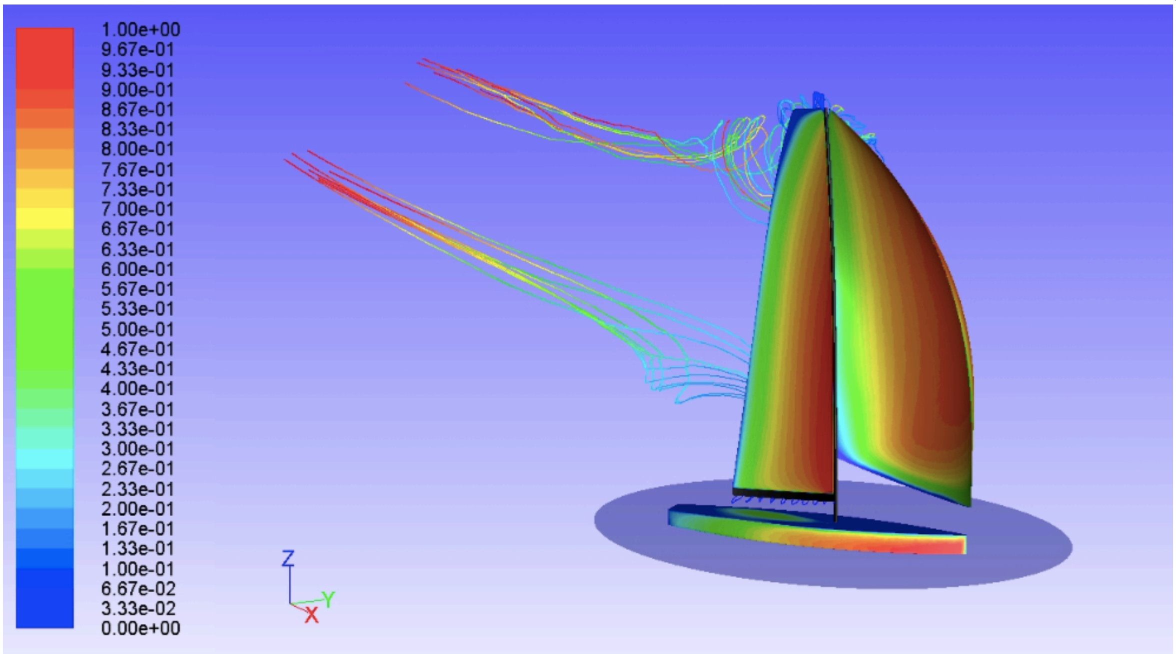

Figure 1: static pressure coefficient distribution on sails and hull ($Cp=(p-p_0)/q$, where p_0 is the outflow undisturbed reference pressure and q is the inflow undisturbed reference dynamic pressure). Path lines colored by time show the boom and the mainsail tip vortexes. The yacht is sailing at 45° of apparent wind angle (i.e. the angle between the hull longitudinal axes and the incident wind at the reference height of 10m full scale) and is 5° leeward heeled. The America's Cup Class (version 5) mainsail and asymmetrical spinnaker for light wind are trimmed to produce the maximum driving force in the boat direction.

3. EXPERIMENTAL MEASUREMENTS

Experimental test has been performed in the Politecnico di Milano Twisted Flow Wind Tunnel. It is a closed circuit wind tunnel with two test sections respectively designed for civil and aerospace applications. On the left of Figure 1 the wind tunnel rendering is presented, airflow is running anti clockwise. On the lower side, aerospace low turbolence test section is showed. On the upper side, the long civil boundary layer test section, 36m length, 14m wide and 4m high, where sail plan tests are performed, which is showed on the right of Figure 12.

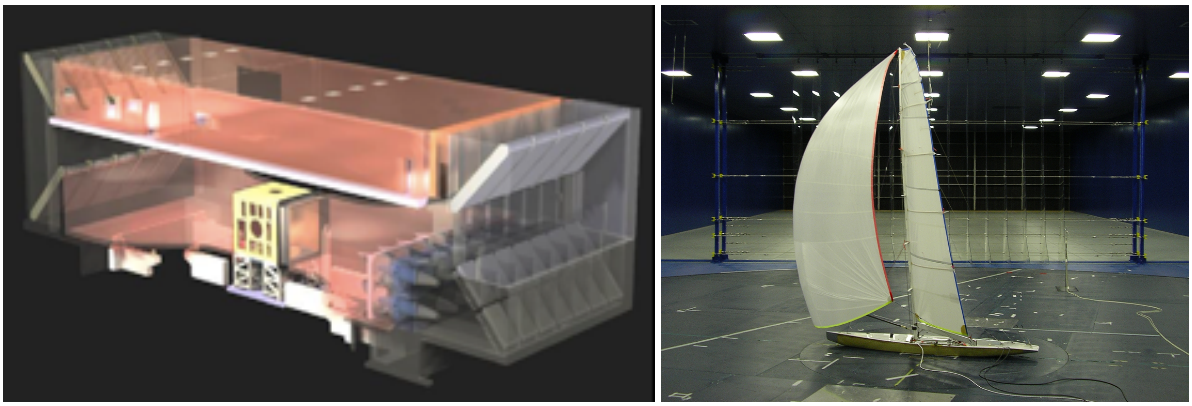

Figure 2, **left**: Politecnico di Milano Twisted Flow Wind Tunnel closed circuit;
right: sail plan test in the boundary layer test section

The 1:12,5 scaled model is fitted on a 6-component dynamometer and it is supplied of 7 drums to trim sails as in real life, operated through a proportional radio control system. Sails are trimmed to produce the maximum aerodynamic force component in boat direction, i.e. driving force. Then actual measurements are obtained by sampling the data over 30 seconds at 100Hz. Coefficients are obtained dividing forces with a reference dynamic pressure and sail area. Reference wind speed is measured 5m windward at the reference height corresponding to 10m in full scale. Wind tunnel tests have been performed with target velocity and twisted profiles according specific situation of an ACC yacht sailing in Valencia atmospheric boundary layer. More details about wind tunnel tests can be found in [12].

4. NUMERICAL ANALYSIS

The commercial code Fluent (Ansys Inc.) with a segregated solver strategy has been used to solve the equations of the flow around the sailing boat without considering time dependence (i.e. steady state), volume forces (i.e. gravity) and density variations and therefore energy equation hasn't been solved. SIMPLE scheme has been solved and first discretization order has been adopted. None turbulence model has been adopted.
All the computations were performed on a Linux Cluster equipped with 74 CPUs AMD Opteron 275 dual-core (2.2 GHz, 2 GB/core) interconnected with Infiniband 4x (10GB/s) and Gigabit Ethernet.
Due to the lack of information for such kind of models we launched the execution of the computation on 128 CPUs according to the maximum degree of parallel processing permitted by the license. The overall computation together with all the input and output operations and file writing took about one week.
During the computation we monitored and analyzed the usage of the CPU using Ganglia (a system able to monitoring and store data concerning the usage of network and CPU in clusters computers); observing that the usage of the CPU was sub-optimal, we decide to perform an accurate benchmark in order to find out the optimal CPU usage. In particular we perform a descending benchmark on 64, 48, 32, 16, 8 and 4 CPUs testing two type of interconnection network, Infiniband and Ethernet Gigabit, and performing 100 iterations starting from the archived data. The benchmarking could not be performed on less then 4 CPUs since it was not possible to allocate in memory the 37 million elements mesh on less then 32GB of memory and wondering to take advantage of the multi-core architecture. For this reason all the results concerning the speed-up evaluation and the efficiency are referred to the 4 CPUs test case. In Figure 3 (left) the total wall time is plotted against the number of the used CPU for the two interconnections considered.

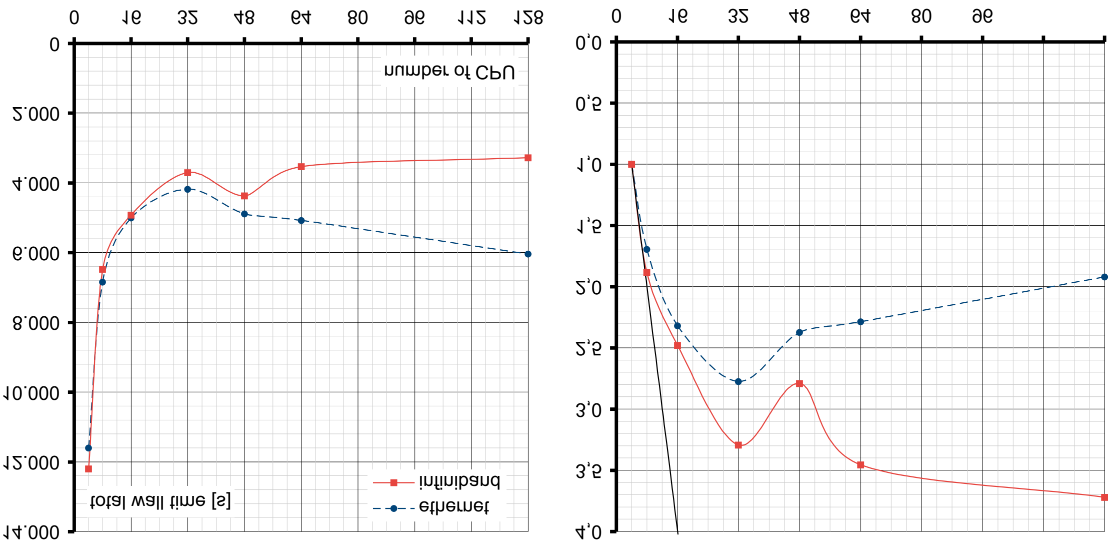

Figure 3, Results on 100 iterations, **left**: Total wall time; **right**: speed-up.

In Figure 3 (rigth) the speed-up with respect to the 4 CPUs test is plotted against the number of the used CPU for the two interconnections considered, in Figure 4 the efficiency, again with respect to the 4 CPUs test case, is plotted against the number of the used CPU for the two interconnections considered.

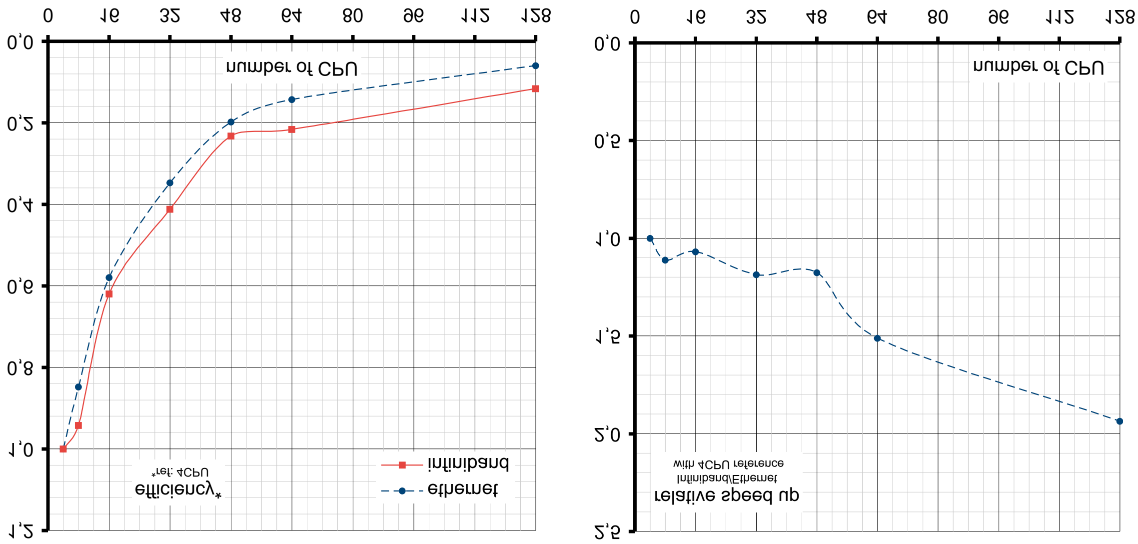

Figure 4, Results on 100 iterations, **left**: efficiency; **right**: gain on speed-up computed as in equation (1)

In order to better appreciate the gain on using a more performing interconnection and due to the fact that we did not had the possibility to compute the bench on less then 4 CPUs, in Figure 4 we plot the relative speed-up computed at a fixed value of used CPU according to:

Relative speed-up = (Infiniband speed-up) / (Ethernet speed-up) *(1);*

The relative speed-up shows that Infiniband speed-up rise up to about double the Ethernet speed-up in the case of 128 CPUs.
All the benchmark results are consistent to the fact that for this case the optimal usage of the CPU is obtained with a degree of parallelism equal to 32, moreover significant advantages are obtainable by means of a high performing interconnection (Infiniband) using higher number of CPU as shown in Figure 4 right.

5. RESULTS

The numerical simulations showed a good agreement with the experimental data, the 37M cells mesh shows differences smaller than 3% in both the global aerodynamic force coefficients lift and drag, defined as following:

$$CD = \frac{drag}{\frac{1}{2}\rho V^2 S} \qquad CL = \frac{lift}{\frac{1}{2}\rho V^2 S} \qquad (2);$$

Where drag and lift are forces along the wind and perpendicular to the wind, respectively, in the horizontal plane acting on the yacht model above the water-plane (included hull rigging and sails), ρ air density, V undisturbed incoming reference wind speed measured at 10m height full-scale, S sail area (sum of the two sail surfaces).

The converging criteria is based on the drag and lift coefficients, which are monitored every iteration until the average values become stable. Figure 5 shows the sensitivity analysis to the mesh dimension: the drag (left) and lift (right) coefficients divided by the experimental values are plotted for the 37M cells mesh together with three other meshes of 0.06M, 1M, 6.5M respectively, obtained in previously validated studies and under the same fluid dynamics conditions. Circle and square marks show the average value and the error bars show the standard deviation of the coefficient oscillations.
Increasing the mesh size of about three order of magnitude an increasing accuracy is obtained: the maximum differences between computed lift and drag with respect to the experimental values is smaller than 8% for the coarser mesh and becomes smaller than 3% for the finer mesh. By the way, the lift coefficients trend comes

across the experimental values: lift is overestimated with coarser mesh and underestimated with finer mesh. Increasing mesh size both drag and lift curve are decreasing monotone. Further researches will be aimed to explore mesh larger than 100 million-cells, which has not been performed up to now because of the computational requirements that would be larger than 100GB of memory usage. Nevertheless the herein discussed work shows that some kind of large scale parallel approaches to RANS code applications in this filed can be a valid candidate to overcome these technical limits.

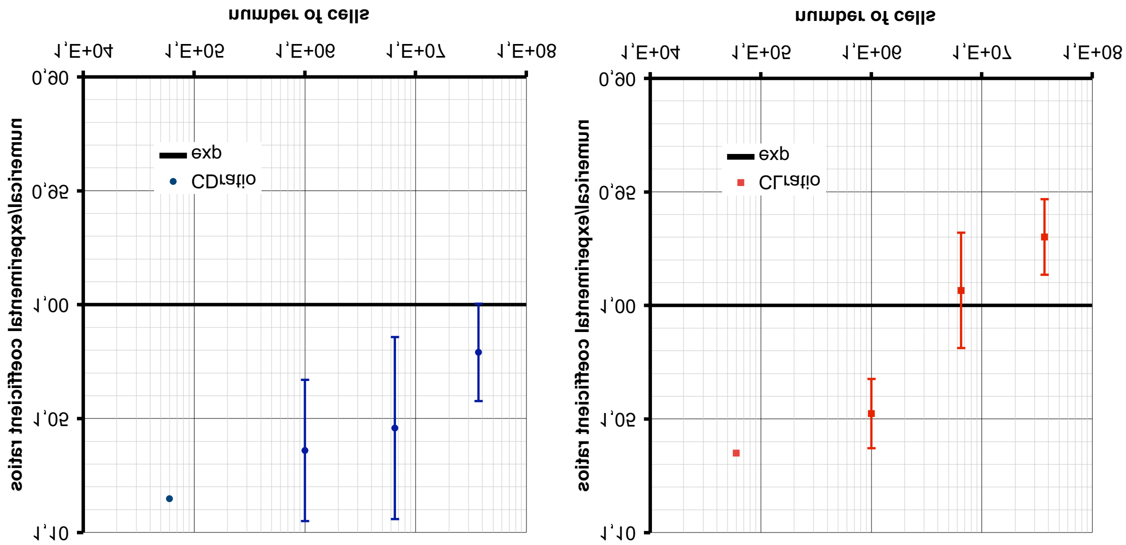

Figure 5: numerical/experimental coefficient ratios are plotted versus the overall number of cells. Circle and squared marks show drag CD (**left**) and lift CL (**right**) average values, respectively, and error bars show the standard deviation of the coefficient signals.

The four meshes are fully tetrahedral and with similar grow rate (the linear dimension ratio between two adjacent cells in the wall-normal direction), hence they are all topologically similar. The wall adjacent tetrahedron dimension, and hence the distance between the tetrahedron centre and the wall (namely the first cell-centre-height $\mathbf{y_1}$) have a dramatic impact onto the resultant overall cells number. In Figure 6 on the left, the ratio between the first cell-centre-height of each mesh and the cell-centre-height of the coarser mesh are plotted versus the resultant overall number of cells.

In Figure 6 on the right, the $\mathbf{y^+}$ values are plotted versus the overall number of cells. An horizontal section at $^1/_3$ height of the yacht model from the water-plane has been considered and the $\mathbf{y^+}$ values are referred to the asymmetric spinnaker leeward edge intersecting the plane.

Values are collected from the cells placed on the asymmetrical spinnaker at $^1/_3$ height of the yacht model from the water-plane at the last iteration stage. In figure the maximum, minimum and average $\mathbf{y^+}$ values are plotted for each mesh.

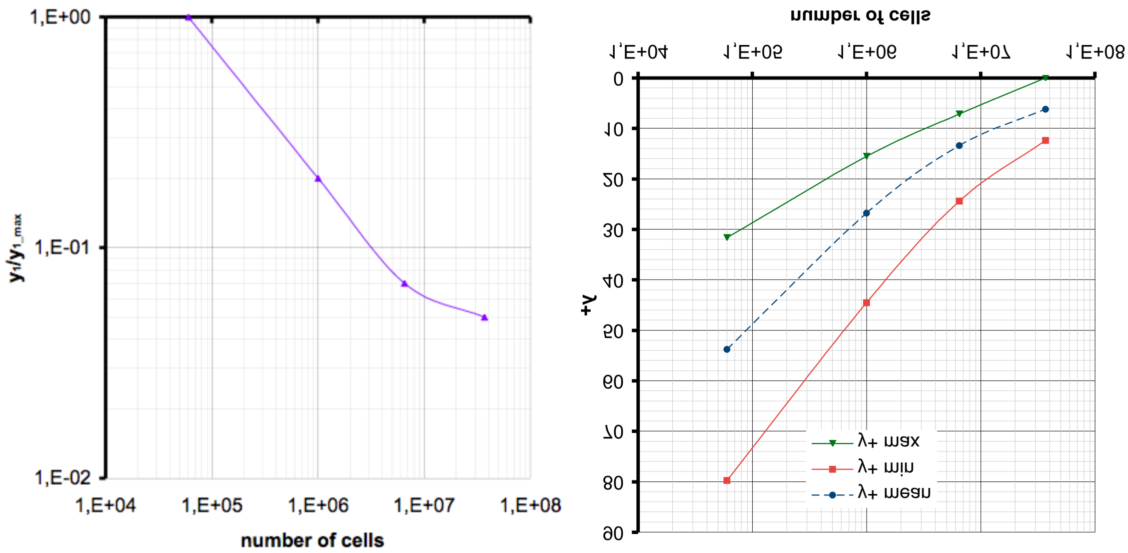

Figure 6, **left**: first cell-centre-height of each mesh divided by the first cell-centre-height of the coarser mesh of 0.06 million cells are plotted versus the overall mesh size; **right**: maximum, minimum and average $\mathbf{y^+}$ values computed at the last iteration stage and collected from the cells of the asymmetrical spinnaker at $^1/_3$ height of the yacht model from the water-plane are plotted versus the overall mesh size.

6. CONCLUSION

In the present work a detailed study of feasibility of CFD approaches on the study of aerodynamics indices in racing sailing yachts is discussed. The main purpose of the study was to understand the usefulness of parallel computational approaches on the evaluation of several typical aerodynamic indices used to design and test in a synthetic manner the performance of a racing sailing yacht. In order to reach this scope a 37 million cells computational model of a ACC-V5 yacht model have been studied on 128 CPUs at the CILEA computer centre, using the parallel version of the commercial code Fluent (Ansys, Inc.) and all the typical aerodynamic factors, such as lift and drag coefficients, as been computed under steady state condition. Computed coefficients have been compared with experimental measurements performed at the Politecnico di Milano Twisted Flow Wind Tunnel, showing very good agreement: differences in both lift and drag smaller than 3%.
In order to evaluate the usefulness of such approach (i.e. using a 37 million mesh) with respect to smaller discretization, we compared the herein obtained results with other pre-computed ones obtained respectively with a 0.06M, 1M, and 6.5M elements and under the same fluid dynamics conditions. An increase in force coefficient computed accuracy has been observed increasing the mesh size.
Finally wondering to understand the better balancing between number of processors, mesh dimension and CPU usage, we performed a benchmark of 100 iteration of the same computational model using 64, 48, 32, 16, 8 and 4 CPUs and with two king of interconnection technologies. In this sense the best configuration is obtained using Infiniband interconnection and 32 CPUs.
In conclusion this work show the feasibility of very large parallel CFD processing with a concrete gain in accuracy that confirm the usefulness of computational approaches as trustable and affordable tools for design and hypothesis testing today more and more complementary to the necessary experimental analysis.

8. AKNOWLEDGEMENT

The Authors are grateful to the Fluent Italia (Ansys) personnel and in particular in the person of Marco Rossi for their support and the confidence accorded during this experimentation.

9. REFERENCES

[1] J.H. Milgram: *The Aeodynamic of Sails*; proceedings of 7th Symposium of Naval Hydrodynamic, pp. 1397-1434, 1968.
[2] Arvel Gentry: *The Application of Computational Fluid Dynamics to Sails*; Proceedings of the Symposium on Hydrodynamic Peformance Enhancement for Marine Applications, Newport, Rhode Island, US, 1988.
[3] H. Miyata, Y.W. Lee: *Application of CFD Simulation to the Design of Sails*; Journal of Marine Science and Technology, **4**:163-172, 1999.
[4] A.B.G. Querard and P.A. Wilson; *Aerodynamic of Modern Square Head Sails: a Comparative Study Between Wind-Tunnel Experiments and RANS Simulations*; In the Modern Yacht, Southampton, UK, 11-12 Oct 2007. London, UK, The Royal Institution of Naval Architects, 8pp, 107-114, 2007. http://eprints.soton.ac.uk/49314/.
[5] P.J. Richards, A. Johnson, A. Stanton: *America's Cup downwind sails - vertical wings or horizontal parachutes?*; Journal of Wind Engineering and Industrial Aerodynamics, **89** 1565–1577, 2001.
[6] William C. Lasher, James R. Sonnenmeier, David R. Forsman, Jason Tomcho: *The aerodynamics of symmetric spinnakers*; Journal of Wind Engineering and Industrial Aerodynamics, **93** 311-337, 2005.
[7] K.L. Hedges: *Computer Modelling of Downwind Sails*; MF Thesis, University of Auckland, New Zealand, 1993.
[8] K.L. Hedges, P.J. Richards, G.D. Mallison: *Computer Modelling of Downwind Sails*; Journal of Wind Engineering and Industrial Aerodynamics **63** 95-110, 1996.
[9] William C. Lasher and Peter J. Richards: Validation of Reynolds-Averaged Navier-Stokes Simulations for International America's Cup Class Spinnaker Force Coefficients in an Atmospheric Boundary Layer; Journal of Ship Research, Vol. 51, No. 1, pp. 22–38, March 2007.
[10] William Lascher & James R. Sonnenmeier: *An Analysis of Practical RANS Simulations for Spinnaker Aerodynamics*; Journal of Wind Engineering and Industrial Aerodynamics, **96** 143-165, 2008.
[11] Challenger of Record and Defender for America's Cup XXXII: *America's Cup Class Rule, Version 5.0*; 15th December 2003;
[12] F. Fossati, S. Muggiasca, I.M. Viola, A. Zasso: *Wind Tunnel Techniques for Investigation and Optimization of Sailing Yachts Aerodynamics*; proceedings of *2nd High Performance Yacht Design Conference*; Auckland, NZ, 2006.