Sailing Yacht Computational Aerodynamics: an investigation in RANS capabilities with large computational resources on up to one billion-cells mesh – the PROJECT

Introduction

Computational Fluid Dynamics (CFD) of racing sailing yachts represents a cutting hedge application field in the marine research technologies. Since 1995 the three-dimensional Navier-Stokes CFD modeling became a complementary tool for any successful design and prototyping . The first Navier-Stokes application to sails has been performed in 1995 by Hedges et al. [1] with a few thousands of cells, six years later Masuyama et al. [2] adopted several tens of thousands of cells. In the recent years racing sailing yachts teams have animated the research in this exciting field and pushed far further the target, considering the necessity to overcome a smaller and smaller gap between the fastest and the slowest boats with more accurate computational modeling. In recent years several millions of cells has been used to model the yacht aerodynamics [3]. Moreover, CFD results are required to match not only the experimental wind tunnel results but also the most demanding benchmark: the full scale performance during the race.

Rationale

In the first half of 2008 the Naval Engineer Ignazio Maria Viola, that during his Ph.D. was involved in the computational and experimental fluid dynamics investigation in sailing yachts for the 32nd America's Cup, sharpened three groups' enthusiasm to investigate the up-today capabilities of the Computational Fluid Dynamics codes in this field. The research team composed by three Ph.D. Engineers, Ignazio Maria Viola, Raffaele Ponzini (CILEA) and Giuseppe Passoni (Politecnico di Milano), have overreached the billion cells frontier investigating the aerodynamic of a yacht sailing downwind with a mainsail and an asymmetrical spinnaker. Regione Lombardia, within the LaborLab project for the Technical-Scientific environment, has financially supported the training grant of PhD. Eng. Viola. CILEA (interuniversity consortium for information and communication technologies) has made available the new super-cluster named Lagrange, 135th in the TOP500

world supercomputer ranking (released in June 2008) [4]. Politecnico di Milano has supported the research with his internationally well recognized know-how in the fluid dynamic science. Ansys Inc. has supported the research providing a 1024 parallel licenses of the world's leader CFD code Fluent.

Materials and methods

At the Politecnico di Milano Wind Tunnel in the 4x14m boundary layer test section, a 1:12.5 scale model of an America's Cup Class sailing yacht [3] has been studied, with flexible sails that can be remotely trimmed operating independently onto 6 drums connected to the sails sheets. A 6-component dynamometer is fixed inside the boat model and the measured forces are displayed in real time to allow the operator to trim the sail optimizing the trust force. When the optimum trim is achieved, forces are recorded at 100Hz for 20 seconds and the mean value is computed. Without changing the trim, pictures of the sail shapes are recorded from several points of views to allow the photogrammetric reconstruction of the sail shapes. The reconstructed geometry has been used in the present work for CFD calculation on up to one billion cells.

The steady uncompressible Navier-Stokes formulation has been solved. The computation has required 179 CPUs and 2,24 TB of RAM for 1 month, equivalent to a total wall time of about 100.000 hours. The cluster has been provided by CILEA, a Hewlett Packard Linux cluster equipped with 208 dual-proc blade nodes with Intel Xeon 3.166 GHz quad-core cpus and 2 GB RAM per core (1664 cores; 3,3 TB RAM total), the total peak performance approaching 22 TFlop/s (thousands of billions of floating point operations per second).

Results and conclusions

The aerodynamic forces and related coefficients computed using CFD modeling has been compared to experimental data obtained during the wind tunnel tests with good agreement confirming the capabilities of CFD in predicting with increasing accuracy the main quantities involved in engineering analysis and design.

In conclusion the present works is a great advancement in the computational fluid dynamic field, being the first concrete engineering application in the world with a complex fully three-

dimensional model performed with a single mesh of more than 1 billion cells. This exciting result has been possible thanks to the cooperation of three researchers with specific and different background and to the overlap of four prestigious partners to which the authors are grateful.

Contacts

For any further information please write to ignazio.viola@tiscali.it or ponzini@cilea.it .

References

[1] 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.

[2] Y. Masuyama, Y. Tahara, T. Fukasawa, N. Maeda: Database of Sail Shapes vs. Sail Performance and Validation of Numerical Calculation for Upwind Condition; The 18th Chesapeake Sailing Yacht Symposium, Annapolis, Maryland, USA, March 2007.

[3] I.M. Viola: Numerical and Experimental Flow Dynamics applied to Sailing Yacht Dynamics, Ph.D. Thesis, Politecnico di Milano, May 2008.

[4] Top500: http://top500.org/lists/2008/06