HPC and CFD in the Marine Industry: Past, Present and Future

Kurt Mizzi
University of Strathclyde,
100 Montrose Street,
Glasgow
+44 (0)141 548 4911
kurt.mizzi@uni.strath.ac.uk

Paula Kellett
University of Strathclyde,
100 Montrose Street,
Glasgow
+44 (0)141 548 3237
paula.kellett@strath.ac.uk

Yigit K. Demirel
University of Strathclyde,
100 Montrose Street,
Glasgow
+44 (0)141 548 4275
yigit.demirel@strath.ac.uk

Richard Martin
University of Strathclyde,
107 Rottenrow East,
Glasgow
+44 (0)141 548 3265
richard.martin@strath.ac.uk

Osman Turan
University of Strathclyde,
100 Montrose Street,
Glasgow
+44 (0)141 548 3211
o.turan@strath.ac.uk

ABSTRACT

This paper explores the use of Computational Fluid Dynamics (CFD) applications on High Performance Computing (HPC) platforms from the perspective of a user engaged in Naval Architecture research. The paper will consider the significant limitations which were imposed on research boundaries prior to present HPC capabilities, how this impacted development in the field and the implications for industry. One particular example is the costly experimental testing which, due to resource constraints, is generally restricted to model scale. It will then present an overview of the numerical simulation capabilities using current HPC performance and capability.

With the increase of computational power and capacity, CFD simulations are proving to be more accurate and reliable. Being relatively cheaper and more time efficient, numerical methods are becoming the preferred choice within the industry compared to traditional experimental tests. Nevertheless, certain experimental procedures cannot be numerically replicated with the current levels of computational capacity.

The future needs and challenges of research and development will be outlined and discussed, highlighting the significant impact exascale computing will have in the field.

Keywords

Computational Fluid Dynamics (CFD), HPC Application, HPC Capability, Capacity Implications

1. INTRODUCTION

The aim of this paper is to provide an understanding of the importance of HPC and its contribution to research development in the area of marine research. Several examples of research fields which are extremely important in the marine sector, both academically and also crucially for the industry, will be used to further these discussions. More specifically, the ability to directly simulate ship resistance, the effects of fouling on ship performance, and underwater noise predictions at model and full scale will be discussed. HPC capabilities that allow the use of CFD as an optimization tool, with the example of Propeller Boss Cap Fin (PBCF) designs, will also be outlined.

Although High Performance Computing has come a long way, current HPC power still limits certain analyses and research, even though some of the current capability would never have been thought to be possible even a few years ago. A typical example would be propeller cavitation which is best predicted using a Detached Eddy Simulation (DES) solver that requires high computational capability. This indicates that computational capability dictates research boundaries as well as processes, which can be significant in an industry which tends to be conservative and reactive.

Although maximising HPC capabilities broadens the horizons for research and analysis, one common problem is convincing industry to rely on numerical simulations. Although these methods may be beneficial financially, while enabling the generation of high levels of useful data, quality assurance of the product is always a prime concern and therefore such new procedures need to be well proven and presented. This paper will also discuss earning the trust of industry because ultimately it is industry that drives and funds research in this sector, enabling it to progress.

2. CFD AND ITS IMPORTANCE IN NAVAL ARCHITECTURE

Generally, during the initial stages of current ship design processes, various design analyses are carried out using numerical approaches typically known as CFD methods. Before CFD procedures were available, all investigations were carried out experimentally in facilities, such as the one shown in Figure 1, which were very time consuming and expensive and which therefore constrained research to the analysis of a limited number of designs. Apart from the complexity of involving a number of stakeholders in experiments, they also incorporate a number of assumptions, errors and tolerances, the most common being the issue of scaling. All experimental studies are carried out at model scale, introducing scaling errors in the extrapolation of the results to full scale. The ability to simulate at full scale is very important as it removes the errors and correction factors associated with model scale testing due to scaling effects, and in cases where improvements in the range of 2-3% are being sought, it is very beneficial to have the sources of error minimised as far as possible. These errors arise due to the fact that the viscous effects of water cannot be scaled, and will therefore be the same at both model and full scale.

CFD technology is capable of predicting various parameters such as resistance, motion, free surface capturing, manoeuvring performance etc. some of which is very hard to predict in experimental procedures due to the need for instantaneous visualization or sophisticated measurement tools.

However, as previously explained, the industry still considers experimental investigation to be more reliable. Therefore, to save on costs, the design process is initially carried out using simulation based design (CFD) analyses followed by experimental studies at the final stages for validation.

High fidelity CFD tools have enhanced research, pushing boundaries in all aspects of the marine industry. For this reason, researchers should focus on developing and improving next generation simulation based design in order to open new horizons and opportunities in marine research. This can generally be achieved by improving the numerical solvers as well as maximising HPC capabilities.

Figure 1. Experimental Tests

3. PAST AND PRESENT COMPUTATIONAL LIMITS

In the very early stages of the application of CFD to Naval Architecture problems, limited computational power and capacity restricted most analyses to potential flow methods. Typically, these methods did not take into account the viscous effects of the fluid which resulted in a loss of accuracy and an over simplification of the problem.

Therefore, the use of more simplified approaches was common and this in turn meant that the capture of complex flow phenomena was much more difficult. In time, with the advance in computational power, Reynolds-Averaged Navier-Stokes (RANS) equations based CFD simulations became more feasible allowing the use of more complex and representative turbulence models. These simulations were time consuming and computationally expensive thus constraining the cell count for meshes to the extent that only model scale could be simulated, which produced similar errors to those observed in experimental results. This also prevented detailed representation of the geometry and proper physical predictions. In addition, simulations took longer to converge, resulting in the fact that analytical and parametric studies were limited and optimization procedures were not really an option. Full scale simulations are still not as common as model scale and when they are applied, certain methods and approaches are typically used (such as mesh refinement regions) to limit the overall size of the mesh and computational domain.

Recent improvements have also led to the introduction of Large Eddy Simulation (LES) and Direct Eddy Simulation (DES) methods which are less common again, but can provide more accurate results through the application of fewer assumptions in certain cases. These kinds of simulation are required for applications such as cavitation, which is discussed later, however they remain time consuming and computationally expensive in comparison to RANS approaches. One promising approach is to blend both the RANS and DES models to produce a compromise between accuracy and execution time.

With the advance towards Exascale Computing, more advanced numerical methods may be introduced. There will likely be a move towards Direct Numerical Simulation (DNS) approaches, where the use of assumptions to simplify the problem are removed, making the results yet more accurate but significantly increasing the calculations that need to be carried out at each time-step.

A comprehensive discussion on the recent advances in CFD, and potential future trends and developments, is presented in [1].

4. CURRENT STATE OF THE ART

The following sections will present the current state of the art in CFD using three case studies from industry-relevant naval architecture problems.

4.1 Naval Architecture Case Studies

The examples discussed in this section outline topics in naval architecture which have been addressed at University of Strathclyde using current CFD capabilities, particularly the commercial Star-CCM+ software. Their advances over past capability as well as their limitations will be highlighted.

4.1.1 Ship Resistance and Fouling

Shipping has been, and still is, one of the most important methods of transport, with more reliance and importance now being placed on this mode of transport as a consequence of advancements in shipping technology and the ability of ships to hold and store increasing capacities of goods. However, these improvements and features bring some problems to the industry due to an increase in fuel consumption, which is detrimental to the environment and which erodes company revenues. Although other forms of fuel power exist, such as wind energy and solar power, carbon-based fuel is currently the only way for ships to run effectively. For this reason, minimising fuel consumption is crucial for shipping companies.

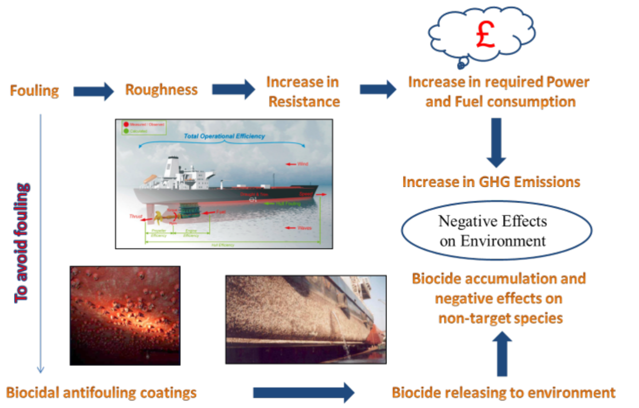

Figure 2. Fouling Analyses Motive

A major challenge is to relate technologies, such as antifouling coatings and the effect of biofouling, to ship resistance and fuel consumption, in order to evaluate their effects on energy efficiency and hence CO_2 emissions (see Figure 2). While retrofitting existing ships with new antifouling paints will improve their energy efficiency, it is equally important to accurately model the potential effects of biofouling on ship resistance and to demonstrate the importance of the mitigation of such effects through scientific research.

Two different CFD models were therefore proposed as outlined in [2] and [3] for the prediction of the roughness effects of antifouling coatings and biofouling on ship resistance. It is worth pointing out that the total number of cells of such CFD simulations is very high since the roughness of the hull surface requires to be represented in the order of micrometers (μm). Additionally, the time-step used in such simulations has to be kept very small due to the very complex and unstable nature of the phenomenon. The capability of the computer system is therefore of great importance for researchers and engineers to be able to carry out reliable and feasible CFD analyses.

The effect of the total cell numbers on the results of such analyses are highlighted in the following table. For each mesh configuration, the frictional resistance coefficients (C_F) of a flat plate of ship length, coated with an antifouling coating are listed in Table 1 [2].

Table 1: C_F results of antifouling coatings at different mesh configurations [2].

Mesh configuration	Total No. of Cells	C_F (CFD)
Coarse	1.8 x10^6	0.001574
Medium	2.5 x10^6	0.001576
Fine	4 x10^6	0.001584
Very Fine	5.5 x10^6	0.001584

From Table 1 it is evident that the results converged very well provided that the total numbers of cells are sufficiently high. Such simulations could not have been run without the existence of High Performance Computers.

It is significant that the modelling used in these simulations was based on available experimental data and was achieved using particular assumptions on the flow properties. Although the proposed CFD model is a reasonable method to predict these effects, the effects of biofouling on ship resistance can only be precisely predicted using a means of geometrical modelling of relevant organisms. Unfortunately, geometrical modelling of such small organisms in detail, along with the other complexities of the ships' systems such as a rotating propeller, is beyond the scope of current HPC capability. Additionally, spatial inhomogeneity of fouling on ship hulls is another challenge for modelling which is still beyond the reach current computational capabilities.

4.1.2 Ship Radiated Underwater Noise

A topic which has recently become important in the marine industry is underwater noise from anthropogenic noise sources, in particular shipping, and its impact on marine wildlife. As outlined in [4], CFD has been used to predict the underwater noise of a moving ship and rotating propeller at given locations. In the past, simulations would not have allowed for a rotating propeller to be modelled. This meant that a steady state approximation was therefore applied to an unsteady problem. However, these kinds of simulations are now feasible, as in Figure 3, and have been carried out at full scale, with the benefit of removing the additional errors associated with scaling. However, given the limits of current computational capability, the vessel had to be simulated moving in calm and very deep water, with the propeller operating in a non-cavitating condition. Therefore, propeller noise and flow noise could be captured but the significant contribution to underwater noise from cavitation could not be addressed in this case.

Cavitation is the phenomenon whereby the water at the propeller effectively boils due to the pressure differential cause by the rotation of the propeller above a certain speed. Almost all propellers will cavitate at certain operational conditions. This phenomenon is of considerable concern in the marine industry as it leads to reduced propeller efficiency, high levels of underwater noise, and in more extreme cases, damage to the propeller and hull. An example of a cavitating propeller can be seen in Figure 4 below. With current CFD capabilities, the higher complexity solvers and the fine computational meshes required for simulations of cavitation only allow them to be carried out at model scale, and generally with only partial hull geometry and a calm water surface represented. The significant additional demands in simulating this phenomenon in more realistic conditions are beyond current computational capability.

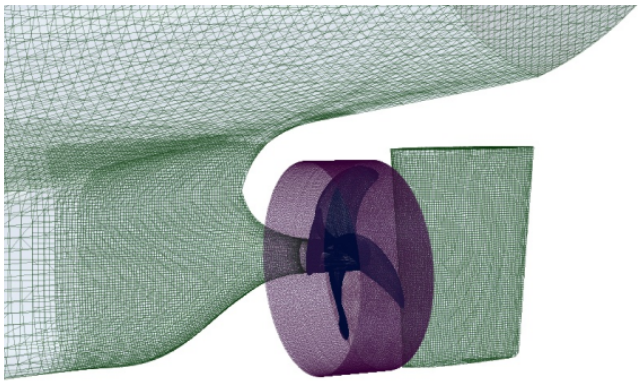

Figure 3. Rotating Propeller Simulation

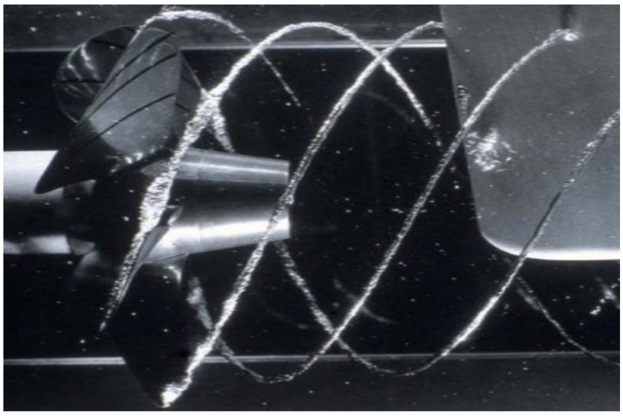

Figure 4. Propeller Cavitation [1]

4.1.3 Propeller Boss Cap Fins Optimisation

As outlined above, there are various methods available to improve the propulsion efficiency of a vessel. With recent development in CFD procedures and HPC capability, designs with improved propulsion efficiency can be achieved by carrying out hull, propeller and retrofit device design optimisation procedures. In particular, one well established retrofit technology is the Propeller Boss Cap Fin (PBCF), which is a post swirl fin that is installed onto the boss cap of the propeller as demonstrated in Figure 7.

Due to the limitation of available tools, previous research was typically conducted by analysing different PBCF design parameters independently and seeking the local optimum by assessing different case studies. However, research was recently carried out seeking to optimise the PBCF design in order to find the global optimum by taking into consideration a number of related parameters [5]. This was made possible by the available capacity of ARCHIE-WeSt, the High Performance Computer at University of Strathclyde, and use of the available software namely Star-CCM+ (the numerical CFD solver) and Friendship-Framework as an optimiser. These advanced numerical tools together with a large-scale computational resource allowed the analysis of 120 different PBCF designs with the optimal fin producing an open water efficiency improvement of 1.3%, which is very significant in this field. Results for the open water efficiency for the different designs can be seen in Figure 5 below. Similar methodology and approaches can be applied to different energy saving devices or case studies to suit different requirements.

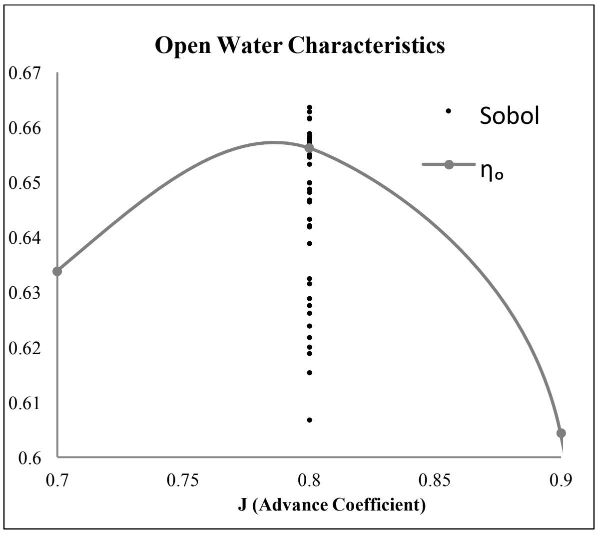

Figure 5. PBCF Optimisation Study

However, optimisation options and procedures are endless, for example, defining one or more constraints or seeking a single or multi-objective approach. Processes might also be computationally expensive and time consuming and therefore careful selection of a robust and efficient system must be taken into consideration. A common preferred approach is to run various designs with a less demanding numerical model, followed by further optimisation on selected designs using more accurate simulations. This demonstrates that optimisation procedures could be further exploited with the development of appropriate tools and resources. With increasing computer capacity, future enhancements could be extended by adding more design variables or, for example, a multi-objective optimisation approach could be used to seek a ship geometry providing maximum energy efficiency and reduction in hub vortex cavitation. Additionally it is conceivable that more detailed analyses could be carried out which might result in different optimal fin geometries altogether (Figure 6).

A particular limitation of these studies is the lack of a suitable cavitation model in the numerical simulation which would require more advanced modelling, numerical approaches and greater computational power. Since cavitation adversely affects propeller characteristics, more effort will be focused on implementing a cavitation model in the open water simulation.

The use of automation in the optimization procedure reduces time and user interaction which can be considered as a benefit, however it also provides less control for the user and requires constant monitoring for quality assurance purposes. Nevertheless, it saves a great deal of time allowing the extensive study of a system or technology within a limited time scale.

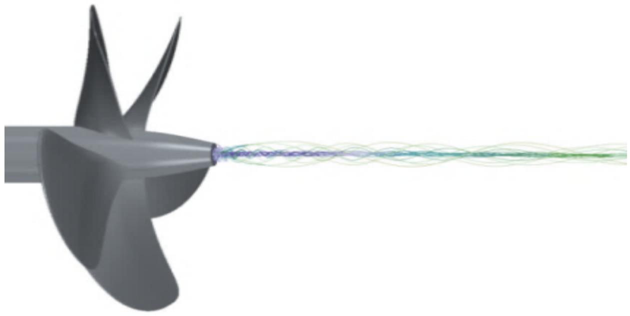

Figure 6. Hub Vortex Cavitation

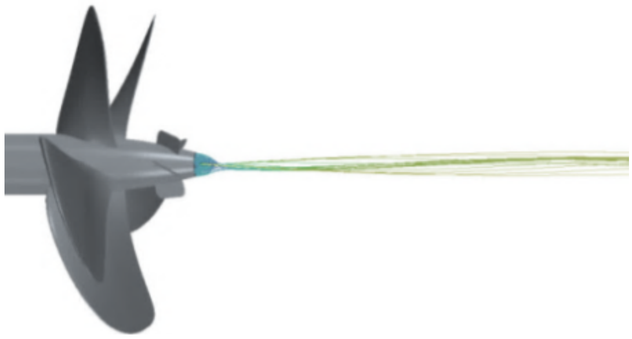

Figure 7. PBCF

5. FUTURE POSSIBILITIES

As has been outlined in the sections above, the advances that have already taken place in computational capability and capacity have enabled much more complex and realistic simulations to be carried out. This has resulted in more accurate predictions in much shorter timescales. Logically this then implies that further increases in computing capability to exascale and even beyond will extend the possibilities for researchers and industry to further improve the simulation of real world problems. Access to much greater capability will allow for the use of larger and more refined meshes within CFD simulations, which in turn leads to more accurate solutions of problems. Increased capacity will also allow for increasingly complex scenarios to be solved, as they will enable more parameters to be calculated directly rather than being assumed or simplified.

A good example of this is the subject of ship propeller cavitation, as discussed in Section 4.1.2 and 4.1.3. At present simulation of the phenomenon is typically only carried out at model scale and with only part of the ship's hull present in order to try to reduce computational complexity wherever possible. However the availability of much higher levels of computational capability would mean that simulations could be carried out in much more realistic conditions with a full scale propeller and full hull present, and operating in a seaway which is representative of an actual sea state. Such simulations would enable researchers, and more crucially industry, to gain a much better understanding of the phenomenon and its implications for real operational conditions in order to take more informed decisions on how to address it.

Further advances could also be directed towards using LES numerical models for cavitation, simulating cavities and eddies. Although it is found to be more accurate for certain engineering applications, LES is not currently very common as it is very computationally expensive.

Although optimization methods are available, their use is not common practice due to the computational capacity required. Hull and propeller optimization using RANS simulations would greatly benefit the industry allowing the analyses of multiple designs at a reasonable expense when compared to experimental procedures. Looking further into the future there is likely to be a venture into optimization methods in real operational conditions and optimization using LES or DNS numerical models.

As explained above, the increase of computational power, both in terms of capability and capacity, introduces endless possibilities to the marine industry and research in general.

However looking at short term improvements, the ability to create mesh configurations with higher cell numbers would increase accuracy, allow more full scale simulations and would make it easier for engineers to satisfy the validation and verification requirements (V&V) of their simulations which in turn would create increased credibility for their work and generate more trust from within industry.

One other significant improvement would be to allow more interaction between the HPC and the user i.e. creating a better graphical user interface allowing for interactive simulations. Some CFD engineers find it best to monitor their work while the job is running ensuring that the simulation is running adequately, and sometimes also making modifications during the run. This is commonly practiced when carrying out minor jobs on a personal computer. However, when running jobs on a HPC, although some visual elements are allowed, this is somewhat limited. Extending and expanding visual capabilities in HPC would definitely help engineers carry out their work more efficiently.

However in the present economic climate, researchers and industry are not at liberty to research and develop whichever techniques and capabilities they choose over an undefined timescale. Nor would access to such computational capability be granted for free. The following sections will discuss the implications of access to such capability from the perspective of research and of industry.

5.1 Technical Implications

Stern et al. outlined three HPC challenges in state of the art CFD simulations; System Memory, Interconnection and Input/output [6]. They explain that 10% of the current system memory (RAM), which is generally 2GB, is dedicated for system usage. In high fidelity complex simulations, a good portion of this is used to store the data to be solved, leaving limited memory for the solver. In some cases, marine applications require the processing of vast amounts of data which do not fit within a single computer node, which may generally be equipped with tens of processor cores. Future advances, such as Exascale computers, may allow transition from a multi–core to many-core systems equipped with many more cores on a single node. Therefore the continuous increase of number of processors within one node will allow more complex simulations with higher volumes of data to be solved within a single shared-memory system. On the other hand, due to the associated costs, a decrease in memory size per core is predicted and therefore it is vital for the CFD engineer to minimise memory usage in CFD simulations and processes.

The interconnection between the nodes is also an issue that requires attention. The network bandwidth and communication latency are the determining factors for network interconnect performance. Therefore while computer engineers are likely to focus on avoiding or reducing latency and increasing network bandwidth, flow simulation engineers should develop solvers that require less communication. In addition, high fidelity flow simulations require processes to read and write vast amounts of data which may result in a great number of nodes or cores to processing in parallel the input/output data. This might make data handling unmanageable and will also be affected by interconnect bandwidth and the performance of the available storage. Consequently, this could deteriorate simulation performance and could be costly.

Such issues require attention not only from the computer engineering perspective but also from the software developers with the former aiming to maximise HPC capability and the latter minimising the need for high data volumes, handling and communication. Together they should aim to improve the system performance and the scalability of the simulations.

5.2 Implications for Research

The direction that research takes is as much dictated by political agendas, funding sources and industry requirements as it is by computational capability. Political agendas tend to dictate “hot topics” which are perceived to be of upmost importance to society at a given juncture. This in turn influences the funding which becomes available for research or facilities which support development in these key areas. These developments can then have an impact in two ways: by producing new techniques or technologies which become available to industry, or through highlighting concerns which impact on industry where they lead to standards and regulations. The process then comes full circle as industry turns to the research community, seeking support in dealing with these new discoveries.

In the particular case of significant advancements in computational capability to Exascale Computing, access to such facilities and the funds to develop applications to take advantage of them would not necessarily be directly linked to the advancements themselves. Instead, it would most likely arise as a means of improving current knowledge and understanding of a key politically-supported topic, or investigating a particular concern raised by industry. This in effect means that just because the capability becomes available, the required developments in CFD and hence the advantages and possibilities outlined within this paper would not necessarily follow directly. For this reason, it is of great importance to all parties that key developments and their possible implications in other fields gain suitable publicity and are appreciated by as wide an audience as possible.

5.3 Implications for Industry

The marine industry has typically been conservative and reactive, meaning that new developments and techniques are not immediately trusted. Even the current state of the art capability in CFD simulation is not widely trusted, and where the results are accepted, they inevitably need to be well supported by costly experimental testing results. Due to the perceived complexity of CFD, where it is not understood, it is not trusted. It is therefore of vital importance that where the capability for sophisticated simulations is available, all possible steps are taken to ensure that the processes are transparent, well validated and can prove to provide accurate and high quality results. Validation and verification (V&V) procedures, e.g. [7] and [8], is a research topic in its own right, with engineers constantly improving the quality of CFD results with continuous developments in tools and computational power leading to new CFD methods.

If this can be achieved and if trust can be developed in the use of these approaches, the benefits for industry would be significant. As has already been outlined, model scale experimental tests are limited in what can be recreated and are also subject to inherent scaling errors. Carrying out such tests is time-consuming and costly, and any design changes would require a new model to be built and a new set of experiments to be run. By contrast, future CFD simulations could be run at full scale, removing any inherent scaling errors. As computational capacity increases, the timescales and costs involved will continue to reduce. Therefore, design changes could be made and analysed more easily, and simulations of realistic scenarios could be conducted. This latter advantage would have a significant benefit in terms of ship safety and, in what is probably more appealing to industry, efficiency. Improved efficiency means a lower fuel bill and the ability to predict the efficiency of a vessel in a range of realistic operational conditions rather than at just one design condition. This will lead to more informed decisions on design and installation, producing vessels that are much better adapted to their typical operating profile than at present.

The challenges which lie ahead for the industry in arriving at this point are significant, with many aspects to consider, but if it can be achieved, the benefits will be universal in the field.

6. CLOSING REMARKS

Numerical solvers have developed significantly over the past few years. This has allowed marine, and more specifically ship hydrodynamics, research to progress in parallel. The continuous improvement of HPC capabilities and the move towards Exascale Computing will allow research methods to be exploited even further, opening up new possibilities previously not possible due to the constraints of the available computing capabilities. With this in mind, numerical solvers should be further developed taking into consideration the power and capacity of Exascale Computing platforms that will open up new windows of opportunity. In time this will result in more robust, accurate, scaleable, high fidelity, state of the art simulations, even when employing optimization procedures.

7. ACKNOWLEDGMENTS

Results were obtained using the EPSRC funded ARCHIE-WeSt High Performance Computer (www.archie-west.ac.uk). EPSRC grant no. EP/K000586/1.

8. REFERENCES

[1] F. Stern, Z. Wang, J. Yang, M. M. H Sadat-Hosseini, S. Bushan, M. Diaz, S. H. Yoon, P. C. Wu, S. Mo, R. S. Thodal and J. L. Grenestedt, "Recent Progress in CFD for Naval Architecture and Ocean Engineering," in *11th International Conference on Hydrodynamics* , 2014.

[2] Y. K. Demirel, M. Khorasanchi, O. Turan, A. Incecik and M. P. Schultz, "A CFD model for the frictional resistance prediction of antifouling coatings," *Ocean Engineering Vol. 89,* pp. 21-31, 2014.

[3] Y. K. Demirel, M. Khorasanchi, O. Turan and A. Incecik, "Prediction of the effect of hull fouling on ship resistance using CFD," in *17th International Congress on Marine Corrosion and Fouling (ICMCF)*, Singapore, 2014.

[4] P. Kellett, O. Turan and A. Incecik, "A Study of Numerical Ship Underwater Noise Prediction," *Ocean Engineering Vol.66,* pp. 113-120, Jul. 2013.

[5] K. Mizzi, Y. K. Demirel, C. Banks and O. Turan, "PBCF design optimisation and propulsion efficiency impact," in *RINA International Conference on Influence of EEDI on Ship Design*, London, 2014.

[6] F. Stern, J. Yang, Z. Wang, H. Sadat-Hosseni, M. Mousaviraad, S. Bhushan and T. Xing, "Computational Ship Hydrodynamics: Nowadays and Way Forward," in *29th Symposium on Naval Hydrodynamics*, Gothenburg, Sweden, 2012.

[7] I. B. Celik, U. Ghia, P. J. Roache, C. J. Freitas, H. Coleman and P. E. Raad, "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications," *Journal of Fluids Engineering,* vol. 130, no. 7, 2008.

[8] F. Stern, R. Wilson and J. Shao, "Quantitative V&V of CFD simulations and certification of CFD codes," *International Journal for Numerical Methods in Fluids,* vol. 50, pp. 1335-1355, 2006.