Procedia Computer Science
Volume 66, 2015, Pages 132–139
YSC 2015. 4th International Young Scientists Conference on Computational Science

Virtual Basin for Simulating Ship Sailing Qualities on HPC Resources

Andrey Aksenov[1], Andrey Pechenyuk[2], Alexey Poyda[3], Eygene Ryabinkin[3], Igor Tkachenko[3], Viacheslav Ilyin[3] and Vasily Velikhov[3]
[1]*JSC "Tesis", Moscow, Russia*
[2]*LLC "Digital Marine Technology", Odessa, Ukraine*
[3]*National Research Centre "Kurchatov Institute", Moscow, Russia*
andrey@tesis.com.ru, rea@grid.kiae.ru, ilyin0048@gmail.com

Abstract
A Virtual Basin project and some features of its implementation for High Performance Computing (HPC) are presented in this paper. There are many attempts to create virtual basin approach for ship hydrodynamic simulations over the world, and the current project is well aligned with this activity. The main features of the Virtual Basin described in this paper in contrast with the analogues are: firstly, developing a simple tool for ship designers without deep expertise in the numerical methods, and, secondly, implementing a simple access to remote HPC resources. The problems of the Virtual Basin implementation on HPC resources are shown in the example project of towing test simulation of the well-known test KCS from Gotherberg-2000 workshop.

Keywords: HPC, CFD, grid meshing, FlowVision, towing test simulation, virtual basin

1 Introduction

Numerical methods for solving hydrodynamic problems started to be extensively used fairly in ship design for developing hull form and studying ship hydrodynamics only recently. The usual practice was to perform real-world model tests using scale models of the ships and then to apply Froude's method (Froude, 1955) to rescale results to obtain the required characteristics. This is the long and costly process which can't always provide reliable results: instrumental and systematic errors in measurements of model properties as well as scale effects in the basin obstruct one to properly capture specifics and obtain needed characteristics of the full-scale ship.

Modelling of the full-scale hull flow that employs numerical simulations and successive extraction of hydrodynamic parameters (including towing resistance) is free of these problems. It allows performing calculations for the full-scale values of Reynolds number and for the designed range of ship velocities (Froude number). Being supplemented with calculation of ship dynamics as the moving

Selection and peer-review under responsibility of the Scientific Programme Committee of YSC 2015

doi:10.1016/j.procs.2015.11.016

body that is driven by various forces (including hydrodynamic ones), such a method provides engineers with information of relative changes of ship position with respect to the water surface at different relative speed.

Aside from the development of the numerical methods in ship hydrodynamics, the independent task is their integration to the design practices. Numerical methods we always considered as the good tool, because they provide an opportunity to avoid long and costly model tests, but yet deliver all necessary information to the design office. At the same time, these relatively new methods require specific approaches to the design workflow. Each numerical simulation requires serious run-up that includes preparation of the geometric data, grid meshing, arrangement of the boundary conditions, etc. The majority of these activities are not directly connected with the crux of the simulation activities and often demand very specific skills. So designers, who know what result they want to get, have difficulties in performing required calculations. On the other hand, people proficient in numerical methods possess the required knowledge, but mostly far from the heart of the matter.

Virtual Basin is the technology for managing numerical simulations on remote HPC resources via Web interface available on the most browsers and with main features as:

- ease and flexibility of Web-based access;
- simplification of pre-, post-processing and control procedures.

The main idea is to make activities of ship designer in user-friendly manner. Ideally, any naval architect should be able to use Virtual Basin tools by himself without external assistance of CFD specialists.

The above-mentioned development originates from sufficiently large experience in numerical simulations: the main trends in simulation parameters (e.g. computational mesh) were analyzed and expressed as pre-defined templates.

Well known KCS benchmark from the Gotherberg-2000 workshop (Larsson, Stern, & Bertram, 2003) was used for verification of the Virtual Basin software.

2 Related works

First public discussion of results obtained via numerical simulation of ship towing tests and their comparison with the experimental data happened in 1980 at a seminar that took place in Gothenburg (Larsson, Stern, & Visonneau, 2011). The main goal of this seminar was to evaluate the current state of the art and further directions in the field of numerical methods in ship hydrodynamics. Most attendees presented results obtained with the help of different species of the panel methods.

10 years later, the second seminar happened where most groups presented simulation results based on the numerical solutions of motion equations for viscous liquid (Navier-Stokes equations with Reynolds averaging, RANS). Main conclusion is that numerical simulation of Navier-Stokes equations gives high reliability of the description for liquid flow in the vicinity of the stern.

Next seminar was in Tokyo in 1994 where new numerical algorithms (along with their applications to the engineering problems) were demonstrated. The physical effects of free surface of a liquid were included in simulations.

Significant development happened at the fourth seminar in 2000 (Larsson, Stern, & Bertram, 2003): there was agreed to use three different ship hull forms as the benchmark objects, supplemented with the detailed 3D models and experimental data for formal checking results. These benchmarks have been started to be publicly accessible, what is now the standard for verification of numerical methods in the ship hydrodynamics.

Table 1 presents a summary of results given by other authors presented at the last (2010) seminar at Gothenburg (Larsson, Stern, & Visonneau, 2011).

Table 1: Summary from last seminar at Gothenburg **(Larsson, Stern, & Visonneau, 2011)**

Authors	Software	Turbulence	Surface	Degree	Mesh
CD-Adapco	STAR-CCM+	*k-ε*	VOF	2	U
CEHINAV-TU	STAR-CCM+	*k-ω, SST*	VOF	1	MU
Chalmers	SHIPFLOW 4.3	*k-ω*, SST, EASM	-	2	S
CSSRC	FLUENT 6.3	*k-ω*, SST, RNG *k-ε*	VOF	2	MS
CTO	STAR-CCM+	*k-ε*	VOF	2	U
ECN/CNRS	ISICFD	*k-ω*, EASM	VOF	2	U
ECN/HOE	ICARE	Wilcox *k-ω*, SST	N-L T	2	S
FLOWTECH	SHIPFLOW-VOF	*k-ω, SST*	VOF	2	MS
FOI	OF	LES	VOF	2	U
FORCE	CFDShip-Iowa	*k-ω*, SST	LF()	2	OS
Uni GL&UDE	Comet OpenFOAM	*k-ε*	VOF	Mixed	U
HSVA	FreCo+	2E, *k-ε*	VOF	3	U
IHI/UniTokyo	WISDAM-UTokyo	B-L, DSGS	DF()	3	OS
IIHR	CFDShip-Iowa	DES + *k-ε/ω*, ARS	LF()	2-4	S, OS
IIHR-SJTU	FLUENT 12.0.16	*k-ε*	VOF	3	MS
IST	PARNASSOS	*k-ω*, SST	-	2	S
UniKyushu	RIAM-CMEN	DNS	THINC	3	S

Used abbreviations in the Table 1:

- N-L T: non-linear tracking,
- LF (): level function,
- DF (): density function,
- B-L: Baldwin-Lomax,
- U: unstructured,
- MU: multiblock-unstructured,
- S: structured,
- MS: multiblock-structured,
- OS: overlapping-structured.

Currently numerical algorithms employ in most cases either *k-ε* or *k-ω* two-parametric models for turbulence (Hirt & Nichols, 1981). Some research groups still use one-parametric and anisotropic models. Methods like LES/DES (Germano, Piomelli, Moin, & Cabot, 1991) are rarely used nowadays. De-facto the standard for free surface modelling is "Volume of liquid" (VOF) scheme (Lawrence Livermore National Laboratory, 2008). Finite-difference or finite volume methods with schemes of 2^{nd}, 3^{rd} and, sometimes, 4^{th}degree is used for solving the Navier-Stokes equations.

The Virtual Basin is based on FlowVision software (Aksenov A. , New release of CFD software FlowVision., 2014). Although FlowVision was not presented at Gothenburg-2010 workshop yet, it includes all the necessary capabilities and technologies for the wide range of applications in the maritime design practice. Not only representative for the towing test simulations, but also a self-propelled simulation of the ship movement, both with actuator disk and the more realistic one, with the propeller model presence can be performed.

The main feature of FlowVision versus other codes is the automatic generation of computational grid. Up to 90 percent of the user’s working time is usually occupied by generation of the computational grid while preparing FV or FE simulations. In FlowVision this problem was resolved by using the sub-grid geometry resolution method for generation of the Cartesian adaptive locally refined grid (CALRG). The essence of this method (Aksenov, Dyadkin, & Pokhilko, Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method., 1998) is a Boolean subtraction of the volume, which is determined by closed surface and borders of the computational domain, from the Cartesian computational grid. In fact the cells of the computational grid, when are crossed by the freeform surface of the computational domain, are converted into complex polyhedrons with an approximation of the solved equations inside them by high-order schemes. In areas near the boundaries with high gradients of the flow parameters, an additional grid resolution is carried out by dynamic adaptation of the computational grid (given contiguous cells are divided into smaller ones). As a result, a user forms general configuration of the initial Cartesian grid only, while the computational cells with complex geometry near boundary conditions are meshed automatically. This unique approach to grid generation provides a natural link with CAD geometry and FV mesh.

In the Virtual Basin the initial computational mesh and conditions for its adaptations are based on the experience in simulating full-scale towing tests of the ships. A methodology of towing test simulation is developed on a base of this experience and this methodology is programmed in the Virtual Basin. First mention about this technology was submitted in paper (Aksenov A. , New release of CFD software FlowVision., 2014).

In modern practice, when the grid reaches the size more the 10^6 cells, simulations with a free surface can be carried out efficiently only by the use of HPC resources. The HPC resources of NRC "Kurchatov Institute" (NRC KI, 2011) were used in this study.

3 Specifics of HPC environments

High-performance computing (HPC) is still challenging both for owners of these resources and for users, especially for engineers. Serious limitations and specifics should be taken into account when one plans complicated CFD calculations on HPC resources.

In this scope for the Virtual Basin project are assumed the following:

- User is a naval engineer thus he needs complicated CFD simulations but he has no serious experience in HPC computing. Thus, there are no requirements to his knowing the particulars of the underlying HPC environment, how to install and configure the software, how to organize the calculation process on the computational field available for him at the supercomputer;
- One of engineer's needs is the visualization of the data obtained during the HPC calculation process, as well visualization of final results. Thus, two processes, calculation and visualization, should be mate to provide user with visual information to understand correctness and fidelity of the simulation process in interactive, or quasi-interactive, mode.

The Web interface technique is chosen to provide engineers with proper solution, in particular to resolve the aspects formulated above.

Virtual Basin takes into account FlowVision license conditions. For Virtual Basin users the FlowVision licensing is hide, in future when Virtual Basin will be used like commercial tool, price of license will be part of payment for using supercomputer resources. But, from other side, user could use now his own FlowVision license for running FlowVision on the supercomputer.

It is worth also to give following restrictions of general character, related to HPC computing specifics, which are in scope of our discussion:

- Working nodes in HPC resources are carried by poor subset of services now being conventional for users. They can be diskless, for example. Also, as a rule, they can have no usual (TCP/IP-based) network access, or this access is seriously restricted;
- HPC systems use specialized high-performance file system, like Lustre one. So, operations with files, conventional for users, are not supported or have only limited options. Thus, user should use special programming techniques and, furthermore,

algorithms used should be arranged accordingly;
- Two (or even more) jobs can be landing to one physical machine (node), these jobs could be run by different users. Thus, user should either have Quality of Service agreements or be ready to deal with node set of non-uniform by performance abilities;
- One could not find, as a rule, two supercomputers with identical (or, even, close) calculation environments. As a result, user should either be ready to work with restricted subset of supercomputing features available at the supercomputer used and/or to work only via high-level interfaces.

All these requirements and limitations were taken into account during developing the Virtual Basin.

4 Simulating turbulent flow around ship hull

FlowVision uses finite volume approach and can deal with both uncompressible and compressible flows of liquids and gases. FlowVision provides fully automatic meshing of a computational grid, and this is very important for main goal of the Virtual Basin – to make a simple instrument for ship designer. The sub-grid geometry resolution method (Aksenov, Dyadkin, & Pokhilko, Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method., 1998) is used for grid meshing. To resolve all peculiarities of the flow, including waves on the free surface and boundary layer a local dynamic adaptation of the grid was performed. This method is based on building Cartesian mesh far from the ship hull and arbitrary polyhedron cells near the ship hull. Figure 1 gives an example of such mesh for KSC ship while Figure 2 depicts the computational domain for the simulation.

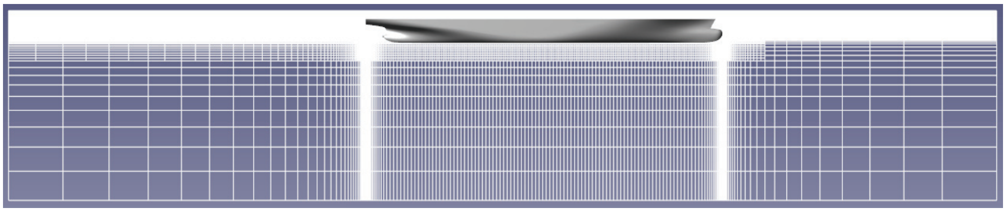

Figure 1: Example of computational mesh

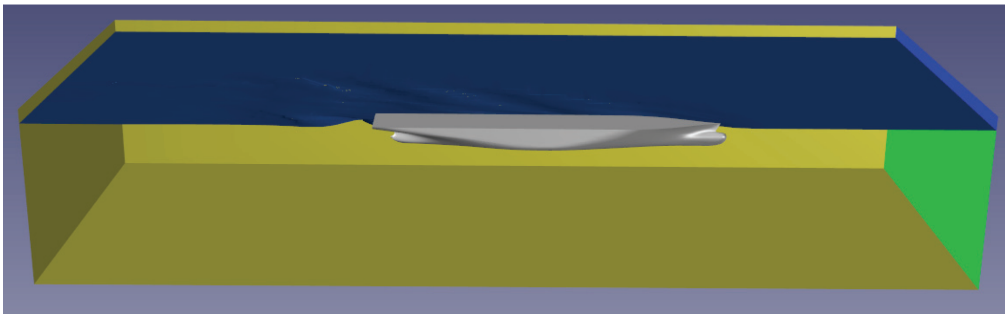

Figure 2: Full modelling volume

When towing speed is constant, it's reasonable to simulate the inverted flow around a hull (like in water tunnel instead of considering water tank conditions). In this case the computational domain is shorter, and the hull is nearly static in respect to the grid. Taking this into account, a box-shaped computational domain was defined, and the corresponding boundary conditions have been set. The transverse sizes of the computational domain were chosen larger than the once recommended for the towing tests, in order to avoid the tunnel wall effects.

FlowVision includes five turbulence models, including ones with small Reynolds numbers: 3 types of k-ε models, SST, Spalart-Almares. During Virtual Basin project a special version of k-e is developed to take into account laminar-turbulent transition on ship hull to get more accurate results for ship drag force (Aksenov, Zhluktov, & Platov, 2013). The standard k-e model was used in KCS simulation.

The turbulence implementation requires rather fine meshing near hull surface that results in large computational 3D-mesh, with number of nodes ranging from million to tenths of millions. To perform such simulation in practice wall time this requires HPC resources with keeping high computational efficiency.

5 The details of HPC infrastructure implementation details

Following the principles outlined in Section 3 we had made a number of steps to adopt the Virtual Basin program components to the HPC environment. For best practices we used the supercomputing resources at NRC KI " (NRC KI, 2011). For use-case discussed the cluster, named as HPC2 in this center, used with the following parameters:

- computing field: 1280 nodes, dual-CPU (Intel Xeon E5450), 2 GB RAM per core, InfiniBand DDR;
- parallel filesystem: Lustre 2.3/2.5, FibreChannel SAN HP P2000 G2;
- infrastructure & services: SLURM, CVMFS, NFS for home file system.

First of all, we had employed the CVMFS (Blomer, Aguado-Snchez, Buncic, & Harutyunyan, 2011) tools developed for delivering High Energy Physics specialized packages. However we deployed Stratum-1 server (primary source of CVMFS data and metadata) to migrate locally-installed software from centralized Lustre file system, used in the NRC KI cluster, to the read-only space. By this users are able to spawn FlowVision software without the need to install or configure it.

Then, we had made a number of changes in the internal HPC infrastructure used to allow nodes from computing field to communicate directly with the user interface node where the FlowVision solver agents and proxy server are placed. The network translation policies were tuned to allow connections with outside networks directly from the working nodes where the FlowVision license manager is installed. This covers the use-cases of externally collocated license server and software own by the research groups.

A Web-based environment for doing in a mate mode the calculations and visualization of the simulation process was created. It is based on the paradigm of task-oriented interface, where user "talks" with the HPC cluster via browser-based GUI with the "natural" task-specific control and options. An additional FlowVision module FvViewer is used for rendering the results. It transforms binary results of simulation to the graphical form that can subsequently be viewed at the Web page. The showing example of this visualization is presented in Figure 3.

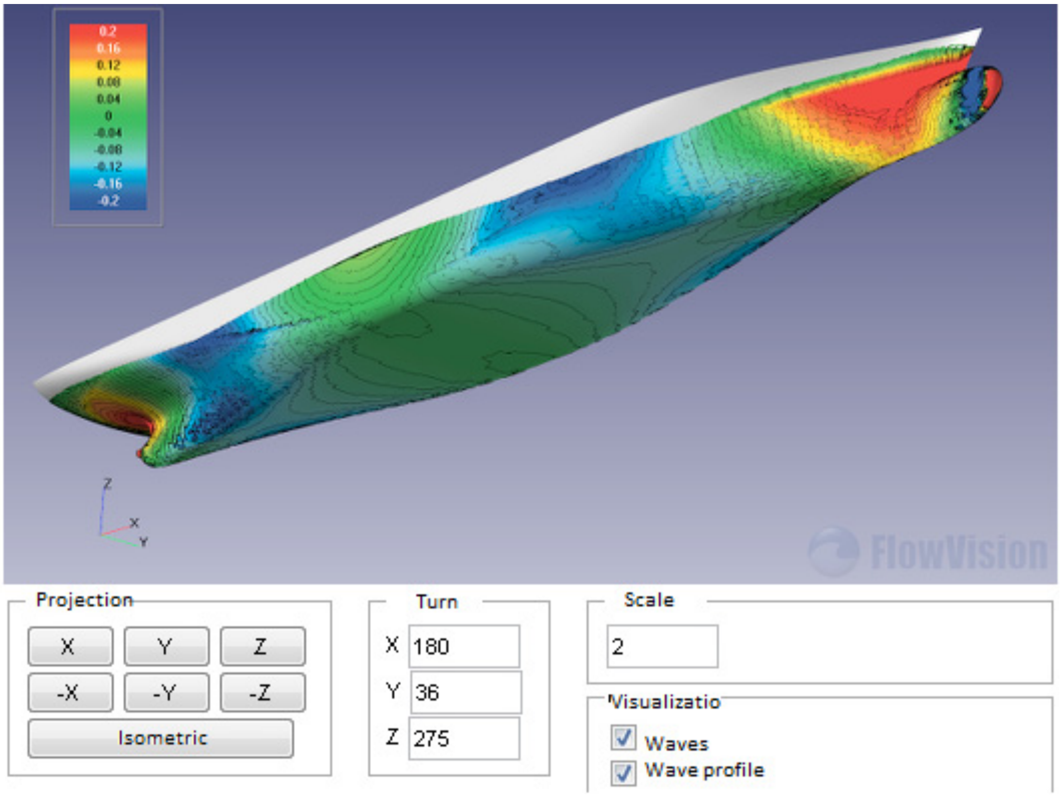

Figure 3: Visualized result via FvViewer

GPU-powered visualization nodes in the NRC KI supercomputer cluster used to share file system that carries the output from the simulations. For better organisation of the user's workflow we plan extending SLURM (resource manager and job scheduler at HPC2 cluster are used) (Lawrence Livermore National Laboratory, 2008) installation with the ability of running complex job that will allow users to spawn the visualization tasks just after completing of the calculations. The goal of this development is extending the current functionality of FvViewer onto the GPU-based clusters for interactive graphical modelling and possibility of visualization of simulation results on various types of screens, including large compound video-walls.

Adaptation of the GPU-based parts of the HPC infrastructure for usage of virtual machines was also started for better matching the current generation of the FlowVision visualization tools.

6 Conclusions and future work

This paper describes some main features of the Virtual Basin project. Simple and intuitive Web interface allows solving ship hydrodynamics problems by users, which are not experienced in the CFD numerical methods. Fully automatic grid meshing by FlowVision software, allows simulating flows around ship hulls with complex geometry and obtaining various flow characteristics. User doesn't take into account meshing the computational grid, but has possibility to control the mesh parameters. Available advanced turbulence models with laminar-turbulent transition provide correct approach to viscous flow simulation. End-user is able to manage remote HPC resources and work via conventional problem-oriented Web interface rather than in the bare-iron HPC computing environment. The use-case described in Section 5 outlined, in particular, the changes made in the HPC infrastructure used that allow using the supercomputers in a variety of ways that will provide wider spectrum of functionality to ship designers.

Acknowledgments

Numerical simulations were performed at supercomputing resources in NRC "Kurchatov Institute", which are supported as the centre for collective usage (project RFMEFI62114X0006, funded by Ministry of Science and Education of Russia).

References

Aksenov, A. A., Zhluktov, S. V., & Platov, S. A. (2013). Numerical modelling laminar-turbulent transition on the ship hull in the FlowVision package. *Sudostroenie, Vol. 4* .

Aksenov, A. (2014). New release of CFD software FlowVision. *CAD & graphics*, (pp. 27-29).

Aksenov, A., Dyadkin, A., & Pokhilko, V. (1998). Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method. *Proc. 1998 ASME Pressure Vessels and Piping Division Conference, Vol. 377-2* (pp. 79-86). San Diego, USA: ASME PVP.

Blomer, J., Aguado-Snchez, C., Buncic, P., & Harutyunyan, A. (2011). Distributing lhc application software and conditions databases using the cernvm file system. *Journal of Physics: Conference Series, 331(4)* .

Carlisle, D. (2010, April). *graphicx: Enhanced support for graphics.* Retrieved from http://www.ctan.org/tex-archive/ help/Catalogue/entries/graphicx.html

Froude, W. (1955). Observations and suggestions on the subject of determining by experiment the resistance of ships. In W. Froude, *The Papers of William Froude, 1810-1879.* London, UK: Royal Institution of Naval Architects.

Germano, M., Piomelli, U., Moin, P., & Cabot, W. H. (1991). A dynamic sub-grid scale eddy viscosity model. *Physics of Fluids, A(3)* , pp. 1760-1765.

Hirt, C. W., & Nichols, B. D. (1981). Volume of liquid (vof) method for the dynamics of free boundaries. *Journal of Computational Physics, 39* , pp. 201-225.

Larsson, L., Stern, F., & Bertram, V. (2003). Benchmarking of computational dynamics for ship flows: the Gothenburg 2000 workshop. *Journal of Ship Research, 47(1)* , pp. 63-81.

Larsson, L., Stern, F., & Visonneau, M. (2011). CFD in ship hydrodynamics – results of the Gothenburg 2010 workshop. *Proc. of the IV International Conference on Computational Methods in Marine Engineering* (pp. 17-36). Barcelona, Spain: CIMNE.

Lawrence Livermore National Laboratory, N. (2008, 07). *SLURM: A highly scalable resource manager. Project site.* Retrieved from SLURM: A highly scalable resource manager. Project site.: https://computing.llnl.gov/linux/slurm/

NRC KI. (2011). *NRC KI computing centre site*. Retrieved from NRC KI computing centre site: https://computing.kiae.ru/

TESIS. (2015, 09 19). *FlowVision. Project site.* Retrieved from FlowVision. Project site.: https://flowvision.ru/

Versteeg, H., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd Edition).* Pearson Education Limited.