Development of an Automatic Analysis Methodology by Integration of Digital Hull Design, Model, Processing, and Evaluation

Xiuqing Xing1,4, Chi Wan Lim1, Joo Hock Ang2,3, Chang Wei Kang1, Peng Cheng1, Jing Lou1

1 Institute of High Performance Computing, Agency for Science, Technology, and Research A*STAR, Singapore, 138632
2 Sembcorp Marine Singapore, 80 Tuas South Boulevard, Singapore 637051
3 University of Glasgow, Singapore
4 Contact Author: xing_xiuqing@ihpc.a-star.edu.sg

Abstract: Starting with the offset data, a methodology is developed to integrate the hull design, CAD model generation, CFD modelling, and hull performance evaluation in this study. The methodology enables an automatic generation of the hull geometry model and direct application in CFD modelling and simulation without manual intervention on the CAD model. The procedure consists of extracting the hull form data in offset table format, reconstructing the offset data to generate the geometry model, defining the simulation domain, discretizing the domain with mesh for CFD simulation, and evaluating the hull performance by the numerical simulation results. A case study is carried out with a container ship hull as an example to demonstrate the automatic analysis procedure, which allows a fully automatic hull form design and makes automated optimization feasible.

1. Introduction
As the marine industry moves towards industry 4.0, ships are getting smarter with autonomous navigation. Digitalization is the first step for autonomous operations and makes decisions based on performance optimization digitally possible. For example, optimum fuel consumption can be achieved by considering the effects of the external factors such as wind, current and other environmental conditions for any specific voyage. Potential applications of digital ship twins can cover the entire life cycle of a ship from design, performance prediction, operation, as well as maintenance.

Currently, the digital hull geometry model is built by using various design methods/software. Most hull geometry models generated from Computer-Aided Design (CAD) are compatible with the pre-processing of Computational Fluid Dynamics (CFD) solvers for hull performance analysis. However, in most of the cases, extensive geometry cleaning and simplification of the Computer Aided Design (CAD) model are required to form a ready-to-use model for CFD pre-processing/meshing because of the following reasons,

- CFD simulation requires a water tight CAD geometry to extract a domain for numerical analysis. Exporting CAD models in various format into a format accepted by CFD solvers causes losses of data and therefore losses of geometry details, leading to so-called ‘dirty’ CAD geometries. Dirty geometries consist of gaps, overlaps, stray points, non-intersecting lines, cross-overs, slivers, and other incompatibilities in the model, preventing the model from being...
valid. Clean-up efforts and compensations for the data losses raised during the data conversion from CAD-kernel to FE-kernel are unavoidable to ensure a watertight geometry model.

- It is needed for simplifying a complex CAD geometry into an appropriate granularity for the CFD simulation model by deleting the spurious details from the view of numerical simulation point, for the ease of meshing and assurance of an efficient simulation without affecting the flow physics.

For a typical case, it takes days or even weeks to produce a quality triangulated surface mesh, along with the required high-quality CFD meshes from a complex and dirty CAD model. It sets an obstacle for the automatic analysis and the automated hull design optimization. This motivates many commercial software developers to work for faster routes and methods from CAD to flow simulation in order to free CFD engineers to concentrate on engineering analysis. For example, a tool called "Surface Wrapper" [1] is included in STAR-CCM+ [2]. It wraps a user's CAD geometry by filling any holes and fixing overlaps or cracks. Still, it requires manual intervention before the geometry can be used for CFD modeling and makes the automated design optimization unachievable.

In this study, a methodology is developed by integrating the hull design, CAD model generation, CFD pre-processing, and hull performance evaluation. It enables automatic generation of the hull geometry and direct application in the CFD model without manual intervention operation on the CAD model. The procedure consists of extracting the hull form data in offset table format, reconstructing the offset data to generate the geometry model, defining the simulation domain and discretizing the domain with meshes for CFD simulation, and evaluating the hull performance by the numerical simulation results. By doing so, an automatic analysis can be achieved for performance evaluation. This allows a fully automatic hull form design and optimization. In addition, it also reduces the dependence on the experiences of the designers.

The paper is organized as follows. Section 2 describes the integrated method and the procedure in detail. A case study is carried out with a container ship as an example in section 3. Applications of the automatic analysis method are demonstrated step by step through the case study. Summary of the test results and the future work are presented in section 4 and section 5, respectively.

2. Integrated method

An integrated method has been developed in order to achieve an automatic procedure from CAD model to CFD modelling without manual intervention operations. The procedure consists of extracting the hull form data in offset table format, reconstructing the offset data to generate the geometry model, defining the simulation domain and discretizing the domain with meshes for CFD simulation, and evaluating the hull performance by the numerical simulation results. The schematic plot of the proposed integrated method is presented in Figure 1.

![Figure 1. Schematic plot of the integrated procedure for hull design, model, processing and evaluation](image-url)

In the method developed in this study, the hull design data from various designs are required to be expressed in an industry-standard offset-table format. The higher the density of the output points, the more accurate the reconstructed surface is. The offset data contains the information to reconstruct one identical half of the ship hull surface. It is presented in a table with a format of \((n \times m)\) containing the points \((y\text{-axis})\) located on the surface of the ship with respect to fixed positions along the \(x\)-axis (stern to bow), \(x_1\) to \(x_n\), and the \(z\)-axis (deck to keel), \(z_1\) to \(z_m\). With the data presented in neat table format as \(P(x_i, z_j)\), one half of the ship hull surface can be easily reconstructed in a quad-like manner \((x_i, z_j - x_{i+1}, z_j - x_i, z_{j+1})\).
\( x_{i+1} z_{j+1} - x_i z_{j+1} \), and thereafter split into two triangles. However, the same efficiency provided by the table format with fixed \( x \) & \( z \)-axis position also produces a staggered effect along the profile of the ship. This is due to the fact that between two points where \( P(xz_j) > 0 \) and \( P(x_{i+1}z_j) = 0 \), it is often unclear at which exact location between \( x_i \) and \( x_{i+1} \) does the ship hull surface actually hit zero point.

To infer the zero-point position for both profile at the stern and bow at each \( z \)-axis interval, a number of points along the \( z \)-axis prior (stern profile) and after (bow profile) are used for interpolation. Using a quadratic second-order function, both zero-points at the stern and bow can be found. Since there is no correlation between the zero-points at different \( z \)-axis locations, an additional smoothing operation is required. A univariate spine curve is interpolated through the set of zero-points at both stern and bow. The resulting spine curve is then used as the profile line to merge both halves of the ship hull together into a watertight geometry model.

With the watertight geometry model, the simulation domain is defined and extracted to simulate the fluid flow without introducing any artificial effects by the domain boundaries. In order to define a feasible domain size, a prior understanding of the fluid dynamics in the flow field is necessary. CFD best practices for different hull shape, hull size, forward speed, as well as environmental conditions have been established through our studies.

Once the simulation domain is determined, SnappyHex method, supplied with OpenFOAM, is then adopted to generate the three-dimensional mesh for CFD evaluations. The snappyHex mesh utility in OpenFOAM generates three-dimensional meshes containing hexahedra (hex) and split-hexahedra (split-hex) automatically from triangulated surface geometries in STL format. The mesh approximately conforms to the surface by iteratively refining a starting mesh and morphing the resulting split-hex mesh to the surface. The mesh options include inserting cell layers for local refinement. The specification of mesh refinement level is very flexible, and the surface handling is robust with a pre-specified final mesh quality. Best practices for mesh size, refinement layers, as well as refinement zones, such as boundary layers, kelvin wave zone, wake zone, bow zone, stern zone, free surface zone, etc. have been established in our system. With the specified mesh settings, the complex hull geometry can be automatically conformed and local refinements can be carried out for capturing the ship-induced waves and assess the resistance on the hull accurately.

The steps described above explain the automatic procedure. A graphic user interface (GUI) will be developed for this procedure in the next phase of the study to establish the platform for hull design optimization.

3. **Case Studies**
A container ship hull design, as an example, is used to test the feasibility of the proposed automatic analysis procedure.

3.1 **Offset data**
A set of offset data is extracted from the hull design. There are 1977 columns data along \( x \)-axis from the stern to the bow, while 193 rows from the keel to the deck along \( z \)-axis. Both rows and columns have non-regular fixed intervals. A 3D view of the data from the offset table is shown in Figure 2.

![Figure 2. Offset table data rendered using points in 3D.](image-url)
3.2 Geometry construction
The zero-point profile line is obtained, first by using the set of offset table with the same z-axis values to get an initial set of zero-points, and then using a univariate spine curve to interpolate through the set, see Figure 3. With the profile line, the ship hull can be easily meshed into a quad grid using the natural row-column format of the offset table. The mesh is then duplicated for the other identical half of the ship and merged together as shown in Figure 4.

![Interpolated Profile Line for Ship Bow](image1.png) ![Interpolated Profile Line for Ship Stern](image2.png)

**Figure 3.** Interpolated profile lines for both bow and stern

![Reconstructed hull geometry](image3.png)

**Figure 4.** Reconstructed hull geometry

![Domain for CFD simulation](image4.png)

**Figure 5.** Domain for CFD simulation

3.3 CFD pre-processing
The CFD simulation domain is defined as 5Lpp in length, 2Lpp in width, and 2Lpp in depth according to our best practices for hull resistance simulation, shown in Figure 5.

Various size of grids has been tested for reducing computational cost with sufficient simulation accuracy. The final grid consists of 2.5 million mesh cells generated according to the best practices delivering a grid-independent result, as shown in Figure 6, with refinement details on the free surface. The mesh size in the boundary layers is selected to satisfy the criterion of $y^+ \leq 50$.

K-ω SST turbulence model is chosen for capturing the turbulence flow. The numerical model is developed with OpenFOAM. Volume of Fluid (VOF) method is adopted to trace the free surface interface between the water and air. With the pressure and shear stress on the hull surface, resistance on the hull surface can be obtained through a surface integral.
3.4 CFD simulation

It is assumed that the hull is in even keel condition and therefore, no dynamic sinkage and trim are considered. The hull resistance simulation is performed by solving the steady Reynolds Averaged Navier-Stokes equations (RANS).

3.5 Evaluation results

Hull resistance under a forward speed of 20 knots, a draft of 9 m condition is simulated. The time history of the resistance on the hull with the mesh shown in Figure 6 is plotted in Figure 7a. The simulated hull resistance is normalized with the experimentally measured resistance value. It converges after about 4200 iterations to a scaled value of 1.068, indicating that the hull resistance obtained from CFD simulation matches the experimental data with a discrepancy of +6.8%. The ship waves, including transverse and divergent waves captured by the simulation are shown in Figure 7b.

During the development of the best practices for mesh size selection, finer meshes and coarser meshes are tested. Bigger or smaller mesh size along X direction and Z direction are tested as wave capture requires enough mesh points along wave height (along Z direction) and wave length (along X direction) directions. The definition of X, Y, and Z directions is shown in Fig.6a. In addition, the mesh size along X and Y direction affects ship wave reflection, while, the mesh size along Z direction has effects on capturing the pressure recover at the stern region, and hence affects the final hull resistance value.

As examples, simulation results from one coarser mesh with a mesh size of 1.2 million (coarser mesh along X direction compared with the baseline case) and one finer mesh with 4.8 million mesh cells (finer mesh along X and Z direction compared with the baseline case) are presented to display the mesh size effect on the hull resistance simulation, as shown in Figure 8 and Figure 9 respectively. It can be seen...
that the difference between the simulated hull resistances is small, although the convergence speeds are different. It takes a longer time (about 8000 iterations) to converge to a steady resistance value of 1.054 with the finer mesh. Whereas, the simulation using a mesh with 1.2 million cells converges after about 3000 iterations, and overestimates the hull resistance 12.1% compared with the experimental data. Comparisons of the simulated hull resistances and the convergence speeds with the three mesh sizes are listed in Table 1. It also can be seen that the hull resistance obtained with a mesh of 2.5 million cells has a satisfied error with reasonable computational costs. The ship waves induced by the hull are captured well with 2.5 million cells and 4.9 million cells. The wave length of the Kelvin is captured well by the coarsest mesh, although the wave height is damped by the coarsest mesh, as shown in Figure 8b. Based on the comparisons, the setup of the mesh with 2.5 million cells is adapted as our best practice for the numerical study of the container hull and its design variations.

The simulation was run in parallel with 12 cores on HP Z800 workstation. It took 16 hours to run 15000 iterations for the fine mesh with 4.9 million cells.

![Figure 8](image8.png)

**Figure 8.** Simulation results with a mesh size of 1.2 million (a) Convergent history of hull resistance and (b) ship induced Kelvin waves

![Figure 9](image9.png)

**Figure 9.** Simulation results with a mesh size of 4.9 million (a) Convergent history of hull resistance and (b) ship induced Kelvin waves

| Mesh Cell Number | Error of Hull Resistance Compared with Experimental Data | Convergence Speed (iterations) |
|------------------|--------------------------------------------------------|-------------------------------|
| Coarse Mesh      | +12.6%                                                  | 3000                          |
| Baseline Mesh    | +6.8%                                                   | 4000                          |
| Fine Mesh        | +5.4%                                                   | 8000                          |
4. Summary
For ship hull design, an automatic analysis procedure, from design to geometry modelling, mesh processing, and performance simulation, has been developed in order to do hull performance evaluation and be ready for the hull design optimization. A case study is carried out to test the methodology. The testing results indicate that the procedure runs smoothly. The method developed in this study enables an automatic generation of the hull geometry and its application in CFD modelling without manual intervention operation on the CAD model. Thus, an automatic procedure from design to performance analysis is achieved. Now, it is ready for a fully automatic hull form design optimization to be carried out.

5. Future work
Currently, the procedure has not been integrated by a GUI into a framework. In order to complete the automatic procedure of the hull performance evaluation, a GUI will be developed to integrate the hull shape design, geometry construction, meshing, and simulation in the next step, as well as to couple with an optimization algorithm, and to provide users options of performance evaluations or design optimizations.

In addition, a method to allow users or an automated algorithm to make modifications to the ship hull for performance improvement might be implemented through a series of control points around the ship hull. Based on the shifted positions of the control points, a radial basis function can be used to make smooth interpolated modifications to the surface of the ship hull. The density of the control point around the hull determines the scale and range of the modification.

References
[1] https://en.wikipedia.org/wiki/CD-adapco
[2] https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html
[3] https://wikiwaves.org/Ship_Kelvin_Wake
[4] OpenFOAM, User Guide v1712.
[5] ITTC Recommended Procedures and Guidelines, 2002, Testing and Extrapolation Methods Propulsion; Cavitation Cavitation-induced Pressure Fluctuations: Numerical Prediction Methods, 7.5-02-03-03.4.
[6] Beall M.W., Walsh J., Shephard M.S., 2003, Accessing CAD geometry for mesh generation, Proceedings of 12th International Meshing Roundtable, Sandia National Laboratories, NM, USA, pp. 33–42.
[7] Sadrehaghighi, I., 2012, CFD Open Series. Revision 1.75. Mesh Generation in CFD. Annapolis, MD.
[8] Tu J., Wong K.K.L., Inthavong K., 2015, Computational Hemodynamics – Theory, Modelling and Applications, Dordrecht ; Heidelberg ; New York : Springer.