Interactive Reduced Order Models for Ship Hull Design and Optimization Coupling an Open-Source CFD Tool with Advanced RBF Mesh Morphing

Emiliano Costa, Benedetto Di Paolo, Apostolos Krassas, Paolo Geremia, ENGYS, Trieste/Italy, e.costa@engys.com
Corrado Groth, Marco Evangelos Biancolini, University of Rome "Tor Vergata", Rome/Italy
Marco Camponeschi, Ubaldo Cella, Emanuele Di Meo, RBF Morph, Monte Compatri/Italy

Abstract

We present the development of the proof-of-concept (PoC) for an effective and efficient numerical procedure conceived to design and optimize ship hulls by leveraging reduced order models (ROMs) gained processing a database of computational fluid dynamics (CFD) fields relevant to the marine sector that is created by performing a design of experiments (DOE) study parametrizing the CFD case using radial basis functions (RBF) mesh morphing. The ROMs, that can be interactively used to get real-time calculations, demonstrated a maximum percentage error on prediction of less than 3.5% for resistance estimates at design points outside the original DOE set, indicating strong predictive accuracy. The resulting ROMs serve as a foundational step toward building Digital Twins of ships.

1. Introduction

The optimization of ship hull forms has traditionally relied on high-fidelity Computational Fluid Dynamics (CFD) simulations to accurately capture the complex hydrodynamic phenomena governing resistance, propulsion, and seakeeping performance. While CFD provides a robust and reliable framework for performance assessment, its application in design optimization is often hindered by the high computational cost associated with repeated simulations across a large design space. This limitation becomes particularly pronounced when gradient-based or evolutionary optimization strategies are employed, as these approaches may require thousands of design evaluations.

Recent advances in Artificial Intelligence (AI) and Reduced-Order Modeling (ROM) techniques have introduced promising alternatives to accelerate hull-form optimization without compromising predictive accuracy. ROMs, constructed from a limited set of high-fidelity CFD solutions, enable the rapid evaluation of new design candidates at a fraction of the computational cost. These surrogate models, which may be based on methods such as Proper Orthogonal Decomposition (POD), Dynamic Mode Decomposition, or Machine Learning regression techniques, capture the dominant flow features while discarding less influential dynamics. When integrated into optimization frameworks, ROMs allow for efficient exploration of the design space, enabling near real-time prediction of hydrodynamic responses.

In the context of simulation-based design optimization in marine engineering, *Serani et al. (2024)*, we performed a proof-of-concept (PoC) to showcase the streamlined and effective numerical procedure conceived to design and optimize ship hulls by leveraging ROMs gained by treating a dataset of results assessing CFD fields relevant to the marine sector. Such dataset is created by carrying out a Design of Experiments (DOE) study parametrizing the CFD case using RBF mesh morphing.

2. Rationale of the numerical procedure

The proposed numerical procedure aims at designing and optimizing ship hull forms by leveraging ROMs obtained processing a database of CFD fields relevant to the marine sector, which is generated by carrying out a DOE study parametrizing the shape of the CFD case using Radial Basis Functions (RBF) mesh morphing techniques, *Biancolini (2017),* and interactive reduced order models, *Biancolini (2020)*.

The whole numerical procedure consists of two sequential stages: an off-line stage and on-line stage. To perform the tasks of such stages, different CAE software tools were employed, while Python scripts were suitably implemented to streamline the processes. The key aspects and steps of those stages are respectively described in the following sections.

2.1. Off-line stage

The off-line stage consists of the following steps:

- Definition of the CFD baseline case: using an open-source CFD software, the reference (i.e., baseline) case is suitably defined to simulate and create the sought numerical output of interest of the PoC.
- Parametrization of the reference CAD model of the hull: the selected CAD model is geometrically parametrized using RBF mesh morphing techniques thus creating the shape variants of the baseline configuration of the ship's hull.
- Generation of the CFD database: the database of the CFD results of interest (e.g., resistance, wall shear stress, free surface elevation) is created performing a DOE study, where each design point (DP) consists of a CFD simulation of calm-water resistance of a CFD case variant, adopting the Linearized Free Surface (LFS) (*Geremia 2019)* solver of the open-source CFD tool used.
- ROMs generation: a ROM is created for all CFD fields of interest such as mesh, resistance and pressure over the hull surface, by processing the CFD results database using the Proper Orthogonal Decomposition (POD) method, *Ostrowski et al. (2008), Dotta et al. (2021),* based on truncated Singular Value Decomposition (SVD), *Stewart (1993)*.

The workflow of the off-line stage is shown in Fig.1.

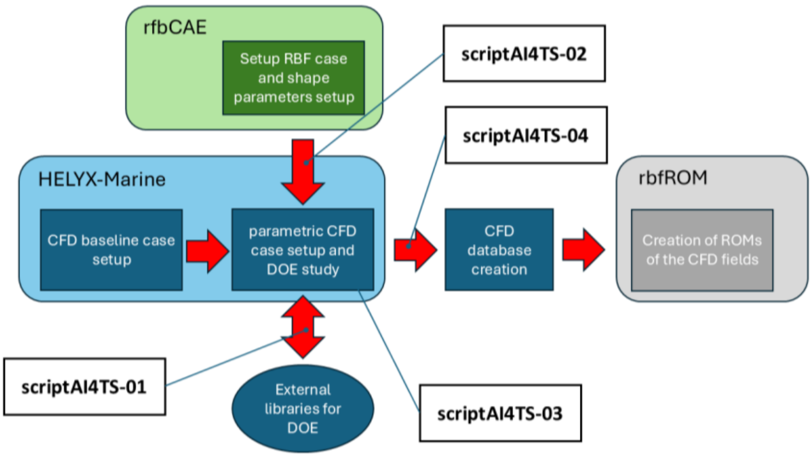

Fig.1: Workflow of the off-line stage

2.2. On-line stage

The on-line stage consists of the following steps:

- Real-time prediction using ROMs: ROMs can accurately predict both the ship resistance and distribution of certain relevant CFD quantities both using text commands and through an interactive dashboard developed using PyVista, https://docs.pyvista.org/, enabling real-time 3D visualization of the CFD results and quantities of interest for new unseen geometries defined with new combinations of parameters not belonging to the training set.
- ROM post validation: identified candidates undergo a full validation by full high fidelity CFD. Thanks to the high level of automation this task can be enabled on-line.

The workflow of the on-line state is shown in Fig.2.

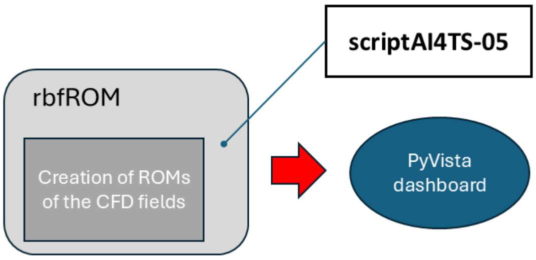

Fig.2: Workflow of the on-line stage

2.3. Automation

In order to make the proposed numerical procedure more efficient while limiting as much as possible the intervention requested to the user, Python scripts were developed and used. This programming language was deliberately chosen, as it enables the automation of CFD and mesh morphing operations through dedicated Application Programming Interfaces (API).

The operations that are automated by these scripts are shown in Fig.1 and Fig.2, respectively. Several stand-alone scripts were implemented to keep the approach more general and flexible; the scripts can be merged or customized depending on specific needs (e.g., in the case that CFD calculations are requested to be accomplished on High-Performance Computing infrastructures).

3. Baseline CFD case generation and results

3.1. CFD methodology

In the field of ship hull hydrodynamics, CFD solvers are commonly built upon either the Volume-of-Fluid (VOF), *Hirt and Nichols (1981),* or level-set approaches, *Yang et al. (2007)*. These techniques are typically integrated within the Reynolds-Averaged Navier-Stokes (RANS) equations using a finite-volume discretization. Both methods have shown excellent accuracy in estimating hull resistance and other critical hydrodynamic characteristics. However, their reliance on high-performance computing and extended runtimes makes them less suitable for use during the early design phases, where rapid feedback is essential.

Additionally, the stability and precision of VOF and level-set solvers depend heavily on tailored mesh generation strategies and the use of very small time steps. These requirements add complexity and further increase computational demands. To mitigate these challenges, a RANS-based Linearized Free surface (LFS) solver that incorporates viscous effects was developed. This solver operates under a steady-state assumption, enabling significantly faster evaluations of hull resistance and related parameters, while maintaining a high level of accuracy, *Rosemurgy et al. (2011)*.

The LFS solver is derived from the linearized form of the unsteady Neumann-Kelvin boundary-value problem, which models ship-generated waves. It assumes that wave amplitudes and slopes are small enough to justify linearizing the free surface boundary condition on the undisturbed water surface. This simplification allows for a double-body representation and the use of a single-phase flow solver, making it possible to analyze multiple hull motions efficiently and with reduced computational effort. Validation of the RANS solver was conducted using the Gothenburg 2010 Case 2.2b, which involves the Kriso Container Ship (KCS) at model scale, http://www.simman2008.dk/KCS/kcs_geometry.htm. Results were compared against a high-fidelity VOF simulation and experimental data. The computational mesh consisted of 1.2 million cells for the LFS case and 3.1 million cells for the VOF case. Fig.3 shows a close-up of the mesh configurations.

The free surface elevation results are presented in Fig.4. The resistance coefficient is shown in Fig.5:

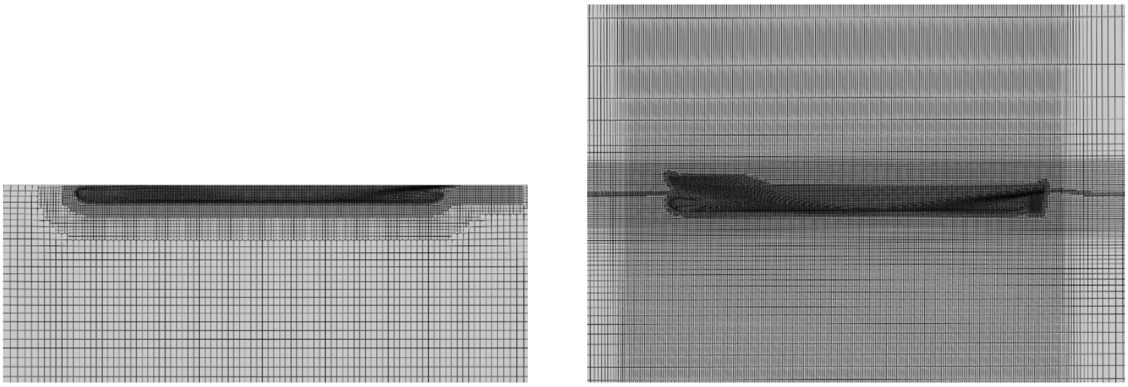

Fig.3: Computational mesh for the validation case: LFS solver (left) and VOF case (right)

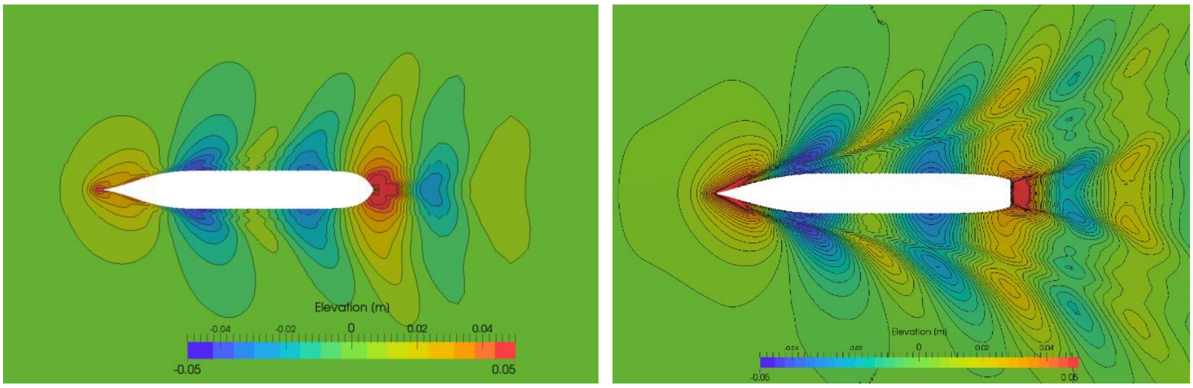

Fig.4: Free surface elevation results for the validation case: LFS solver (left) and VOF case (right)

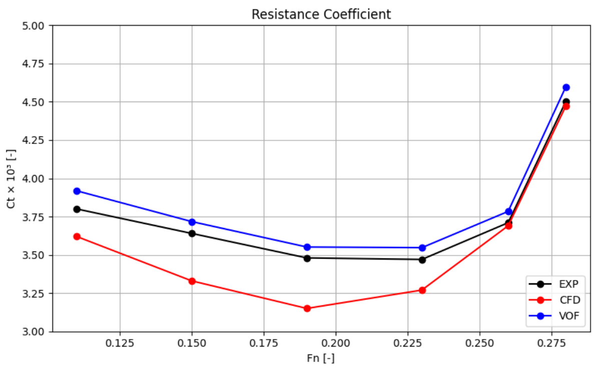

Fig.5: Resistance coefficient for the validation case

Although the LFS method is an approximation, the resistance force trend is accurately captured across the full range of Froude numbers. The key advantage of the LFS solver lies in its computational efficiency: each simulation runs in approximately 20 minutes on a 64-core AMD Genoa system, compared to 5.6 hours required by the VOF solver. This makes the LFS approach particularly attractive for parametric studies and hull-form optimization during the early design phase.

3.2. Strategy to set up the RBF case

The baseline CFD case of the KCS container ship was created in the CFD software to calculate the results of interest, which are required for the subsequent development of the ROMs. This process was carried out semi-automatically through the combined use of Python and the CFD software's API.

The procedure involves loading the discretized CAD model of the KCS in STL format, which represents half of the ship model at full scale, and generating the mesh which is properly detailed to capture the areas of the surfaces with higher curvature. The generated volume mesh consists of approximately 2.5 million cells and includes multiple wall-cell layers to enable the correct application of wall functions in the vicinity of the hull surfaces. Standard water properties were assumed, and the k–ω SST turbulence model was adopted.

To compute the quantities of interest in steady state conditions corresponding to a Froude number (Fn) of 0.26, a RANS simulation was carried out with 5,000 iterations, sufficient to achieve convergence of the calm-water resistance with two degrees of freedom, namely heave and pitch angle, using the LFS solver available in the CFD tool.

Once the steady-state solution was obtained, the CFD results of interest were extracted, namely the total resistance of the hull, the pressure distribution on the hull surface, and the wave elevation field, saved in VTK (*Schroeder et al. 2006)* ASCII format. The evolution of the total force acting on the KCS hull as a function of the CFD iterations is shown in Fig.5. The steady state resistance, calculated as the average of the total force over the final 1,000 iterations, was found to be 102.45 kN. This quantity is the main CFD scalar field of interest considered in the development of the PoC.

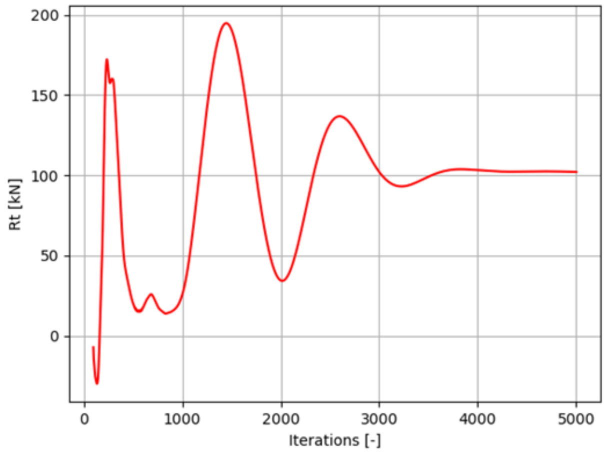

Fig.5: Total force profile as a function of CFD iterations

Fig.6 shows the two main three-dimensional outputs of interest in the PoC, i.e. the steady state pressure distribution over the KCS hull and free surface elevation on the left and right, respectively.

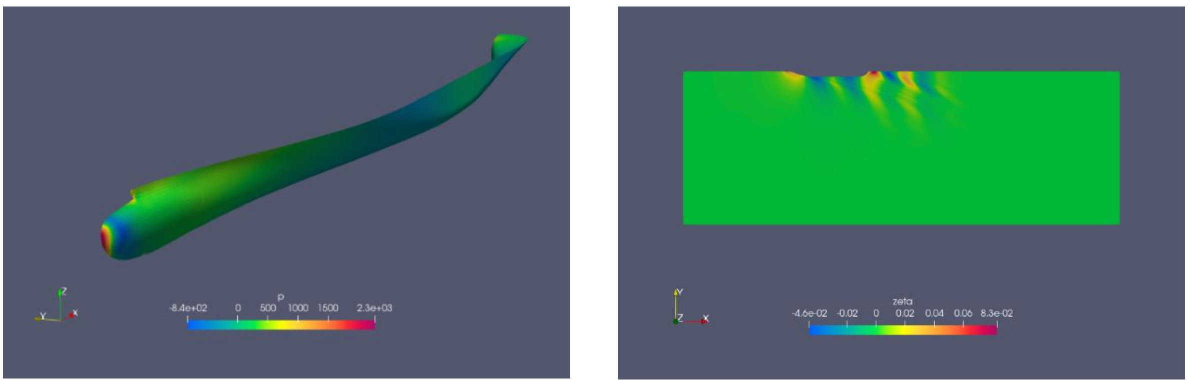

Fig.6: Distribution of pressure over the hull and free surface elevation

Given the need to build a database in the subsequent steps of the numerical procedure, the relevant CFD outputs for the PoC - such as pressure distribution over the hull, hull's resistance and free surface elevation - are configured to be exported in VTK ASCII format upon completion of the computation.

4. Geometrical parametrization using RBF mesh morphing and design points generation

4.1. Strategy to set up the RBF case

In the context of the tools used in the PoC, two morphing strategies can be considered for generating the DOE DPs, namely surface-morphing and volume-morphing.

In the surface-morphing approach, new hull geometries are obtained by applying RBF modifications to a discretized STL reference model. Each variant is then used to create a separate case in the CFD tool. This method is generally more robust, as it modifies only the surface mesh, with no direct impact on the quality of the volume mesh which will have, in general, a different topology for the different DPs.

In contrast, the volume-morphing approach directly alters the baseline CFD mesh by applying combinations of RBF parameters. While this method avoids regenerating a new volume mesh for each DOE DP and can therefore be faster, it is less robust because mesh quality deteriorates as deformations increase, limiting the allowable extension of modifications.

Considering the need for POD mathematical framework to process arrays of CFD fields of consistent length, volume-morphing was employed to ensure the mesh topology remains consistent across the different DPs of the DOE. This guarantees that the CFD results contain the same number of surface data points.

When the validity of volume mesh cannot be guaranteed, a hybrid approach can be used (not covered in this study): both methods are applied simultaneously, so that the new geometry is represented both as a deformed mesh and as a newly generated highly quality mesh. Once CFD convergence is achieved, the results are mapped onto the deformed mesh, which is then ready for POD compression.

4.2. RBF case setup

The adopted strategy for implementing RBF-based shape modifications focused on altering both the bulbous bow and the stern region of the KCS hull, as these areas were identified as having the most significant impact on hydrodynamic performance. The modifications included shape variations both within the hull's symmetry plane and in directions outside of it (i.e., out-of-plane).

To ensure a high degree of flexibility in creating such modifications by means of the RBF mesh morphing tool, the following items were generated and provided as input: (i) a discretized CAD model of the entire ship in STL format, serving as one of the primary inputs for generating the baseline mesh; (ii) a neutral CAD model of the bulbous bow and stern region in STEP format, used as input to the mesh-morphing software for creating the cases defining the RBF shape modifications; and (iii) a discretized CAD model of the bulbous bow and stern region in STL format, used as input to the morphing tool for the same purpose as the neutral CAD model. In such a way both CAD entities such as vertex, edges and areas, as well as mesh entities such as nodes can be used to create RBF shape modifiers.

The discretized CAD model of the entire hull features a high spatial resolution, allowing for an accurate representation of surfaces with significant curvature. In contrast, the discretized CAD models of the bulbous bow and stern region were generated with a lower resolution, to simplify the setup and facilitate the application of shape modifications on the baseline CFD mesh.

The geometric parameterization of the bulbous bow and stern region of the KCS hull was investigated. The mesh-morphing case setup was created using the GUI provided with the morphing software. The CAD models of the bulbous bow and stern region are shown in Fig.7 and Fig.8 respectively, where the neutral CAD model in STEP format is presented on the left, and the corresponding discretized STL model is presented on the right.

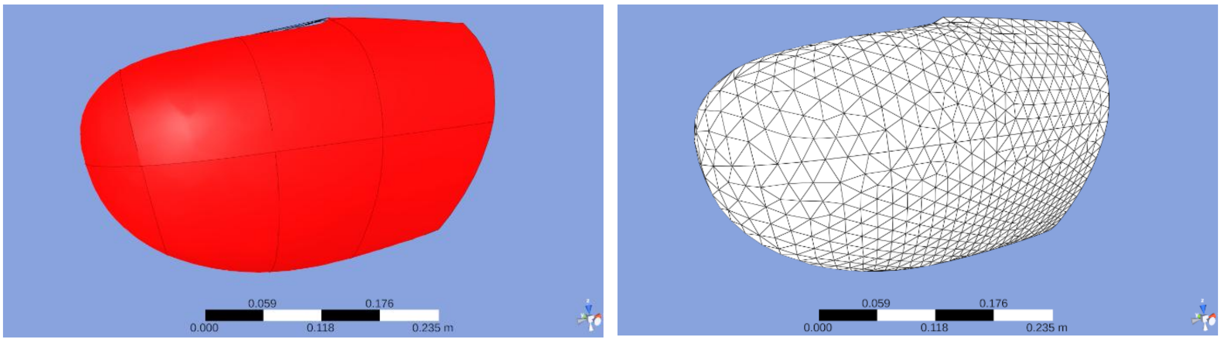

Fig.7: Neutral and discretized CAD models of the bulbous bow

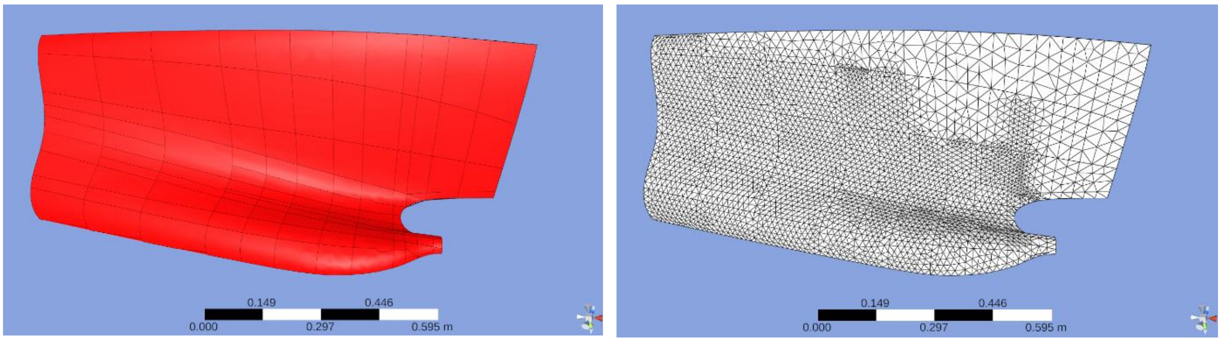

Fig.8: Neutral and discretized CAD models of the stern region of the hull

Utilizing the CAD models mentioned above, a morphing-tool case was generated that includes five different RBF shape modifications: three applied to the bulbous bow and two to the stern region. To ensure continuity of the modifications with respect to the hull, all RBF shape parameters of the first three modifications were constrained to preserve the position of the edges shared with the hull by imposing a zero-displacement condition.

For one of the RBF shape modifiers that acts outside the “symmetry” plane of the model in Fig.9 (left) the positions of the RBF points before and after the application of morphing are shown in green and blue respectively. This modification combines RBF points that keep fixed—including the end nodes of the node set highlighted on Fig.9 (right) —with the scaling action illustrated on the left for a given amplification factor. The scaling operation thus makes it possible to widen or reduce the bulb’s cross-sectional area while preserving, as much as possible, curvature continuity in the transverse direction relative to the ship’s forward motion.

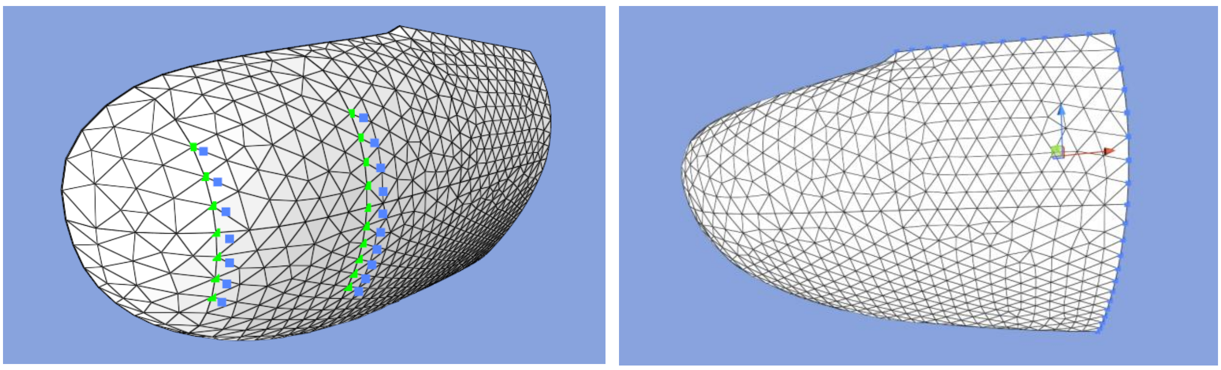
Fig.9: Main settings of one of the out-of-plane shape modifications

Fig.10 shows the effect of the shape modification (right) compared to the baseline bulb configuration (left), from two different views. Once the CFD baseline case and RBF solution to parametrize the hull are prepared, the main inputs needed by the numerical procedure are ready to be used to run the semi-automated processes described below.

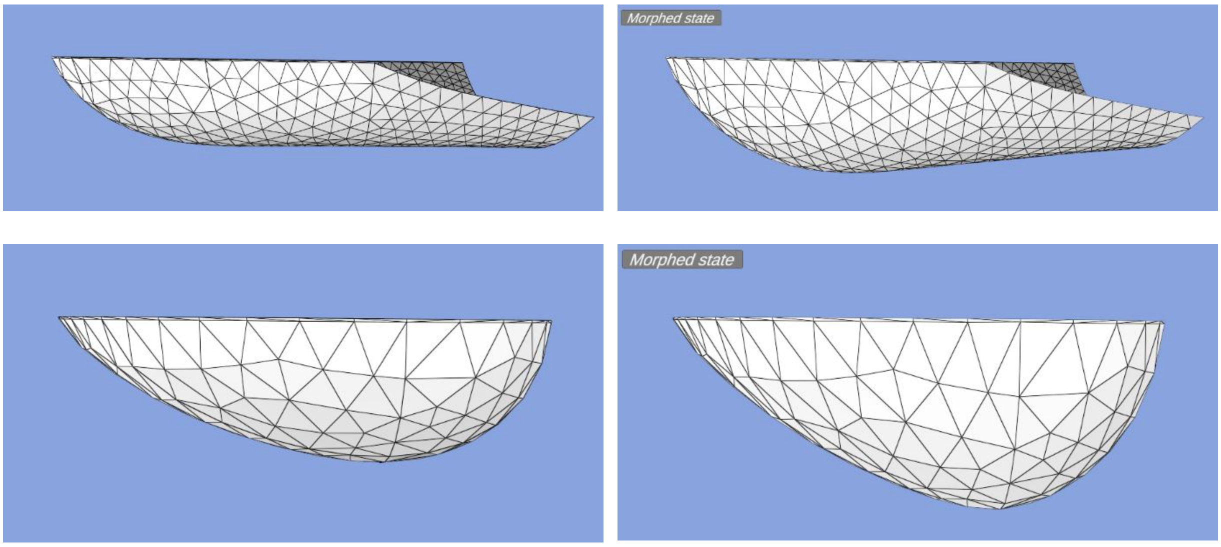

Fig.10: Morphing action of one of the out-of-plane shape modifications

5. CFD database generation and creation of ROMs of CFD fields

5.1. Database generation

A first Python script, Fig.1, automatically generates the DOE table in a format suitable for the mesh morphing tool by reading the main data defining the number of RBF parameters, their name and range of variation, as well as the number of DPs and the sampling method in an input file. The DOE study carried out for the PoC was generated via Latin Hypercube Sampling (LHS) and finally composed by 101 DPs including the baseline configuration.

The second Python script automatically creates the DPs as CFD morphed cases configurations, by reading the DOE table generated in the previous step adopting the volume morphing approach.

5.2. Database generation

The third Python script manages the simulation of all DPs that are run in sequence. The values of the hull's resistance calculated for all DPs vary between -1.43% and +10.14% with respect to the one determined for the baseline configuration of the KCS hull. The machine to perform this task was equipped with 2x EPYC 9354 (Genoa) 32-Core with a clock frequency of 3.5 GHz. After the CFD computation, the fourth and final Python script collects all the CFD results of interest and generates the CFD database. This includes a VTK file containing the pressure distribution over the hull and free surface elevation, as well as the DOE table enriched with resistance values for each DP.

5.3. ROMs creation

The first task for ROMs creations consists of the setup of the framework suitable for POD processing according to which the CFD fields of interest, that is mesh, resistance, pressure and free surface elevation, are structured as arrays.

Once this framework is established, the weights and parameters of the CFD fields are determined. Subsequently, the POD processing is performed to compute the modes of the CFD fields. ROMs can then be constructed by retaining a selected number of modes—an operation commonly referred to as modes truncation. This number of modes can be chosen by the user based on graphs showing the maximum error associated with the model basis, which is computed during the POD processing. An example of these graphs is shown in Fig.11 referring to both the mesh and pressure field.

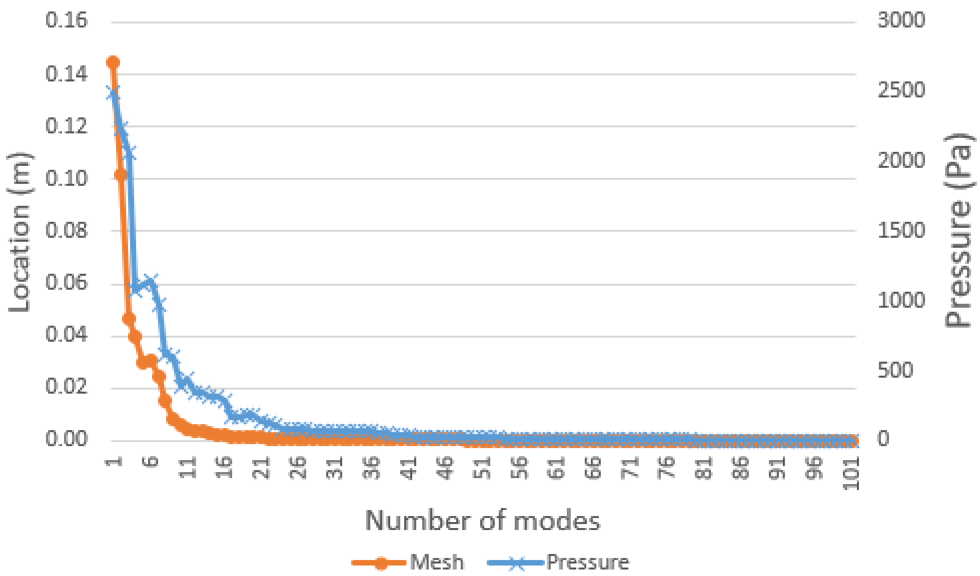

Fig.11: Maximum error of mesh and pressure against number of modes

Considering the low computing demands related to the update of the CFD fields through ROMs, all the computed modes were used and the full SVD decomposition retained.

6. Results

To assess the accuracy of the ROMs during the on-line phase, the evaluation first focused on the prediction of pressure and resistance fields. These verifications were performed for two design points within the DOE as well as for two additional points outside the DOE table using both text commands and the interactive dashboard.

6.1. Verification using DOE design points and additional design points

Table I reports the results of the resistance prediction accuracy verification for the selected DPs, showing that the error is negligible; this is a direct consequence of having used all the modes as the SVD decomposition is complete and no truncation has been imposed.

Table I: Validation of ROM accuracy for resistance at the four DOE DPs

ID	RBF parameters combination	CFD	ROM	Error %
0	0, 0, 1, 0, 1	102.4538300	102.45383	0.000000E+00
21	-0.0347, 0.0327, 1.19, 1.6, 1.25	108.5159485	108.51595	-1.382285E-06

Fig.12 and Fig.13 present, respectively, a visualization of the pressure distribution and of the free surface elevation predicted by the interactive ROM and computed with the full CFD. As shown, the dashboard consists of a panel displaying the mesh, the scalar field value of hull resistance in the top-right corner, and the distribution of three-dimensional CFD outputs such as pressure and free surface elevation. Additionally, a set of five sliders on the left allows users to adjust the amplitude of each RBF parameter within its variation range to update the CFD fields accordingly.

To complete the verification of the ROM prediction accuracy, two DPs outside the DOE set were identified. The combinations of the RBF parameter values defining these points are collected in Table II. The maximum error is quantified as the capability to predict the resistance variation; it is equal to 3.5% for the additional study point with ID1, and equal to 0.03% for the additional study point with ID2.

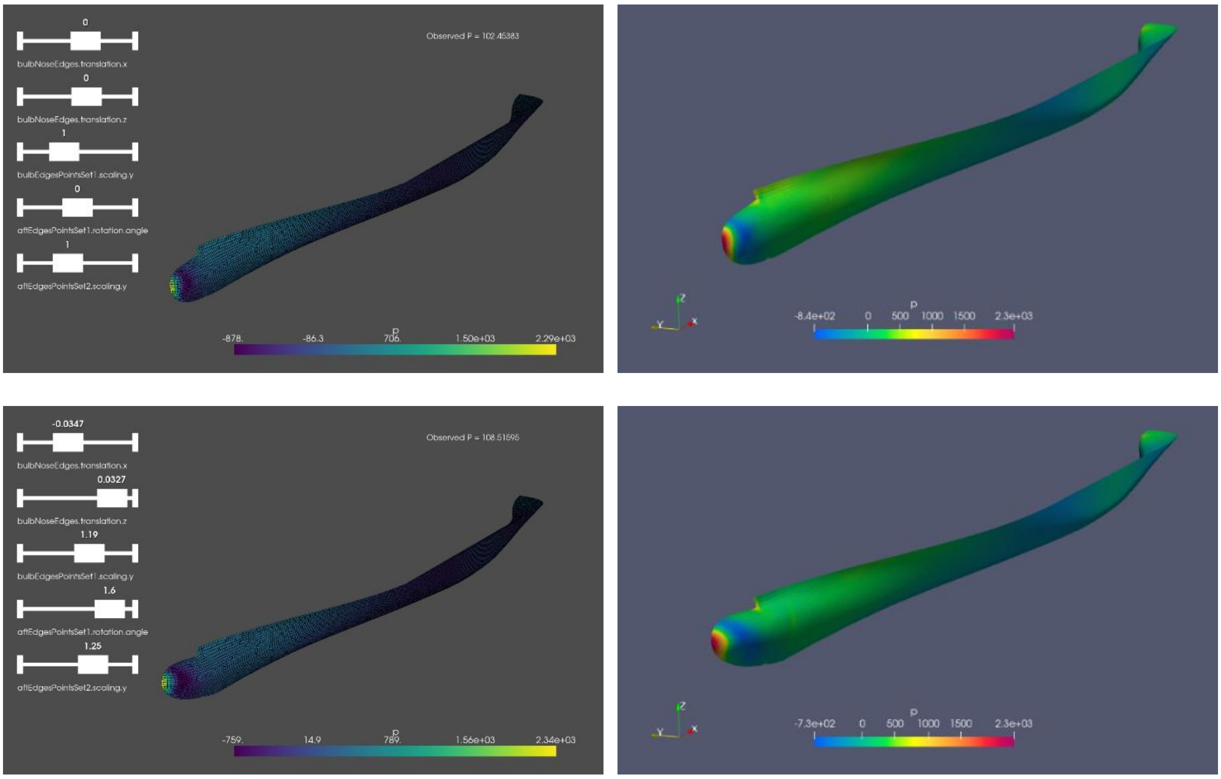

Fig.12: Validation of ROM accuracy for resistance and pressure at two DOE DPs, ROM (left) and CFD (right)

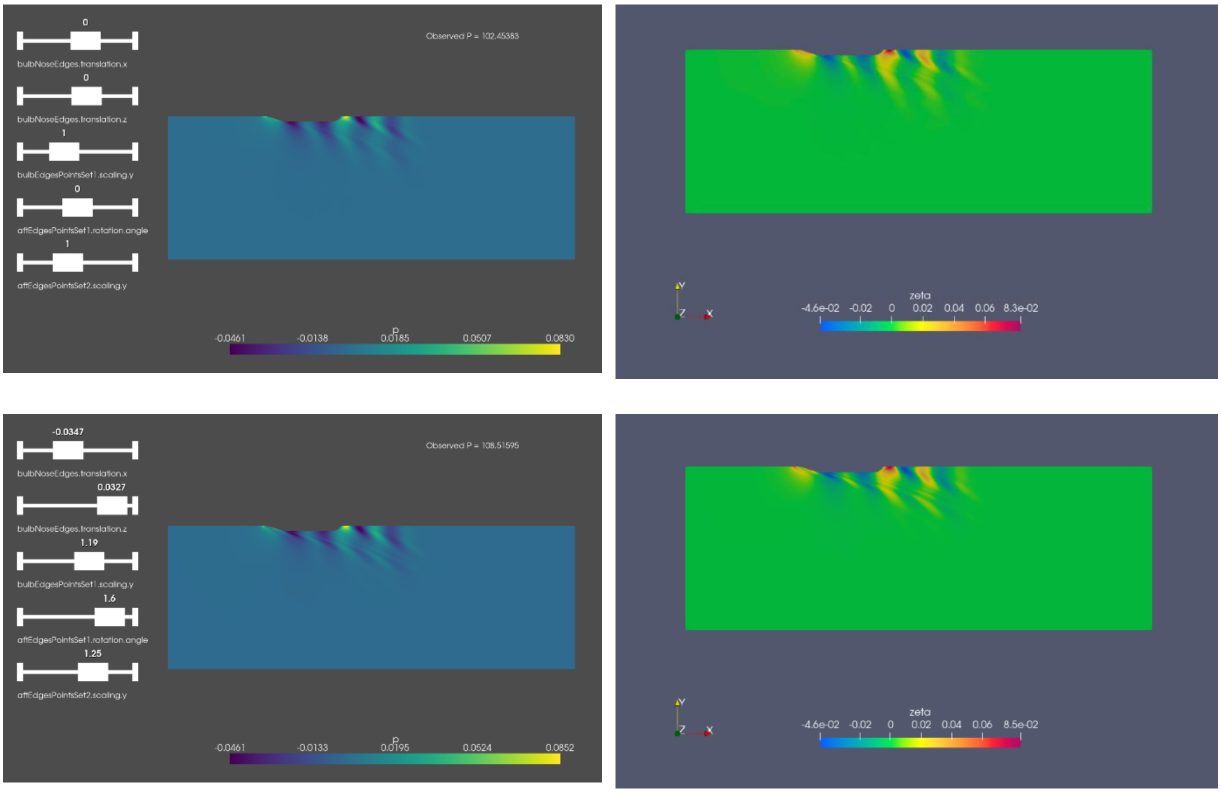

Fig.13: ROM accuracy verification for resistance and free surface elevation for two DOE DPs, ROM (left) and CFD (right)

Table II: Validation of ROM accuracy for resistance at the two additional DPs

ID	RBF parameters combination	deltaCFD	deltaROM	Error %
1	-0.05, 0.0392, 0.9, 1, 0.9	2.58	2.49	3.48
2	0.0209, -0.0312, 1.04, 1.82, 1.16	1.14	1.14	0.03

Fig.14 shows the pressure distribution on the hull and the resistance values for the selected out-of-DOE points respectively predicted by ROMs compared with the full CFD.

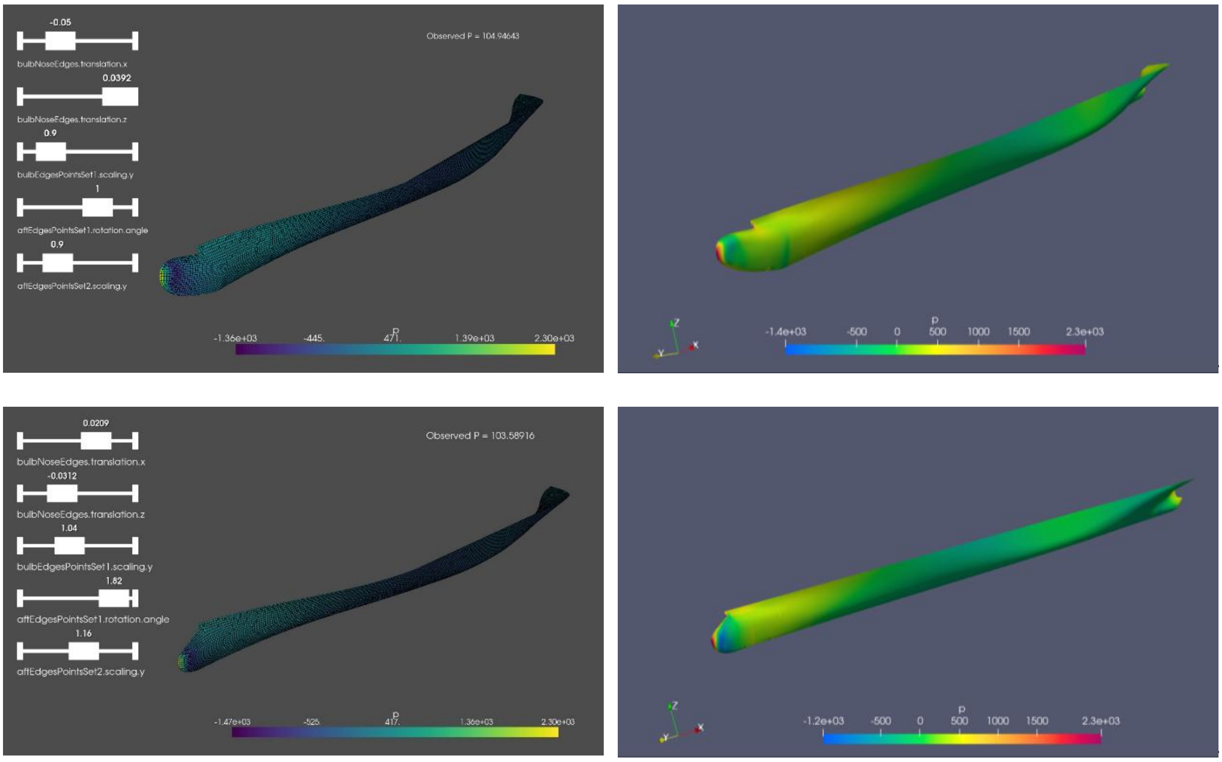

Fig.14: Validation of ROM accuracy for two DPs outside of the DOE

6.2. Real-time computations and interaction with the dashboard design points

The final evaluation of the numerical procedure is focused on the user experience during the on-line stage, specifically in terms of interactivity and response times when visualizing the CFD fields provided by the ROMs through the dashboard, as a function of different combinations of RBF shape parameters. Overall, the user experience proved to be highly satisfactory, as demonstrated by the animations available online on a proprietary YouTube channel, Interactive ROM predictions of pressure over the KCS hull and KCS hull resistance, Interactive ROM predictions of free surface elevation and KCS hull resistance.

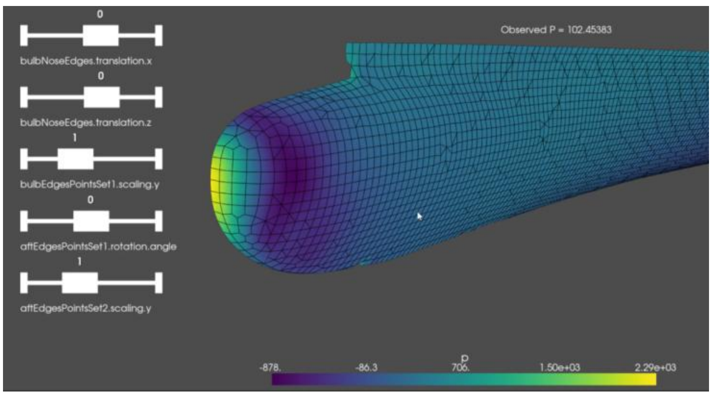

Fig.15: Snapshot of on-line stage to compute hull resistance and pressure distribution interactively

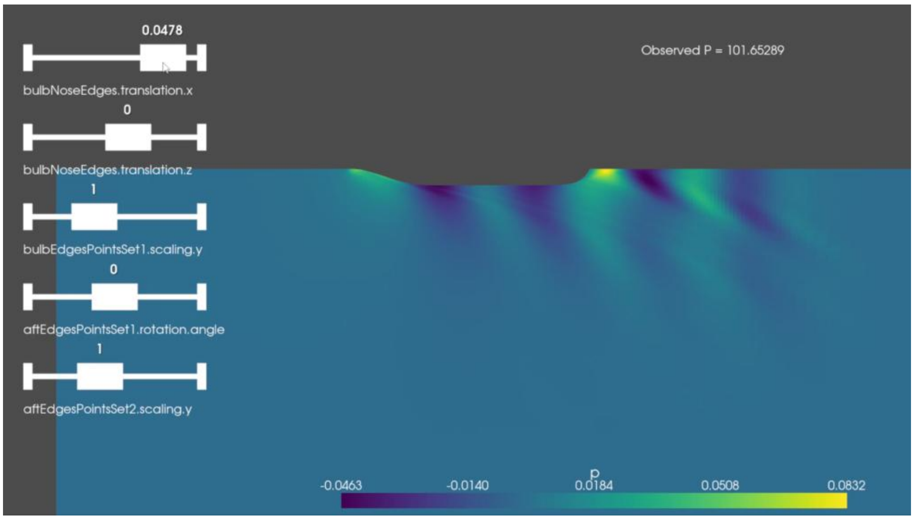

Fig.16: Snapshot of on-line stage to compute hull resistance and free surface elevation interactively

Two snapshots captured from these animations are shown in Fig.15 and Fig.16 for the pressure distribution over the KCS hull and free surface elevation, respectively. For both hull resistance of the current RBF parameters combination is reported as well.

The assessment of interactivity with the ROMs was carried out using a laptop computer equipped with an AMD Ryzen 7 PRO 6850U processor operating at a clock frequency of 2.70 GHz and integrated with an AMD Radeon 680M graphics card.

7. Conclusions

In this paper, the development of a Proof-of-Concept demonstrating a streamlined numerical procedure for ship hull design and optimization was presented. The procedure leverages Reduced-Order Models (ROMs) constructed through the Proper Orthogonal Decomposition (POD) method, applied to a dataset of Computational Fluid Dynamics (CFD) field data relevant to the marine sector. The database was generated by conducting a Design Of Experiments (DOE) study, consisting of 101 design points obtained by parameterizing the CFD model of the KCS hull with five shape parameters, implemented via Radial Basis Function (RBF) mesh morphing. Each design point was simulated using a steady-state RANS solver within an open-source CFD tool to compute calm-water resistance, thereby assembling the full database of morphed hull shapes.

The ROMs enabled real-time predictions, accessible through either text commands or an interactive dashboard that can be run even on a personal laptop smoothly, with high accuracy at design points outside the DOE set, the maximum relative error in resistance prediction was below 3.5%. Wherever possible, the workflow was automated through Python scripting, further enhancing the efficiency and reproducibility of the procedure.

RBF mesh morphing proved to be a versatile tool for handling complex shape parameterization, offering a high level of control over surface modifications, while POD processing demonstrated to be fast, robust, and computationally efficient. The resulting ROMs open the way for multi-physics studies such as fluid–structure interaction (FSI) and represent a key step toward the development of Digital Twins of ships. In this context, the adoption of open-source CFD tools for large-scale computations makes the overall approach economically attractive, as no additional licensing costs are incurred. This aspect is particularly relevant when exploiting HPC resources to generate large datasets for data-driven modeling.

Acknowledgements

This work was developed within the framework of the Italian research project AI4TwinShip, that was co-funded by the European Union, the Italian government and the Friuli-Venezia Giulia region in the context of the PR FESR 2021-2027 program.

References

BIANCOLINI, M.E. (2017), *Fast Radial Basis Functions for Engineering Applications*, Springer Cham, https://doi.org/10.1007/978-3-319-75011-8

BIANCOLINI, M.E.; CAPELLINI, K.; COSTA, E.; GROTH, C.; CELI, S. (2020), *Fast interactive CFD evaluation of hemodynamics assisted by RBF mesh morphing and reduced order models: the case of aTAA modelling*, Int. J. Interact. Des. Manuf. 14, pp.1227–1238

DUTTA, S.; FARTHING, M. W.; PERRACCHIONE, E.; SAVANT, G.; PUTTI, M. (2021), *A greedy non-intrusive reduced order model for shallow water equations*, J. Computational Physics 439, 110378

GEREMIA, P.; MAKI, K.J.; ALEXIAS, P. (2019), *A linearized free surface RANS method for self-propulsion and maneuvering*, 11th Int. Workshop on Ship and Marine Hydrodynamics (IWSH2019)

HIRT, W.C.; NICHOLS B.D. (1981), *Volume of fluid (VOF) method for the dynamics of free boundaries*, J. Computational Physics 39, pp.201-225

OSTROWSKI Z., BIAŁECKI R.A.; KASSAB A.J. (2008), *Solving inverse heat conduction problems using trained POD-RBF network inverse method*, Inverse Problems in Science and Engineering 16:1, pp.39-54

ROSEMURGY, W.J.; EDMUND D.O.; MAKI K.J.; BECK R.F. (2011), *A Method for Resistance Prediction in the Design Environment*, 11th Int. Conf. Fast Sea Transportation

SCHROEDER, W; MARTIN, K.; LORENSEN, B. (2006), *The Visualization Toolkit* (4th ed.), Kitware

SERANI, A.; SCHOLCZ, T.P.; VANZI, V. (2024), *A Scoping Review on Simulation-Based Design Optimization in Marine Engineering: Trends, Best Practices, and Gaps*, Arch. Computat. Methods Eng. 31, pp.4709–4737

STEWART, G.W. (1993), *On the early history of the singular value decomposition*, SIAM Review 35 (4), pp.551–566

YANG, J.; SAKAMOTO, N.; WANG, Z.Y.; CARRICA, P.; STERN, F. (2007), *Two Phase Level-Set/Immersed-Boundary Cartesian Grid Method for Ship Hydrodynamics*, 9th Int. Conf. Numerical Ship Hydrodynamics, Ann Arbor