Asociación Argentina
de Mecánica Computacional

Mecánica Computacional Vol XLII, pp. 343-352
A. Caggiano, G. Etse, P. Folino, M. Goldschmit, M. Pucheta, M. Storti (Eds.)
L. Battaglia, H. Castro, M. Coussirat, R. Paz, S. Sarraf, A. Scarabino (Issue eds.)
Buenos Aires, November 11-14, 2025

APPLICATION OF DYNAMIC MESH TECHNIQUES TO THE MODELLING OF DENSITY CURRENTS IN THE PANAMA CANAL MIRAFLORES LOCKS

Nicolás D. Badano, **Foteini Kyrousi**, **Mahdi Alemi**, **Boudewijn Decrop**

International Maritime Dredging Consultants (IMDC). Antwerp, Belgium, https://imdc.be
nicolas.badano@external.imdc.be

Keywords: dynamic mesh, density currents, CFD, OpenFOAM.

Abstract. As part of a broader study aimed at investigating the hydrodynamic and density-related effects within the Panama Canal locks, several Computational Fluid Dynamics (CFD) models of the Miraflores Lock complex have been developed using OpenFOAM. These models incorporate multiple moving components—including vessels, miter gates, and variable free surface levels—as well as the interaction between saltwater and freshwater, which generates stratified density currents. To accurately capture the buoyancy-driven effects associated with the salt-water – fresh-water mixing, a custom flow solver was developed to handle two miscible phases with dynamic mesh support. Modifications to the turbulence model were also implemented to incorporate buoyancy effects Several dynamic mesh techniques were evaluated to accurately simulate flow conditions inside the lock chambers during different lockage stages. The advantages and limitations of these techniques are discussed, and the implementation of a hybrid approach—combining mesh deformation with periodic remeshing—is presented. This method was found to be the most effective in terms of flexibility and ease-of-use.

ISSN: 2591-3522 DOI: 10.70567/mc.v42.ocsid8450

1 INTRODUCTION

The Panama Canal is one of the most critical waterways in the world, enabling maritime traffic between the Atlantic and Pacific Oceans without the need to circumnavigate South America. Each year, thousands of vessels of different sizes and types transit the canal, making its continuous operation and efficiency vital for global trade (Rodríguez, 2015). The Panama Canal is operated and managed by the *Authority of the Panama Canal* (ACP), an agency of the government of Panama.

The Miraflores Locks, located at the Pacific entrance of the canal, play a particularly important role in regulating ship passage. They consist of two lock chambers that lift or lower vessels by approximately 16 meters. As in other lock systems, hydraulic processes inside the chambers are highly complex, involving transient filling and emptying flows, moving gates, and vessel dynamics. In addition, due to the proximity of the Pacific Ocean, salt-water intrusions occur, resulting in density-driven currents that interact with vessel maneuvers and the lock infrastructure.

Density currents within the Miraflores Locks pose significant operational challenges. When the miter gates open, lock-exchange flow driven by density differences develop. This flow produces forces over the vessels entering or exiting the lock complex, complicating navigation.

The combination of multiple moving boundaries (ships, miter gates, water levels) with density-driven processes necessitates the use of advanced Computational Fluid Dynamics (CFD) techniques. In particular, dynamic mesh techniques are essential for simulating these transient processes with sufficient accuracy.

This study evaluates several dynamic mesh strategies within the open-source CFD framework OpenFOAM, identifies their limitations, and proposes a flexible hybrid approach combining mesh deformation with periodic remeshing.

2 NUMERICAL FRAMEWORK

2.1 Governing Equations

The simulations are based on the incompressible Reynolds-Averaged Navier–Stokes (RANS) equations under the assumption of constant density within each phase, but allowing for spatially varying density across the mixture. The continuity equation ensures mass conservation of the mixture and can be expressed as:

$$\nabla \cdot U = 0 \tag{1}$$

where U is the velocity field of the mixture.

The momentum conservation equation, solved for the velocity field, takes the form (Badano, 2021):

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) - \nabla \cdot [\rho(\upsilon + \upsilon_t)(\nabla U + \nabla U^T)] = -\nabla p + \rho g \tag{2}$$

where p is the pressure, ρ the local mixture density, ν the kinematic viscosity, υ_t is the eddy viscosity, and g the gravitational acceleration. Mixture properties such as ρ and ν are determined through the local volume fraction of each constituent liquid, for example:

$$\rho = \alpha\, \rho_a + (1 - \alpha)\, \rho_b \tag{3}$$

$$\upsilon = \alpha\, \upsilon_a + (1-\alpha)\, \upsilon_b \tag{4}$$

where α is the volume fraction of liquid a.

The transport of the volume fraction is modeled by an advection–diffusion conservation equation for the α scalar field, which allows the solver to track the spatial and temporal evolution of the concentration field, providing the coupling between scalar transport and hydrodynamics:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha U) - \nabla \cdot [(D + D_t)\nabla \alpha] = 0 \tag{5}$$

where D is the molecular inter-phase diffusivity, and D_t is the eddy diffusivity, which is modelled as proportional to the eddy viscosity:

$$D_t = \frac{1}{Sc_t}\, \upsilon_t \tag{6}$$

with the proportionality constant being the inverse of the turbulent Schmidt number.

The combination of the mass, momentum and phase fraction conservation equations provides a framework capable of representing both the hydrodynamic field and the scalar mixing process between two liquid species, with buoyancy forces arising due to density differences.

Pressure–velocity coupling is handled through the PIMPLE algorithm, that merges the controls of SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) and PISO (Pressure-Implicit with Splitting of Operators), in particular the iterative loops and under-relaxation. Mixture properties are updated at each iteration from the evolving volume fraction field.

2.2 Custom Solver Development

OpenFOAM includes a built-in solver, called *twoLiquidMixingFoam*, that solves the governing equations described in the previous sections. However, this solver does not include dynamic mesh capabilities. Hence, a new solver, *panamaMixingDyMFoam*, was developed, extending the standard solver with the same dynamic mesh capabilities already available on other standard Volume-of-Fluid solvers, such as *interFoam*. This allows coupling of buoyancy-driven flows with moving meshes.

2.3 Turbulence modelling

Standard turbulence models often fail in stratified flows because buoyancy suppresses turbulence generation in stable layers. A buoyancy-modified version of the k–ω SST model, namely *kOmegaSSTBuoyancy* (Devolder, 2020), was implemented. This model introduces an additional buoyancy production/destruction term, enabling more accurate predictions of stratified mixing.

A validation study was carried out against a laboratory scale lock-exchange experiment (Kesong, 2019). Results showed that incorporating buoyancy effects significantly improved predictions of front propagation and salinity fields compared to the standard *kOmegaSST* model (Figure 1).

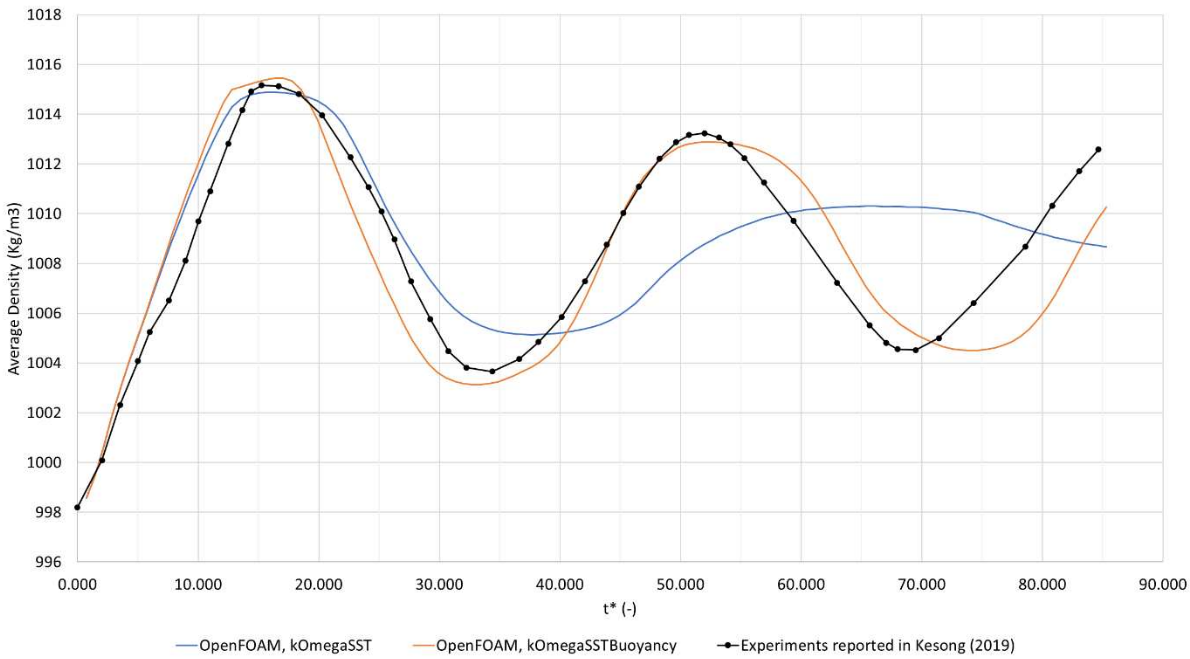

Figure 1: Laboratory scale validation run, simulating lock-exchange currents.

3 DYNAMIC MESH TECHNIQUES

Several techniques were tested in OpenFOAM to represent moving boundaries. Each has advantages and limitations.

3.1 Sliding regions with AMI Couplings

The first approach considered was the use of sliding regions with Arbitrary Mesh Interface (AMI) couplings. This strategy was tested for scenarios in which a ship entered the chamber, and it involved both sliding mesh regions and dynamic addition and removal of cell layers to accommodate the large displacements. An example setup for this case is presented in Figure 2.

While the method was able to represent vessel motion along a straight trajectory, the setup was cumbersome. Moreover, it could not be generalized to situations where the ship motion was not rectilinear, nor could it accommodate the rotational movement of miter gates or scenarios where vessel motion and gate opening occurred simultaneously

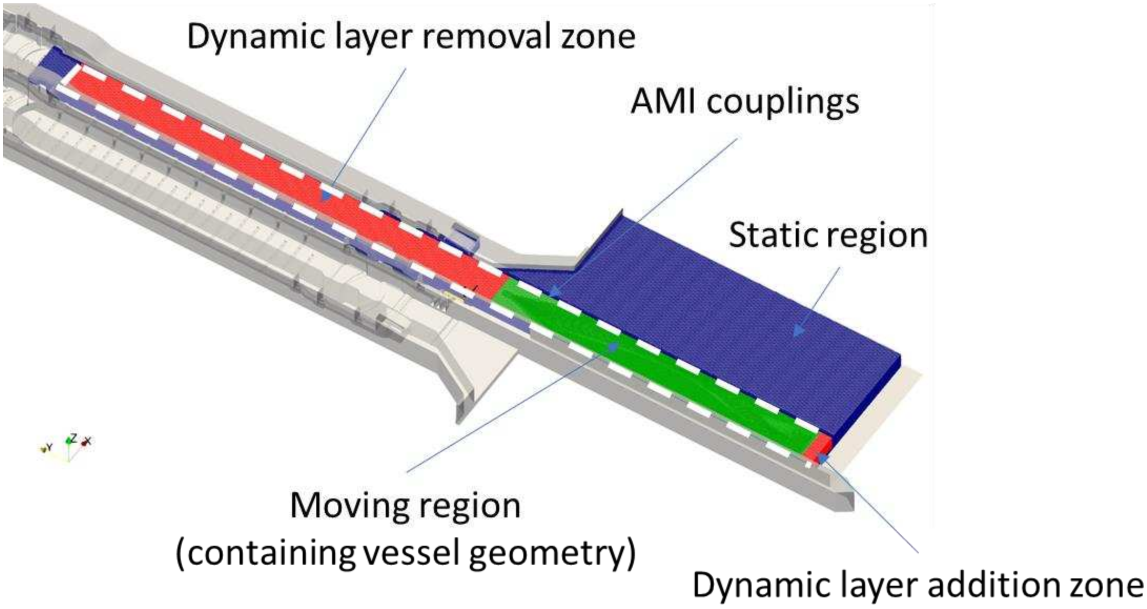

Figure 2: Sliding interface setup with AMI couplings.

3.2 Overset Mesh

A second strategy involved the use of overset meshes, in which multiple overlapping meshes were defined and interpolated. In principle, overset methods are particularly suitable for problems with complex moving geometries, as they allow for independent motion of embedded meshes.

However, in the present application, difficulties arose when moving meshes came into close contact with solid boundaries, as occurs naturally in gate–wall interactions (Figure 3). In these situations, continuity errors developed in the flow field, leading to non-physical solutions and loss of accuracy. This limitation reduced the applicability of overset meshes for the lock simulations.

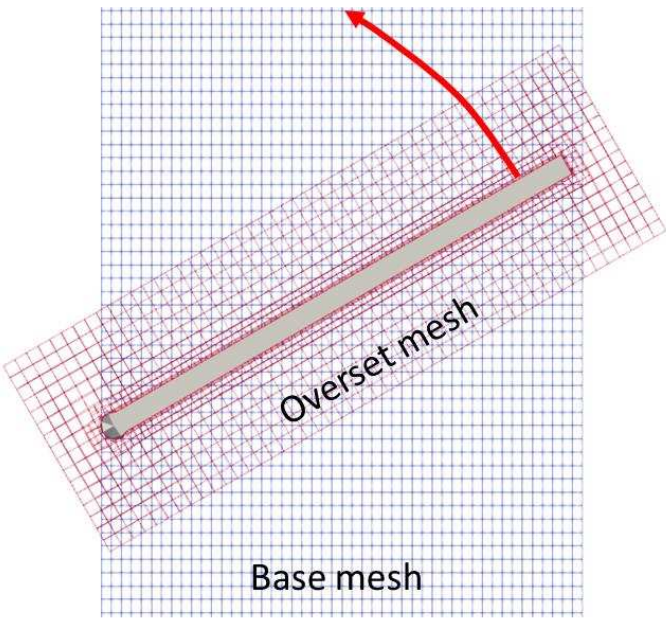

Figure 3: Overset mesh arrangement for miter gate.

3.3 Mesh Deformation

The use of mesh deformation was also considered. In this case, the computational mesh deformed continuously in response to boundary motion, which is propagated to the interior nodes by solving a diffusion equation.

For small displacements, the method works satisfactorily and is straightforward to implement. However, for larger motions, mesh quality rapidly degraded, producing distorted cells and ultimately causing solver instability. The method was therefore deemed unsuitable for realistic lock operations, which involve large-scale motions of both vessels and gates.

3.4 Node-per-Node Prescribed Motion

A more specialized technique was developed for the simulation of miter gates, in which a node-per-node prescribed motion function was used to deform the mesh according to the gate kinematics. This allowed accurate representation of gate rotation and produced high-quality meshes throughout the motion.

Nevertheless, the approach required significant implementation effort and was not flexible enough to accommodate additional simultaneous motions, such as vessel translation occurring during gate opening. Figure 4 presents a sequence of mesh configurations for gate opening using this prescribed-motion approach.

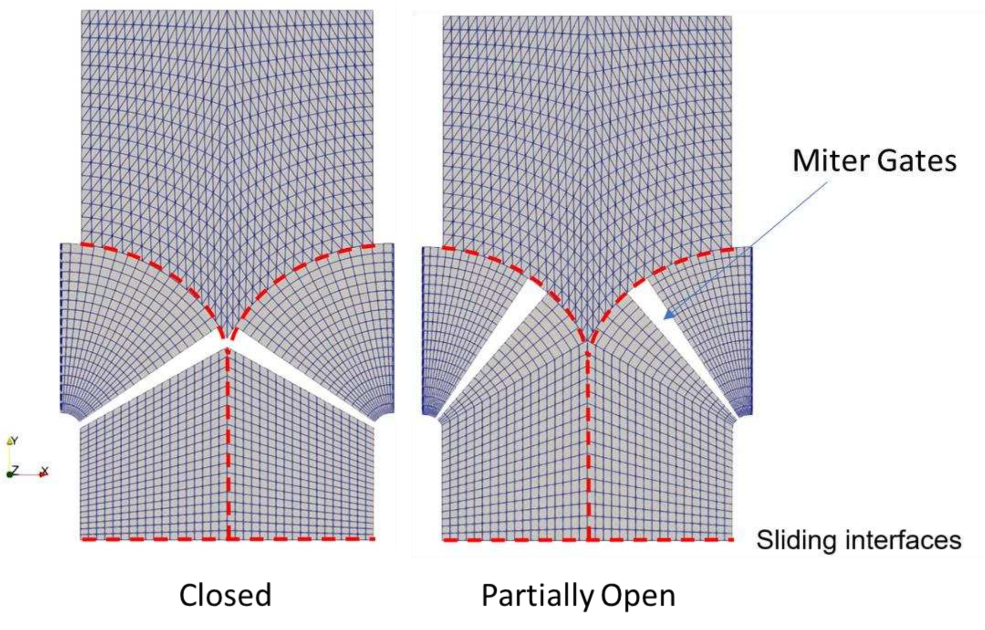

Figure 4: Sequence of mesh deformation for gate opening.

3.5 Flexible Hybrid Approach: Mesh Deformation with Periodic Remeshing

Given the limitations of the above methods, a more flexible strategy was proposed, combining mesh deformation with periodic remeshing.

In this hybrid approach, the simulation is divided into a certain number of stages, that typically last a few seconds each. Each of these stages constitutes an independent OpenFOAM run.

1. For each stage a new mesh is generated (in this case with *snappyHexMesh*, but any other automatic meshing tool could be also used).

2. During each stage, the current mesh is deformed following the motion specified on the boundaries (motion in this case was prescribed, but could also be computed with a rigid body solver on other cases).
3. Solution is mapped with *mapFields* from the current mesh to the next one after each stage.

The outer loop of this algorithm was implemented for this project as a python script, that runs as job within a cluster scheduler. An outline of this outer loop is schematized in pseudo-code in Figure 5. It is worth pointing out that for each stage of the simulation, the script creates a new run folder, based on a common run template. This template contains most of the setup for each individual case.

```
# For each stage 1 to n
for i = 1:n
    Duplicate run_template as run[i]
    Set initial geometry displacement for run[i]
    Generate mesh for run[i]
    Set mesh boundary velocities for run[i]
    if i > 1:
        Map Solution from run[i-1] to run[i]
    Run run[i], with mesh deformation
```

Figure 5: Outline of the mesh deformation with periodic remeshing technique, in pseudo-code.

It was found that this technique offers a few comparative advantages and disadvantages, namely:

- Advantages:
  - Handles arbitrarily complex geometry and motions.
  - Any mesh quality issue can be solved splitting the run in more stages (more meshes).
  - Once the general automation of the process has been developed, the setup of a simulation is relatively simple.
  - The complexity of the case setup is independent of the complexity of the motion.

- Disadvantages:
  - Increased computational cost due to the need for multiple mesh generations and mappings. For the application case described below, the generation/mapping process takes longer than the actual simulation time.
  - The solution mapping introduces interpolation errors that might accumulate behaving as additional diffusion. Since this effect is dependent on the mesh size, it can be assessed by doing mesh sensitivity tests, and eventually controlled by using finer meshes.

4 APPLICATION TO THE MIRAFLORES LOCKS

The hybrid dynamic mesh approach was then applied to the simulation of salt-water intrusion into the Miraflores Locks. For the operation presented in this paper, the computational domain included the entire lower lock chamber and part of the adjacent water body on the Pacific side (Figure 6).

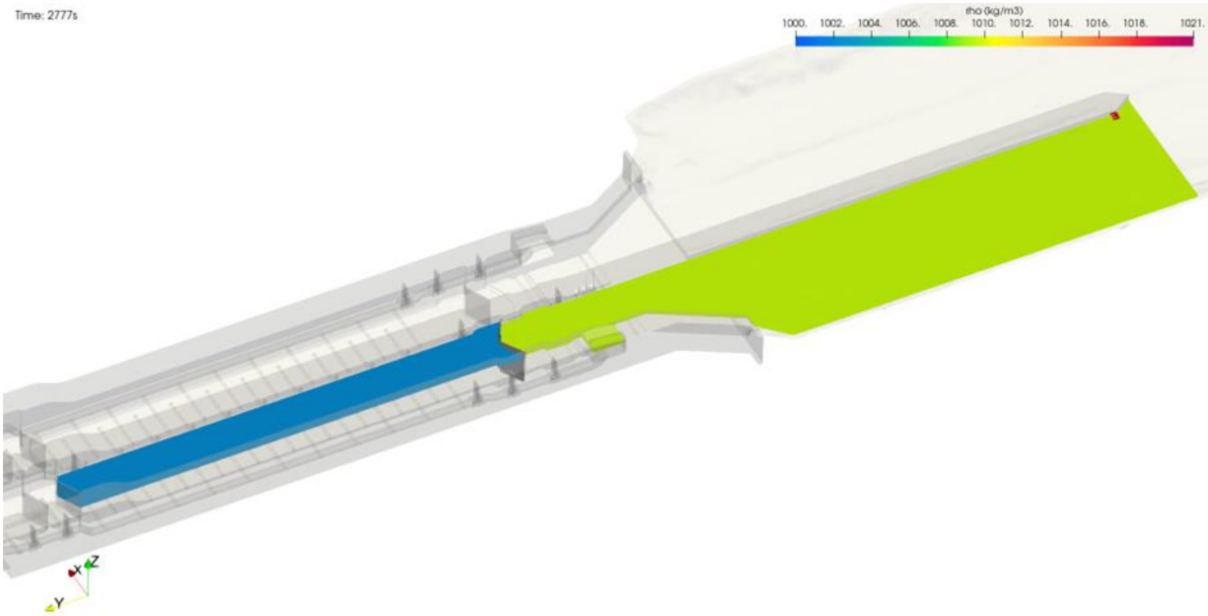

Figure 6: Computational domain of lock chamber and Pacific connection.

The initial salinity profile outside the lock was defined based on results from a far-field TELEMAC-3D model that extends toward the Pacific Ocean (OPTIMAL Consortium, 2025). Meanwhile, the initial salinity profile inside the chamber was defined based on in-situ measurements, performed by ACP. Fresh-water (0 PSU) and Salt-water (25 PSU) were used as phases, with any other concentration in between being represented as mixtures.

The opening and closing process of the miter gates were simulated using prescribed kinematics and the hybrid mesh method described earlier. The rigid-lid approximation was applied to the free surface, which is acceptable given that most of the relevant dynamics are density-driven and internal to the water column.

The simulations revealed the development of a bottom-propagating density current that advanced into the chamber after the gate opened. Simultaneously, a fresh water outflow develops on the topmost part of the water column. The salinity interface (called *pycnocline*) remains sharp during the process. Figure 7 illustrates the evolution of velocity and salinity fields on the midplane of the lock chamber during the simulation, while Figure 8 shows the deformation of the computational mesh around the miter gates at different stages.

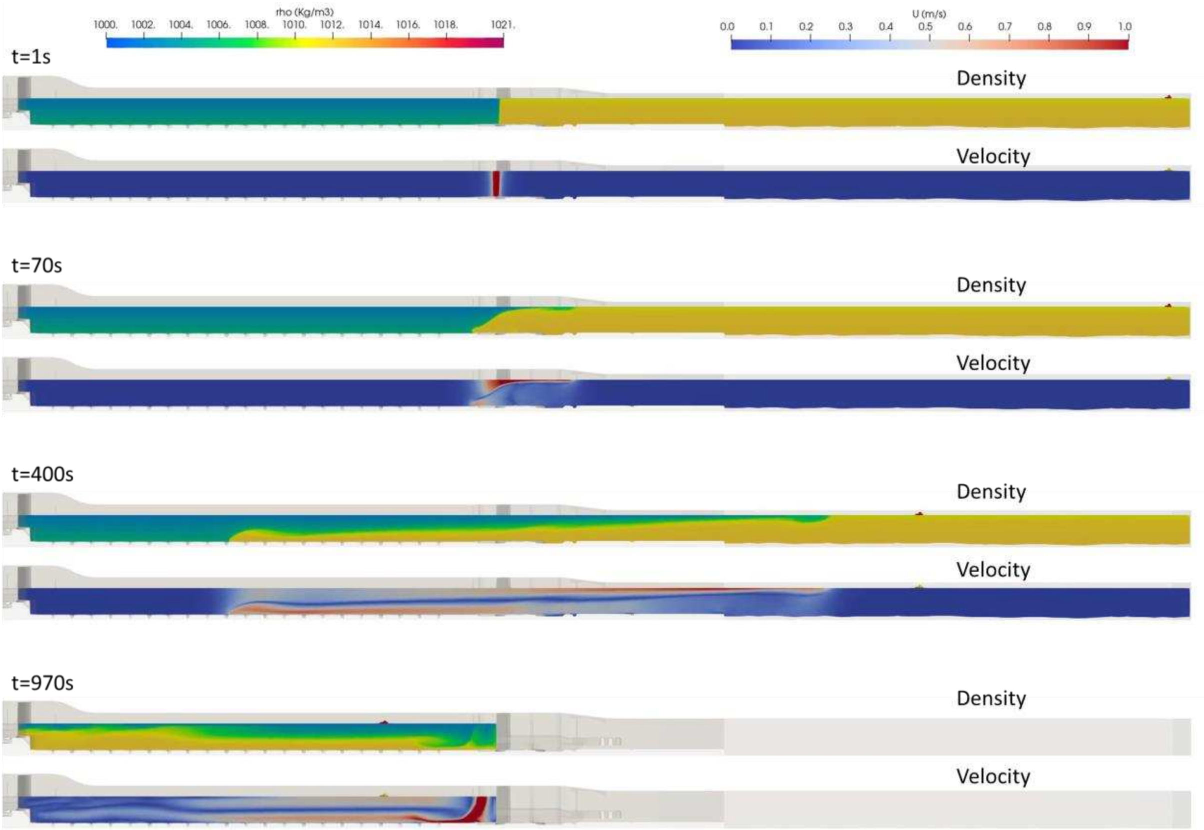

Figure 7: Evolution of velocity and salinity fields during the simulation.

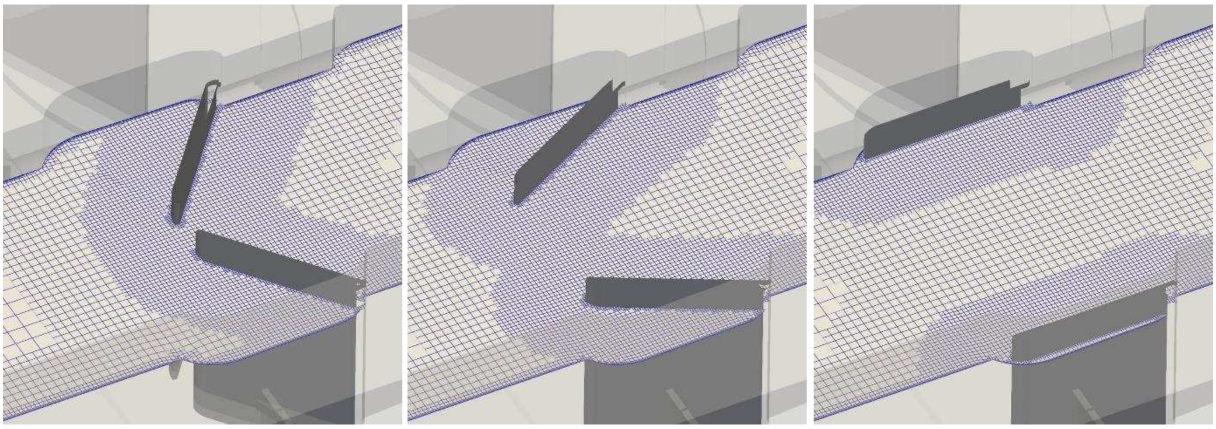

Figure 8: Deformation of the computational mesh at different stages.

The model results were validated against data from Acoustic Doppler Current Profiler (ADCP) surveys and Conductivity-Temperature-Depth (CTD) sensors deployed during sailing campaigns, performed by ACP. The agreement between simulations and measurements was satisfactory, particularly in terms of current velocities and salinity stratification. These comparisons demonstrated that the hybrid dynamic mesh method, combined with buoyancy-modified turbulence modeling, is capable of reproducing the complex hydrodynamic processes within the locks.

5 CONCLUSIONS

This study presented the application of dynamic mesh techniques for the CFD modeling of density currents in the Miraflores Locks of the Panama Canal. Several approaches were evaluated, including sliding interfaces, overset meshes, pure deformation, and node-prescribed motions. Each method offered particular advantages but also critical limitations. The most effective strategy proved to be a hybrid approach combining mesh deformation with periodic remeshing, which enabled accurate simulations of arbitrarily complex geometries and motions.

A custom solver, *panamaMixingDyMFoam*, was developed to combine two-phase miscible flow capabilities with dynamic mesh motion. The inclusion of buoyancy effects in the turbulence model was found essential for accurate representation of stratification. Validation against both laboratory experiments and field data confirmed the reliability of the approach.

The proposed methodology provides a robust framework for simulating density currents in lock chambers, offering improved predictive capability for salt-water intrusion and hydrodynamic forces on vessels.

REFERENCES

Badano, N.D. Estudio de efectos de escala en estructuras hidráulicas mediante modelación numérica. Doctor of Engineering. Thesis, University of Buenos Aires, 2021.

Devolder, B. Buoyancy-modified turbulence models library. GitLab repository: https://gitlab.com/brecht.devolder/buoyancyModifiedTurbulenceModels. 2020.

Kesong, L. Experimental study of lock-exchange gravity currents. PhD Thesis, Delft University of Technology, 2019.

OPTIMAL.OP1-P1-01 Far Field modelling report, 2025.

Rodríguez, J. The Panama Canal expansion: a strategic project for global trade. Maritime Policy & Management, 42(7), 647–660, 2015.

Simpson, J. E. Gravity Currents in the Environment and the Laboratory. Cambridge University Press, 1997.