2025

Computation of Hydrodynamic Coefficients for Submerged Spheres: A Comprehensive Analysis

PAULO JUSTINO

Computation of Hydrodynamic Coefficients for Submerged Spheres: A Comprehensive Analysis

Paulo Justino

LNEG, Março 2025

Report #1/2025

Computation of Hydrodynamic Coefficients for Submerged Spheres: A Comprehensive Analysis

1. Introduction: Defining Hydrodynamic Coefficients for Submerged Spheres

Hydrodynamic coefficients are essential parameters in fluid mechanics that serve to quantify the complex interaction between a submerged body and the surrounding fluid . Specifically in the context of submerged spheres, these coefficients provide a measure of the forces and moments exerted by the fluid on the sphere, or conversely, the resistance the sphere encounters as it moves through the fluid . Understanding and accurately determining these coefficients is paramount for a wide range of applications, including the design and control of underwater vehicles, the analysis of offshore structures, and the prediction of the behavior of marine organisms. The accurate prediction of wave loads on submerged structures also relies heavily on the evaluation of these coefficients.

The significance of hydrodynamic coefficients lies in their ability to simplify the complex physics of fluid-structure interaction into manageable parameters. For instance, the added mass coefficient reflects the inertia of the fluid that is accelerated along with the moving sphere, effectively increasing the sphere's apparent mass. This phenomenon is critical in dynamic analyses, as the inertial forces are directly influenced by this added mass. Similarly, the drag coefficient quantifies the resistance force that opposes the sphere's motion, a force primarily attributed to viscous effects and the pressure distribution around the body. In certain flow regimes, particularly involving non-uniform flow fields such as those induced by waves or in the presence of rotation, a lift coefficient may also become relevant, representing a force acting perpendicular to the direction of motion. The accurate determination of these coefficients, which can be linear or nonlinear depending on the complexity of the fluid-structure interaction, is a crucial step in the design and analysis of any system involving submerged spheres.

The interaction between a submerged sphere and the surrounding fluid exemplifies a fundamental fluid-structure interaction problem. When a solid object like a sphere is placed in the path of a fluid, the fluid exerts pressure and viscous forces on the sphere's surface, potentially leading to its motion or deformation. Conversely, the presence and motion of the sphere alter the flow field of the fluid. Hydrodynamic coefficients serve as the critical link in understanding this bidirectional influence, allowing engineers and researchers to predict the dynamic response of the sphere to fluid forces and the impact of the sphere's motion on the fluid environment. For example, in the context of underwater explosions, the structural response of a submerged body significantly affects the dynamics of the explosion bubble, highlighting the intricate nature of fluid-structure interaction where hydrodynamic coefficients play a vital role.

2. Analytical Methods: Potential Flow Theory

Potential flow theory provides a foundational analytical framework for understanding the hydrodynamic characteristics of submerged bodies, including spheres. This theory simplifies the complex Navier-Stokes equations by making several key assumptions about the fluid flow: it is assumed to be irrotational (meaning fluid particles do not rotate), inviscid (possessing no viscosity), and incompressible (having a constant density) . Under these assumptions, the fluid motion can be described by a scalar velocity potential, which satisfies Laplace's equation. This linear partial differential equation is amenable to analytical solutions for certain geometries, including the sphere.

The application of potential flow theory to submerged spheres often involves solving boundary value problems for Laplace's equation, subject to specific conditions on the sphere's surface, the free surface (if present), and the seabed (in cases of finite water depth) . Techniques such as multipole expansions are frequently employed to obtain analytical expressions for the velocity potential and subsequently derive hydrodynamic coefficients. For instance, when considering the diffraction of water waves by a submerged sphere, the total velocity potential can be decomposed into the sum of the incident wave potential and the diffraction potential, which represents the scattering of the wave by the sphere. By applying appropriate boundary conditions, such as the condition that the fluid velocity normal to the sphere's surface must be zero, analytical expressions for hydrodynamic coefficients like added mass and components of the exciting force (surge and heave) can be derived. These analytical solutions often involve series expansions using special functions like Legendre polynomials, which are particularly suited to spherical coordinate systems.

Despite its utility in providing fundamental insights and closed-form solutions for idealized scenarios, potential flow theory has inherent limitations stemming from its core assumptions. The neglect of viscosity means that potential flow cannot accurately predict drag forces arising from skin friction or account for energy dissipation due to viscous effects. This limitation also leads to D'Alembert's paradox, which states that the drag on a body moving through an infinite, inviscid fluid is zero, a clear contradiction with real-world observations. Furthermore, the assumption of irrotational flow precludes the theory from capturing phenomena associated with vorticity, such as the formation of boundary layers and wakes behind the sphere, which are crucial in determining hydrodynamic forces, especially at higher Reynolds numbers. While potential flow can sometimes predict complex phenomena like negative added mass due to free surface effects, the underlying mechanisms in real fluids may be influenced by viscous effects that the theory does not encompass. Even nonlinear potential flow solvers struggle to describe phenomena like wave breaking, which involves significant energy dissipation and free surface reconnection, again due to the inviscid and irrotational flow assumptions. In practical applications, potential flow solutions are sometimes augmented with empirical corrections to account for viscous effects, acknowledging the theory's limitations in fully representing real fluid behavior.

3. Numerical Methods: Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) offers a powerful alternative to analytical methods for determining hydrodynamic coefficients of submerged spheres, particularly in situations where the assumptions of potential flow theory are not valid or for complex flow regimes. CFD involves discretizing the flow domain into a computational mesh and numerically solving the governing Navier-Stokes equations (or their Reynolds-averaged or filtered forms for turbulent flows) using methods such as the finite volume method (FVM), finite difference method, or finite element method. The finite volume method, which is based on dividing the flow domain into a finite number of control volumes and applying conservation principles to each volume, is a widely used approach in CFD for hydrodynamic analysis. The use of unstructured grids in conjunction with FVM provides flexibility in handling the spherical geometry and allows for local mesh refinement in critical flow regions.

Setting up a CFD simulation for a submerged sphere requires careful consideration of the computational domain, the mesh, and the boundary conditions. The domain should be sufficiently large to minimize the influence of the outer boundaries on the flow around the sphere. For symmetrical flows, symmetry boundary conditions can be employed to reduce the computational domain and hence the computational cost. Generating an appropriate mesh is crucial for the accuracy of the simulation. For flows around a sphere, mesh refinement is often necessary in the vicinity of the sphere's surface to resolve the boundary layer and in the wake region behind the sphere to capture flow separation and vortex shedding. Prism layers, a type of structured mesh near walls, are often used to accurately model the steep velocity gradients within the boundary layer. Appropriate boundary conditions must be applied at the domain boundaries and on the sphere's surface. Typical boundary conditions include specifying the inlet velocity, outlet pressure, and applying a no-slip condition on the sphere's surface to ensure that the fluid velocity matches the velocity of the sphere.

For many practical applications involving submerged spheres, especially at higher Reynolds numbers, the flow becomes turbulent. In such cases, turbulence models must be incorporated into the CFD simulations to account for the effects of turbulence on the flow field and the resulting hydrodynamic coefficients. Various turbulence modeling approaches exist, ranging from Reynolds-Averaged Navier-Stokes (RANS) models like k-epsilon and k-omega SST, to more computationally intensive Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) models. RANS models are based on time-averaging the Navier-Stokes equations and introduce additional transport equations for turbulence quantities. LES models resolve the large-scale turbulent eddies directly and model only the smaller scales, while DES models are hybrid approaches that combine RANS and LES in different regions of the flow. For flow around a sphere at high Reynolds numbers, studies have shown that LES and DES models generally provide better agreement with experimental data compared to RANS models, particularly in capturing unsteady flow features and predicting drag coefficients. The choice of turbulence model depends on the specific Reynolds number range, the computational resources available, and the level of accuracy required for the hydrodynamic coefficients. Commercial CFD software packages like ANSYS Fluent, ANSYS CFX, and SimScale offer a range of turbulence models and solvers suitable for hydrodynamic analysis. It is important to note that using a turbulence model for a laminar flow simulation can lead to inaccurate results.

4. Experimental Techniques for Determining Hydrodynamic Coefficients

Experimental techniques provide crucial real-world data for determining hydrodynamic coefficients of submerged spheres and for validating the results obtained from analytical and numerical methods. Several experimental approaches are commonly employed, including towing tank tests and free decay tests.

Towing tank tests involve moving a model of the submerged sphere through a body of still water at a controlled velocity. The forces acting on the sphere, such as drag and lift, are measured using force balances attached to the model. By varying the towing speed, the hydrodynamic coefficients can be determined as a function of the flow velocity, and consequently, the Reynolds number. Specialized towing tank setups can also be used to investigate specific aspects of the hydrodynamic interaction, such as the measurement of wave drag on submerged spheres near the free surface. Planar Motion Mechanism (PMM) tests and Circular Water Channel (CWC) tests, while often used for more complex underwater vehicles like ROVs, can also be adapted for spheres to measure hydrodynamic forces and moments under controlled translational and rotational motions.

Free decay tests offer an alternative, often simpler, experimental method for estimating hydrodynamic coefficients, particularly added mass and damping. In a free decay test, the submerged sphere is displaced from its equilibrium position (e.g., by giving it an initial velocity or angular displacement) and then released to move freely under the influence of fluid forces. The time history of the sphere's motion (displacement, velocity, acceleration) is recorded using sensors such as accelerometers or Inertial Measurement Units (IMUs) . By analyzing the damped oscillations or the decay of the motion, parameters like the natural frequency and damping ratio can be extracted, which are directly related to the added mass and damping coefficients. A variation of this technique is the free decay pendulum test, where the submerged sphere is attached to a pendulum and allowed to oscillate. The analysis of the pendulum's motion provides estimates for hydrodynamic coefficients associated with the sphere's movement. These tests are particularly useful for scaled models and can serve as a cost-effective alternative when large-scale facilities are not available .

Other experimental techniques, such as forced oscillation tests, where the sphere is oscillated at a known frequency and amplitude, and the resulting forces are measured, can also be used to determine hydrodynamic coefficients, especially those related to oscillatory flows. Techniques like Particle Image Velocimetry (PIV) can provide detailed measurements of the flow field around the sphere, allowing for the inference of hydrodynamic forces and a deeper understanding of the fluid-structure interaction.

5. Key Hydrodynamic Coefficients Relevant to Submerged Spheres

For submerged spheres, several key hydrodynamic coefficients are particularly relevant for characterizing their interaction with the surrounding fluid. These include the added mass coefficient, the drag coefficient, and, under certain conditions, the lift coefficient.

The added mass coefficient (often denoted as C_m or C_A) quantifies the amount of fluid that effectively moves with the accelerating sphere. It represents the ratio of the added mass to the mass of the fluid displaced by the sphere. Added mass arises because when a submerged body accelerates, it must also accelerate some of the surrounding fluid. This results in an effective increase in the inertia of the body. For a sphere accelerating in an unbounded, inviscid fluid, the theoretical added mass coefficient is 0.5. However, this value can be significantly affected by factors such as the proximity to boundaries, the free surface, and the presence of waves. Near the free surface, complex interactions with surface waves can even lead to the phenomenon of negative added mass under certain frequency conditions. Added mass is a tensorial quantity in general, reflecting the directional dependence of the fluid inertia, although for a sphere, it simplifies due to symmetry. Accurate knowledge of the added mass coefficient is crucial for predicting the dynamic response of submerged spheres in various applications, including the design of marine vehicles and offshore structures.

The drag coefficient (C_d) is a dimensionless parameter that represents the resistance to motion experienced by the sphere due to the fluid. It is defined as the ratio of the drag force to the dynamic pressure multiplied by a reference area (typically the frontal projected area of the sphere) . The drag force on a submerged sphere originates from two primary sources: skin friction due to the viscosity of the fluid acting on the sphere's surface, and form drag, which is related to the pressure difference between the front and rear of the sphere caused by flow separation. The drag coefficient is strongly dependent on the Reynolds number, which characterizes the ratio of inertial to viscous forces in the flow. For a sphere, the drag coefficient exhibits a well-known dependence on the Reynolds number, including a significant drop in C_d around a critical Reynolds number of approximately 2.5 * 10^5, a phenomenon known as the drag crisis, which is associated with the transition to a turbulent boundary layer. For submerged spheres near the free surface, an additional component of drag, known as wave drag, can become significant due to the energy radiated away in the form of surface waves. The drag coefficient is essential for calculating the power required to move a submerged sphere at a certain velocity and for predicting its trajectory under the influence of fluid resistance.

The lift coefficient (C_l) represents the force acting perpendicular to the direction of motion. For a perfectly smooth sphere in a uniform, steady flow, the lift coefficient is ideally zero due to the symmetry of the flow around the sphere. However, lift forces can arise if there is an asymmetry in the flow field. This asymmetry can be induced by factors such as the rotation of the sphere (Magnus effect), the presence of shear flow, or the interaction with waves. For instance, a submerged sphere advancing in a regular water wave can experience a net lift force due to the non-uniform pressure distribution caused by the wave motion. While the lift coefficient is often less significant than the drag coefficient for non-rotating spheres in simple flows, it can play a crucial role in specific scenarios, such as the motion of spheres in complex wave environments or in the context of fluid-structure interaction with other bodies.

6. Analyze How Various Parameters Influence Hydrodynamic Coefficients

The hydrodynamic coefficients of submerged spheres are not constant values but are significantly influenced by various parameters related to the flow conditions, the sphere's properties, and the surrounding environment.

The Reynolds number (Re = rho U D / mu), where rho is the fluid density, U is the flow velocity, D is the sphere diameter, and mu is the dynamic viscosity, is a primary parameter that governs the flow regime around the sphere and has a profound impact on the drag coefficient. As the Reynolds number increases, the flow transitions from laminar (at low Re) to turbulent (at high Re), leading to significant changes in the boundary layer development, flow separation, and wake formation, which in turn affect the drag coefficient. The well-documented drag crisis for a sphere occurs at a Reynolds number around 2.5 * 10^5, where the drag coefficient suddenly drops due to the transition to a turbulent boundary layer that delays flow separation. The Reynolds number also influences other hydrodynamic coefficients, as it dictates the relative importance of inertial and viscous forces in the fluid. Studies have been conducted over a wide range of Reynolds numbers, from 10^3 to 10^6 and beyond, to characterize the behavior of hydrodynamic coefficients for submerged spheres in different flow regimes. In flows involving a free surface, the Froude number (Fr = U / sqrt{gD}), which represents the ratio of inertial forces to gravitational forces, also becomes important, particularly for floating or shallowly submerged spheres, as it influences the wave patterns generated by the sphere's motion and hence the wave drag .

In oscillatory flow regimes, such as those encountered in waves, the Keulegan-Carpenter number (KC = U_m T / D), where U_m is the maximum velocity of the oscillation and T is the period of oscillation, becomes a relevant dimensionless parameter. The Keulegan-

Carpenter number represents the ratio of the distance a fluid particle travels during one oscillation period to the characteristic length scale of the body. It is particularly important in determining the relative contributions of drag and inertia forces in oscillatory flows around submerged structures. While the provided snippets do not explicitly detail the influence of the Keulegan-Carpenter number on the hydrodynamic coefficients of submerged spheres, it is generally known that at low KC numbers, inertia forces dominate, while at high KC numbers, drag forces become more significant.

The surface roughness of the sphere can also influence the hydrodynamic coefficients, particularly the drag coefficient, especially in turbulent flow regimes. A rougher surface can promote earlier transition of the boundary layer from laminar to turbulent, which can affect the point of flow separation and the size of the wake, leading to an increase in drag compared to a smooth sphere at the same Reynolds number. Experiments often specify the surface finish of the sphere to ensure consistency and to allow for comparisons with other studies.

The proximity of the submerged sphere to boundaries, such as the seabed, walls, or the free surface, can significantly alter the flow field around the sphere and consequently affect its hydrodynamic coefficients. For instance, the added mass of a sphere generally increases as it gets closer to a boundary because the presence of the boundary restricts the flow of fluid around the sphere. Similarly, the free surface introduces complex interactions with surface waves, which can lead to variations in added mass, damping, and the emergence of wave drag, especially when the sphere is shallowly submerged. The submergence depth of the sphere relative to the free surface is a critical parameter influencing these effects.

7. Investigate Different Models and Empirical Formulas

Given the complexity of fluid flow around a submerged sphere across various Reynolds number regimes and environmental conditions, numerous models and empirical formulas have been developed to estimate the hydrodynamic coefficients. These models range from purely analytical solutions based on simplified theories to fully empirical formulas derived from experimental data.

For very low Reynolds number flows (Re << 1), known as Stokes flow, analytical solutions can be obtained for the drag coefficient. Stokes' law gives the drag force F_d = 6 pi mu R U, where R is the sphere radius, U is the velocity, and \mu is the dynamic viscosity. This leads to a drag coefficient of C_d = 24 / Re. As the Reynolds number increases, corrections to Stokes' law have been developed to account for inertial effects.

In the moderate Reynolds number range where potential flow theory might be applicable as a first approximation, the added mass coefficient for a sphere in an unbounded fluid is theoretically 0.5. However, as discussed earlier, this value can change significantly in the presence of boundaries or a free surface. For drag in this regime, empirical formulas based on experimental data are often used, as potential flow theory itself predicts zero drag. These formulas typically express the drag coefficient as a function of the Reynolds number.

For high Reynolds number flows (Re > 10^3), where turbulence becomes dominant, empirical formulas for the drag coefficient are widely used in engineering practice. These formulas are often based on fitting curves to experimental data obtained from towing tank tests and wind tunnel experiments. A well-known example is the standard drag curve for a sphere, which shows the variation of C_d with Re across several orders of magnitude, including the drag crisis around Re = 2.5 * 10^5 where C_d drops from about 0.5 to 0.1. The specific value of the drag coefficient in this regime can also be influenced by surface roughness.

Semi-empirical models also exist, which combine theoretical frameworks with empirical data or correction factors. For example, the Morison equation, commonly used in offshore engineering to estimate wave forces on structures, includes a drag term with a drag coefficient (C_d) and an inertia term with an inertia coefficient (C_m), both of which are often determined empirically based on the Reynolds number and the Keulegan-Carpenter number. Similarly, a semiempirical relation for the dipping of a floating sphere as a function of the Froude number has been derived based on a potential flow approximation for the downward force .

The applicability and accuracy of empirical formulas are inherently tied to the range of parameters for which they were developed and the quality of the experimental data used to derive them . While these formulas provide practical tools for estimating hydrodynamic coefficients, their accuracy may be limited when extrapolated to conditions outside their validated range or for scenarios not well represented in the original experiments. Therefore, it is crucial to select the appropriate model or empirical formula based on the specific flow regime, the geometry of the sphere, and the environmental conditions of interest.

8. Compare and Contrast Results from Analytical, Numerical, and Experimental Methods

Analytical, numerical (CFD), and experimental methods each offer unique strengths and weaknesses in determining the hydrodynamic coefficients of submerged spheres . The choice of method depends on the specific requirements of the application, including the desired accuracy, the complexity of the flow and geometry, and the available resources.

Analytical methods, primarily based on potential flow theory, provide fundamental insights into the underlying physics and can yield exact or closed-form solutions for idealized scenarios involving simple geometries like spheres. They are computationally inexpensive and can be valuable for preliminary design and theoretical studies. However, their reliance on simplifying assumptions such as inviscid and irrotational flow limits their applicability to real-world scenarios, especially at higher Reynolds numbers where viscous and turbulent effects become significant. Analytical methods often struggle to accurately predict drag and damping coefficients, which are strongly influenced by viscosity.

Numerical methods, particularly CFD, offer the capability to handle complex geometries and boundary conditions and can model viscous and turbulent flows with the appropriate selection of turbulence models. CFD simulations can provide detailed information about the flow field around the sphere, allowing for a deeper understanding of the fluid-structure interaction. However, CFD simulations can be computationally expensive, and the accuracy of the results depends heavily on the quality of the computational mesh and the appropriateness of the chosen turbulence model. Setting up and validating CFD simulations requires expertise, and the results often need to be validated against experimental data to ensure their reliability.

Experimental methods, such as towing tank tests and free decay tests, provide real-world data that can capture complex physical phenomena that are difficult to model analytically or numerically. Experimental data are essential for validating numerical models and for developing empirical formulas. However, experiments can be expensive and time-consuming to conduct, and controlling all the relevant parameters precisely can be challenging. Measurements may also be subject to errors, and the amount of detailed flow field information obtained from experiments is often limited compared to CFD simulations.

In many cases, a combination of these approaches is most effective. Analytical methods can provide initial estimates and theoretical understanding, while CFD simulations can offer more detailed insights and handle more complex scenarios. Experimental studies are crucial for validating both analytical and numerical results and for providing data for empirical models. For example, a study on the added mass coefficients of an underwater vehicle showed good agreement between results obtained from a suggested CFD method and those from analytical and experimental methods. This highlights the importance of cross-validation between different approaches to ensure the accuracy and reliability of the determined hydrodynamic coefficients.

The choice of method ultimately depends on the specific application and the trade-offs between accuracy, complexity, cost, and the type of hydrodynamic coefficients required. For instance, in early design stages or for simple flow conditions, analytical methods might suffice. For detailed analysis of complex flows or geometries, CFD simulations are often necessary. And for obtaining real-world data and validating models, experimental techniques are indispensable.

Table with:

Method	Strengths	Weaknesses	Typical Applications
Analytical Methods (Potential Flow)	Provide fundamental insights, yield exact solutions for simple geometries, computationally inexpensive	Rely on simplifying assumptions (inviscid, irrotational), limited applicability to complex flows and high Reynolds numbers, may not accurately predict drag and damping	Preliminary design, theoretical studies, low Reynolds number flows
Numerical Methods (CFD)	Handle complex geometries and boundary conditions, model viscous and turbulent flows, provide detailed flow field information	Computationally expensive, accuracy depends on mesh and turbulence model, requires expertise, results may need experimental validation	Complex geometries, high Reynolds number flows, detailed flow analysis, design optimization
Experimental Methods	Provide real-world data, capture complex phenomena, validate numerical models	Expensive and time-consuming, challenging to control parameters, measurements subject to errors, limited flow field information	Validation of numerical models, development of empirical formulas, complex flow regimes

9. Conclusions

The computation of hydrodynamic coefficients for submerged spheres is a multifaceted problem that can be addressed through analytical, numerical, and experimental methods. Each approach offers distinct advantages and limitations, making the choice of method dependent on the specific requirements of the application. Analytical methods based on potential flow theory provide a foundational understanding but are limited by their simplifying assumptions. Numerical methods, particularly CFD, offer a powerful tool for analyzing complex flows and geometries but require careful setup and validation. Experimental techniques provide essential real-world data for validation and for capturing phenomena that are difficult to model theoretically. The accurate determination of hydrodynamic coefficients such as added mass, drag, and lift is crucial for the design and analysis of various systems involving submerged spheres, ranging from underwater robots to offshore structures. Future advancements in computational power and experimental techniques will continue to enhance our ability to predict and understand the intricate hydrodynamic interactions of submerged spheres across diverse flow regimes and environmental conditions.

References

Cambridge Press

MDPI Press

WitPress

Ansys©

Wamit©