XLVI CILAMCE
VITÓRIA - ES - BRAZIL - 2025
XLVI IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGENEERING

The discretization error analysis for simulation of transient flow in a water reservoir

Sara C. Silva[1], Ana Paula S. Vargas[2], Daiane Zanatta Author[3], Marcio A. V. Pinto[4]

[1]*Dept. of Mathematics - DAMAT, Technological Federal University of Parana - UTFPR*
Rosalina Maria dos Santos St, 1233, 87301-899, Campo Mourão/Paraná, Brazil
sarasilva@utfpr.edu.br
[2]*Dept. of Mathematics - DAMAT, Technological Federal University of Parana - UTFPR*
Marcílio Dias St, 635, 86812-460, Apucarana/Paraná, Brazil
anavargas@utfpr.edu.br
[3]*Department of Mathematics, State University of Centro-Oeste - UNICENTRO*
Professora Maria Roza Zanon de Almeida St, 84505-677, Irati/Paraná, Brazil
daiane@unicentro.br
[4]*Department of Mechanical Engineering, Federal University of Parana - UFPR*
Cel. Francisco H. dos Santos Ave, 100, 81530-000, Curitiba/Paraná, Brazil
marcio_ villela@ufpr.br

Abstract. The study of hydrodynamic and transport models that focus on their behavior and on examining the results obtained with numerical and computational simulations seeks to ensure the best performance in the analysis and decision-making process for real problems. Obtaining accurate and reliable numerical solutions from models that involve interaction between advective and diffusive processes is a matter of great relevance due to the difficulties commonly experienced in solving these models. This work presents a simulation of a shallow water hydrodynamic model by means of a numerical solution that uses the finite volume method (FVM) with a hybrid approximation scheme (central/upwind) for discretizing the advective-diffusive terms. A priori and a posteriori error analysis were performed to estimate discretization errors. The implemented code allows a temporal analysis of the level, volume, and flow in reservoirs that have devices at their outlets. An adequate graphical and numerical interpretation of the physical phenomenon studied was obtained with a high level of accuracy and a stable solution with low computational cost.

Keywords: Shallow water equations, Finite volume, Implicit scheme, Discretization error estimators, Apparent order

1 Introduction

In this article, the effectiveness of a computational model developed to simulate transient 2D water flow in a reservoir with slope and friction at the bottom is analyzed. The model utilizes a comprehensive mathematical formulation of the shallow water equations, as used in Mahmood and Yevjevich [1] and Silva et al. [2]. Shallow water equations represent a simplification of the Navier-Stokes equations and are applicable to flows in which vertical stresses and accelerations can be neglected without significant loss of accuracy. These equations are derived by integrating the governing equations (Navier-Stokes and Continuity) in the vertical direction.

Obtaining an analytical solution for the shallow water equations system is a complex task, as the mathematical model is described by a set of nonlinear hyperbolic partial differential equations. Valiani et al. [3] used experimental data to verify the accuracy, stability, and reliability of a numerical solution of the shallow water equations obtained through a finite volume scheme. This numerical model achieved second-order accuracy in both space and time but at a high computational cost.

An unstructured finite volume scheme for triangular grids to simulate unstable 2D shallow water flows over complex topography is presented in Nikolos and Delis [4]. In this work, the explicit Runge-Kutta temporal method and unstructured grids are used. The efficiency of an implicit scheme is explored in Fernández-Pato et al. [5] for 2D shallow water fluid simulation, in which authors adopted a complex treatment for spatial discretization, utilizing a flexible unstructured mesh to achieve a better fit for irregular geometries. However, reservations in Nikolos and

Delis [4] and Fernández-Pato et al. [5] are made regarding the computational time required to obtain the solutions.

Silva et al. [2] presented a hybrid and fully implicit finite volume scheme for discretizing the shallow water equations to obtain a stable numerical solution for simulating transient 2D flows over slopes and subject to flow resistance. They proposed an optimized iterative algorithm with fast convergence and low computational cost. However, no analysis of the error of the obtained numerical solution is provided.

In another study Behzadi and Newman [6], a numerical model based on a semi-discrete Streamline Upwind Petrov-Galerkin scheme was presented for solving the 2D shallow water equations, considering arbitrary bed topography and wetting-drying fronts. However, Behzadi and Newman [6] did not perform error analysis or verify the order of the discretization error.

One of the greatest challenges in Computational Fluid Dynamics (CFD) is achieving a high level of accuracy in numerical solutions. Among the various sources of numerical error, the most significant one is typically the discretization error (Eh) (Marchi and Silva [7]). To mitigate this error, estimators were proposed in Marchi et al. [8] and applied to problems whit 1D and 2D equations.

In general, reducing the discretization error is closely related to refining the grid size. In other words, as the grid becomes more refined, the discretization error decreases. However, highly refined grids result in large-scale linear systems that require significant computational effort to obtain a numerical solution da Silva et al. [9].

The primary objective of our study is to achieve accurate results. According to Marchi et al. [10], this can be accomplished by using the finest grid ever employed to solve the problem, applying Richardson extrapolation to reduce the discretization error, verifying the order of accuracy, and employing an iterative process until the machine round-off error is reached. In this work, we have chosen to verify the order of accuracy.

In this context, the aim of this work is to ensure the effectiveness of the scheme presented in Silva et al. [2], by conducting an analysis of the error in the obtained numerical solution to achieve accurate results.

The structure of this work is as follows: Section 2 presents the mathematical and numerical models adopted for the 2D shallow water cases. Section 3 introduces the concepts of numerical errors. Section 4 presents the simulation results. Finally, in Section 5, we provide the conclusions.

2 Mathematical and numerical models

2.1 Mathematical model

In the analysis of transient $2D$ flow in a channel, where the bottom topography is described by a surface $z_0(x, y)$, the objective is to determine the fluid level on the free surface $h(x, y)$ and the flow velocity $U(u, v)$ at each time instant t.

The system of equations governing transient 2D flow, considering friction and slope on the bottom surface, is presented in Mahmood and Yevjevich [1] as follows:

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(uh) + \frac{\partial}{\partial y}(vh) = 0, \tag{1}$$

$$\frac{\partial}{\partial t}(uh) + \frac{\partial}{\partial x}(u^2h + \frac{1}{2}gh^2) + \frac{\partial}{\partial y}(uvh) = gh(S_{0x} - S_{fx}), \tag{2}$$

$$\frac{\partial}{\partial t}(vh) + \frac{\partial}{\partial x}(uvh) + \frac{\partial}{\partial y}(v^2h + \frac{1}{2}gh^2) = gh(S_{0y} - S_{fy}), \tag{3}$$

where $h(m)$ is the depth of the channel, $u(m/s)$ and $v(m/s)$ are the velocity components, $g(m/s^2)$ is the acceleration due to gravity, $t(s)$ is the time, z_0 is the elevation of the bottom surface, and S_{0x} and S_{0y} are the slopes of this surface defined as:

$$S_{0x} = -\frac{\partial z_0}{\partial x}, \qquad S_{0y} = -\frac{\partial z_0}{\partial y}, \tag{4}$$

while the slopes of flow resistance (energy loss)in the x and y directions, respectively, are given by:

$$S_{fx} = \frac{\eta^2 u\sqrt{u^2+v^2}}{h^{\frac{4}{3}}}, \qquad S_{fy} = \frac{\eta^2 v\sqrt{u^2+v^2}}{h^{\frac{4}{3}}}, \tag{5}$$

where $\eta(m/s^{\frac{1}{3}})$ is the Manning's roughness coefficient, which quantifies the flow friction with the bottom surface.

2.2 Discretization and linearization

The numerical solution of the hydrodynamic model described by eq. (1) - eq. (3) is obtained using the finite volume method (FVM), presented in Hirsch [11].

To obtain the numerical solution of the system of eqs. (1) - (3), we use the integral formulation of the shallow water equations over the rectangular generic control volume (CV) centered at P.

In this article, we adopt the fully implicit temporal formulation. Although its implementation is more complex, the obtained solution is unconditionally stable, in according with Burden and Faires [12].

In spatial integration, we use the Gauss divergence theorem followed by the fundamental theorem of calculus Kreyszig [13] to rewrite the model as a system of equations constituted by the values of the variables at the west (w), east (e), south (s), and north (n) faces, which are obtained using interpolation functions. For calculating these values, we adopt a combination of central differencing scheme (CDS) and upwind differencing scheme (UDS) interpolations. In the combination of these schemes, CDS and UDS, we adopt UDS for the main variable of the equation and CDS for the secondary variables (known values).

Adopting a fully implicit formulation in the temporal integration and the combination of CDS and UDS schemes described earlier, the discretization obtained for the model can be written in a general form as:

$$A\phi_P = b_{\phi_P}, \tag{6}$$

with coefficients calculated in detail in Silva et al. [2] for the variables h, u and v.

3 Numerical errors

One of the greatest challenges in Computational Fluid Dynamics (CFD) is related to achieving a high level of accuracy in numerical solutions. The numerical error (E) for a specific variable of interest is defined as the difference between the exact analytical solution Φ and its numerical approximation ϕ. This error can arise from various sources, including truncation errors (E_T), iteration errors (E_I), and rounding errors (E_π) as described in Rodrigues et al. [14]. While it is impossible to completely eliminate numerical errors, they should be minimized as much as possible. When the errors (E_I) and (E_π) are minimized or negligible, the remaining error (E_T) is referred to as discretization error (Eh).

When (Eh) is the only source of numerical error, it can be represented using the Taylor series as follows:

$$Eh = E(\phi) = c_0 h^{p_0} + c_1 h^{p_1} + c_2 h^{p_2} + \ldots = \sum_{V=0}^{\infty} c_V h^{p_V}, \tag{7}$$

where $c_0, c_1, c_2, \ldots$ are real numbers that can be functions of the dependent variable and its derivatives, but are independent of the grid spacing h. The exponents $p_0, p_1, p_2, \ldots$ represent the true orders of $E(\phi)$. This set is denoted by $p_V = \{p_0, p_1, p_2, \ldots\}$ and consists of positive integers that typically follow the relation $1 \leq p_0 < p_1 < p_2 < \ldots$, forming an arithmetic progression. The first element of p_V, p_0, is referred to as the asymptotic order of $E(\phi)$.

The asymptotic order of accuracy for each numerical method can be approximated using the apparent orders pU Marchi and Silva [7]. The analysis of p_0 a posteriori of the numerical solution, based on the calculation of pU, is performed when the analytical solution is unknown. The expression for pU, described in Marchi and Silva [7], is given by:

$$pU = \frac{log\left(\dfrac{\phi_2 - \phi_1}{\phi_3 - \phi_2}\right)}{log(r)}, \tag{8}$$

where ϕ_1, ϕ_2, and ϕ_3 represent the numerical solutions for the variable of interest on the grids Ω^{h_1} (coarse), Ω^{h_2} (fine), and Ω^{h_3} (superfine), respectively. The refinement ratio $r = \dfrac{h_1}{h_2} = \dfrac{h_2}{h_3}$, is the ratio of grid refinement between the grids.

When the analytical solution Φ is unknown, it is not possible to directly calculate the discretization error. Instead, the concept of uncertainty (U) is used. The uncertainty of a numerical solution is determined by the difference between the estimated analytical solution (ϕ_∞) for the variable of interest and its numerical approximation (ϕ) Marchi and Silva [7]:

$$U(\phi) = \phi_\infty - \phi. \tag{9}$$

To estimate the discretization error, the Richardson's error estimator and the Grid Convergence Index (GCI) based on the apparent order are commonly used and widely reported in the literature Marchi et al. [10]. The Richardson's error estimator is given by:

$$U_{Ri}(pU) = \frac{\phi_1 - \phi_2}{q^{pU} - 1}. \tag{10}$$

The GCI estimator is given by:

$$GCI(pU) = S_f \frac{\phi_1 - \phi_2}{q^{pU} - 1}, \tag{11}$$

where S_f is the safety factor typically set to 3 for general applications, according to Marchi and Silva [7].

4 Results

4.1 Transient flow in a water reservoir

To simulate the flow in a rainwater containment reservoir, a square reservoir $(7\,m \times 7\,m)$ with rigid and impermeable walls was considered. The reservoir has two discharge devices, and its topography is given by:

$$z = 1 - \frac{1}{7}y, \tag{12}$$

with Manning friction coefficient $\eta = 0.013\,m/s^{1/3}$ (adapted from Porto [15]) that is used to quantify the flow resistance at the bottom of the channel for cement surfaces.

The presented case study has a practical appeal, aiming to simulate flows in rainwater containment reservoirs that have some slope on the bottom surface, facilitating drainage when necessary.

Initially, the water in the reservoir is at rest, and the water surface level throughout the reservoir is $5\,m$ high.

Since the reservoir walls are impermeable, the outflow of stored water occurs through discharge control devices at the east and north outlets.

Regarding the boundary conditions, we consider ghost volumes around the boundaries and adopt the zero Neumann condition for h on the domain boundaries.

For the flow variables (u, v), the walls are considered impermeable, and all velocity components normal to the boundaries are set to zero. On the other hand, the tangential velocity components are calculated considering zero derivatives for each component, in other words, a zero Neumann condition for each tangential component.

The simulations presented in this section were performed using Microsoft Visual Studio 2003 Fortran Language on a 64-bit PC with an Intel Core i7 - 7700HQ 2.8GHz processor, 16GB RAM, running on the Windows 10 Home Single Language operating system.

The numerical parameters are: mesh size $(7\,m \times 7\,m)$; $\Delta x = \Delta y = 0.1\,m$ and $\Delta t = 0.001\,s$.

To analyze the complete flow of the stored water, descript in Fig. 1 in approximately 120 seconds, we computed the accumulated volume at each time step. For this purpose, we calculated the water column height (the difference between $h(x, y, t)$ and $z_0(x, y, t)$) at each point (x, y) in the mesh accumulatively. Therefore, the volume stored in the reservoir at each moment in time is given by:

$$V(t) = \sum \Delta V(t) = \sum_{x,y} [h(x, y, t) - z_0(x, y, t)] \Delta x \Delta y. \tag{13}$$

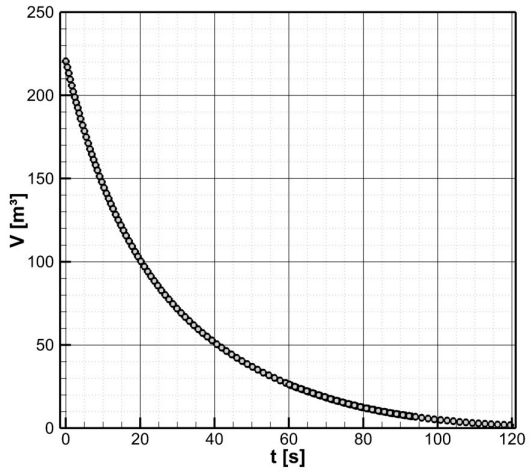


Figure 1. Variation of the stored water volume in the reservoir over the first $120\,s$.

4.2 The algorithm

The physical domain D, defined as $[x_a, x_b] \times [y_a, y_b]$], is discretized into a grid of N_x by N_y control volumes, with dimensions Δx and Δy in the x and y directions, respectively. The temporal domain $[t_0, t_f]$ is partitioned into N_t time steps of size Δt. The bottom surface $z_0(x, y)$, as well as the initial conditions h_0, u_0, and v_0, are inputted into the simulation. As the temporal discretization employs a fully implicit scheme, an iterative approach is employed to solve the resulting linear system obtained from the finite volume method (MVF) discretization. Specifically, the Gauss-Seidel method (GS) is employed as the iterative solver, with the maximum number of iterations itmax or the global residual R serving as stopping criteria, given by

$$R = \frac{1}{(N_x - 2)(N_y - 2)} \left[\frac{r_h + r_u + r_v}{3} \right] \leq tol, \tag{14}$$

where $(N_x - 2)(N_y - 2)$ is the total number of inner control volumes, tol is the tolerance, and the residual r_ϕ, $\phi = h, u, v,$, represents the sum of individual residuals for each control volume of the variable ϕ. The residual is calculated incrementally as follows:

$$r_\phi = \sum_j (\sum_i (r_\phi)_{i,j}), \tag{15}$$

with $i = 1, \cdots, N_x - 2$ and $j = 1, \cdots, N_y - 2$ where $(r_\phi)_{i,j} = (b_\phi)_{i,j} - (A\phi)_{i,j}$, and $\phi = h, u, v$, as defined in Eq. (6).

4.3 Error analysis

To analyze the discretization error of the method, we used the volume V described in eq. (13) as the variable of interest and performed the flow simulation in the time interval from 0 to 1 second, refining both the temporal and spatial domains.

Since the calculation of the variable of interest is dependent on the values of h, Table 1 displays the residual value r_h obtained for different grids.

From Table 1, we observe that as the grids become more refined, the residual value in the calculation of the variable of interest decreases. We also note that even for coarse grids, the residual tends to approach rounding-off error, indicating a high level of accuracy in the calculations.

For the analysis of the discretization error of the model, we calculated the apparent order, pU, described by eq. (8)), using the volume V as the variable of interest according to eq. (13).

Table 1. Convergence analysis of the adopted iterative method.

N_t	Δt	N_x	N_y	$\Delta x = \Delta y$	Residual r_h
100	0.01	14	14	0.5	1.113×10^{-11}
200	0.005	28	28	0.25	7.853×10^{-12}
400	0.0025	56	56	0.125	6.259×10^{-12}
800	0.00125	112	112	0.0625	1.371×10^{-12}

We used a combination of the UDS and CDS (with asymptotic orders of first and second order, respectively) for spatial discretization and a fully explicit method (first order) for temporal discretization. According to Ferziger and Peric [16], it is known that this combination should yield a method with an asymptotic order of first order, i.e., $p_0 = 1$. This result is evident in Fig. 2.

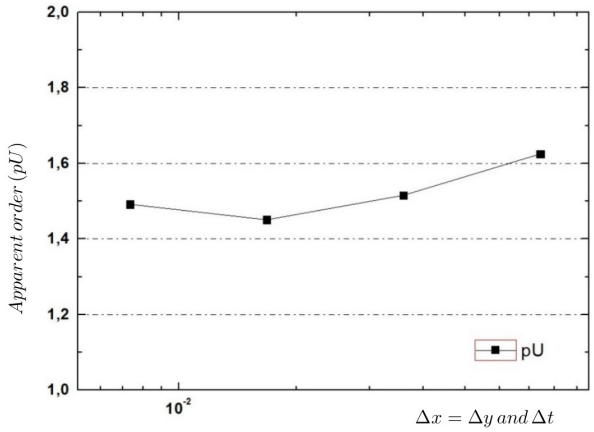


Figure 2. pU versus $\Delta x = \Delta y$ and Δt values as shown in Table 1.

Due to the high computational cost, we did not use highly refined grids to emphasize that $pU \to 1$. Nevertheless, we can still observe this trend with grid refinement. Another powerful tool for analysis, when the solution is unknown, is error estimation. For this purpose, we can estimate the discretization error based on the apparent order using Richardson and Grid Convergence Index (GCI) estimators (eq. (10) and eq. (11), respectively) for the simulation intervals of 0 to 1 s and 0 to 3 s. The results obtained for these estimators using the grids from Table 1 are described in Fig. 3.

It is noteworthy that the highest estimated error occurred for the longest simulation time interval $t = 3s$). However, regardless of the simulated time interval, the discretization error tends to zero with grid refinement.

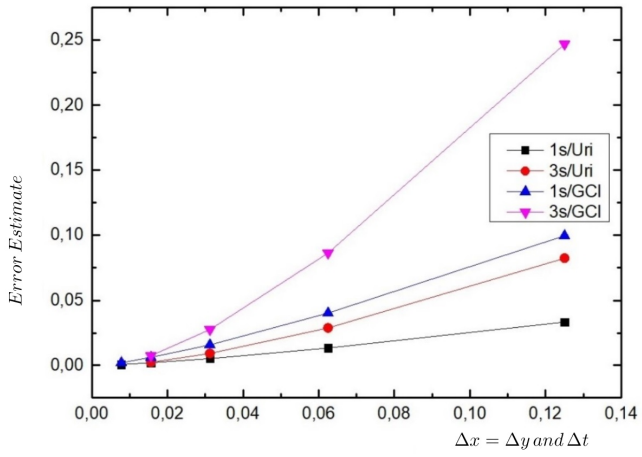


Figure 3. Discretization error estimate using Richardson (Uri) and Grid Convergence Index (GCI) estimators.

5 Conclusions

The numerical simulation of the shallow water hydrodynamic model, along with its error analysis, aims to achieve optimal performance and facilitate the accurate interpretation of the underlying physical phenomenon while maintaining a low computational cost.

The computational model presented in this study has a high capacity to accurately simulate complete reservoir drainage.

Based on the presented results, the obtained numerical solution has demonstrated stability and the absence of non-physical behavior, even when considering activated source terms. Furthermore, the presence of varying bottom slope within the reservoir did not pose any issues for the code, which successfully handled the complete shallow water equations.

The Richardson and GCI estimators demonstrated their suitability and provided an accurate numerical interpretation of the studied physical phenomenon, ensuring a good balance between accuracy and computational cost.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] K. Mahmood and V. Yevjevich. *Unsteady flow in open channels*. Water Resources Publications, Colorado, USA, 1975.

[2] S. C. Silva, E. Kaviski, L. M. Gramani, A. M. Lobeiro, R. V. P. Rezende, and R. A. Almeida. Hybrid numerical scheme for simulation of transient flow in a pluvial water reservoir. *Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería*, vol. 36, n. 2, pp. 1–22, 2020.

[3] A. Valiani, V. Caleffi, and A. Zanni. Case study: malpasset dam-break simulation using a two-dimensional finite volume method. *J. Hydraul. Eng.*, vol. 128, pp. 460–472, 2002.

[4] I. K. Nikolos and A. I. Delis. An unstructured node-centered finite volume scheme for shallow water flows with wet/dry fronts over complex topography. *Comput. Methods. Appl. Mech. Engrg.*, vol. 198, pp. 3723–3750, 2009.

[5] J. Fernández-Pato, M. Morales-Hernández, and P. García-Navarro. Implicit finite volume simulation of 2d shallow water flows in flexible meshes. *Comput. Methods. Appl. Mech. Engrg.*, vol. 328, pp. 1–25, 2018.

[6] F. Behzadi and J. Newman. An exact source-term balancing scheme on the finite element solution of shallow water equations. *Comput. Methods. Appl. Mech. Engrg.*, vol. 359, pp. 1–21, 2020.

[7] C. H. Marchi and A. F. C. Silva. Unidimensional numerical solution error estimation for convergent apparent order. *Numerical Heat Transfer, Part B: Fundamentals*, vol. 42, n. 2, pp. 167–188, 2002.

[8] C. H. Marchi, L. K. Araki, A. C. Alves, R. Suero, S. F. T. Gonçalves, and M. A. V. Pinto. Repeated richardson extrapolation applied to the two-dimensional laplace equation using triangular and square grids. *Applied Mathematical Modelling*, vol. 37, pp. 4661–4675, 2013.

[9] L. P. da Silva, B. B. Rutyna, A. R. S. Righi, and M. A. V. Pinto. High order of accuracy for poisson equation obtained by grouping of repeated richardson extrapolation with fourth order schemes. *Computer Modeling in Engineering & Sciences*, vol. 128, n. 2, pp. 699–715, 2021.

[10] C. H. Marchi, C. D. Santiago, and C. A. R. Carvalho Jr. Lid-driven square cavity flow: A benchmark solution with an 8192x8192 grid. *Journal of Verification, Validation and Uncertainty Quantification*, vol. 6, pp. 1–23, 2021.

[11] C. Hirsch. *Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics*. John Wiley & S, 4 th edition, 2007.

[12] R. L. Burden and J. Faires. *Numerical Analysis*. Cengage Learning, California, USA, 2010.

[13] E. Kreyszig. *Advanced Engineering Mathematics*. John Wiley and Sons, New Jersey, USA, 2010.

[14] S. Rodrigues, M. A. V. Pinto, M. A. Martins, and S. R. Franco. Reducing the discretization error for a poroelasticity problem in variables having extreme values. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 44, n. 147, pp. 1–11, 2022.

[15] R. M. Porto. *Hidráulica Básica*. EESC-USP, São Paulo, Brazil, 4 th edition. (in Portuguese), 2006.

[16] J. H. Ferziger and M. Peric. *Computational methods for fluid dynamic*. Springer, New York, USA, 2002.