Alternatives for flow solution at the leading edge of gravity currents using the shallow water equations

THOMAS M. HATCHER, PhD Student, Department of Civil Engineering, Auburn University, Auburn, AL 36849, USA.
Email: tmh0019@auburn.edu

JOSE G. VASCONCELOS, Assistant Professor, Department of Civil Engineering, Auburn University, Auburn, AL 36849, USA
Email: jvasconcelos@auburn.edu (author for correspondence)

Gravity current leading edge modeling alternatives
Alternatives for flow solution at the leading edge of gravity currents using the shallow water equations

ABSTRACT

The shallow water equations are implemented in gravity current models because of the good balance between accuracy and computational effort. In this context appropriate treatment is required at the gravity current leading edge that is analogous to a moving boundary condition. To date, solution strategies used at the leading edge have not been systematically compared to determine the best alternative. This work aims to analyze three strategies, comparing results with experiments and Reynolds-averaged Navier-Stokes models. Two of these alternatives are implemented using method of characteristics theory, while the third alternative, proposed the authors, constitutes a newer conservative shock-tracking approach. Despite some of the differences in continuity error and computational effort, these solution strategies accurately predicted front trajectories albeit with much less computational effort when compared to Navier-Stokes models. As anticipated, gravity current shapes yielded by one-layer shallow water equation models don’t compare well particularly at earlier flow stages.

Keywords: Boundary conditions; Characteristic equations; Gravity currents; Numerical methods; Shallow water equations.

1 Introduction

Gravity currents (GC) are driven by a density difference between two or more fluids and generally travel in a quasi-horizontal plane. This work focuses on high Reynolds GC flows characterized by small density differences ($\Delta \rho$) between fluid layers. Such flows can be caused by differences in ambient and current sediment concentrations, temperature or salinity levels (Simpson 1997). In laboratory settings, these flows are often generated by the release of a certain volume of fluid into an ambient fluid with $\Delta \rho$ (herein referred to as constant-volume GC flow) or by forcing a known influx of fluid with $\Delta \rho$ at a system boundary (referred to as density intrusion GC flow). Both conditions are illustrated in Fig. 1.

Figure 1: Diagram of initial conditions of GC flows: (a) constant-volume GC and (b) density intrusion GC.
Numerical solution alternatives for simulating GC flows typically include: integral, shallow water equation (SWE) and Navier-Stokes (NS) models. Integral models (Huppert and Simpson 1980) represent the simplest alternative to simulate GCs and assume a horizontal free-surface for both fluid layers. These models focus on the front trajectory and average GC depth. With increased availability of improved computers, more complex GC flows have been simulated with NS models including complex geometries, interactions with environment and sophisticated turbulent formulations. While undoubtedly more accurate tools, NS models are far more complex than SWE models and thus require much larger computational effort, which may be unfeasible in certain cases. The use of simpler SWE models can be justified for instance when the focus of the modeling effort is simply the kinematics of the propagation of the turbidity currents. Such cases could include numerical studies where large data ensembles from simulations are required and/or studies involving the variation of flow parameters such as Reynolds and densimetric Richardson numbers, depth ratios, etc. Even so, the accuracy of GC flows yielded by SWE models with respect to the shape of the GC head may be limited.

SWE models require special handling at the leading edge of GCs since flow conditions there violate the assumptions used in these models. At the leading edge, there are strong vertical accelerations and mixing that contribute to a curved interface between GC and ambient that invalidates some of the hypotheses formulated in the SWE derivation. To overcome the difficulties at these locations, SWE models adopt front conditions that relate the GC depth and correspondent velocity at the leading edge region. An early example of these front conditions was proposed by von Karman (1940):

\[
\frac{u_{LE}}{(g' h_{LE})^{1/2}} = F_{LE}
\]

in which \(g'\) is the reduced gravity defined as \(g' = \Delta \rho / \rho_0\), and \(u_{LE}, h_{LE}\) and \(F_{LE}\) are the velocity, depth and Froude number at the leading edge, respectively. Defining \(\phi = h_0/H\) as the initial depth ratio between current and ambient fluids, Von Karman applied this front condition to the deep ambient scenario (\(\phi \approx 0\)) in which \(F_{LE} = \sqrt{2}\); Benjamin (1968) extended Eq. 1 to the range \(0 < \phi \leq 1\) in a more comprehensive analysis. Because both of these theoretical front conditions overestimate the GC front velocity, significant research has focused on empirical front conditions that account for entrainment, friction, dissipation, etc. (Huppert and Simpson 1980, Rottman and Simpson 1983). The Huppert and Simpson front condition (referred to as HS) is utilized in this work and is expressed as:

\[
F_{HS} = \begin{cases} 
\frac{1}{2} \phi_{LE}^{-1/3} & (0.075 \leq \phi_{LE} < 1) \\
1.19 & (\phi_{LE} \leq 0.075)
\end{cases}
\]
After substituting Eq. 2 into Eq. 1, there is one equation for the two unknowns \((u_{LE} \text{ and } h_{LE})\) where \(\phi_{LE} = h_{LE}/H\) at the GC leading edge, so another equation must be provided. A traditional method to provide this equation has been through the method of characteristics (MOC), whereby a relevant characteristic equation that is valid within a trajectory in the space-time solution domain (a characteristic line) is used to provide closure to the flow calculation. Within the MOC, there are two alternatives to provide this equation:

- **Grid-of-Characteristics (GOC):** Described by Lai (1988), this method involves explicit tracking of characteristic lines to determine flow conditions at locations where they intercept each other or intercept a boundary condition (BC), such as GC leading edges.

- **Specified time intervals (STI):** Also known as Hartree MOC (Sturm 2010), this popular approach involves back-projecting of characteristic lines in the space-time grid and spatial interpolation of variables between the leading edge coordinate and the adjacent node.

A third method referred to as Dual-Cell (DC) has been proposed by Hatcher and Vasconcelos (2013) and attempts to provide a set of equations that ensure observance of continuity and momentum equations at the leading edge region. In addition to these equations, the DC method uses a front condition (e.g. Eqs. 1 and 2) to determine flows at the GC leading edge. This method has been proven accurate to simulate both one and two-layer SWE models when compared with experimental results presented by the authors. These comparisons involved constant-volume GC flows, but not constant-influx GC flows.

While there has been significant progress in the development of improved front conditions to simulate GC flows, a more detailed study on the advantages between these different strategies of solving the flow at the GC leading edge has not received as much attention. Some important questions are still open and require clarification. First, the STI approach is a practical and relatively cheap alternative to solve flows at GC leading edges, but one question is to what extent the interpolation it introduces affects simulation accuracy. Second, while the GOC approach does not use interpolation, a relevant question is how expensive is the explicit tracking of characteristic lines that were generated at the upstream BC, particularly for highly-discretized solution domains. Third, the relative performance of the DC method when compared to the MOC methods has not been assessed to this date. Finally, another pertinent question is whether there are applicability limits to the outlined strategies to solve flows at GC leading edge in the context of constant-volume and density intrusion GC. The main purpose of this work is to address these questions.

With the recent improvements of computing power, computational fluid dynamics (CFD) models have become a popular alternative for various flow simulations. Among the
studies on GC using CFD, Reynolds-averaged Navier-Stokes (RANS) models have been applied to the description of such flows. Laterally-averaged RANS models have been utilized to simulate compositional (Mehdizadeh and Firoozabadi 2009) and particle-driven GC flows (Mehdizadeh et al. 2008). In these studies the $v^2-f$ and variations of $k-\varepsilon$ turbulence models were implemented. The authors concluded that the $v^2-f$ model simulated best the height and velocity structure of the GC. This improved accuracy came at a computational cost of 4.5 times the standard $k-\varepsilon$ model because of two additional equations in addition to reduction methods that were required for stability. The work by An et al. (2000) also explored the potential of RANS models for GC flows. They found minor differences in a comparison between RNG $k-\varepsilon$ and Large Eddy Simulation (LES) turbulence models in the case of intrusive and particle-driven GCs.

In recent efforts to improve upon the accuracy of traditional RANS models, LES and Direct Numerical Simulation (DNS) models have gained increased research focus particularly for turbulent GC flows. Ilicak et al. (2008) determined that the $k-\varepsilon$ model was not sufficient in accurately modeling the Red Sea overflow. To overcome this issue and to avoid large computational times, they combined RANS and LES approaches using the Very Large Eddy Simulation (VLES) model. Tokyay et al. (2011) performed LES simulations for GC flows over an array of obstacles. In the case of lock exchange GCs, Hartel et al. (2000) performed high-resolution DNS simulations, which gained insight into the GC turbulence and the location of the leading edge stagnation point. Cantero et al. (2007) performed high-resolution simulations for a larger range of Reynolds numbers and analyzed the transition between and the three-dimensionality of the different phases. While not yet applicable to large-scale field applications, these high-resolution NS models have been important research tools and will continue to gain popularity.

2 Objectives

The objective of the proposed work is two-fold. The first one is to present a comparative study on three different strategies (STI, GOC and DC methods) to solve flows at the leading edge of Boussinesq GCs using SWE models. To attain this goal, two different GC flows are considered: flows generated by the release of a fixed volume of dense fluid; and flows caused by the constant discharge of a dense fluid at the bottom of a tank. These cases were selected because they provide a means to compare the modeling results with previously published experimental works presented by Huppert and Simpson (1980) and Wright and Paez-Rivadeneira (1996). For each strategy different grid sizes are considered, and the overall continuity error is quantified alongside the computational effort associated with the simulations.
The second goal of this work is to qualitatively compare SWE simulation results with those obtained by a 2-D RANS model and experimental results for constant-volume GCs. Such CFD models have less computational effort than high-resolution alternatives (e.g. LES, DNS) and thus shorter computational times. The comparison involves the GC front trajectory, shape and computational effort.

3 Methodology

3.1 SWE modeling
This comparative study is performed in the context of the single-layer SWE, which are represented in a divergent, conservative and homogenous format:

$$\frac{d\mathbf{U}}{dt} + \frac{\partial F(\mathbf{U})}{\partial x} = S(\mathbf{U})$$

where $h$, $u$ correspond to the local depth and velocity of the GC, $\mathbf{U}$ is the vector of conserved variables, $F(\mathbf{U})$ is the flux of conserved variables and $S(\mathbf{U})$ is the source term vector. This relatively simple mathematical model is solved within the finite volume framework, yielding the following expression that performs the updates on the conserved variables for each computational cell $i$ at time step $n+1$:

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n + \frac{\Delta t}{\Delta x} \left( F(\mathbf{U})_{i+1/2}^{n+1/2} - F(\mathbf{U})_{i-1/2}^{n+1/2} \right) + S(\mathbf{U}) \Delta t$$

In this work the non-linear HLL scheme was selected, and details of this implementation are provided in Toro (2001). Other schemes (Roe, HLLC, WAF, Lax-Wendroff) were also tested, but the HLL scheme performed better when accounting for both accuracy and computational efficiency.

At the boundaries of the solution domain (either at a physical boundary or at the GC leading edge), numerical schemes cannot be used to determine the two flow variables $\mathbf{U} = [h, uh]^T$, and BC calculations are thus required. The solution alternatives at physical walls are straightforward: either enforce zero flow ($u = 0$) along with a relevant characteristic equation or apply a reflective BC with the aid of a virtual cell, as presented in Toro (2001). The former alternative was used in this work.

The treatment of the moving BC at the GC leading edge is one of the two primary focuses, and the formulation and implementation details of each one of the three strategies outlined in the introduction are presented below. Solutions involved iterative calculations of a
system of nonlinear equations to solve the flow at the GC leading edge using the bisection method with a tolerance of $10^{-5}$ for convergence. For all SWE modeling, the Huppert and Simpson (1980) front condition was used.

For constant-flux GC simulations, the following parameters were known regarding the initial conditions (IC): initial GC length ($x_0$) and depth ($h_0$) as well as the ambient depth ($H$). Compared to the constant-volume implementation, the primary difference for the density intrusion IC was the initial GC volume where a relatively small value for $x_0$ was used at the start of the simulations, following Bonnecaze et al. (1995).

**Grid-of-Characteristics (GOC) approach**

The GOC approach has the simplest concept of all the methods. It involves the combination of a front condition and a $C+$ characteristic equation that is tracked within a characteristic line originated at a system boundary. This provides two equations that allow for the solution of the two unknowns $[h_{LE}, uh_{LE}]^T$ at the GC leading edge cell ($LE$ subscript). When the model requires the solution at specified intervals, one method to implement the GOC approach is to generate the characteristic lines at every time step (e.g. at a physical boundary) and track them so that the corresponding characteristic equation can be used in the BC computation (see Figure 2). This is an important distinction from the STI method, which generates these characteristic lines at the vicinity of the GC leading edge. While the continuous generation and tracking of characteristic lines is computationally costly, particularly for finer grids, the GOC method benefits from no loss of accuracy caused by interpolation that is required in the STI method.

![Figure 2 Diagram of the GOC approach](attachment:image.png)

Figure 2 Diagram of the GOC approach, where $C+$ characteristic lines (not to scale and can be nonlinear) are generated at each time step at the upstream boundary. These lines travel towards and eventually reach the GC leading edge.
For constant-volume GCs, the depth varies over time at the upstream wall (cell 1), but the velocity is always zero. Tracking the characteristic lines over time brings the Riemann invariant values \((u+2c)\) from the wall to the leading edge of the GC. The resulting set of equations is thus:

\[
u_{\text{cell:1}} + 2\sqrt{g h_{\text{cell:1}}} = u_{\text{LE}} + 2\sqrt{g h_{\text{LE}}} (5)\]

\[
u_{\text{LE}} = F_{\text{LE}} \sqrt{g h_{\text{LE}}} (6)\]

in which \(u_{\text{cell:1}}\) is 0 at the upstream physical boundary, and \(h_{\text{cell:1}}\) is the depth at the wall that was observed when the characteristic line was generated, at an earlier time step. The value of \(h_{\text{cell:1}}\) will be constant and equal to the initial depth \(h_0\) until the depression wave generated by the initial gate release arrives at the wall (for constant volume simulations). Up to that moment in the simulation, flow conditions are consistent with a simple wave condition (Sturm 2010) and there is no need to generate/track characteristic lines. As the flow depth drops at the wall, characteristic lines are created each time step and the trajectory of this line is updated during the simulation using the \(C+\) characteristic equation \((dx/dt = u+c)\) evaluated with the local flow conditions between the wall and the leading edge of the GC. Once a new characteristic line reaches the GC front, Eqs. 5 and 6 are solved iteratively for \(h_{\text{LE}}\) and \(u_{\text{LE}}\). The position of the front advance \((\Delta x_{\text{LE}}^{n+1})\) is updated according to the kinematic condition:

\[
\Delta x_{\text{LE}}^{n+1} = \Delta x_{\text{LE}}^n + \Delta t \cdot u_{\text{LE}} (7)
\]

Once \(\Delta x_{\text{LE}}^{n+1}\) equals the internal cell length \(\Delta x\), one more computational cell of length \(\Delta x\) is added to the solution domain and \(\Delta x_{\text{LE}}^{n+1}\) and \(\Delta x_{\text{LE}}^n\) are reset to zero. In the case of density intrusion GCs tested in this work, both \(h_{\text{cell:1}}\) and \(u_{\text{cell:1}}\) are predefined by the problem conditions. Unlike constant-volume GC, density intrusion problems involve non-zero velocity conditions at the upstream boundary. Since \(h_{\text{cell:1}}\) and \(u_{\text{cell:1}}\) (and respective Riemann Invariants) are constant throughout the problem, there would be no need to track the characteristic lines. However, this tracking is performed to consistently quantify the computation effort between the three tested approaches.

**Specified Time Intervals (STI) approach**

The STI approach, similarly to GOC, also utilizes the MOC. However, instead of tracking characteristic lines in the space-time grid from the line origin, interpolation between neighbor computational cells is utilized. It thus allows the calculation of the relevant Riemann invariant used in the computation of flow conditions along with the front condition much more rapidly, decreasing computational effort. The approach is consistent with traditional implementation of the MOC-Hartree method at BCs (Sturm 2010) except that the location of the BC is
updated with time. Using Fig. 3 as reference, integration of the $C+$ characteristic equation and the corresponding trajectory of the characteristic line yields:

$$u_P - u_R + \frac{g'}{c_R}(h_P - h_R) = 0 \quad (8)$$

$$x_P - x_R = (u_R + c_R)\Delta t \quad (9)$$

The subscript $P$ corresponds to a point $P$ point immediately upstream from the leading edge in the next time step where the flow solution $(u_P, h_P)$ at the leading edge is obtained. Point $R$ is the origin of the $C+$ characteristic line that arrives at point $P$, bringing the information on the Riemann invariants (Eq. 8) to be applied in the solution of the flow in the BC. It is assumed that flow conditions at point $P$ correspond to the conditions at the leading edge, less than $\Delta x$ away. This approach is tested with good accuracy in comparison with the other front BC approaches with time steps ranging from 0.0022 - 0.95 seconds (see Results 4.1).

While conditions at points $A$ and $B$ (previous time step) are known, flow depth, velocity, and celerity at point $R$ are unknown, and this is where interpolation is introduced. A potential drawback of STI interpolation is the generation of inaccuracies in the solution, especially in locations where there are strong gradients. Assuming that the previous time step index is denoted by $n$, the computation of $u_R$, $c_R$ and $h_R$ follows the expressions presented in Sturm (2010):

$$u_R = \frac{u_R^n + r(u_A^n + u_B^n)\frac{c_A^n + u_B^n}{c_A^n}}{1 + r(u_A^n - u_B^n)} \quad (10a)$$

$$c_R = \frac{c_B^n + r(u_A^n c_B^n - c_A^n)}{1 + r(c_B^n - c_A^n)} \quad (10b)$$

$$h_R = h_B^n - r(h_B^n - h_A^n)(u_R + c_R) \quad (10c)$$

Once the parameters in Eqs. 10a-10c have been computed, Eq. 8 is solved alongside a front condition to determine $u_P = u_{LE}$ and $h_P = h_{LE}$. The determination/update of the GC leading edge location uses the same kinematic condition that was presented for the GOC front BC (Eq. 7).
Figure 3. The numerical solution for the STI method in which the characteristic grid is superimposed onto a rectangular grid for a single time step. Adapted from Sturm (2010).

**Dual-Cell (DC) approach**

As pointed out, despite the relative simplicity behind MOC-based methods to provide closure to the flow calculation at the leading edge of GCs, both GOC and STI approaches have limitations. The GC approach promotes explicit tracking of characteristic lines that may become computationally intensive in some cases. On the other hand, the STI approach involves interpolation that may impact model accuracy. In addition, characteristic equations for two-layered flows (Rottman and Simpson 1983) are significantly more complex and difficult to implement.

These combined factors led to the development of an alternative method (Hatcher and Vasconcelos 2013) that does not depend on characteristic equations, but rather on flow conditions at two computational cells located at the leading edge region of the flow. This approach, referred to as Dual-Cell (DC) method, can be applied to one and two-layer GC flows without alterations. Like MOC-based approaches, it also requires a front condition to achieve flow closure at the leading edge of the GC. The DC method combines the enforcement of continuity, linear momentum, and the kinematic condition at the GC leading edge assuming that the front spans over two cells: (1) cell \( LE \) corresponds to a computational cell fully occupied by the dense fluid and (2) cell \( LE+1 \) undergoes the advance of the current and is not yet completely filled (see Fig. 4 for reference).
Because the leading edge spans over two computational cells, the DC method requires five equations to solve for the unknown variables: $h_{LE}$, $uh_{LE}$, $h_{LE+1}$, $uh_{LE+1}$ and $\Delta x_{LE}$. The continuity equations at the two leading edge cells, linear x-momentum equation at cell $LE$, front condition, and kinematic condition yield the required five equations to achieve closure. Details are omitted here for brevity but can be found in Hatcher and Vasconcelos (2013).

### 3.2 Laterally-averaged RANS modeling

To perform a comparative analysis on the applicability of the SWE to simulate GC flows, 2D RANS models were constructed using OpenFOAM 2.1. Both the standard $k$-$\varepsilon$ (using wall functions to reduce computational time) and $\nu^2$-$f$ (without wall functions) turbulence models were implemented in this analysis. The mean velocity vector field ($\bar{u}$) satisfies the incompressible RANS mass and momentum equations:

\[
\nabla \cdot \bar{u} = 0
\]

\[
\frac{\partial}{\partial t} \bar{u} + \bar{u} \cdot \nabla \bar{u} = -\nabla P + \nabla \cdot [(\nu + \nu_t) \nabla \bar{u}]
\]

where $P$ is mean pressure, $\nu$ is kinematic viscosity and $\nu_t$ is turbulent viscosity. The turbulent viscosity definition is different between turbulence models:

\[
\nu_t = C_{\mu} \frac{k^2}{\varepsilon} \quad (k$-$\varepsilon \text{ model})
\]

\[
\nu_t = C_{\mu} \nu^2 T \quad (\nu^2$-$f \text{ model})
\]

In the above expressions, $k$ is the turbulent kinetic energy, $\varepsilon$ is the turbulent dissipation, $\nu$ is the turbulent stress normal to streamlines, $T$ is the turbulent time scale and $C_{\mu}$ and $C_{\nu ke}$ are coefficients described ahead in this section. The turbulent viscosity expression for the $\nu^2$-$f$ model in Eq. 12 is more accurate for near-wall turbulence according to DNS and experiments (Mortensen et al. 2010). In both RANS models implemented in this work, the following $k$-$\varepsilon$ equations are used:
\[
\begin{align*}
\partial_t k + \bar{u} \nabla k &= P_k - \varepsilon + \nabla \cdot \left[ \left( \nu + \frac{\nu^s}{\sigma_k} \right) \nabla k \right] \\
\partial_t \varepsilon + \bar{u} \nabla \varepsilon &= \frac{C_{\varepsilon 1} P_k + C_{\varepsilon 2} \varepsilon}{T} + \nabla \cdot \left[ \left( \nu + \frac{\nu^s}{\sigma_\varepsilon} \right) \nabla \varepsilon \right]
\end{align*}
\] (13)

in which \( P_k \) is the production term and \( C_{\varepsilon 1}, C_{\varepsilon 2}, \sigma_k \) and \( \sigma_\varepsilon \) are standard \( k-\varepsilon \) coefficients. The main difference for the \( \bar{v}^3-f \) turbulence model (Durbin 1991, 1995) is that it contains an additional expression for \( \bar{v}^3 \) as seen in Eq. 12 which is used near wall boundaries. There the model performs a nonlocal suppression of \( \bar{v}^3 \), so an elliptic relaxation equation is utilized for a new parameter (\( f \)). Both \( \bar{v}^2 \) and \( f \) expressions are provided below:

\[
\partial_t \bar{v}^2 + \bar{u} \nabla \bar{v}^2 = kf - 6\bar{v}^3 \frac{\varepsilon}{k} + \nabla \cdot \left[ \left( \nu + \frac{\nu^s}{\sigma_\varepsilon} \right) \nabla \bar{v}^2 \right]
\]

\[
L_k^2 \frac{\partial^2 f}{\partial x_i \partial x_j} = f + \frac{1}{T} \left[ \left( C_1 - 6 \right) \frac{\bar{v}^3}{k} - \frac{2}{3} (C_1 - 1) \right] \cdot C_2 \frac{P_k}{k}
\] (14)

where \( L_k \) is the turbulent length scale, \( kf \) is a pressure-strain term, and \( C_1 \) and \( C_2 \) are coefficients provided in Lien and Kalitzin (2001). The production term (\( P_k \)) is computed from the expression (see Durbin 1995):

\[
P_k = \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}
\] (15)

and the turbulent time (\( T \)) and length (\( L_k \)) scales are provided below (Lien and Kalitzin 2001):

\[
T = \max \left[ \frac{k}{\varepsilon}, C_T \left( \frac{\nu}{\varepsilon} \right)^{1/2} \right] \quad L_k = C_L \max \left[ \frac{k^{3/2}}{\varepsilon}, C_{\eta} \left( \frac{\nu}{\eta} \right)^{1/4} \right]
\] (16)

The coefficients for the RANS models were adopted from Lien and Kalitzin (2001):

\[
\begin{align*}
C_{\mu\varepsilon} &= 0.09, \quad C_{\mu} = 0.22, \quad C_{\varepsilon 1} = 1.4 \left( 1 + 0.05 \sqrt{k} \bar{v}^2 \right), \quad C_{\varepsilon 2} = 1.9 \\
\sigma_k &= \sigma_\varepsilon = 1.3, \quad C_1 = 1.4, \quad C_2 = 0.3, \quad C_T = 6, \quad C_L = 0.23, \quad C_{\eta} = 70
\end{align*}
\] (17)

A total of 20,000 cells were used in the \( k-\varepsilon \) simulations: 500 longitudinal cells and 40 vertical cells (both uniform). The \( \bar{v}^2-f \) model (without wall functions) requires a much finer grid than the \( k-\varepsilon \) model near the bed (\( y^+ \approx 1 \)). For the \( \bar{v}^2-f \) simulations, there were 500 computational cells in the longitudinal direction (\( \Delta x = 6.0 \text{ mm} \)) and 80 cells in the vertical direction (40,000 total cells). The grid spacing in the vertical direction was increased by a factor of 20 toward the channel bottom while the longitudinal spacing was uniform. Rigid lid conditions were assumed for both 2D-RANS models, and the following expressions were used for wall BCs in the \( \bar{v}^2-f \) model (Mehdizadeh and Firoozabadi 2009):

\[
\bar{u} = 0, \quad k = 0, \quad \bar{v}^2 = 0, \quad \varepsilon = \frac{2
\nu}{y^+}, \quad f = 0
\] (18)
4 Results

To attain the objectives of this work, this section is structured in three parts. The two initial sections focus on the assessment of SWE modeling strategies in terms of continuity error and computational time for constant volume and constant flux GC flows. The third and last part compares SWE predictions to 2D-RANS modeling performed with OpenFOAM and with experimental results by Marino et al. (2005). Parts one and two also include a comparison with other SWE model predictions by Ungarish (2009) and with experimental results of constant flux GC flows by Wright and Paez-Rivadeneira (1996).

4.1 Constant-volume gravity currents

To analyze the front BC solution strategies for constant-volume GC, $h_0$ was adjusted so that $(0 < \phi \leq 0.5)$ while the following IC were constant: $H = 1 \text{ m}$, $x_0 = 1 \text{ m}$, $L = 10 \text{ m}$. For Boussinesq, constant volume GCs, the propagation is a function of the depth ratio (Ungarish 2009). Thus, the deep ambient ($\phi \approx 0$) and critical condition ($\phi = 0.5$) scenarios were chosen to generate the data in Tables 1 and 2, whereas to perform the direct comparison with the results by Ungarish (2009) the value of $\phi = 0.33$ was used. These IC were used since the single-layer shallow water framework is better suitable for $\phi < 0.5$.

In order to compare the front BC solution strategies, continuity errors, time steps and computational times were quantified. The continuity error (negative values equate to mass lost) is computed from the difference between initial ($x_{LE} = x_o$) and final ($x_{LE} = L$) GC volumes. Results for continuity error and computational times are shown in Table 1 ($\phi \approx 0$) and Table 2 ($\phi = 0.5$) for the tested range of discretization sizes. Because the front location is changing within the solution domain, the number of discretization cells is a maximum value that represents the number of cells at $x_f = L$. Computations were carried out until the leading edge interface was at position $x/L_0 = 10$ or until the computational time reached one hour. The number of time steps for each front BC solution strategy was almost identical regardless of discretization size, which allowed for a more accurate computational time comparison. In addition, GC front trajectory results were very similar between each BC strategy.
Table 1: Comparison between measured continuity errors and computational time for STI, GOC and DC strategies – constant volume GC flows, deep ambient condition ($\phi \approx 0$).

<table>
<thead>
<tr>
<th>Method</th>
<th>No. of cells</th>
<th>100</th>
<th>200</th>
<th>500</th>
<th>1000</th>
<th>5000</th>
<th>10000</th>
</tr>
</thead>
<tbody>
<tr>
<td>STI</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cont. error (%)</td>
<td>-7.66</td>
<td>-3.77</td>
<td>-1.49</td>
<td>-0.74</td>
<td>-0.15</td>
<td>-0.08</td>
<td></td>
</tr>
<tr>
<td>Comp. time (s)</td>
<td>0.44</td>
<td>0.69</td>
<td>1.78</td>
<td>3.87</td>
<td>38.02</td>
<td>129.47</td>
<td></td>
</tr>
<tr>
<td>No. of time steps</td>
<td>180</td>
<td>363</td>
<td>911</td>
<td>1824</td>
<td>9131</td>
<td>18265</td>
<td></td>
</tr>
<tr>
<td>GOC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cont. error (%)</td>
<td>-0.48</td>
<td>-0.10</td>
<td>+0.15</td>
<td>+0.12</td>
<td>+0.05</td>
<td>NA</td>
<td></td>
</tr>
<tr>
<td>Comp. time (s)</td>
<td>0.61</td>
<td>1.22</td>
<td>5.04</td>
<td>19.48</td>
<td>2469.0</td>
<td>&gt; 1 hr</td>
<td></td>
</tr>
<tr>
<td>No. of time steps</td>
<td>182</td>
<td>365</td>
<td>913</td>
<td>1826</td>
<td>9133</td>
<td>NA</td>
<td></td>
</tr>
<tr>
<td>DC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cont. error (%)</td>
<td>-6.17</td>
<td>-2.56</td>
<td>-1.02</td>
<td>-0.51</td>
<td>-0.10</td>
<td>-0.05</td>
<td></td>
</tr>
<tr>
<td>Comp. time (s)</td>
<td>0.17</td>
<td>0.96</td>
<td>2.33</td>
<td>5.11</td>
<td>44.01</td>
<td>144.09</td>
<td></td>
</tr>
<tr>
<td>No. of time steps</td>
<td>181</td>
<td>363</td>
<td>911</td>
<td>1825</td>
<td>9132</td>
<td>18265</td>
<td></td>
</tr>
</tbody>
</table>

The results for both IC indicate a similar trend for each BC approach. The GOC approach consistently yielded the smallest continuity error, yet this method is clearly the most computationally intensive. The STI approach resulted in the largest continuity error, particularly for larger depth ratios, but had the smallest computational time. The DC approach computational time was slightly larger than the STI method, albeit with much smaller continuity errors especially for larger depth ratios.

Table 2: Comparison between measured continuity errors and computational time for STI, GOC and DC strategies – constant volume GC flows, critical condition ($\phi = 0.5$).

<table>
<thead>
<tr>
<th>Method</th>
<th>No. of cells</th>
<th>100</th>
<th>200</th>
<th>500</th>
<th>1000</th>
<th>5000</th>
<th>10000</th>
</tr>
</thead>
<tbody>
<tr>
<td>STI</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cont. error (%)</td>
<td>-10.20</td>
<td>-5.90</td>
<td>-3.64</td>
<td>-2.84</td>
<td>-2.34</td>
<td>-2.27</td>
<td></td>
</tr>
<tr>
<td>Comp. time (sec)</td>
<td>0.32</td>
<td>0.66</td>
<td>1.77</td>
<td>3.90</td>
<td>38.81</td>
<td>137.48</td>
<td></td>
</tr>
<tr>
<td>No. of time steps</td>
<td>195</td>
<td>394</td>
<td>990</td>
<td>1984</td>
<td>9939</td>
<td>19882</td>
<td></td>
</tr>
<tr>
<td>GOC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cont. error (%)</td>
<td>-4.08</td>
<td>-1.53</td>
<td>-0.46</td>
<td>-0.19</td>
<td>-0.02</td>
<td>NA</td>
<td></td>
</tr>
<tr>
<td>Comp. time (sec)</td>
<td>0.39</td>
<td>0.97</td>
<td>4.84</td>
<td>20.64</td>
<td>2586.0</td>
<td>&gt; 1 hr</td>
<td></td>
</tr>
<tr>
<td>No. of time steps</td>
<td>198</td>
<td>397</td>
<td>993</td>
<td>1988</td>
<td>9942</td>
<td>NA</td>
<td></td>
</tr>
<tr>
<td>DC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cont. error (%)</td>
<td>-6.96</td>
<td>-3.58</td>
<td>-1.44</td>
<td>-0.71</td>
<td>-0.15</td>
<td>-0.07</td>
<td></td>
</tr>
<tr>
<td>Comp. time (sec)</td>
<td>0.35</td>
<td>0.72</td>
<td>1.87</td>
<td>4.18</td>
<td>40.25</td>
<td>127.44</td>
<td></td>
</tr>
<tr>
<td>No. of time steps</td>
<td>196</td>
<td>396</td>
<td>992</td>
<td>1986</td>
<td>9941</td>
<td>19884</td>
<td></td>
</tr>
</tbody>
</table>

In Figure 5 the single-layer model outlined in the methodology using the three BC methods is compared to the single-layer model in Ungarish (2009) for the case when $\phi = 0.33$ and the discretization of 500 cells. In the Ungarish (2009) model, the internal cells were computed with the Lax-Wendroff second-order-accurate finite difference scheme, and the GOC BC approach was used. The single-layer SWE model in the present work utilizes the first-order-accurate nonlinear HLL scheme.
Figure 5 Constant-volume GC using each BC solution approaches superimposed with the model predictions by Ungarish (2009) for the case when $\phi = 0.33$, 500 cell computation.

There are no significant differences between the results obtained with the current model work using either one of the three tested approaches to solve flows at the leading edge of GC in Fig. 5. While results between the models are very similar (as shown in Figure 5), the minor differences observed at early times in the simulation ($t^* = t/x_0/\sqrt{g H_0} \leq 4$) can be attributed to the differences in the numerical scheme between the two models.

4.2 Density intrusion gravity currents

For the density intrusion GC simulations, the three BC approaches were tested for the conditions presented in the experimental work by Wright and Paez-Rivadeneira (1996). The experimental apparatus length was 10 m, and the advancing front velocity was experimentally measured at station $x = 4$ m from the dense fluid inlet. Various conditions were tested involving many injection flow rates per unit width ($u_{h_{in}}$), depth ratios at the inlet ($\phi_{in}$), and reduced gravity ($g'$). In the present study, a comparison between the approaches to solve flows at the leading edge is performed for 20 of the tested conditions. The ranges of experimental values considered in this comparison were: $\phi_{in}$ from 0.10 - 0.417, $g'$ from 3.60 - 8.82 cm s$^{-2}$, and $u_{h_{in}}$ from 23.85 - 49.87 cm$^2$s$^{-1}$. The resulting range of front velocities observed in this study was 3.81-6.43 cm s$^{-1}$.

For all tested conditions, the continuity errors were relatively small with the largest around 3% and in most cases 1%. To be consistent with the reported experimental conditions, both depths and flow rates were enforced at the apparatus boundary. Continuity errors and computational times are shown for two of the simulated cases in Fig. 6. Results follow approximately the same general trend observed in the previous example involving constant-
volume GC flows. Continuity errors were in general largest for the STI approach and smallest for the GOC approach. Computational time, on the other hand was smallest for the STI approach, but results with the DC approach are comparable especially for finer discretization. GOC results required much larger time when the number of grid cells were 5,000 or above.

One may notice that the increase of computational cells for case b) in Fig. 6 has not resulted in zero continuity errors in any of the tested solutions. This lack of convergence has to do with the enforcement of $h$ and $uh$ at the system boundary. In the context of open-channel flow problems, it is known that such practice leads to accuracy issues when a characteristic line from the domain arrives at the system boundary. An alternative solution at the system boundary that enforced only the inflow and used a $C$- characteristic line was used in such cases, and continuity errors were much reduced (results not shown for brevity). However, to maintain consistency between the 20 cases selected for comparison, as well as with the reported experimental conditions, all results shown enforced these two flow variables.

A comparison between experimentally measured front velocities and model predictions using the three solution approaches for the flow at the leading edge is shown in Fig. 7. All simulations were obtained with a 100-cell grid, as these front velocity results were not too dependent on the discretization size. One notices that there is generally good agreement between predicted and measured front velocities, which were not dependent on the selected approach to solve flow velocities at the GC leading edge. The one-layer SWE results displayed little bias compared to the experimental measurements.

4.3 Comparison with 2D RANS simulations

Results presented thus far indicated that computational effort for GC flows solved with the SWE is generally small even for fine discretization, particularly when DC or STI methods to solve flows at the leading edge are used. While continuity was mostly preserved in calculations, a pertinent question is how well those models represent the advance of GC flows over time. SWE models are clearly unable to describe the detailed flow structure for GC flows due to their one-dimensional nature, so one anticipates that the GC shape cannot be accurately represented. However, one question is whether the trajectory of the leading edge is well predicted over the slumping and self-similar stages of the GC flow. To address this question a three-way comparison is performed for constant-volume GC between experimental results presented by Marino et al. (2005), predictions from the SWE model (using DC strategy) and 2D-RANS modeling strategies presented in the methodology. For the CFD modeling, the choice of $k$-$\varepsilon$ turbulence model was motivated by simplicity and popularity, whereas $\overline{u^2}$-$f$ model was chosen because of improved accuracy in comparison with other
RANS models (Mehdizadeh and Firoozabadi 2009). Only constant-volume GC flows are used in this comparison between SWE and 2D-RANS models.

Figure 6 Continuity error and computational time comparison for density intrusion GC flows using the DC, GOC and STI approaches to compute flows at leading edge region. For the experimental conditions, (a) $\phi = 0.16$, $g' = 6.75 \text{ cm s}^{-2}$, $u_{in} = 7.28 \text{ cm s}^{-1}$ and (b) $\phi = 0.40$, $g' = 6.17 \text{ cm s}^{-2}$, $u_{in} = 5.22 \text{ cm s}^{-1}$.

Figure 6 Front velocity (ms$^{-1}$) comparison between single-layer SWE model and density intrusion experiments conducted by Wright and Paez-Rivadeneira (1996).
The partial depth experimental results from Marino et al. (2005) were used for comparison and are displayed in Figure 7, Figure 8, and Figure 9. They were able to determine the equivalent GC depth with a lighting technique discussed in Dalziel (1993) in which the saltwater mass was measured within 2% accuracy. GC flow conditions represent the case for \(x_0 = 0.1\, \text{m}, L = 3\, \text{m}, h_0 = 0.16\, \text{m}, H = 0.4\, \text{m}, \phi = 0.4\) and \(g' = 0.098\, \text{m/s}^2\).

According to the GC profiles displayed in Figure 7, the SWE model predicts the early stages of front location with fair accuracy in comparison with Marino et al. (2005) experiments, with better accuracy than the 2D RANS simulation using the \(\overline{v^2-f}\) model. As shown in Fig. 8d for \(t = 40.0\, \text{s}\), 2D-RANS predictions at the leading edge of the GC exceed the SWE prediction by a slight amount. As indicated in Fig. 9, the same trend is observed when the \(k-\varepsilon\) model is considered, except that at latter stage there is better agreement between this 2D-RANS model and the SWE prediction. However, as indicated in the trajectories presented in Fig. 10, the best representation of the leading edge location measured in the experiment at \(t = 40.0\, \text{seconds}\) (\(t^* = 50.1\)) was yielded by the \(\overline{v^2-f}\) model.

As indicated in Figs. 8 and 9, SWE predictions fail to present an accurate depiction of gravity current head throughout the different stages of the flow propagation, particularly at the earlier stages. Both \(\overline{v^2-f}\) and \(k-\varepsilon\) models yield a fair representation of the shape of the GC leading edge. The former presented a better representation of the flow conditions behind the leading edge, as the \(k-\varepsilon\) model (with wall functions) seemed to excessively diffuse this region upstream of the GC head.

The front trajectories are compared in Figure 9 for the two numerical modeling alternatives and the same Marino et al. (2005) experiment analyzed in Figure 7 and Figure 8. All of the numerical models predict the location of the front reasonably well before the end of the initial slumping stage (\(t^* \approx 18\)). In comparison with the experiments, the \(\overline{v^2-f}\) model clearly is the most accurate within the later self-similar stage. The SWE model simulated accurately the initial slumping stage and fairly accurately the unsteady self-similar phase (a closer approximation than \(k-\varepsilon\) model results).
Figure 7 Gravity current propagation comparison between (solid line) SWE and 2D RANS ($\bar{u}^2$-$f$) simulations and experiments (dashed line) by Marino et al. (2005) for (a) $t = 4.5$ s, (b) $t = 6.9$ s, (c) $t = 9.9$ s, and (d) $t = 40.0$ s: $x_0 = 0.1$ m, $L = 3$ m, $h_0 = 0.16$ m, $H = 0.4$ m, $\phi = 0.4$ and $g' = 0.098$ ms$^{-2}$.

For this constant-volume GC flow, the computational times were compared on the same computer with a 2.40 GHz clock CPU. The SWE model simulation time was 6.4 seconds. The $k$-$\varepsilon$ model with wall functions yielded results in slightly over 20 minutes (1,215 seconds) while the $\bar{u}^2$-$f$ model had the longest computational time of almost 5 hours (17,251 seconds). These times indicate that depending on the modeling objectives there may be some advantages in using a simpler SWE simulation for GC flows. This could be the case when a large number of simulations are required, as exemplified in parametric studies such as the ones conducted by Ungarish and Huppert (2002) and Hogg et al. (2005).
Figure 8 Gravity current profile comparison between (solid line) SWE and 2D RANS ($k$-$\varepsilon$) simulations and experiments (dashed line) by Marino et al. (2005) for (a) $t = 4.5$ s, (b) $t = 6.9$ s, (c) $t = 9.9$ s, and (d) $t = 40.0$ s: $x_0 = 0.1$ m, $L = 3$ m, $h_0 = 0.16$ m, $H = 0.4$ m, $\phi = 0.4$ and $g' = 0.098$ ms$^{-2}$.

Figure 9 GC front trajectory comparison between SWE and 2D RANS simulations and Marino et al. (2005) experiment.
5 Conclusions

This paper presents results of an investigation on different BC solution strategies for GC flows using the one-layer SWE, as well as the adequacy of this modeling framework in terms of the prediction of the GC leading edge trajectories. The three BC solution strategies (referred to as GOC, STI and DC approaches) constitute in variations of how flows can be solved at the leading edges of GCs. STI and GOC methods apply characteristic equations and Riemann invariants alongside with a front condition. The DC method enforces explicit mass and momentum conservation for two cells at the leading edge in addition to a front condition. While earlier works have indicated that these strategies were applicable to Boussinesq GC flows, there were no previous studies providing a systematic comparison between these approaches in terms of computational effort and accuracy.

With respect to mass conservation accuracy and computational time, all three approaches performed well. The STI strategy has the simplest implementation and smallest computational effort but presented the largest continuity errors. These errors increased with the depth ratio for constant volume GCs, but were in general smaller for constant-flux simulations. These observations were also valid for the DC method, except that the method has more complex implementation. This Dual-Cell BC method generally had the same computational time as the most efficient approach (STI) but presenting superior accuracy in terms of continuity errors. As anticipated, the GOC model clearly yielded the best accuracy, particularly for deep ambient conditions. Yet, the GOC has the greatest computational cost, about three times more than STI and DC methods using a 500-cell mesh. For larger discretizations, the GOC computational time was so large that it offset the advantages of using SWE to solve GC flows.

This work also assessed the ability of SWE models to present an accurate representation of a GC advance over time by means of comparison with experiments and 2D-RANS simulations. Two CFD models were developed based on $k$-$\epsilon$ and $\tau^2$-$f$ turbulence models, and numerical results were compared to experimental results from Marino et al. (2005) and to respective SWE simulation results. While the 2D-RANS models yielded a fairly accurate representation of the GC head, both the RANS and SWE models performed reasonably well in predicting front trajectories from experimental measurements. Each of the methods were fairly accurate in the slumping stage, but the $\tau^2$-$f$ model was clearly superior in the self-similar stage. With computational efforts orders of magnitude smaller than even the simplest 2D-RANS alternatives, SWE models may constitute an attractive means to obtain large sets of modeling results if the modeling objectives are focused on the description of GC trajectories or average depths.
Notation

CFD = Computational Fluid Dynamics
DC = Dual-Cell front BC solution approach
F(\overline{U}) = vector for the fluxes of conserved variables
F = Froude number at the gravity current front (-)
g' = reduced gravity (ms²)
GC = gravity current
GOC = grid-of-characteristics front BC solution approach
h = gravity current depth (m)
h₀ = initial depth of denser fluid for constant-volume scenario (m)
h_m = depth at inlet for density intrusions (m)
H = ambient depth (m)
k = turbulent kinetic energy (m²s⁻²)
k-ε = k-epsilon turbulence model
L = length of channel/tank
LE = leading edge of gravity current
L_{ke} = turbulent length scale (m)
\text{P}_k = Production of turbulent kinetic energy
u = velocity for SWE implementation (ms⁻¹)
\overline{u} = mean velocity vector for RANS implementations (ms⁻¹)
\overline{U} = vector of conserved variables
q = flow rate per unit width (m²s⁻¹)
RANS = Reynolds-averaged Navier-Stokes equations
STI = specified time intervals front BC solution approach
S(\overline{U}) = source term vectors
\overline{\nu}² = turbulent stress normal to streamlines
\overline{\nu}²-f = eddy viscosity turbulence model without damping functions
xₙ = initial length of denser fluid for constant-volume scenario (m)
x_{LE} = distance from upstream boundary to GC leading edge (m)
Δt = dimension of time discretization (s)
Δx_{LE} = distance between GC front and adjacent-filled cell (m)
Δx = dimension of space discretization (m)
\Delta \rho = difference in fluid densities (kg m⁻³)
\varepsilon = turbulent dissipation (m²s⁻³)
v = kinematic viscosity (m²s⁻¹)
v_t = turbulent viscosity (m²s⁻¹)
\[ \rho = \text{density of fluid (kg m}^{-3}\text{)} \]
\[ \phi = \text{initial depth ratio } h_0/H (-) \]
\[ \phi_{in} = \text{depth ratio at the inlet for density intrusion GCs (-)} \]
\[ \phi_{LE} = \text{depth ratio at GC front (-)} \]

**References**


