Numerical simulations of concrete flow: A benchmark comparison

Roussel, Nicolas; Gram, Annika; Cremonesi, Massimiliano; Ferrara, Liberato; Krenzer, Knut; Mechtcherine, Viktor; Shyshko, Sergiy; Skocec, Jan; Spangenberg, Jon; Svec, Oldrich

Published in:
Cement and Concrete Research

Link to article, DOI:
10.1016/j.cemconres.2015.09.022

Publication date:
2016

Document Version
Peer reviewed version

Link back to DTU Orbit

Citation (APA):

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.
Abstract: First, we define in this paper two benchmark flows readily usable by anyone calibrating a numerical tool for concrete flow prediction. Such benchmark flows shall allow anyone to check the validity of their computational tools no matter the numerical methods and parameters they choose. Second, we compare numerical predictions for these two benchmark flows obtained by various research teams around the world using various numerical techniques. Our results show that all numerical techniques compared here give very similar results suggesting that numerical simulations of concrete flow have reached a technology readiness level allowing them to move from the lab to the industrial practice.
NUMERICAL SIMULATIONS OF CONCRETE FLOW: A BENCHMARK

COMPARISON

Nicolas Roussel¹, Annika Gram², Massimiliano Cremonesi³, Liberato Ferrara³, Knut Krenzer⁴, Viktor Mechtcherine⁵, Sergiy Shyshko⁵, Jan Skocec⁶, Jon Spangenberg⁷, Oldrich Svec⁷, Lars Nyholm Thrane⁸, Ksenija Vasilic⁹

¹ Université Paris Est, IFSTTAR, laboratoire Navier, France.
² Swedish Cement and Concrete Research Institute, Stockholm, Sweden.
³ Department of Civil and Environmental Engineering, Politecnico di Milano, Italy.
⁴ IAB Weimar - Institut für Angewandte Bauforschung Weimar GmbH, Germany.
⁵ TU Dresden, Institute of Construction Materials, Dresden, Germany.
⁶ Heidelberg Cement Technology Center GmbH, Leimen, Germany.
⁷ Department of Mechanical Engineering, Technical University of Denmark, Denmark.
⁸ Danish Technological Institute, Denmark.
⁹ BAM, Federal Institute for Material Research and Testing, Berlin, Germany.

Highlights

We define two benchmark flows for comparison of numerical simulations for concrete flow.
We describe the three most used numerical methods for concrete flow simulations.
We compare the predictions of these methods when used by various research teams.
We conclude on the technological readiness level of these numerical tools.

Abstract
First, we define in this paper two benchmark flows readily usable by anyone calibrating a numerical tool for concrete flow prediction. Such benchmark flows shall allow anyone to check the validity of their computational tools no matter the numerical methods and parameters they choose. Second, we compare numerical predictions for these two benchmark flows obtained by various research teams around the world using various numerical techniques. Our results show that all numerical techniques compared here give very similar results suggesting that numerical simulations of concrete flow have reached a technology readiness level allowing them to move from the lab to the industrial practice.

Keywords
Fresh Concrete (A) ; Rheology (A) ; Workability (A) ; Modeling (E) ; casting

1. Introduction

The first use of numerical simulations of concrete flow by Mori and Tanigawa traces back to 1992 [1]. Since then, the increasing use of Self Compacting Concrete (SCC) and the growing interest for rheology and processing has led to a very strong increase in the academic activity in this field along with an increasing number of publications dealing with concrete flow simulations using various numerical techniques. This topic is of a complex nature as it involves the non-steady free surface flow of a non-Newtonian fluid. Therefore, it requires the knowledge of modern computational techniques, non-Newtonian fluid mechanics and the specific concrete casting processes used in civil engineering. A lot of progresses have however been made in recent years. They are gathered in a recent RILEM State of the Art report [2].
Despite this academic activity, the use of concrete flow simulations in industrial practice is still sporadic. It can of course be expected that no one should consider simulating the casting of a residential concrete slab. It is however surprising that the optimization of pre-cast factory processes or the possibility to numerically forecast a critical phase of a concrete construction process in the case of advanced super-structures has not drawn much attention from the industry yet. We can note however that the use of these tools seem to have been steadily increasing in the field of litigation as they allow, in many case, to distinguish between the responsibility of the contractor and the responsibility of the concrete supplier.

It is our belief that the use of these advanced engineering tools is hampered by their diversities and the fact that, as for all computational tools, one always get a result but, without experience, one has no clue about the meaningfulness of the obtained prediction. This paper focuses therefore on demonstrating that numerical simulations of concrete flow are now fully able to predict accurately concrete flows are now a mature technology fully able to accurately predict concrete flows and that these scientific and engineering tools are now ready to be used for a wide range of either academic or industrial purposes.

Our objectives here are two-fold. First, we define two benchmark flows readily usable by anyone calibrating a numerical tool for concrete flow prediction. Such benchmark flows shall allow anyone to check the validity of their computational tools no matter which numerical methods and parameters are chosen. Second, we compare numerical predictions for these two benchmark flows obtained by various research teams around the world using various numerical techniques. Our results suggest that numerical simulations of concrete flow have reached a technology readiness level allowing them to move from the lab to the industrial practice.
2. The studied benchmark flows

We choose in this work to study and compare numerical predictions for two benchmark flows, namely the slump flow and the channel flow. These tests were chosen because approximate analytical solutions for the final shape of the concrete sample do exist. It can moreover be kept in mind that two benchmark flows are necessary, as all numerical methods compared here are not using the same input parameters. For instance, whereas Computational Fluid Dynamics methods (CFD) are using standard rheological parameters such as yield stress and plastic viscosity as input, Distinct Element Methods (DEM) are using interaction parameters between constitutive particles chosen to mimic the behavior of a given concrete (Cf. section 3). We use therefore in this paper the slump flow as a calibrating benchmark flow for all methods while we compare the numerical predictions in the case of the channel flow.

2.1. The virtual concrete

The virtual concrete, which is studied here and implemented in the codes of the various CFD tools that are compared, can be described by a Bingham model. It has a yield stress of 50 Pa, a plastic viscosity of 50 Pa.s and a density of 2300 kg/m$^3$. This concrete is virtually mimicked by the DEMs by tuning the particle interaction laws until the behavior of the resulting virtual material is similar to the one of a 50 Pa yield stress material with a plastic viscosity of 50 Pa.s.
2.2. The slump flow

We are considering here the standard Abrams cone geometry, which corresponds to $H_0 = 300$ mm, $R_{\text{min}} = 50$ mm and $R_{\text{max}} = 100$ mm using the notations in Fig. 1.

Roussel and Coussot [3] described the various relations existing between slump/slump flow and yield stress from an analytical point of view for two asymptotic situations, namely $H >> R$ and $H << R$ ($H$ and $R$ being the height and radius of the sample, respectively). In these two situations, the governing flow equations can be significantly simplified in order to obtain approximate analytical solutions. We focus here on the case $H << R$, which corresponds to the slump flow regime of interest in this paper. The three-dimensional flow problem simplifies then to a one-dimensional equation. The yielding criterion becomes mono-dimensional and flow simply stops when shear stress in the sample becomes equal or lower than the yield stress $\tau_c$. It was shown in [3] that the final shape can then be computed from:

$$h(r) = \left(\frac{2\tau_c(r-H)}{\rho g}\right)^{\frac{1}{2}}$$

where $h$ is the height, $r$ the radial coordinate, $\rho$ the density and $g$ the gravity. Knowing the sample volume $V$ and neglecting inertia and surface tension [4], the following equation relates the yield stress to the final radius of the sample:

$$\tau_c = \frac{225\rho H^2}{128\pi^2 h^3}$$

2.3. The channel flow

The geometry we consider here is the one suggested in [5] and shown in Fig. 2. The channel width is 200 mm. Its length is higher than 900 mm and the height of the lateral walls is higher than 150 mm. This geometry allows for a quick and easy measurement of the yield
stress of SCC as there exists an analytical relation between yield stress and final channel flow length \( L \) when concrete is not reaching the end of the channel [5, 6]. This relation writes:

\[
L = \frac{\eta u_0}{i_c} + \frac{i_c}{2\Lambda} LN \left( \frac{u_0}{l_u + 2h_u} \right)
\]

(3)

Where \( h_u \) is the thickness of the deposit at \( x = 0 \), \( l_u \) is the width of the channel and \( \Lambda = \frac{2\tau_c}{\rho g l_u} \).

3. **Numerical methods**

Table 1 gathers the various numerical methods and their variants used by the researchers involved in the present study. More details on the methods can be found below or in [2,7, 8].

4. **Computational Fluid Dynamics**

Computational Fluid Dynamics (CFD) is the part of fluid mechanics that refers to the use of numerical methods and algorithms to solve and analyze fluid flow problems. Many researchers have carried out simulations of concrete flow within the framework of CFD [2, 8]. Simulations have mostly been used to model the flow of fresh concrete during testing including both standard test methods and viscometers. However, a few examples of computational modeling of mixing and full-scale castings can also be found [2, 8].

4.1. **Governing equations**

The solution to a concrete flow problem involves the computation of properties such as velocity, pressure, density and temperature as functions of space and time from conservation of momentum, mass, and energy. The variation in temperatures can often be
neglected though, as it is for these two benchmark flows. It can be noted that concrete, within this frame, is often assumed to be incompressible.

4.2. Behavior law

When using CFD for concrete, the constitutive equation allowing for the description of the fresh material properties is usually derived from the generalized Newtonian model [9, 10]. It can either take the form of the Bingham model or of the Herschel Buckley model, which both involves the input of a yield stress that must be exceeded for flow to occur. Examples of more sophisticated material models of thixotropic nature can also be used as presented in [11-13]. From a numerical point of view, implementing a pure yield stress fluid model (i.e. perfect elasto-visco-plastic behavior) is at the origin of many numerical issues. Indeed, in non-flowing zones or in plug-flow zones, the material average strain rate is zero and the apparent local viscosity shall then be infinite and lead to computational divergence. Therefore, approximations based on a smoothing of constitutive relationships often replace the exact behavior laws [14-16]. Using these approximations, the apparent local viscosity is often capped by a maximum value below a critical strain rate. The drawback from such approximations is that the simulated material never really stops flowing. It becomes therefore sometimes necessary to define a flow stop criterion, which allows for the definition of the final shape of the free surface of the material.

4.3. Specificities of each team

Volume of fluid method – DTU / IFSTTAR / CBI / BAM

The volume of fluid method is based on the donor-acceptor approach first introduced in [17]. In a first step, the algorithm computes the volumetric fluxes by geometrically
reconstructing the interface using the values of the fluid fraction function at the previous
time step in and around a given control volume. In the second step, the Navier-Stokes
equations are solved to compute the velocity field. The fluid is then advected accordingly
and the local fluid fraction function is updated. The discretization of the Navier-Stokes
equations is carried out with the Finite Volume Method (FVM) in order to obtain the velocity
field. In addition, the Volume Of Fluid (VOF) method [17] is used to advect the fluid. In a first
step, the VOF algorithm geometrically reconstructs the interface using the values of the fluid
fraction function at the previous time step in and around a given control volume. In the
second step, the fluid is then advected accordingly by updating the fluid fraction function
based on the reconstructed interface and the previously obtained velocity field. The codes
used to solve the flow are FLOW3D® for IFSTTAR and DTU, OPENFOAM for CBI and FLUENT®
for BAM. The models developed here mostly differ by the approximations used to soften the
constitutive laws as described above.

Particle finite element method - Politecnico di Milano
In the approach developed by the group at Politecnico di Milano [18, 19], the governing
equations are discretized according to the Lagrangian approach, which, by tracking the
motion of the material particles, is able to automatically capture the evolution of the free
surface configuration, without any a-priori imposed limit to the description of the fluid
movement. The FEM mesh remains therefore attached to the material nodes, and a re-
meshing may be necessary to remedy the resulting severe mesh distortion. The constitutive
law is the Papanastasiou exponential approximation of a Bingham fluid [16].

5. Distinct element methods
Discrete numerical simulations of granular materials or Distinct Elements Methods (DEM) give access to mesostructure or even microstructure at the scale of the grains and their contacts using various methods such as the ones described in [2]. These approaches provide therefore an opportunity to study both the flow of the concrete mixture as a whole and the displacement of its individual components. The concrete mixture is modeled by a large number of particles, which are interacting through contact laws. The definition of these contact laws allows for the creation of a virtual concrete mimicking the behavior of the real material.

5.1. Governing equations

In DEM, calculations alternate between the application of Newton’s Second Law with respect to the motion of particles and the force-displacement law at the contact points between particles [20-22]. The contact force vector can be divided into normal and shear components with respect to the contact plane. The force-displacement law relates these two force components to the corresponding components of the relative displacement [20-22].

The computation time is directly proportional to the number of particles and increases therefore with decreasing particle size for a given material volume in the simulation.

5.3. Specificities of each team

IAB Weimar

The IAB Weimar (former IFF Weimar) developed a user defined DEM contact model both in PFC 3D and in EDEM for the representation of fresh concrete. The approach is based on three main aspects: local shear rates, two-layer particle representation and shear angle
dependent normal force. When two particles are in contact, the distance of their centers of mass and their relative velocities are used to compute an approximated local shear rate. Using this shear rate and an approximated contact area, based on the particle sizes and the amount of contacts, the contact force is computed based on the Bingham model and its related parameters: yield stress and plastic viscosity. Using the standard parameters for the Bingham model is easing the calibration of the DEM for a given fresh concrete and also enables a particle size independency of the parameters.

In the IAB Weimer approach, the particles are represented as two-layer particles with an inner core representing the real coarse aggregate and an outer core representing the surrounding mortar layer. The outer layer is obviously softer than the inner core and the Bingham model described above is only applied when the outer cores are in contact. As soon as the inner cores get in contact, a normal friction model is applied. The two-layer approach decreases significantly the amount of particles to be computed in comparison to a mortar representation by smaller particles. The computation time is therefore reduced to the same extent.

Finally, to reach a smoother movement of the particles and improve the representation of the mortar layer in the outer core, a shear angle dependent normal force is also implemented in the case of a layer contact. This results in a reduction of the additional overlap force in normal direction, depending on the tangential degree of the relative particle movement.

TU DRESDEN

The interaction laws for the normal and tangential direction used by the team at TU Dresden are shown in Figure 3. They consist of the basic rheological elements spring, dashpot, and
slider, which respectively represent the elastic, viscous and friction components of the particles interaction. A contact element enables the definition of the strength of the contact, the simulation of the loss of an old interaction due to the reaching of a certain distance between two particles, and the formation of a new interaction [22].

The force-displacement curve in tension mode is defined for small deformations by a very steep ascending branch (i.e. there is practically no deformation until a given force value (here “yield force”) is reached). After reaching this force level, there is only a slight increase in tensile force up to a defined ultimate force (called bond strength) and then a linear decrease to zero in a kind of softening regime. When the tensile force becomes zero, particles lose contact. It can be kept in mind that bond strength is the main parameter defining the interaction of neighboring particles. The bond strength value corresponding to a fresh SCC with yield stress equal to 50 Pa was computed using the methodology presented in [23].

Concrete was modeled by particles of different sizes in such a way that a realistic grading curve of aggregates could be represented with good approximation. The particles were first randomly distributed over a given volume and then compacted under the effect of gravity.

6. **Lattice Boltzmann methods**

The Lattice Boltzmann Method (LBM) used at DTU originated around the end of 1980s when cellular automata began to be applied to the field of fluid dynamics. Originally, researchers dealt with lattice gas cellular automata and, only in the beginning of 1990s, LBM was developed and applied to the field of fluid dynamics, see e.g. [24, 25] for an overview of the history of the method. LBM, contrary to the aforementioned traditional methods formulating the problem by means of macroscopic quantities such as macroscopic velocity
or pressure fields, is based on Boltzmann equations and thus on the theory of ideal gases. It
defines the problem in terms of particle distribution functions, which can be seen as clouds
of microscopic particles, e.g. molecules, chaotically moving in space. Space is commonly
discretized into square (2D domain) or cubic Eulerian cells forming a fixed Cartesian grid (see
Fig. 4)). Eulerian nodes are placed into the centre of each cell and a lattice is formed by
connecting the nearest Eulerian nodes. The movement of the microscopic particles is
restricted onto the lattice directions, which results in a discretization of continuous particle
distribution functions into only a few particle distributions associated with lattice velocities
(Fig. 4)).

The spacing of nodes, the reference density of the fluid and the length of the time step are
usually set to unity for the most simple and common lattice types and the simulated
problem is scaled accordingly. Therefore, all quantities are assumed to be dimensionless in
this Section. All the equations presented in this section and their derivations can be found
e.g. in [26] and references herein. An evolution of the particle distributions in space and
time is expressed by the lattice Boltzmann equation, which consists in:

A streaming part, which represents propagation of particle distributions from one
node to another along the lattice links. The lattice velocities are chosen in such a way that a
particle distribution streams from one node to another during one unit time step.
A forcing term accounting for a modification of particle distributions due to the effect
of an external force, see e.g. [27].
A collision part, which mimics real collisions between particles. The collisions are
difficult to express explicitly and, therefore, an approximation in the form of a collision
operator was introduced into the LBM. The so-called BGK operator, see e.g. [24], linearly
deforming the actual particle distributions towards an equilibrium state, is one of the most
widely used collision operators. The equilibrium state is represented by equilibrium particle
distributions, which are derived from the Maxwell-Boltzmann equation. The local densities
and velocities of the fluid are computed as the first two moments of the particle
distributions. The rate of the deformation of the particle distributions towards the
equilibrium state is introduced by a relaxation time computed from local macroscopic
apparent viscosity (note that kinematic and dynamic viscosity are equal due to the
dimensionless form). For non-Newtonian fluids, viscosity is a function of the shear rate and
stress deviator tensor, which can be approximated from local off-equilibrium parts of
particle distributions [28].
Due to the kinematic nature of the LBM and to accuracy and stability issues for high values
of relaxation times, behavior of the Bingham plastic cannot be modeled exactly. Recently, a
lot of effort has been given into investigations of modeling of Bingham rheology by the LBM,
see e.g. [29-32]. Here, the Bingham rheology is approximated by essentially a tri-viscosity
fluid, for which we consider that flow has stopped when relaxation time has reached a
critical value in a given fraction of the computation cell.
We used the Mass Tracking Algorithm [33] to model the free surface. The adopted mass-
tracking algorithm is applied directly at the level of the LBM, so the algorithm mimics the
free surface by modifying the particle distributions. Lattice cells are differentiated into fluid,
gas and interface cells. Gas cells do not contain any particle distributions and thus carry no
information, i.e. the LBM is not applied in the gas cells. Fluid cells behave as ordinary LBM
cells. Interface cells form a boundary layer between gas cells and fluid cells. Interface cells
are treated in the same manner as fluid cells with additional information, namely a local
mass of the fluid in the cell. An exchange of mass between cells is calculated based on
streamed-out and streamed-in particle distributions. The interface cell becomes a fluid cell
when the mass reaches its density and, vice versa, the interface cell becomes a gas cell when
the mass drops down to zero. Particle distributions that would be streamed from gas cells to
interface cells do not exist. Therefore, they are reconstructed from particle distributions
streamed from interface cells to gas cells based on mass, momentum and pressure
equilibriums. Hence, the MTA conserves mass precisely. In the presented benchmarks, we
used D3Q15 lattice – a three dimensional lattice with 15 lattice velocities - and spacing of
lattice nodes equal to 5 mm.

7. Comparison between numerical predictions

We only focus here on the final shape of the studied concrete volume when flow stops. This
information is indeed of first order importance when one is interested in checking if a
casting process leads to the proper filling of a given mold or formwork.

7.1. Slump flow

We plot in Fig. 5 the various numerical predictions for the benchmark slump flow test
studied here. It can first be kept in mind that the approximate analytical solution plotted in
Fig. 5, which predicts a slump flow diameter of 60 cm, is not correct on the symmetry axis.
This solution, to be kept analytical and simple, is indeed based on a scalar plasticity criterion,
which only involves shear stress [3]. On this symmetry axis, the stress tensor turns, however,
into a purely diagonal matrix, typical of an extensional or elongational flow and the plasticity
criterion shall involve the three terms of the diagonal [34]. The shear stress being equal to
zero on the axis forces the surface slope of the sample locally to be equal to zero, which is
predicted by most numerical techniques shown in Fig. 5 but not by the approximate analytical solution.

It can moreover be noted that CFD methods and LBM, without any fitting, give very similar results and predict a slump flow diameter of average 59.1 cm. The relative standard deviation between these predictions is 1.8 %. The DEM methods parameters and particle interaction laws of TU Dresden are fitted on this first benchmark test. After this fitting phase, all methods predict very similar slump flow diameters with an average value of 58.8 cm and a relative standard deviation of 2.29 %.

7.2. Channel flow

We plot in Fig. 6 the numerical predictions of the channel flow for the various numerical methods presented above. It can be reminded that the exact same input parameters and numerical method as in the previous section are used here. All methods predict very similar profiles at flow stoppage with an average flow length value of 69.7 cm and a relative standard deviation of 2.95 %. The results in Fig. 6, obtained with very different techniques and input parameters, show that these numerical methods are all able to predict concrete flow and the conditions, under which it stops.

8. Conclusions

This paper focused on demonstrating that numerical simulations of concrete flow are now fully able to predict accurately concrete flows and that these scientific and engineering tools are ready to be used widely for either academic or industrial purposes.

First, we defined two benchmark flows readily usable by anyone calibrating a numerical tool for concrete flow prediction. Such benchmark flows shall allow anyone to check the validity
of their computational tools regardless of the numerical methods and parameters they choose. Second, we compared numerical predictions for these two benchmark flows obtained by various research teams around the world using various numerical techniques. Our results showed that all numerical techniques compared here give very similar results suggesting that numerical simulations of concrete flow have reached a technology readiness level allowing them to move from the lab to the industrial practice.

REFERENCES


Table 1. Numerical methods and softwares used by the researchers involved in the present study.

<table>
<thead>
<tr>
<th>Contact name</th>
<th>Country</th>
<th>Research group</th>
<th>Commercial Software</th>
<th>Numerical method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liberato Ferrarra</td>
<td>Italy</td>
<td>Politecnico di Milano</td>
<td>None</td>
<td>CFD - Particle Finite Element method</td>
</tr>
<tr>
<td>Nicolas Roussel</td>
<td>France</td>
<td>IFSTTAR</td>
<td>FLOW 3D ©</td>
<td>CFD - Volume of Fluid method</td>
</tr>
<tr>
<td>Ksenija Vasić</td>
<td>Germany</td>
<td>BAM</td>
<td>ANSYS Fluent ©</td>
<td>CFD - Volume of Fluid method</td>
</tr>
<tr>
<td>Annika Gram</td>
<td>Sweden</td>
<td>CBII</td>
<td>OpenFOAM</td>
<td>CFD - Volume of Fluid method</td>
</tr>
<tr>
<td>Jon Spangerberg</td>
<td>Denmark</td>
<td>DTU</td>
<td>FLOW 3D ©</td>
<td>CFD - Volume of Fluid method</td>
</tr>
<tr>
<td>Knut Kröner</td>
<td>Germany</td>
<td>IAB Weimar</td>
<td>PFC 3D ©</td>
<td>Distinct Element method</td>
</tr>
<tr>
<td>Viktor Mechtcherine</td>
<td>Germany</td>
<td>TU Dresden</td>
<td>PFC 3D ©</td>
<td>Distinct Element method</td>
</tr>
<tr>
<td>Jan Skocek</td>
<td>Denmark</td>
<td>DTU</td>
<td>None</td>
<td>Lattice Boltzmann method</td>
</tr>
</tbody>
</table>
Figure 1. Initial cone shape and cylindrical coordinates.
Figure 2. The channel flow and its geometry.
Figure 3. Model for particle interaction: (a) normal direction and (b) tangential direction [22].
Figure 4. Scheme of so-called D2Q9 lattice for Lattice Boltzmann method. Square marks stand for nodes, grey lines for cell boundaries and dashed lines for the lattice. b) Set of corresponding lattice velocity vectors in a node.
Figure 5. Comparison of the various simulations for the slump flow
Figure 6. Comparison of the various simulations for the channel flow.