The design of slushflow barriers: OpenFOAM simulations

Halldór Pálsson*, Ásdís Helgadóttir and Rebecca Anne Jones

University of Iceland, Hjarðarhagi 2–6, IS-107 Reykjavík, ICELAND
**Corresponding author, e-mail: halldorp (at) hi.is*

ABSTRACT

Understanding the dynamics of slushflows and snow avalanches plays a major role in estimating their flow in natural terrain and their effect on obstructions and man-made structures and, and more importantly, risk assessment regarding people's safety. Several factors contribute to the high complexity of such flows, among them the geometrical complexity of the flow path (ground), the physical behaviour of free-surface flows where complex hydraulic jumps occur, and the non-Newtonian fluid properties in the case of snow avalanches. Finally, the understanding of the flow dynamics is fundamental in the design of flood mitigation structures, both in terms of their strength and effectiveness in directing floods away from sensitive structures and people.

Experiments with scaled-down models have been used for decades to visualize floods and estimate their effect on sensitive structures. The scaling itself must be carefully conducted in order to preserve the fundamental behaviour of floods, which can be difficult in some cases, e.g. when both the dynamic similarity of the Reynolds and Froude numbers should be preserved. Nevertheless, experiments and measurements are considered the best method of acquiring accurate results, but they are in most cases quite time-consuming and costly to perform.

Computational fluid dynamics (CFD) have become an important tool in flow simulations because of increased number-crunching abilities of modern computers and the use of clusters for large-scale computational problems. Despite this, modelling of complex phenomena such as turbulent flow and free-surface flows still poses a great challenge. Nevertheless, many free-surface flow problems have been investigated using CFD methods, some of which resembling slushflows and to some extent snow avalanches.

In the current work, two CFD models have been constructed, using the public domain OpenFOAM CFD software, in order to simulate results from a slushflow laboratory experiment where different set-ups of barriers were tested. The purpose was to determine an efficient design for a slushflow mitigation structure (see the paper by Hákonardóttir and others in this volume). One of the models assumes a wide uniform channel, and is therefore implemented as a 2D problem, but the other is fully three-dimensional. Both models simulate the full Navier–Stokes equations, with two phases present (liquid and air), and using a surface-capturing algorithm to model the interface between the two phases.

The results show that the CFD models can replicate some of the actual results from the laboratory experiments remarkably well, which indicates that three-dimensional CFD models could be a valuable tool in the designs of slushflow mitigation structures and in the design of experiments. It appears possible to conduct initial laboratory experiment to calibrate a suitable CFD model, which is then used in a series of numerical experiments to optimize the design of the structure being considered, and finally perhaps verify the optimized design with a series of laboratory experiments. Further work could involve simulating a non-Newtonian fluid with properties that resemble the granular rheology of snow in a dry-snow avalanche.