Tightly Coupled Computational Fluid and Crowd Dynamics

Rainald Löhner[1] and Fernando Camelli[1]

[1] Center for Computational Fluid Dynamics
George Mason University, Fairfax, VA 22030, USA
e-mail: rlohner@gmu.edu, fcamelli@gmu.edu, web page: https://cfd.gmu.edu/

ABSTRACT

Advances in computational fluid and crowd dynamics (CFD, CCD), as well as computer hardware and software, have enabled fast and reliable simulations in both disciplines. A natural next step is the coupling of both disciplines. This would be of high importance for evacuation studies where fire, smoke, visibility and inhalation of toxic materials influence the motion of people, and where a large crowd can block or influence the flow in turn. Cases where this may occur are subway stations, gangways, cinemas or dance clubs. In each of these the crowd can block a considerable portion of the passage area, thereby influencing the flow.

The present work considers a tight, bi-directional coupling, whereby the flow and the motion of the crowd are computed concurrently and with mutual influences. Enabling technologies that led us to consider this tight coupling as feasible include: a) Development of immersed boundary methods; b) Implementation of fast search techniques for information transfer between codes; c) Strong scaling to tens of thousands of cores for CFD codes.

The coupling methodoly proceeds as follows: The CFD code computes the flowfield, providing such information as temperature, smoke and toxic substance concentration, and any other flow quantity that may affect the movement of pedestrians. These variables are then interpolated to the position where the pedestrians are, and are used with all other pertinent information (e.g. will-forces, targets, exits, signs, etc.) to update the position, velocity, inhalation of smoke and/or toxic substances, state of exhaustion or intoxication, and any other pertinent quantity that is evaluated for the pedestrians. The position, velocity and temperature of the pedestrians is then transferred to the CFD code and used to modify and update the boundary conditions of the flowfield in the regions where pedestrians are present.

Figure 1 shows pedestrians and the resulting flowfield in narrow, winding corridor in the presence of pedestrians. The CFD mesh had 6.7 Mels. The case was run on an SGI ICEx machine using 48 cores (6 mpi-domains x 8 omp-cores for each domain). Note the wake of the pedestrians, as well as the change in the overall flow pattern as a result of the presence of pedestrians in the flowfield.

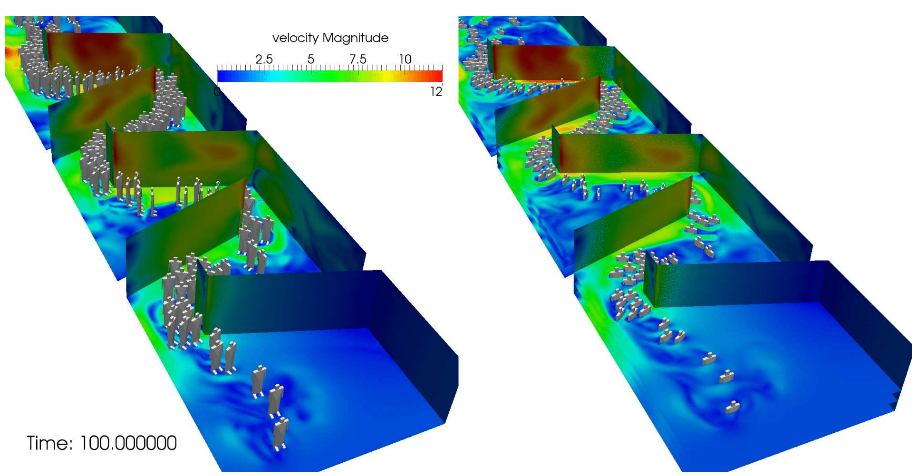

Figure 1 Pedestrians and Flowfield at Time $T = 100$ sec