VII International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2017
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

TIGHTLY COUPLED COMPUTATIONAL FLUID AND CROWD DYNAMICS VIA IMMERSED BOUNDARY METHODS

RAINALD LÖHNER[1], FERNANDO CAMELLI[1] AND EUGENIO OÑATE[2]

[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/

[2] International Center for Numerical Methods in Engineering (CIMNE)
Universidad Politécnica de Cataluña
Campus Norte UPC, 08034 Barcelona, Spain
e-mail: onate@cimne.upc.edu, web page: http://www.cimne.com/

Key words: Computational Fluid Dynamics, Computational Crowd Dynamics, Pedestrian Flows, Coupled Problems, Fire and Rescue Operations, Event Planning, Supercomputing

Abstract. A methodology to couple computational fluid and computational crowd dynamics (CFD, CCD) has been developed. Technological advances that made this possible include: a) Mature CFD and CCD codes/solvers; b) Development of immersed boundary methods for moving bodies (CFD); c) Strong scaling to tens of thousands of cores (CFD); and d) Implementation of fast search techniques for information transfer between codes (CFD, CCD).
We consider that tightly coupled simulations such as the ones presented here will lead to more realistic 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 metro-stations, high-rise buildings and indoor sports arenas, where a crowd can block a considerable portion of the passage area, thereby influencing the flow.

1 INTRODUCTION

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 metro-stations or high-rise buildings, where a crowd can block a considerable portion of the passage area, thereby influencing the flow. To date, most of the coupling attempts have been uni-directional: in a first step, the flow is precomputed; thereafter, the crowd moves under the influence of this precomputed fire and smoke field. This type of coupling has several justifications: a) In many cases the presence of pedestrians has a negligible influence on the flow; b) Historically, CFD runs were orders of magnitude more expensive than CCD runs; having to run CFD codes to 30-60 mins of simulation time was prohibitive. 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; and c) Strong scaling to tens of thousands of cores for CFD codes.

2 COUPLING METHODOLOGY

The coupling methodoly used is shown in Figure 1. 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 relevant quantities that are 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.

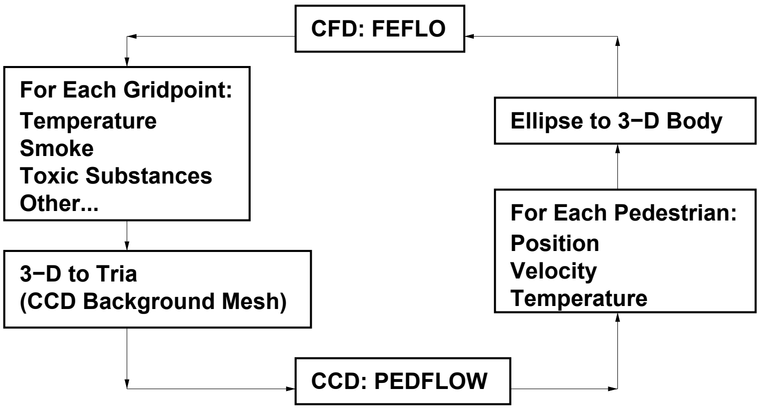


Figure 1 Coupling CFD and CCD Codes

Of the many possible coupling options (see e.g. [1, 13, 2]), we have implemented the simplest one: loose coupling with sequential timestepping ([7, 11]). This is justified, as the timesteps of both the flow and pedestrian solvers are very small, so that possible coupling errors are negligible.

3 COMPUTATIONAL FLUID DYNAMICS CODE: FEFLO

The FEFLO code has been in development and use for over two decades ([6, 14, 8, 9]). It solves the the incompressible Navier-Stokes and an arbitrary number of transport equations. The code has had a long history of relevant applications involving incompressible flows of various types, from external flow of automobiles to wind engineering and climate control. FEFLO has been ported to vector, shared memory, distributed memory and GPU-based machines.

The information required from CCD codes such as PEDFLOW consists of the pedestrians in the flowfield, i.e. their position, velocity and temperature. As the CCD codes describe the pedestrians as points, circles or ellipses, a way has to be found to transform this data into 3-D objects. Two possibilities have been pursued here:

a) Transform each pedestrian into a set of (overlapping) spheres that approximate the body with maximum fidelity with the minimum amount of spheres;

b) Transform each pedestrian into a set of tetrahedra that approximate the body with maximum fidelity with the minimum amount of tetrahedra.

The reason for choosing spheres or tetrahedra is that one can perform the required interpolation/ information transfer much faster than with other methods.

In order to 'impose' on the flow the presence of a pedestrian the immersed boundary methodology is used. The key idea is to prescribe at every CFD point covered by a pedestrian the velocity and temperature of the pedestrian. For the CFD code, this translates into an extra set of boundary conditions that vary in time and space as the pedestrians move. This is by now a mature technology. Fast search techniques as well as extensions to higher order boundary conditions may be found in [9]. Nevertheless, as the pedestrians potentially change location every timestep, the search for and the imposition of new boundary conditions can add a considerable amount of CPU as compared to 'flow-only' runs.

4 COMPUTATIONAL CROWD DYNAMICS CODE: PEDFLOW

The PEDFLOW code ([10, 12]) has been in development and use for more than a decade. It uses a combination of force-based and agent-based methods. Individuals move according to Newton's laws of motion; they follow (via will forces) 'global movement targets'; at the local movement level, the motion also considers the presence of other individuals or obstacles via avoidance forces (also a type of will force) and, if applicable, contact forces. A complete description of the model, as well as verificaton and validation studies may be found in [10, 3, 4, 15, 5]. Over the last two years the code has been parallelized for both shared (via OpenMP at the loop-level) and distributed (via MPI and domain decomposition) memory architectures. This has allowed real-time micro-

modelling of a million pedestrians ([12]). The geographic information required, such as terrain data (inclination, soil/water, escalators, obstacles, etc.), climate data (temperature, humidity, sun/rain, visibility), signs, the location and accessability of guidance personnel, as well as doors, entrances and emergency exits is stored in a so-called background grid consisting of triangular elements. This background grid is used to define the geometry of the problem. At every instance, a pedestrian will be located in one of the elements of the background grid. Given this 'host element' the geographic data, stored at the nodes of the background grid, is interpolated linearly to the pedestrian. The closest distance to a wall δ_w or exit(s) for any given point of the background grid evaluated via a fast ($O(N \ln(N))$) nearest neighbour/heap list technique ([9, 10]). For cases with visual or smoke impediments, the closest distance to exit(s) is recomputed every few seconds of simulation time.
The information required from CFD codes such as FEFLO consists of the spatial distribution of temperature, smoke, or other toxic or movement impairing substances in space. This information is interpolated to the (topologically 2-D) background mesh at every timestep in order to calculate properly the visibility/ reachability of exits, routing possibilities, smoke and toxic substance inhalation, and any other flowfield variable required by the pedestrians. As the tetrahedral grid used for the CFD code and the triangular background grid of the CCD code do not change in time, the interpolation coefficients need to be computed just once at the beginning of the coupled run. While the transfer of information from CFD to CCD is voluminous, it is very fast, adding an insignificant amount to the total run-times.

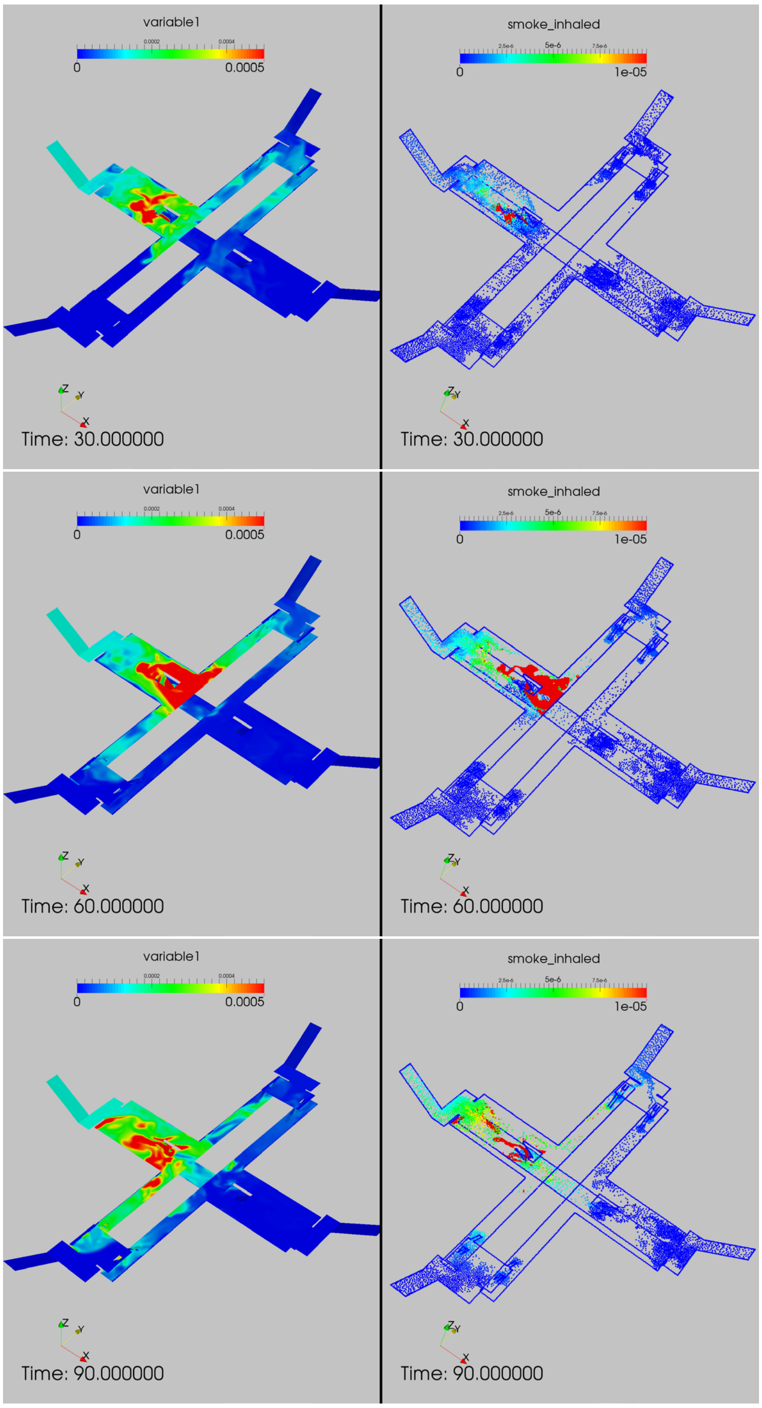
5 COUPLING CODE: FEMAP

The transfer of information is guided by the general coupling code FEMAP. FEMAP calls each of the codes as a subroutine, and performs the timestepping as specified by the user. FEMAP is presently linked to several computational structural dynamics (CSD) codes (among them FEEIGEN, DESOL, ASICSD, DYNA3D, SIMPACT, FEAP, NASTRAN), several CFD codes (among them FEFLO and FDFLO), several computational thermodynamics (CTD) codes (among them FEHEAT and NASTRAN) and, as of 2016, also PEDFLOW. FEMAP allows for the use of different levels of parallelism and processors for the different codes, and allows the concurrent use of several CSD, CFD and CTD at the same time.

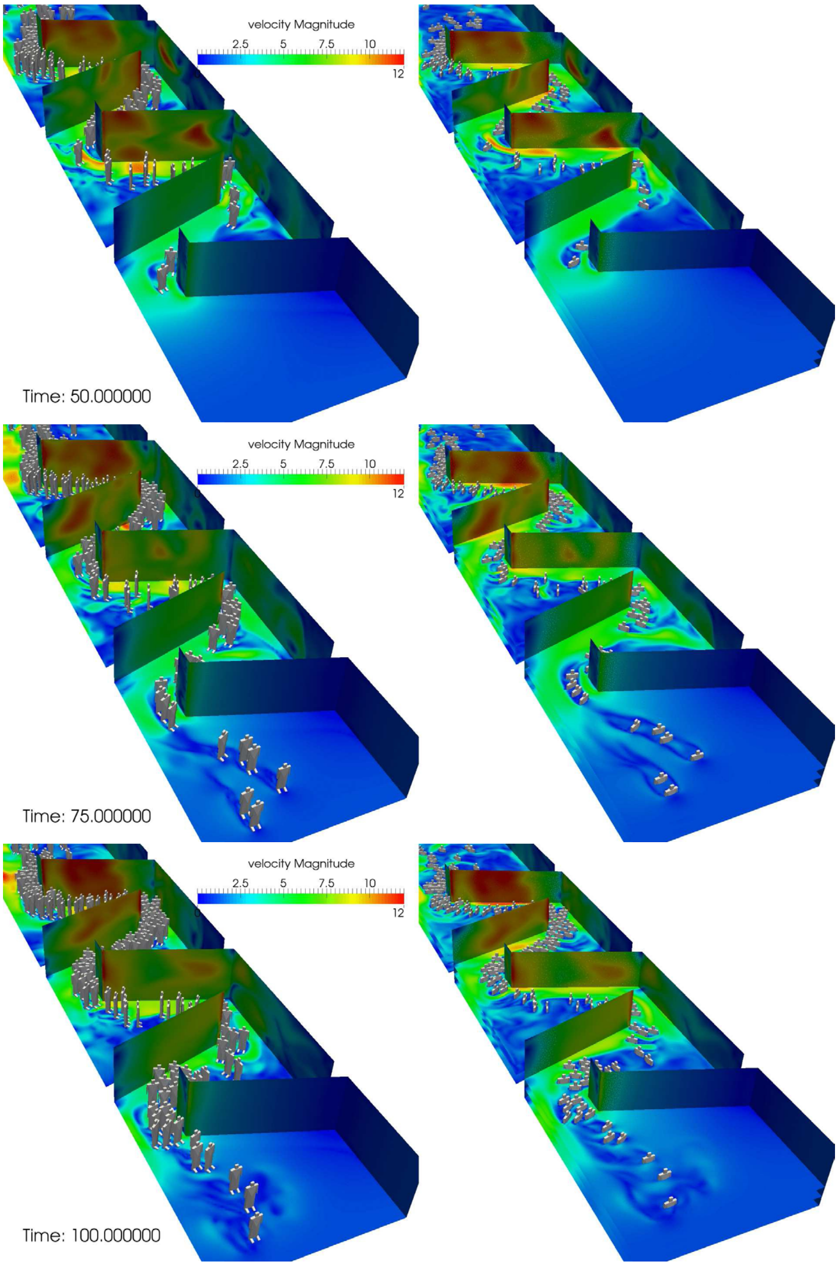
6 EXAMPLES

1. Metro Center

This example considers an evacuation case due to fire in the Metro Center metro station of Washington, D.C. A fire was assumed in one the the escalators. The flow was computed, together with the production of smoke.



Figures 2.1-2.3: Metro Station Evacuation:
Left: Smoke Density, Right: Pedestrians and Limit of Visibility at Times $T = 30, 60, 90$ *sec*



Figures 3.1-3.3: Corridor: Pedestrian and Flowfield at Times $T = 50, 75, 100$ *sec*

The geometry, pedestrians and flowfield may be inferred from Figures 2.1-2.3. The number of pedestrians at the beginning of the run was set to $N_p = 1660$. The perceived distance to closest exit, which is based on visibility, was recomputed every second. Figures 2.1-2.3 show the distribution of smoke (left), as well as the pedestrians colored according to the amount of smoke inhaled (right).

2. Corridor

This example considers the flow inside a narrow, winding corridor in the presence of pedestrians. The geometry, pedestrians and flowfield may be inferred from Figures 3.1-3.3. The pedestrians are entering from the far side (top of figure) at a rate of $f_p = 2\ p/sec$. The flow is opposite to the movement direction of the pedestrians: it enters from the near side (bottom of figure) with a speed of $v_{in} = 1\ m/sec$. 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). Before starting the coupled run, the CFD code was run without pedestrians until a quasi-steady flowfield was established. 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.

7 CONCLUSIONS AND OUTLOOK

A methodology to couple computational fluid and computational crowd dynamics (CFD, CCD) has been developed. Enabling technologies that made this possible include:

a) Mature CCD and CFD codes/solvers;
b) Development of immersed boundary methods for moving bodies (CFD);
c) Strong scaling to tens of thousands of cores (CFD); and
d) Implementation of fast search techniques for information transfer between codes (CFD, CCD).

We consider that tightly coupled simulations such as the ones presented here will lead to more realistic 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 metro-stations, high-rise buildings and indoor sports arenas, where a crowd can block a considerable portion of the passage area, thereby influencing the flow.

Another interesting application area is the study of airborne transmitted diseases, where the detailed modeling of the flow near pedestrians will lead to a higher accuracy in the predictions.

REFERENCES

[1] H.-J. Bungartz and M. Schäfer (eds.) *Fluid-Structure Interaction*, Springer Lecture Notes in Computational Science and Engineering, Springer (2006).

[2] J.R. Cebral and R. Löhner - On the Loose Coupling of Implicit Time-Marching Codes; *AIAA*-05-1093 (2005).

[3] M. Isenhour and R. Löhner - Verification of a Pedestrian Simulation Tool Using the NIST Recommended Test Cases; *The Conference in Pedestrian and Evacuation Dynamics 2014 (PED2014), Transportation Research Procedia* 2, 237-245 (2014).

[4] M. Isenhour and R. Löhner - Verification of a Pedestrian Simulation Tool Using the NIST Stairwell Evacuation Data; *The Conference in Pedestrian and Evacuation Dynamics 2014 (PED2014), Transportation Research Procedia* 2, 739-744 (2014).

[5] M. Isenhour - Simulating Occupant Response to Emergency Situations; *PhD Thesis*, George Mason University, Fairfax, VA (2016).

[6] R. Löhner - A Fast Finite Element Solver for Incompressible Flows; *AIAA*-90-0398 (1990).

[7] R. Löhner, C. Yang, J. Cebral, J.D. Baum, H. Luo, D. Pelessone and C. Charman - Fluid-Structure Interaction Using a Loose Coupling Algorithm and Adaptive Unstructured Grids; *AIAA*-95-2259 [Invited] (1995). -

[8] R. Löhner, Chi Yang, J.R. Cebral, F. Camelli, O. Soto and J. Waltz - Improving the Speed and Accuracy of Projection-Type Incompressible Flow Solvers; *Comp. Meth. Appl. Mech. Eng.* 195, 23-24, 3087-3109 (2006).

[9] R. Löhner - *Applied CFD Techniques, Second Edition*; J. Wiley & Sons (2008).

[10] R. Löhner - On the Modeling of Pedestrian Motion; *Appl. Math. Modelling* 34, 2, 366-382 (2010).

[11] R. Löhner - Coupling Several CFD and CSD Codes in One Application; pp. 1 - 16 in Special Edition *Int. J. of Multiphysics* (2011).

[12] R. Löhner, M. Baqui, E. Haug, B. Muhamad - Real-Time Micro-Modelling of a Million Pedestrians; *Engineering Computations* 33, 1, 217-237 (2016).

[13] M. Schäfer and S. Turek (eds.) - *Proc. Int. Workshop on Fluid-Structure Interaction: Theory, Numerics and Applications*, Herrsching (Munich), Germany, Sept. 29 - Oct 1 (2008).

[14] R. Ramamurti and R. Löhner - A Parallel Implicit Incompressible Flow Solver Using Unstructured Meshes; *Computers and Fluids* 5, 119-132 (1996).

[15] J. Zhang, D. Britto, M. Chraibi, R. Löhner, E. Haug and B. Gawenat - Qualitative Validation of PEDFLOW for Description of Unidirectional Pedestrian Dynamics; *The Conference in Pedestrian and Evacuation Dynamics 2014 (PED2014), Transportation Research Procedia* 2, 733-738 (2014).