# Case Study # 2: OpenFOAMLid-Driven Cavity Flow

**John Karasinski**
Graduate Student Researcher
Center for Human/Robotics/Vehicle Integration and Performance
Department of Mechanical and Aerospace Engineering
University of California
Davis, California 95616
Email: karasinski@ucdavis.edu

## 1 Problem Description

The flow of an incompressible fluid induced in a square cavity by a moving upper boundary, or "lid-driven" cavity flow, is a classic validation case for Computational Fluid Dynamics (CFD) solvers. This case study involves performing numerical simulations of this flow problem for various flow regimes using an Open Source PDE solver: OpenFOAM.

## 2 Solver Setup

## 3 Numerical Solution

## 4 Discussion and Conclusions

## References

[1] Bruneau, C.-H., and Saad, M., 2006. "The 2d lid-driven cavity problem revisited". *Computers & Fluids,* **35**(3), pp. 326–348.

[2] Ghia, U., Ghia, K. N., and Shin, C., 1982. "High-re solutions for incompressible flow using the navier-stokes equations and a multigrid method". *Journal of computational physics,* **48**(3), pp. 387–411.

[3] Jasak, H., Jemcov, A., and Tukovic, Z., 2007. "Openfoam: A c++ library for complex physics simulations". In International workshop on coupled methods in numerical dynamics, Vol. 1000, pp. 1–20.