A PHASEFIELD APPROACH TO MODELING FLUID-FLUID INTERFACES IN AN EULERIAN FRAMEWORK

J. Hill[a], **N. Walkington**[b], and **O. Ghattas**[c]

[a]Mechanics, Algorithms, and Computing Laboratory
Carnegie Mellon University
Pittsburgh, Pennsylvania 15213
jhill@cs.cmu.edu

[b]Department of Mathematical Sciences
Carnegie Mellon University
Pittsburgh, Pennsylvania 15213
noelw@andrew.cmu.edu

[c]Mechanics, Algorithms, and Computing Laboratory
Carnegie Mellon University
Pittsburgh, Pennsylvania 15213
oghattas@cs.cmu.edu

The modeling of the motion of fluid interfaces with deformable boundaries is a research area of great interest to the computational science and engineering community. Physical examples of such interfaces, such as the spreading of a fluid across a solid surface and the mixing of two immiscible fluids such as oil and water, abound. One challenge for simulations of fluids with deforming boundaries is the accurate and efficient tracking of the interfaces.

This work is motivated by the need for microstructural, or cell-scale, models of blood flow. Existing macroscopic models of blood flow homogenize the microscopic properties in the continuum. Because these macroscopic models parameterize the microscopic details at large length scales, developing accurate microscopic models is essential. Cell-scale models can resolve individual cell deformations and interactions, key components to understanding blood damage.

In this talk, we will present a phasefield model for describing deforming fluid-fluid interfaces. Our approach is twofold:

1. The phasefield, represented by a scalar convection equation, is discretized using a discontinuous Galerkin method.

2. The phase variable, describing the two fluids, is coupled into the Navier-Stokes equations through the fluid properties. The coupled Navier-Stokes equations are solved using a standard Galerkin method.

We will show results from two- and three-dimensional simulations of the flow of immiscible fluids and the scalability of this method on parallel computers.