Simultaneous Coupling of Fluids and Deformable Bodies

Nuttapong Chentanez Tolga G. Goktekin Bryan E. Feldman James F. O'Brien
University of California, Berkeley

Introduction

In this sketch, we present a novel method for simulating the two-way interaction between fluids and deformable bodies. The key to this method involves setting up a system of equations that simultaneously conserves the momentum of the fluid and deformable bodies while also enforcing conservation of mass.

Previous approaches use a time splitting procedure. They alternately fix the fluid pressure while simulating the solid, then fix the solids velocity while simulating the fluid. While this approach can work well for physical systems with non-stiff coupling, it can lead to instability and visual artifacts for more stiff systems. These problems occur because while solid velocities are fixed they will ignore arbitrarily large fluid pressures, and the converse when the fluid velocities are fixed. Time splitting also requires non-physical fixes for systems that include fluid regions completely surrounded by a deformable body. By enforcing simultaneous coupling our method avoids these artifacts and allows for substantially larger time steps.

Overview

The interaction between a fluid and a deformable solid occurs at the interface where the fluid applies pressure forces to the solid's boundary while the solid imposes boundary velocities on the fluid. Therefore, in order to simulate the complex interactions between a solid and a fluid we need to augment both simulations to take the other system into account and create a combined coupled system. The key idea is that when performing pressure correction on the fluid and implicitly solving for the solid's node velocities, we need to simultaneously account for how fluid pressure changes will affect the deformable solid and how the solid's motion will affect the fluid.

Using a finite element or finite difference method and an implicit Newmark time integration scheme the dynamics of an elastic solid body deforming under pressure forces can be fully discretized in the following form:

$$\mathbf{A}u^{n+1} - \mathbf{J}p^{n+1} = b \tag{1}$$

where $\mathbf{A}=\frac{1}{h}\mathbf{M}+\mathbf{C}+\frac{h}{2}\mathbf{K}'$ and $b=\frac{1}{h}\mathbf{M}u^n-\frac{h}{2}\mathbf{K}'u^n-\mathbf{K}(d^n)+f_e$. The solid's node displacement and velocity are denoted by d and u respectively. $\mathbf{M}$ and $\mathbf{C}$ are the mass and damping matrices. $\mathbf{K}$ is the non-linear stiffness matrix function and $\mathbf{K}'$ is its tangent matrix evaluated at d^n. Forces due to the pressure of the fluid are computed and mapped by the $\mathbf{J}$ matrix. The term f_e adds any additional external forces. Finally, h and n denote the timestep and time index respectively.

The fluid, on the other hand, needs to account for the boundary fluxes imposed by the solid while satisfying the incompressibility condition. The resulting Pressure Poisson equation can be formulated as:

$$-\mathbf{D_1}\mathbf{H}u^{n+1} + \frac{h}{\rho}\mathbf{D_2}\mathbf{G}p^{n+1} = \mathbf{D_2}v^* \tag{2}$$

where $\mathbf{D}_1$ and $\mathbf{D}_2$ are matrices for computing the divergence due to the boundary and internal fluid fluxes respectively.

Figure 1: A frame from an animation of a jet interacting with two thin rubber sheets.

The matrix $\mathbf{G}$ is the gradient matrix for internal fluxes and $\mathbf{H}$ is a matrix that converts solid's boundary node velocities into fluxes on the fluids's boundary. Finally, v^* is the intermediate fluid velocity computed from the fluid simulation that needs to be projected onto its divergence free component v^{n+1}.

In order to simulate full coupling between the fluid and the solid we combine Equation (1) and Equation (2) to solve for u^{n+1} and p^{n+1} simultaneously:

$$\begin{bmatrix} \mathbf{A} & -\mathbf{J} \\ -\mathbf{D}_1\mathbf{H} & \frac{h}{\rho}\mathbf{D}_2\mathbf{G} \end{bmatrix} \begin{bmatrix} u^{n+1} \\ p^{n+1} \end{bmatrix} = \begin{bmatrix} b \\ \mathbf{D}_2 v^* \end{bmatrix} \tag{3}$$

Finally, we can compute the divergence free fluid velocities v^{n+1} as follows:

$$v_i^{n+1} = \begin{cases} (v^* - \frac{h}{\rho}(\mathbf{G}p))_i & \text{if } i \text{ is not a boundary face} \\ (\mathbf{H}u^{n+1})_i & \text{if } i \text{ is a boundary face} \end{cases} \tag{4}$$

Results

Figure 1 shows a frame from an animation demonstrating the interaction between a jet of smoke and two rubber sheets. The supplemental video includes several examples demonstrating the effectiveness of the method. Additionally, the stability of the method is demonstrated in an example where with a time step of $\frac{1}{30}$ sec the simultaneous coupling method presented in this paper remains stable while the time-splitting method goes unstable. The time-splitting method is stable only after reducing the time step to $\frac{1}{120}$ sec. Even though the resulting system in Equation (3) is larger than those used in time-splitting, it is very sparse and by solving it with pre-conditioned bi-conjugate gradient stabilized method the advantage of using large timesteps significantly outweighs the overhead of the larger system.