

# Technical Report

Toffalini Leonardo

## FIRST DRAFT

at first i will ramble on about anything, then i will trim the fat

All the code can be found at <https://github.com/leonardo-toffalini/viscous>

## 1 Introduction to the equations of fluids

The advective form of the **Navier-Stokes equations** for an incompressible fluid with uniform viscosity are as follows:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = \nu \Delta \mathbf{u} - \frac{1}{\rho} \nabla p + \frac{1}{\rho} \mathbf{f}. \quad (1)$$

Meaning of the variables:

1.  $\mathbf{u}$  is the velocity vector field.
2.  $\mathbf{f}$  is the external forces.
3.  $\rho$  is the scalar density field.
4.  $p$  is the pressure field.
5.  $\nu$  is the kinematic viscosity.

We will not go into the details of deriving the above equations, instead we will just take it as granted that they truly formulate the evolving velocity of a viscous incompressible fluid.

**TODO** this notation introduction is janky. Refer the reader to Besenyei, Komornik, Simon Parciális differenciál egyenletek 4.5.

For some, that have not yet encountered the differential operators used in the above formulation, we give a short summary:

The  $\nabla$  operator formally represents the  $(\partial_1, \partial_2, \dots, \partial_n)$  vector, thus

$$\nabla u = (\partial_1 u, \partial_2 u, \dots, \partial_n u)$$

$$\mathbf{u} \cdot \nabla = \sum_{i=1}^n u_i \partial_i$$

$$(\mathbf{u} \cdot \nabla) u = \sum_{i=1}^n u_i \partial_i u$$

The  $\Delta$  operator, called the **Laplace operator**, sometimes written as  $\nabla \cdot \nabla = \nabla^2$ , acts as follows:

$$\nabla \cdot \nabla u = \Delta u = \sum_{i=1}^n (\partial_i^2 u)$$

Explanation of the different parts of Equation 1:

$$\frac{\partial \mathbf{u}}{\partial t} + \overbrace{(\mathbf{u} \cdot \nabla) \mathbf{u}}^{\text{Advection}} = \underbrace{\nu \Delta \mathbf{u}}_{\text{Diffusion}} \overbrace{-\frac{1}{\rho} \nabla p}^{\text{Internal source}} + \underbrace{\frac{1}{\rho} \mathbf{f}}_{\text{External source}}.$$

In broad strokes the parts can be described as follows:

1. Advection – How the velocity moves.
2. Diffusion – How the velocity spreads out.
3. Internal source – How the velocity points towards parts of lesser pressure.
4. External source – How the velocity is changed subject to external intervention, like a fan blowing air.

## 2 Equations for fluid simulation

It is best to mention here, in this section, that the described method is not exact. This, however, will not be to our detriment, as our aim here is to show interesting and realistic visuals as opposed to precise measurements.

For our purposes we will rearrange Equation 1 such that we have only the time derivative of the velocity on the left and omit the internal force part, as we will recover this part later. So we arrive at the equation which we will solve:

$$\frac{\partial \mathbf{u}}{\partial t} = -(\mathbf{u} \cdot \nabla) \mathbf{u} + \nu \Delta \mathbf{u} + \frac{1}{\rho} \mathbf{f}.$$

The evolving of the velocity field in and of itself is not that interesting to see, because in real life we do not see the velocity field, we only see its effect. To provide a more stimulating visual experience we must let the velocity field act upon a density field and in turn visualize said density field. We present the equation for the density field

$$\frac{\partial \rho}{\partial t} = -(\mathbf{u} \cdot \nabla) \rho + \kappa \Delta \rho + S.$$

The astute reader might find this equation offily similar to that of the velocity field, which is by no means a coincidence as the same broad strokes apply to the density field too, which are: advection, diffusion, and external sources. The only thing we must be careful of, is that the density is a scalar field, in contrast to the velocity, which is a vector field.

The problem in this state is incomplete, we must also specify a boundary condition. In our endeavour to simulate a fluid, we will work with the Dirichlet-boundary condition, which prescribes that the values on the boundary must vanish. We can formulate the previous statement as follows:

$$\begin{aligned} u|_{\partial\Omega} &= 0 \\ \rho|_{\partial\Omega} &= 0, \end{aligned}$$

where  $\Omega$  represents the domain on which we are searching for the solution.

## 3 Simulating fluids

The equations presented in the previous sections hold on the entire  $n$ -dimensional space in which the fluid resides. However, when numerically solving a partial differential equation one often discretizes the space into small rectangles and only bothers to calculate the solution on these rectangles. For our purposes we will confine ourselves to the 2-dimensional plane, but everything mentioned hereafter can be easily extended to higher dimensions.

We will make another simplification, which is to only consider a rectangular domain. This makes the calculations easier to handle as matrix operations, however, one is free to extend the domain to more complex shapes by exercising proper caution when handling the

boundary. Figure 1 shows a schematic diagram of how one must imagine a grid on a rectangular domain. Notice, how we introduced a 0-th and an  $(N + 1)$ -th row and column to handle the boundary, this way it is clear that we will be simulating the fluid on an  $N \times N$  grid on the inside.

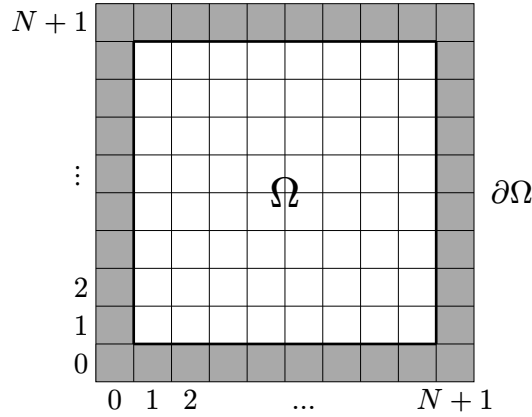


Figure 1: Discretization of the domain.

As mentioned in the previous sections, to simulate the change in time of both the velocity and the density, we must simulate three steps: advection, diffusion, and external sources.

## 4 Moving densities

In our simulation, we will solve the parts of the simplified Navier-Stokes equations for the density field one by one. Starting with the simplest, adding additional density, then diffusion, and lastly advection. The simulation steps can be seen in Figure 2, where one must imagine applying the steps in order from left to right, and repeating the last three steps, while the simulation is running.

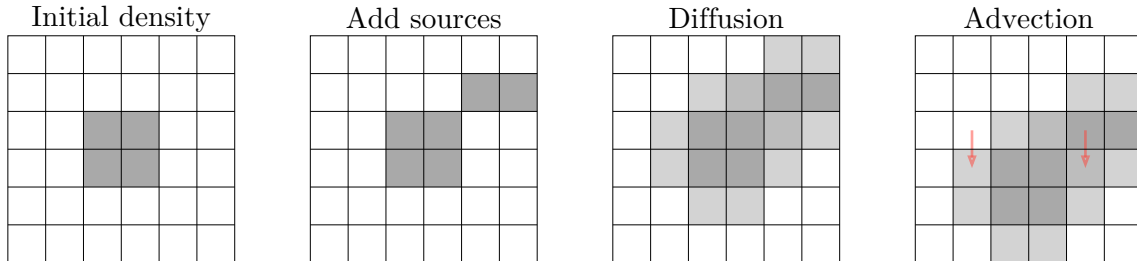


Figure 2: Rough outline of the density solver

### 4.1 Adding sources

Possibly the simplest step of the simulation is to add new sources to the density field, or to the velocity field for that matter. For a rectangular grid this step can be simplified to a simple matrix addition  $\rho_h + S_h$ , where  $\rho_h$  is the density field on the discretized domain, and  $S_h$  is the matrix containing the sources at each grid cell.

## 4.2 Diffusion

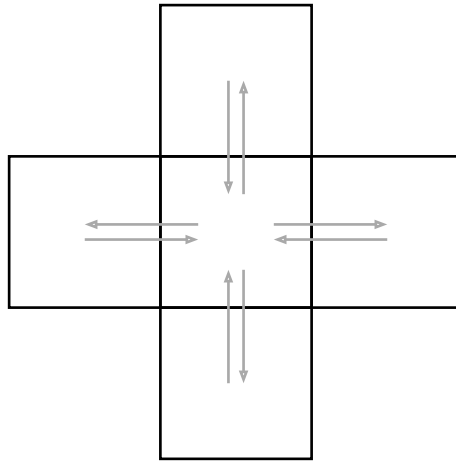


Figure 3: Five point stencil intuition

In essence this is just a finite difference method of a five point stencil.

Here is the discretization matrix  $A_h$  for this five point stencil.

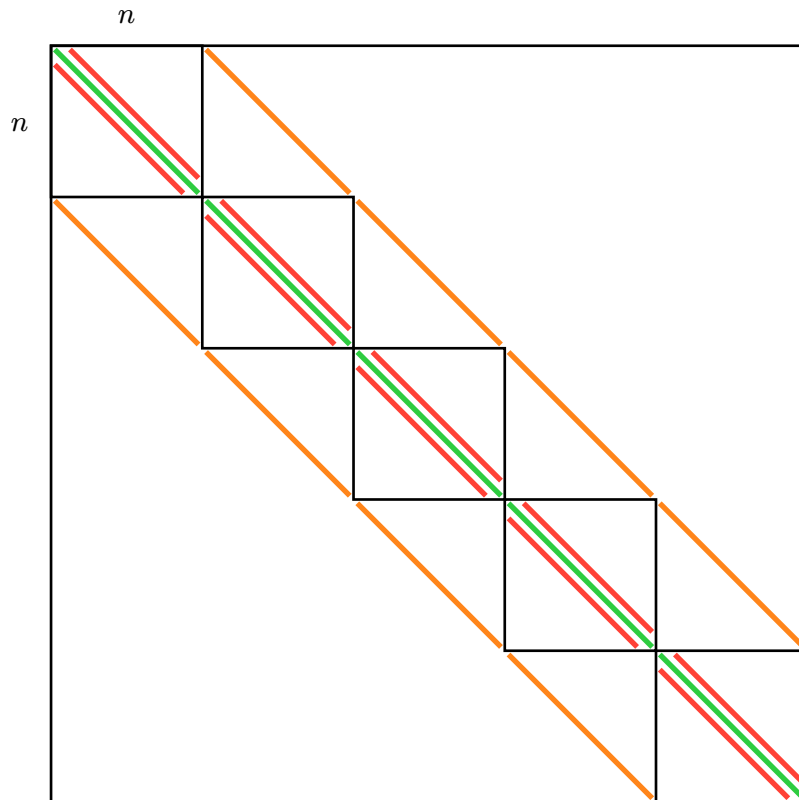


Figure 4: Five point stencil matrix

Let us denote the Kronecker product of two matrices as  $A \otimes B$  and let  $B = \text{tridiag}(-1, 2, -1)$ . Then, the above discretization matrix of the five point stencil can be achieved with the following succinct formula:  $I \otimes B + B \otimes I$ .

However, this matrix is so sparse, that we need not even construct it, as we can just solve the resulting linear system of equations with an iterative method without constructing the full matrix.

### 4.3 Advection

Make it clear that this is the first crucial idea of the method. The idea is that instead of solving the advection equation as a partial differential equation, we trace a path backwards to where a fluid particle could have come from.

The problem with solving the advection equation is that it is dependant on the velocity vector, in contrast to the diffusion step where it was only dependant on the previous state of the density field.

After tracing back the possible locations where fluid particles could have come from we might get a particle that came from not the exact center of a grid. Remember, that we established that we shall think of the fluid as point masses centered at the middle of the grid cells. If a particle came from not the dead center then we must somehow give meaning to it too. In this case we will take the linear interpolation of the four closes neighbours of where the particle came from.

Fluid simulations methods that solve a partial differential equation on a discretized space are called Lagrangian, whereas methods that simulate fluids as a collection of particles are called Eulerian. For this reason this method is sometimes called semi-Lagrangian.

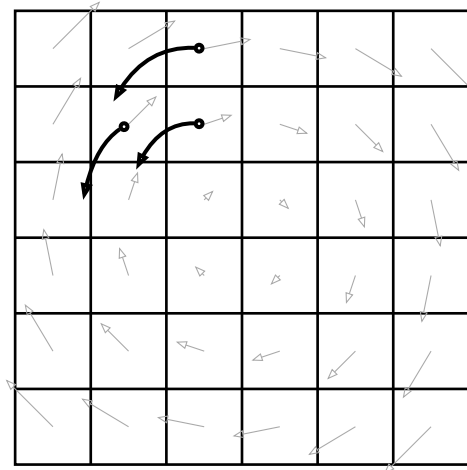


Figure 5: Semi Lagrange

## 5 Evolving velocities

Recall, that the velocity equation is almost the same as the density equation.

This is where the second novel idea comes into play. Remember in the second section we mentioned that we shall omit a part of the equation to make it simpler and handle it later, now is the time to do so. The part we left out made sure that the velocity field was divergence free, meaning that it was mass conserving. This is intuitive about fluids, that a fluid can not just fluid outward from a single point, if some fluid flows out from a point, then an equal amount must flow into said point.

Since we did not take care to hold the divergence free property during the diffusion and advection steps we quite possible end up with a velocity field which has non zero divergence. To combat this we rely on a result from vector calculus which states that a vector field can be decomposed as a sum of a field with no divergence and one which is the gradient of a scalar potential. This result is called the Helmholtz–Hodge decomposition.

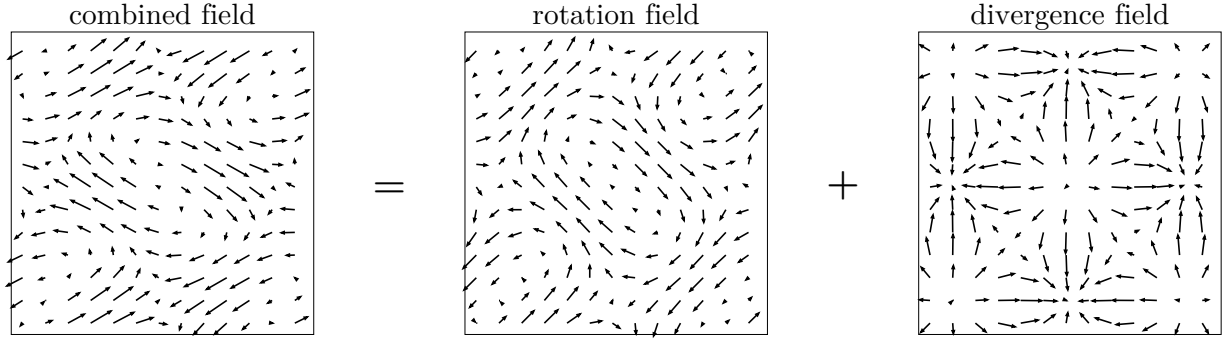


Figure 6: Helmholtz–Hodge decomposition

The Helmholtz–Hodge decomposition states that any vector field  $\mathbf{w}$  can be uniquely decomposed into the sum of a divergence field and a rotation field, more concisely

$$\mathbf{w} = \mathbf{u} + \nabla q,$$

where  $\nabla \cdot (\mathbf{u}) = 0$ , and  $q$  is a scalar field. Figure 6 illustrates the idea of the decomposition.

Formally taking the dot product with the  $\nabla$  operator of both sides, we get

$$\begin{aligned}\nabla \cdot \mathbf{w} &= \nabla \cdot \mathbf{u} + \nabla \cdot \nabla q \\ \nabla \cdot \mathbf{w} &= 0 + \nabla \cdot \nabla q \\ \nabla \cdot \mathbf{w} &= \Delta q\end{aligned}$$

The relation between  $\mathbf{w}$  and  $q$  we just derived is a simple Poisson equation for  $q$ , which can be solved with the finite difference method we outlined in the previous section. After solving for  $q$ , we can extract  $\mathbf{u}$  as

$$\mathbf{u} = \mathbf{w} - \nabla q.$$

With this result in our hands we can finally resolve the mass conserving property of the velocity of the simulated fluid by decomposing the resulting field after the last step into a divergence free field.

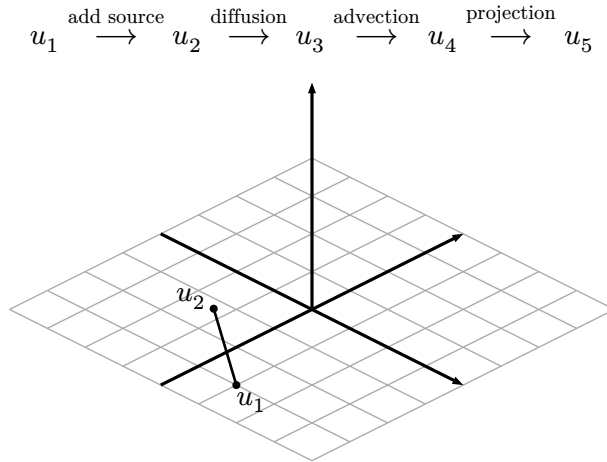


Figure 7: Projection