Simulation of Smoke in Two Dimensions using Finite Difference Method in MATLAB

Thanakorn Chanthanasaro and Chaiwoot Boonyasiriwat

ABSTRACT

In this work, smoke from a candle is modeled as an incompressible, viscous fluid. The governing equations including the continuity equation, the Navier-Stokes equations, the advection-diffusion equations for heat and smoke concentration were numerically solved using the finite difference method on a staggered grid and the pressure-correction method. A simulation program was developed in MATLAB based on the code of Seibold (2008). Our program was successfully used to simulate smoke in a 2D rectangular domain. Numerical results show that symmetric flows were obtained when kinematic viscosity was higher than 10^{-4} while asymmetric flows occurred when kinematic viscosity was lower than the critical value. Flow patterns also depend on the time lag between the releases of heat and smoke sources as expected but the simulated patterns are not consistent with real observations. Further work must be performed to improve the simulation results.

INTRODUCTION

Simulating realistic fluids is a challenging problem both in computer graphic and fluid dynamics studying Fedkiw et al. (2001), Stam (1999). Though there are many software that can simulate things easily nowadays, but understanding its physics behind is important. All we have to do when simulate water is solving Navier-Stokes equations. In case of smoke, advection-diffusion equation of heat and smoke are needed to be solved together. In this work, we study the effect of viscosity and the existence of fluid flow to the flow pattern. Since these equations are hard to be solved analytically, numerical method is used to numerically solve these equations. My MATLAB code is modified from code of Seibold (2008). In the theory part, I follow the book Fluid Simulation for Computer Graphics Bridson (2008).

THEORY

When the flow's velocity is well below the speed of sound. The compressible effect of the flow can be neglected. Flow behavior is described by incompressible Naiver-Stokes equations given by

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f} \tag{2}$$

where $\mathbf{u} = (u, v, w)$ or $(u, v, 0)$ in two-dimensional is the fluid's velocity, p is its pressure, ρ is its density, ν is its kinematic viscosity, and $\mathbf{f}$ is an external force. Navier-Stokes equations state that the velocity should conserve both mass (Equation (1)) and momentum (Equation (2)). Momentum equation is solved from Newton's equation. LHS of momentum equation is rate of change of velocity. Its second term is the rate due to moving of the fluid. Where RHS of momentum equation is the force accounted. Its first term is from the gradient of the pressure and its second term is from the viscosity effect.

In case of smoke simulation, there are two more equation needed for the evolution of two scalar variables temperature T and concentration of the smoke s given by

$$\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla)T = k_T \nabla^2 T + r_T(T_{source} - T) \tag{3}$$

$$\frac{\partial s}{\partial t} + (\mathbf{u} \cdot \nabla)s = k_s \nabla^2 s + r_s \tag{4}$$

where k_T and k_s are diffusion coefficient of temperature and concentration of smoke respectively, r_T is the rate that the heat is released, r_s is the rate that smoke is released, and T_{source} is the temperature of the source point. Equation (3) and (4) are called advection-diffusion equation of heat and concentration of smoke. They account rate of change of each variable due to moving of the fluid (the second term of LHS) and the diffusion effect (the first term of RHS). The last terms of equations (3) and (4) are source terms.

Next, the fluid density is linearly approximated to be function of temperature T and concentration of smoke s as

$$\rho = \rho_0[1 + \alpha s - \beta(T - T_{amb})] \quad (5)$$

where ρ_0 is the smoke-free air density at ambient temperature, α and β are two positive constants, and T_{amb} is ambient temperature. Plugging this into momentum equation. The static pressure $\rho_0\mathbf{g}$ is added to the variation pressure to new pressure. Using the *Boussinesq* approximation, assume that $|\alpha s - \beta\Delta T| \ll 1$, leading to

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u} = -\frac{1}{\rho_0}\nabla p + \nu\nabla^2\mathbf{u} + (\alpha s - \beta\Delta T)\mathbf{g} \quad (6)$$

where $\Delta T = T - T_{amb}$. The last term of equation (6) is the external force. Smoke falls down due to gravity force and rises up due to hot air. Note that when $s = 0$ and $T = T_{amb}$, this force is zero.

METHODS

2D smoke is simulated on a rectangular domain using finite difference method, discretized on staggered grid shown in figure 1, and projection method for the pressure. The solution of next time step ($u^{n+1}, v^{n+1}, p^{n+1}, T^{n+1}$, and s^{n+1}) is solved by splitting the whole process by compute intermediate velocities (u^*, u^{**}, v^*, v^{**}, and v^{***}), temperature (T^*, and T^{**}), and smoke concentration (s^*, and s^{**}). The velocity is solved first follow by the temperature and smoke concentration.

Advection terms of momentum equation

$$\frac{u^* - u^n}{\Delta t} = -\frac{\partial (u^n)^2}{\partial x} - \frac{\partial (u^n v^n)}{\partial y}$$
$$\frac{v^* - v^n}{\Delta t} = -\frac{\partial (u^n v^n)}{\partial y} - \frac{\partial (v^n)^2}{\partial y}$$

Intermediate velocity $u^*$and v^* are solved explicitly be rearranging the advection term of momentum equation as above. This introduces a CFL condition which limits the time step by a constant times the spacial resolution.

Viscosity terms

$$\frac{u^{**} - u^*}{\Delta t} = \nu(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2})$$
$$\frac{v^{**} - v^*}{\Delta t} = \nu(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2})$$

Intermediate velocities (u^{**} and v^{**}) are solved implicitly. Thus, we have to solve two linear systems in each time step.

Force term

There is a force only in y-direction. It is solve explicitly.

$$\frac{v^{***} - v^{**}}{\Delta t} = -(\alpha s - \beta\Delta T)g$$

Pressure term

The fluid is made incompressible by compute the pressure needed to make the divergence of the velocity at next time step to be zero. In the vector notation

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^{***}}{\Delta t} = -\frac{1}{\rho_0}\nabla p$$

Applying the divergence to both sides yields the linear system

$$\nabla^2 p = \frac{\rho_0}{\Delta t}\nabla\cdot\mathbf{u}^{***}$$

we solve this Poisson's equation to find the pressure and use it to calculate the velocity at next time step $\mathbf{u}^{n+1}$. The temperature and smoke concentration are then solved after the velocity is solved.

Advection terms of advection-diffusion equations

Advection terms are solved explicitly to obtain intermediate temperature T^* and smoke concentration s^*.

$$\frac{T^* - T^n}{\Delta t} = -(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y})$$
$$\frac{s^* - s^n}{\Delta t} = -(u\frac{\partial s}{\partial x} + v\frac{\partial s}{\partial y})$$

Source terms

Intermediate temperature T^{**} and smoke concentration s^{**} are solved explicitly.

$$\frac{T^{**} - T^*}{\Delta t} = r_T(T_{source} - T)$$
$$\frac{s^{**} - s^*}{\Delta t} = r_s$$

Diffusion terms

$$\frac{T^{n+1} - T^{**}}{\Delta t} = k_T\nabla^2 T$$
$$\frac{s^{n+1} - s^{**}}{\Delta t} = k_s\nabla^2 s$$

Diffusion terms are solved explicitly to obtain temperature and smoke concentration at next time step T^{n+1} and s^{n+1}

FINITE DIFFERENCE METHOD

The first derivative of spatial domain is approximated using central finite difference.

$$\frac{\partial u}{\partial x} \approx \frac{u(x+\Delta x) - u(x-\Delta x)}{2\Delta x}$$

The approximation of the second derivative is given by

$$\frac{\partial^2 u}{\partial x^2} \approx \frac{u(x+\Delta x) - 2u(x) + u(x-\Delta x)}{\Delta x}$$

In time domain, forward finite difference is used

$$\frac{\partial u}{\partial t} \approx \frac{u(t+\Delta t) - u(t)}{\Delta t}$$

STAGGERED GRID

Staggered grid is the grid where different variables are stored at different location. The pressure, temperature, and smoke concentration are sampled at the middle of the cells. The fluid velocity in x-direction u is sampled at the middle of the vertical cell faces. The fluid velocity in y-direction v is sampled at the middle of the horizontal cell faces. It is shown in Figure 2.1. If the value required is not at the location, it is interpolated.

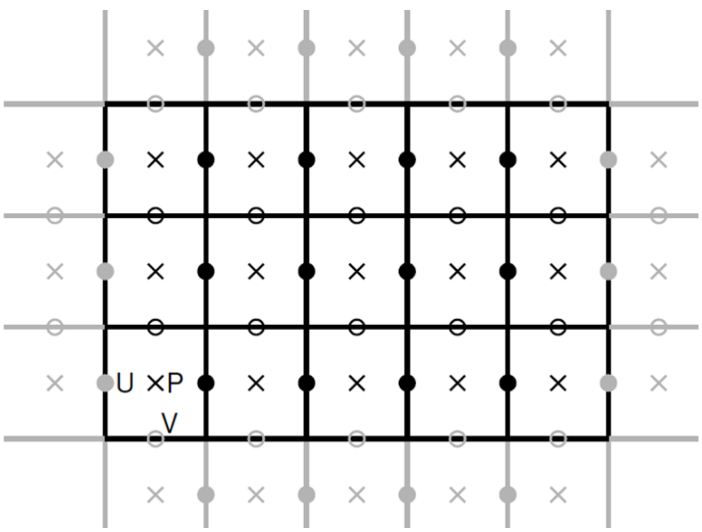

Figure 1: Staggered grid used. x at the middle of the cell is the location of pressure, temperature, and smoke concentration. Black circle the fluid velocity in x-direction u. White circle is the location of fluid velocity in y-direction v.

DISCRETIZATION AND INDEX

The rectangular domain is discretized into many square cells. L_x , the horizontal length of domain, is discretized into $n_x \Delta x$. Where n_x is integer and Δx is grid spacing. Discretization is the same for $L_y = n_y \Delta y$. At location $(i\Delta x, j\Delta y)$, the variable is indexed by subscript (i, j), for example, temperature is indexed $T_{i,j}$. The velocity in x-direction u is indexed by half integer, e.g., $u_{i+1/2,j}$. In time domain, the time is discretized into the time step times integer $t_n = n\Delta t$. The variable is indexed by superscript, for example, smoke concentration at time t_n is s^n.

BOUNDARY CONDITIONS

Boundary conditions are needed in solving differential equation. In this work, smoke is simulated in a 2D box. The boundary condition for the velocities both in x and y-direction is Dirichlet boundary condition, i.e.,the velocity is zero at the boundary. The boundary condition for temperature is at the boundary, temperature is equal to ambient temperature. And the boundary condition for smoke is at the boundary, smoke concentration is zero. The boundary condition for pressure is Neumann boundary condition, i.e., $\frac{\partial p}{\partial \mathbf{n}} = 0$. Where $\mathbf{n}$ is the normal direction to the boundary.

RESULTS

First, we study the effect of viscosity to the flow pattern at $T_{amb} = 300$ K, $T_{source} = 400$ K, $\alpha = 0.001$, $\beta = 0.0033$, $\rho_0 = 1.289$ km/m^3, $k_T = k_s = 0.0001$, $r_T = 0.01$, $r_s = 1$, $\Delta t = \Delta x = \Delta y = 0.01$. The result is shown in Figure 2 and 3 At high kinematic viscosity ($\nu > 10^{-4}$), the flow is symmetric. The big vortex appears at the top of the flow. When the kinematic viscosity is lower than the critical value 10^{-4}. The symmetry is broken. The small vortex at the top of the flow is observed. Since the system is asymmetric (the source is not placed rightly at the middle of domain but slightly left) , this may cause the flow asymmetric. The other reasons are the algorithm of computing and numerical error.

Second, we study the effect of time lag between the releasing of heat and smoke concentration to the flow. The result is shown in Figure 4 and 5

The case of there is no time lag of the releasing of heat and smoke concentration, the big vortex appear at the top of the flow. For the case there is time lag between the releasing of heat and smoke concentration, the vortex is not quite big. The smoke of the latter rises up faster at early time and look sharper than the former. Because the heat released first cause the fluid to flow before the smoke is released.

CONCLUSION

1. Smoke can be simulated using Navier-Stokes equations and advection-diffusion equations of heat and smoke concentration

2. The flow symmetry is broken when the kinematic viscosity is less than 10^{-4}.

3. Higher kinematic viscosity cause the vortex at the top of the flow bigger. The existence of fluid flow also affect the vortex appeared.

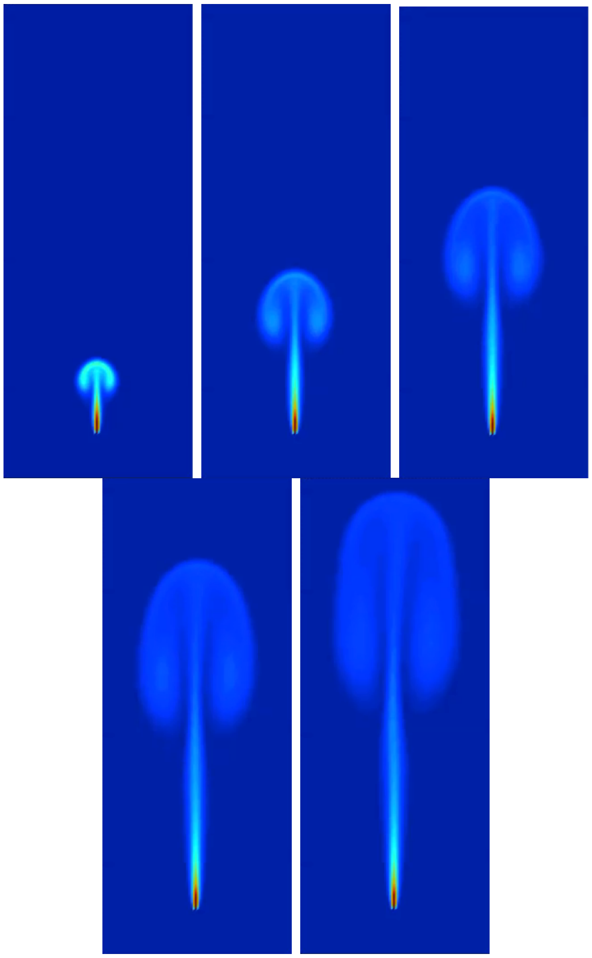

Figure 2: Plot of smoke concentration at different time: 8, 16, 24, 32, and 40 s respectively of fluid with kinematic viscosity 10^{-3}

.

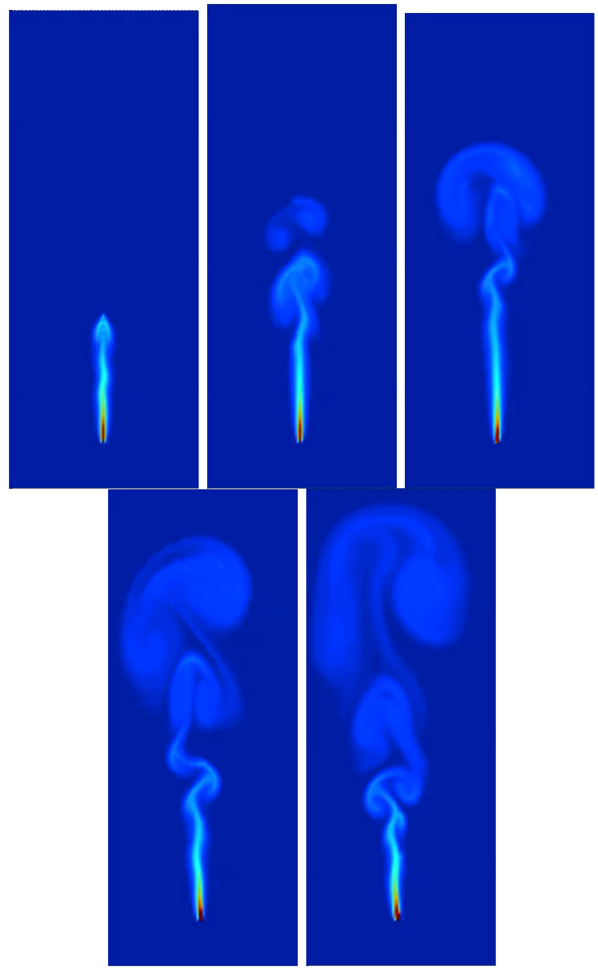

Figure 3: Plot of smoke concentration at different time: 8, 16, 24, 32, and 40 s respectively of fluid with kinematic viscosity 10^{-5}

.

ACKNOWLEDGMENTS

First of all, I would like to thank my advisor, Dr.Chaiwoot Boonyasiriwat, for all advises and attention. And thank you Development and Promotion of Science and Technology Talents Project (DPST) scholarship for receiving me and all of financial support since the first year. And thank you my family for letting me study science. Thank you Benjamin Seibold for his code used in my project. Finally, thank you my friends for what we have been through.

REFERENCES

Bridson, R., 2008, Fluid simulation for computer graphics: A K Peters.

Fedkiw, R., J. Stam, and H. W. Jensen, 2001, Visual simulation of smoke: In proceedings of the 28th annual conference on Computer graphics and interactive techniques ACM, 15–22.

Seibold, B., 2008, A compact and fast matlab code solving the incompressible navier-stokes equations on rectangular domains: Massachusetts Institute of Technology.

Stam, J., 1999, Stable fluids: In Proceedings of SIGGRAPH 99, 121–128.

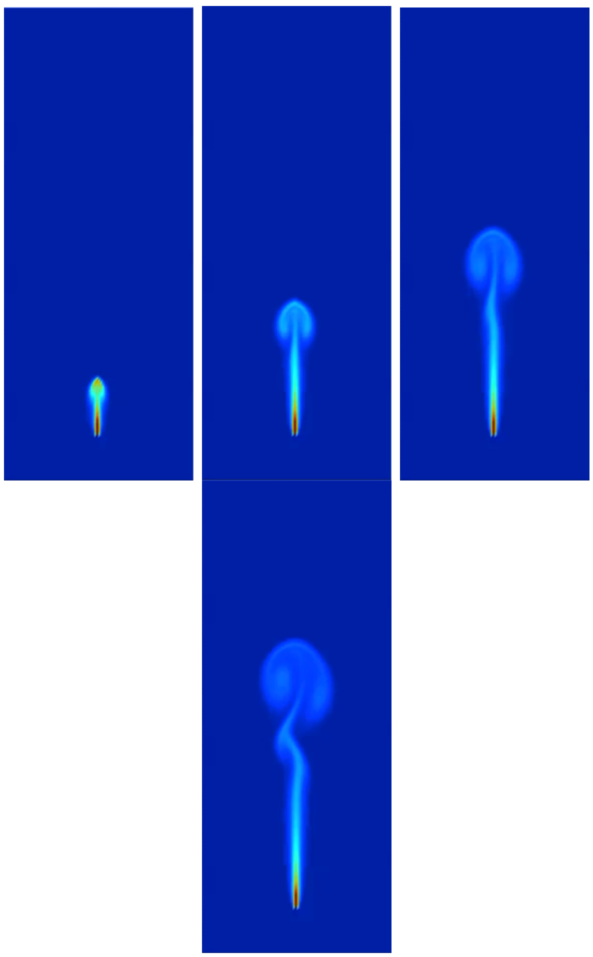

Figure 4: Plot of smoke concentration at different time: 5, 10, 15, and 20 s respectively. Time lag of releasing heat and smoke concentration is 0 s.

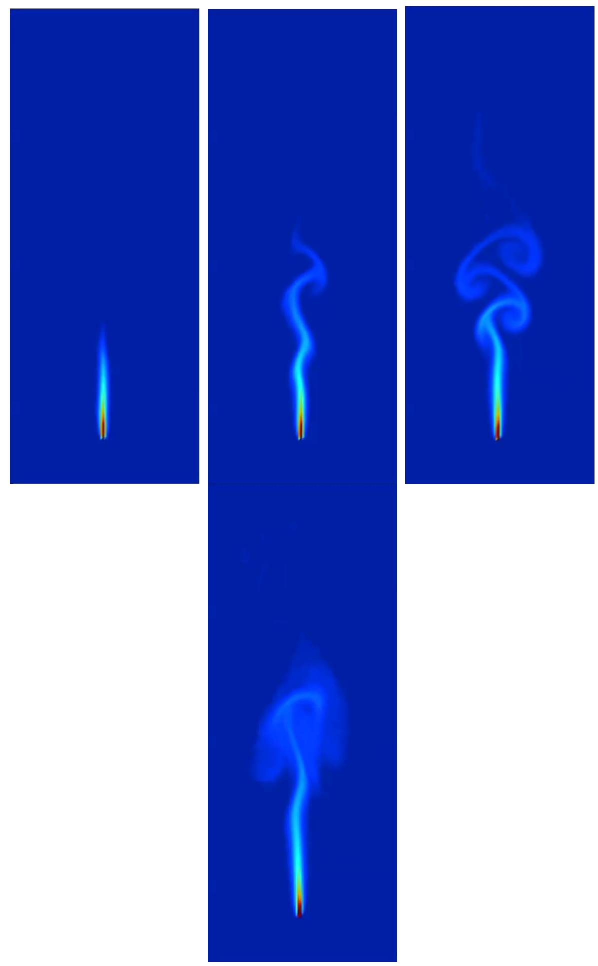

Figure 5: Plot of smoke concentration at different time: 5, 10, 15, and 20 s respectively. Time lag of releasing heat and smoke concentration is 20 s. The heat is released before the smoke is released.