Smoothed Particle Hydrodynamics in Engineering and Astrophysics, A Two-Dimensional Study of Cloud-Cloud Collisions Using SPH

Submitted by-

Samanpreet Singh Lang

Department of Physics, Panjab University, Chandigarh, India

Under the supervision of-

Dr Sandeep Sahijpal

Department of Physics, Panjab University, Chandigarh, India

Abstract

This project investigates the use of Smoothed Particle Hydrodynamics (SPH), a mesh-free, Lagrangian computational method, in both engineering and astrophysical fluid dynamics. By representing fluids as discrete particles and using kernel-based interpolation, SPH provides a flexible framework for simulating complex flow phenomena without the constraints of traditional grid-based methods.

In the engineering context, the project models the behaviour of a fluid inside a rigid container. The simulation accounts for key physical forces such as pressure, viscosity, and boundary interactions, demonstrating SPH's ability to capture realistic fluid motion and confinement effects within a closed environment.

In the astrophysical context, the SPH method is applied to simulate cloud-cloud collisions—interactions between large, self-gravitating gas clouds in space. These simulations focus on the role of pressure, gravity, and hydrodynamic shocks in the evolution of colliding clouds, capturing key features such as compression, fragmentation, and potential star-forming structures.

The project showcases the versatility and power of SPH across vastly different physical regimes, highlighting its relevance for problems in both terrestrial engineering and astrophysical modelling. The implementation is carried out in C++, with detailed attention to numerical methods, force calculations, and particle dynamics in diverse simulation settings.

Table of Contents

1. INTRODUCTION

1.1 Historical Background

The modelling and simulation of fluid flows have long been central topics in both classical and modern physics. Historically, such simulations relied heavily on grid-based methods like Finite Difference (FD), Finite Element (FE), and Finite Volume (FV) techniques. These methods involve discretizing the fluid domain into a fixed mesh or grid and solving the governing equations at these discrete points.

However, grid-based methods come with notable limitations—especially when dealing with problems involving large deformations, free surfaces, or moving boundaries. In astrophysical contexts, such as star formation or galaxy collisions, the physical systems often exhibit complex, highly dynamic behaviours with irregular geometries. Grid-based approaches struggle to resolve these dynamics efficiently and accurately.

In this context, Smoothed Particle Hydrodynamics (SPH) emerged as a revolutionary technique. Introduced independently by Lucy (1977) **[1]** and Gingold & Monaghan (1977) **[2]**, SPH was initially developed to simulate astrophysical phenomena. Its key innovation lies in representing fluids not with a fixed grid but with particles that carry properties such as mass, velocity, and density. These particles move with the fluid flow, providing a Lagrangian perspective that naturally handles large displacements and deformable bodies.

Since its inception, SPH has evolved significantly and found applications far beyond its astrophysical roots. Researchers have adapted it for engineering problems, including hydrodynamics, solid mechanics, impact simulations, and more. Its meshless nature and adaptability make it a powerful tool in any scenario where traditional methods may falter.

1.2 Why Smoothed Particle Hydrodynamics (SPH)?

Smoothed Particle Hydrodynamics stands out for several reasons. First and foremost, its meshless structure **[3]** allows it to tackle problems involving free surfaces, interfaces, or fragmentation—situations where mesh-based methods may suffer from distortion or require complex re-meshing algorithms.

In SPH, each particle serves as a moving interpolation point, and physical quantities are estimated using a smoothing kernel. This approach allows SPH to simulate fluids in a more physically intuitive manner, avoiding artificial numerical diffusion and better conserving physical properties like mass and momentum **[4]**.

From a computational standpoint, SPH simplifies the implementation of complex geometries. The particles naturally follow the motion of the fluid, so boundary interactions, deformation, and large strain regions can be resolved more effectively. This feature is especially beneficial in astrophysical simulations, where systems evolve dynamically over large spatial and temporal scales.

In astrophysics, SPH is widely used for modelling interstellar cloud dynamics, accretion disks, galaxy formation, and stellar collisions. In engineering **[5]**, it is applied to simulate splashing, sloshing, dam breaks, and other complex fluid behaviors. Its robustness in multi-phase flows and adaptability to parallel computing frameworks further contribute to its popularity.

Moreover, SPH integrates well with modern numerical frameworks and can be coupled with gravitational solvers, radiative transfer models, or thermodynamic modules. These features make it a versatile choice for multi-physics simulations.

In this project, we apply SPH to two distinct problems:

1. **Fluid in a Fixed Container** – A test case to understand the fluid dynamics and verify the stability and performance of our SPH implementation.

2. **Cloud-Cloud Collision** – An astrophysical application that models the interaction between two gas clouds, providing insights into shock formation, compression, and potential star-forming regions.

This report aims to present not only the theoretical foundations of SPH but also its practical realization. By exploring these two cases, we will highlight both the strengths and limitations of the SPH method and assess its potential for future work in computational astrophysics and fluid dynamics.

---

2. Formulation

2.1 Smoothing Function

At the heart of Smoothed Particle Hydrodynamics lies the concept of **kernel interpolation**. Unlike traditional grid-based methods, SPH approximates field variables using a **weighted sum over neighbouring particles**, where the weights are given by a smoothing kernel. This interpolation technique transforms the continuous equations of fluid dynamics into a particle-based discrete form.

2.1.1 Kernel Approximation

Consider a scalar function **A(r)**. Its value at a point **r** can be approximated using:

$$A(\mathbf{r}) \approx \int A(\mathbf{r}')W(\mathbf{r}-\mathbf{r}',h)\,d\mathbf{r}'$$

Here, **W (r − r′, h)** is the smoothing kernel function, and **h** is the **smoothing length**, which determines the region of influence around **r**. The kernel must satisfy the following conditions:

Normalization:

$$\int W(\mathbf{r}-\mathbf{r}',h)\,d\mathbf{r}' = 1$$

Delta-function behaviour as **h → 0**:

$$\lim_{h\to 0} W(\mathbf{r}-\mathbf{r}',h) = \delta(\mathbf{r}-\mathbf{r}')$$

Compact support: **W** is non-zero only within a finite radius, ensuring computational efficiency.

2.1.2 Discrete Approximation

Replacing the integral with a sum over particles:

$$A(\mathbf{r}) \approx \sum_j \frac{m_j}{\rho_j} A_j W(\mathbf{r} - \mathbf{r}_j, h)$$

where:

- $\boldsymbol{m_j}$ is the mass of particle j
- $\boldsymbol{\rho_j}$ is its density
- $\boldsymbol{A_j}$ is the value of the field at $\boldsymbol{r_j}$
- W is the kernel

This forms the basis for approximating gradients, divergences, and other derivatives needed to model fluid dynamics.

2.2 Lagrangian Formulation of SPH

SPH operates in a **Lagrangian framework**, meaning the computational nodes (particles) move with the fluid flow. This makes it well-suited for problems with evolving boundaries and interfaces.

2.2.1 Governing Equations

The standard fluid equations in Lagrangian form include:

Continuity Equation:

$$\frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{v}$$

Momentum Equation:

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho} \nabla P + \mathbf{f}$$

Energy Equation (optional, depending on the simulation):

$$\frac{du}{dt} = \frac{P}{\rho^2} \frac{d\rho}{dt}$$

where:

- **ρ** is the density,
- **v** is velocity,
- **P** is pressure,
- **u** is internal energy,
- **f** includes body forces (e.g., gravity).

2.2.2 SPH Discretization

Using SPH interpolation, the **density** of a particle is estimated as:

$$\rho_i = \sum_j m_j W(\mathbf{r}_i - \mathbf{r}_j, h)$$

The **momentum equation** becomes:

$$\frac{d\mathbf{v}_i}{dt} = -\sum_j m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} + \Pi_{ij} \right) \nabla W_{ij}$$

Here, **Πij** is an **artificial viscosity** term introduced to handle shocks and prevent particle interpenetration. A common form is Monaghan's viscosity:

$$\Pi_{ij} = \begin{cases} \frac{-\alpha c_{ij}\mu_{ij} + \beta\mu_{ij}^2}{\bar{\rho}_{ij}}, & \text{if } \mathbf{v}_{ij} \cdot \mathbf{r}_{ij} < 0 \\ 0, & \text{otherwise} \end{cases}$$

where **μij** is a function of the velocity and distance between particles, **α**, **β** are tunable parameters, and **c_ij** is the average sound speed.

2.2.3 Pressure and Equation of State

In SPH, pressure is usually derived from an **equation of state** (EOS). A common choice is the **ideal gas law** or a **Tait equation**:

$$P = k(\rho^\gamma - \rho_0^\gamma)$$

where:

- **k** is a stiffness constant,
- **γ** is the polytropic index (typically around 7 for water-like fluids),
- **ρ_0** is reference density.

The EOS introduces compressibility, which is useful even in weakly compressible SPH formulations (WCSPH) to stabilize the system numerically.

This formulation enables SPH to evolve a system in time by updating particle positions and velocities at each time step using numerical integration (typically leapfrog or Runge-Kutta). Boundary conditions, damping terms, and neighbour search algorithms further refine this basic structure.

---

3. Applications

Smoothed Particle Hydrodynamics (SPH) is a highly versatile method that has found widespread use across many scientific disciplines. Its particle-based, Lagrangian nature makes it particularly effective for simulations involving large deformations, free surfaces, and complex geometries. While it originated in astrophysics, its flexibility has led to widespread adoption in engineering, geophysics, medicine, and other fields. This section highlights SPH applications in two major domains: **astronomy and astrophysics**, and **engineering**.

3.1 Astronomy and Astrophysics

3.1.1 Star Formation and Interstellar Cloud Collapse

One of the earliest and most impactful uses of SPH was modelling **gravitational collapse** of **interstellar molecular clouds**, which are the birthplaces of stars. These clouds, composed primarily of gas and dust, evolve under self-gravity, hydrodynamic forces, turbulence, and magnetic fields. SPH allows researchers to model the collapse, fragmentation, and core formation processes with adaptive resolution, automatically following regions of high density.

SPH can handle the **multi-scale nature** of star formation — from parsec-scale clouds to astronomical unit (AU)-scale protostellar disks. By coupling SPH with radiative transfer or magnetohydrodynamic (MHD) models, researchers have simulated:

- Protostellar accretion disks

- Binary star formation
- Effects of radiative feedback on star-forming cores

Such simulations are crucial for understanding the **initial mass function (IMF)**, stellar multiplicity, and the conditions leading to planet formation.

3.1.2 Galaxy Collisions and Mergers

Galaxy interactions and mergers play a major role in cosmic structure formation. SPH is well-suited for simulating these events due to its ability to model both **collisionless (dark matter, stars)** and **collisional (gas)** components. When two galaxies collide, tidal forces redistribute stars, compress gas, and often trigger intense starburst activity.

SPH simulations of galaxy mergers can reproduce observable features such as:

- Tidal tails and bridges
- Gas inflows toward galactic centres
- Formation of elliptical galaxies from spiral mergers

Coupling SPH with **N-body dynamics**, feedback from supernovae, and chemical evolution enables realistic modelling of galactic evolution over cosmic time.

3.1.3 Cloud-Cloud Collisions and Shock Fronts

In the interstellar medium, **supersonic turbulence** and **cloud-cloud collisions** can compress gas to the point of gravitational instability. SPH enables high-resolution tracking of shock fronts, density evolution, and post-shock cooling.

Simulating cloud collisions with SPH helps understand:

- Conditions for triggered star formation
- Evolution of magnetic fields during compression
- Clump fragmentation and filamentary structure formation

These applications highlight the importance of SPH in modelling dynamic, compressible gas flows.

3.1.4 Accretion Disks and Planet Formation

SPH has been used to simulate **accretion disks** around young stars, black holes, and compact objects. In protoplanetary disks, SPH helps track dust aggregation, gap opening by forming planets, and migration. For high-energy astrophysics, SPH models accretion around neutron stars and black holes, often involving relativistic corrections and intense feedback processes.

3.2 Engineering

Although SPH originated in astrophysics, its application to engineering problems has grown rapidly due to its ability to handle **complex boundaries, multi-phase flows, and large deformations**.

3.2.1 Free Surface and Fluid Flow Simulations

SPH is especially powerful in simulating **free-surface flows [6]**, which are difficult to handle with grid-based methods due to interface tracking challenges. Common examples include:

- **Dam break problems**
  SPH accurately models the collapse of a fluid column and the resulting wave propagation, including splashing and impact forces on boundaries.

- **Wave–structure interaction**
  Used in coastal engineering to study how tsunamis and storm surges interact with barriers, sea walls, or structures like oil platforms.

- **Sloshing in tanks**
  Important for aerospace and automotive fuel tank design, SPH handles the motion of fluid within containers subject to vibration or acceleration.

3.2.2 Solid Mechanics and Deformable Bodies

With extensions like **Elastic SPH** or **Total Lagrangian SPH**, the method can model **deformable solids** and **impact dynamics**. Applications include:

- **Metal forging and cutting**
  SPH can capture plastic deformation, crack propagation, and tool–material interaction in manufacturing simulations.

- **Ballistics and impact modelling**
  In defence and materials science, SPH simulates the penetration of projectiles into armour, accounting for high-strain-rate deformations and fragmentation.

- **Soft tissue modelling**
  In biomechanics and medical physics, SPH is used to simulate soft tissue behaviour under compression or surgical manipulation.

3.2.3 Multi-Phase and Thermal Flows

SPH naturally models **multi-phase systems**, such as:

- Air–water interaction in bubble dynamics
- Sediment transport in rivers and coastal environments
- Melting, freezing, and phase change phenomena

Thermal SPH variants incorporate **heat conduction and energy transport**, allowing simulations of thermal stress, laser heating, or cooling in electronic devices.

3.2.4 Additive Manufacturing and Granular Flows

In modern manufacturing techniques like **3D printing**, SPH models the behavior of molten droplets, powder sintering, and layer-by-layer deposition. It is also used in simulating **granular media** like sand, powders, and grains — especially under vibration or flow conditions.

Together, these applications demonstrate the **broad relevance of SPH** in both theoretical and practical domains. Whether modelling cosmic gas clouds or industrial metal forming, SPH's strength lies in its adaptability to diverse, complex, and evolving systems.

---

4. Implementation

4.1 Liquid Inside a Fixed Container

In this implementation, Smoothed Particle Hydrodynamics (SPH) is applied to model the motion of an idealized liquid confined within a rectangular, closed container. This setup allows us to verify the physical correctness and numerical stability of our SPH model before proceeding to more complex systems like cloud dynamics. The simulation consists of 600 particles initialized randomly in the lower half of a 10×10 cm^2 domain. Each particle represents a discrete fluid parcel and follows equations of motion governed by SPH-based pressure forces and gravity.

4.1.1 Numerical Discretization and Particle Representation

The fluid is modelled as a Lagrangian ensemble of particles. Each particle carries with it all the required physical quantities — position, velocity, and density. The positions (xi, yi) are updated explicitly using velocity and time step values. Unlike Eulerian grid-based solvers, SPH avoids any mesh and instead computes local field values through a kernel-weighted sum over neighbours.

The governing idea is that the value of a field (such as density or pressure) at any particle location is estimated by integrating contributions from neighbouring particles within a radius h, the smoothing radius. Due to the particle-based nature of SPH, it is naturally adaptive in regions of higher density and can simulate free surfaces without additional tracking.

The smoothing kernel function W(r, h) chosen here is a simplified quadratic form with compact support, vanishing outside r = h. This ensures computational efficiency by ignoring particles beyond the influence zone.

4.1.2 Density Computation

The SPH density approximation is a cornerstone of the formulation:

$$\rho_i = \sum_j m_j W(|\vec{r}_i - \vec{r}_j|, h)$$

This summation over neighbouring particles gives an instantaneous measure of how compressed or rarefied the local fluid element is. In this simulation, each particle is assigned a unit mass **m=1**, simplifying the summation to a pure kernel-weighted count.

The kernel used in this simulation is a 2D variant:

$$W(r, h) = \frac{(h - r)^2}{V}$$

This form ensures that **W (r ,h) → 0** smoothly as **r → h**, preserving continuity and differentiability, which are essential for stability in SPH simulations.

4.1.3 Pressure Calculation and Forces

Once density $\boldsymbol{\rho_i}$ is evaluated, pressure is calculated using a linear equation of state:

$$P_i = k(\rho_i - \rho_0)$$

where **k=2.5** is a tunable pressure multiplier, and $\boldsymbol{\rho_0}$ **= 1.0** is the reference rest density. This linear relation is computationally inexpensive and sufficient for simple compressible fluid behaviour.

The inter-particle pressure force is derived from the momentum conservation equation:

$$\vec{f}_{ij}^{\text{pressure}} = -\frac{P_i + P_j}{2} \nabla W(|\vec{r}_i - \vec{r}_j|, h)$$

To compute this, the gradient of the kernel function is evaluated as:

$$\nabla W(r, h) = -\frac{2(h-r)}{V} \cdot \hat{r}$$

This force is repulsive when densities are higher than the rest value, naturally producing pressure-driven expansion and stabilization. This is a core advantage of SPH — the fluid self-regulates its local density and avoids artificial compression.

4.1.4 Gravity and Damping

An external body force, gravity, is applied to each particle along the negative y-direction:

$$\vec{a}_i^{\text{gravity}} = (0, -g)$$

This value is scaled to accelerate simulation behaviour and ensure that particles settle quickly into the bottom of the container. The net acceleration combines pressure and gravitational terms:

$$\vec{a}_i = \vec{a}_i^{\text{pressure}} + \vec{a}_i^{\text{gravity}}$$

To prevent perpetual oscillation and maintain numerical stability, a damping factor γ=0.9 is applied:

$$\vec{v}_i \leftarrow \gamma \cdot (\vec{v}_i + \vec{a}_i \cdot \Delta t)$$

This mimics internal viscous dissipation without explicitly modelling viscosity, which is adequate for a containerized liquid simulation.

4.1.5 Integration Scheme

The simulation uses an explicit time-stepping scheme based on the forward Euler method. Although simple, this method is stable enough for small Δt and well-tuned parameters:

Position update:

$$\vec{r}_i(t + \Delta t) = \vec{r}_i(t) + \vec{v}_i(t) \cdot \Delta t$$

Velocity update (after force application):

$$\vec{v}_i(t + \Delta t) = \gamma \cdot [\vec{v}_i(t) + \vec{a}_i \cdot \Delta t]$$

Here, **Δ t =0.01 s**, which was chosen after trial and error to ensure stability and smooth motion. This time step is small enough to resolve the force fluctuations and to keep particles within the container bounds throughout the simulation.

4.1.6 Boundary Conditions

Reflective boundaries are enforced at all four walls of the container. After each position update, particle positions are clamped to remain inside the box of size 10×10 cm and the velocity component normal to the wall is inverted:

- If a particle hits the left or right wall:

$$\mathbf{Vx} \rightarrow -\mathbf{Vx}$$

- If it hits the top or bottom:

$$\mathbf{Vy} \rightarrow -\mathbf{Vy}$$

These boundary conditions prevent energy loss through the wall while enforcing spatial confinement.

Collisions with the wall effectively act like perfectly elastic collisions, while damping ensures that repeated bouncing doesn't lead to chaotic particle behaviour.

4.1.7 Initialization and Output

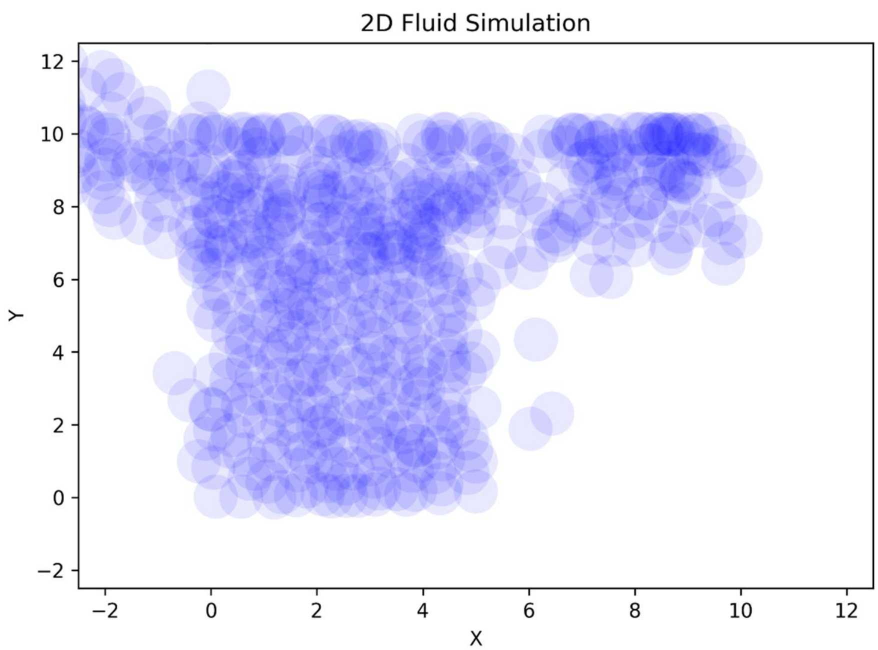
Particles are randomly initialized within a smaller subregion (half the box) to allow for observable spreading under gravity and pressure. The output consists of periodic dumps of the particle positions into a file simulation_output.txt. This data can be later visualized using Python, Gnuplot, or other tools to inspect fluid evolution and validate simulation accuracy.

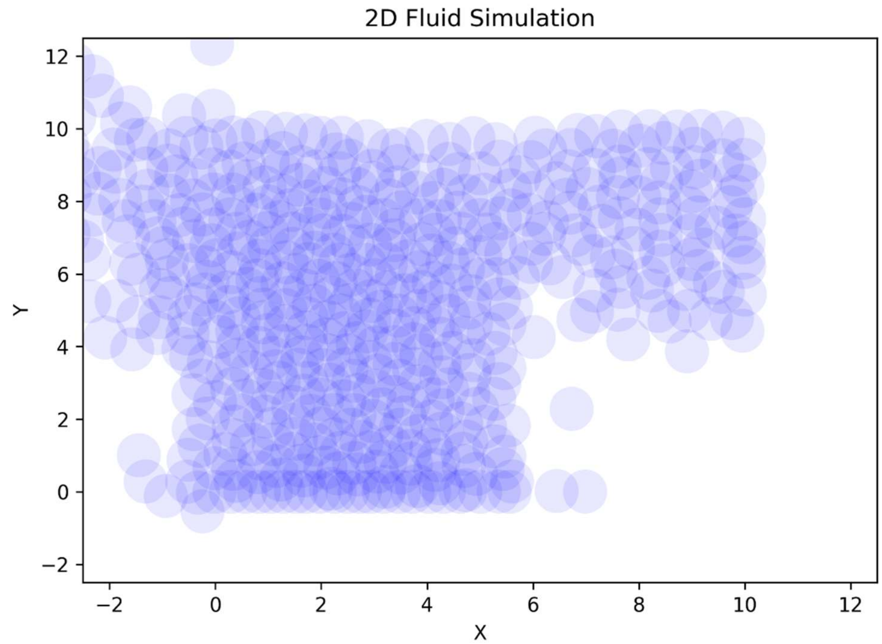
Each simulation runs for 3000 steps, and every 10th step is recorded, generating a smooth animation when rendered. Particles tend to settle into a compressed configuration at the bottom due to gravity, and small-scale ripples emerge from pressure corrections, giving a realistic fluid-like appearance.

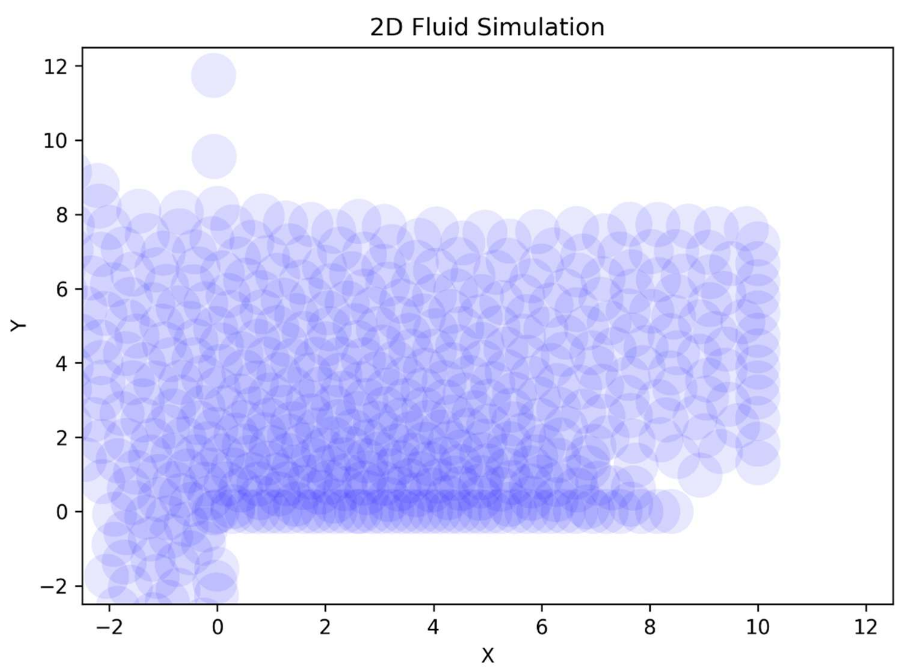
4.1.8 Physical and Numerical Considerations

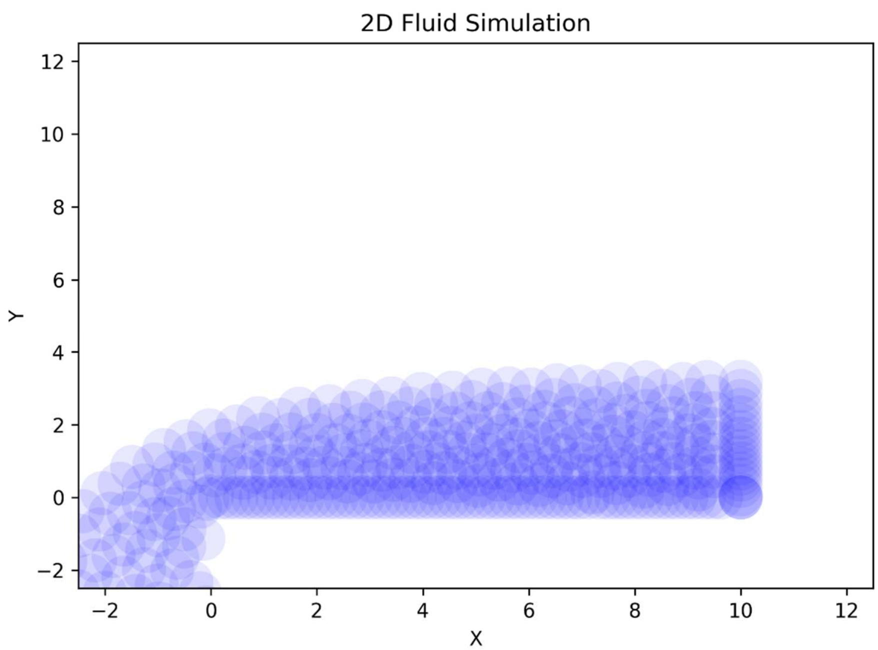
- The code does not implement viscosity or surface tension explicitly but still exhibits fluid-like behaviour due to damping and pressure feedback.
- The kernel function is non-negative and compact, preventing numerical instability from long-range interactions.
- There is no neighbour search optimization (e.g., spatial hashing), so the current $\mathbf{O}(\boldsymbol{N}^2)$ complexity is manageable only because of the small particle count.

- No adaptive time-stepping is employed, so stability is manually ensured by choosing a small **Δt**.
- This implementation is ideal for testing fundamental SPH mechanics in a controlled setup.




2D Fluid Simulation
Y
X
12
10
8
6
4
2
0
−2
−2
0
2
4
6
8
10
12


2D Fluid Simulation
Y
X
12
10
8
6
4
2
0
−2
−2
0
2
4
6
8
10
12


2D Fluid Simulation
Y
X
12
10
8
6
4
2
0
−2
−2
0
2
4
6
8
10
12

4.2 Cloud-Cloud Collision

In this section, we describe the implementation of a Smoothed Particle Hydrodynamics (SPH) simulation designed to model the dynamics of colliding interstellar gas clouds. These collisions are astrophysically significant, as they often lead to the formation of dense regions conducive to star formation **[7] [8]**. The approach taken here uses a particle-based method to capture the fluid-like behaviour of the clouds and their interaction under pressure and gravitational forces.

Particle Representation and Initialization

Each cloud is modelled as a collection of discrete particles, where each particle carries position, velocity, density, and pressure information. Two distinct clouds are initialized:

- **A V-shaped cloud**, consisting of two arms symmetrically inclined, mimicking shock-prone structures observed in cloud interactions.
- **A rod-shaped cloud**, with particles arranged in a linear bar structure, positioned to move toward the V-shaped cloud.

Particle placement is executed with slight random perturbations to reflect natural irregularities in gas density and to reduce artificial symmetry. The rod is rotated to align with the intended axis of approach using a standard 2D rotation transformation:

$$
\begin{aligned}
x' &= x\cos\theta - y\sin\theta,\\
y' &= x\sin\theta + y\cos\theta.
\end{aligned}
$$

The particles are assigned initial velocities such that the two clouds move toward each other.

4.2.1 Compute Density and Pressure

Each particle's density is computed using the SPH summation formula:

$$\rho_i = \sum_j m_j W(r_{ij}, h),$$

where:

- $\boldsymbol{\rho_j}$ the density of particle **i**,
- $\boldsymbol{m_j}$ is the mass of neighbouring particle **j** (assumed constant),
- **W** is a smoothing kernel,
- $\boldsymbol{r_j}$ is the distance between particles **i** and **j**,
- **h** is the smoothing radius.

The smoothing kernel used here is a compact polynomial kernel with support within the radius h, and its volume normalization ensures mass conservation. It takes the form:

$$W(r,h) = \begin{cases} \frac{(h-r)^2}{\frac{\pi h^4}{6}}, & r < h, \\ 0, & r \geq h. \end{cases}$$

Once the density is known, pressure is computed from a linear equation of state:

$$P_i = K(\rho_i - \rho_0),$$

4.2.2 Compute Forces

For each particle **[9]**, the following forces are calculated:

a) Gravitational Force

An attractive gravitational interaction is introduced between all particles, softened to prevent singularities at very small separations:

$$\mathbf{F}_{\text{grav}} = -G \frac{\hat{\mathbf{r}}_{ij}}{r_{ij}^2 + \epsilon^2},$$

where:

- **G** is the gravitational constant (scaled),
- $\hat{\boldsymbol{r}}_{\boldsymbol{ij}}$ is the unit vector from particle **i** to **j**,
- ϵ is a softening length to avoid infinite forces.

b) Pressure Force

SPH computes pressure gradients symmetrically using both particles' pressures and densities:

$$\mathbf{F}_{\text{pressure}} = -\sum_j m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} \right) \nabla W(r_{ij}, h),$$

where the gradient of the smoothing kernel is:

$$\nabla W(r, h) = \begin{cases} -\frac{2(h-r)}{\frac{\pi h^4}{6}} \hat{\mathbf{r}}, & r < h, \\ 0, & r \geq h. \end{cases}$$

This ensures momentum conservation and a symmetric response between interacting particles.

c) Artificial Damping

To mimic energy dissipation and avoid numerical instabilities due to high-velocity collisions, a velocity-based damping force is applied when particles approach each other within the smoothing radius:

$$
\mathbf{F}_{\text{damping}} = \begin{cases} \lambda(\mathbf{v}_i - \mathbf{v}_j) \cdot \hat{\mathbf{r}}_{ij}, & \text{if } r_{ij} < h \text{ and } (\mathbf{v}_i - \mathbf{v}_j) \cdot \hat{\mathbf{r}}_{ij} < 0, \\ 0, & \text{otherwise,} \end{cases}
$$

where $\boldsymbol{\lambda}$ is a damping coefficient.

To avoid unphysically large accelerations, a capping mechanism is introduced to limit the magnitude of the net acceleration on any particle.

4.2.3 Update Particle Velocities and Positions

Using the net acceleration $\boldsymbol{a_i}$ obtained from all the above forces, the velocities and positions are updated with explicit time integration:

$$
\mathbf{v}_i(t + \Delta t) = \mathbf{v}_i(t) + \mathbf{a}_i \Delta t,
$$
$$
\mathbf{x}_i(t + \Delta t) = \mathbf{x}_i(t) + \mathbf{v}_i(t + \Delta t) \Delta t.
$$

This completes one full timestep of the simulation.

Cloud-Cloud Collision Dynamics

As the simulation progresses, the rod cloud moves into the V-shaped cloud, creating zones of enhanced pressure and density at the interface. These interactions reproduce key physical phenomena such as:

- **Compression fronts** at the collision interface.
- **Shock formation**, where pressure and density rise rapidly.
- **Shear flows** between differently moving regions.
- **Gravitational contraction**, leading to clump formation in high-density zones.

The system evolves dynamically, with particles interacting both hydrodynamically (via SPH forces) and gravitationally. This framework captures a simplified but effective model for studying gravitational fragmentation and turbulence resulting from cloud-cloud interactions.

This implementation effectively models the interaction of two gas clouds with different initial geometries and velocity distributions using SPH. It handles complex dynamics including fluid compression, gravitational interactions, and energy dissipation. Although simplified, it serves as a foundation for more advanced astrophysical simulations involving magnetic fields, radiative feedback, or multi-phase media.

---

5. Inference

The Smoothed Particle Hydrodynamics (SPH) simulation of the cloud-cloud collision yields several important insights into the dynamics of interstellar gas interactions. By modelling the physical processes of pressure, gravity, and momentum exchange between particles, the simulation reveals how such collisions can lead to complex fluid behaviour, density enhancement, and potentially star-forming conditions. This section discusses the key observations and physical interpretations drawn from the results.

5.1 Formation of Dense Interaction Zones

One of the most prominent outcomes of the simulation is the formation of high-density regions at the interface between the two colliding clouds. As the rod-shaped cloud advances into the stationary V-shaped cloud, particles from both clouds converge, leading to a compression of material. The pressure forces calculated through the SPH algorithm respond dynamically to the local increase in density, resisting further compression and helping to define a compact interaction front.

This compression effect mimics real astrophysical processes where shock fronts form during cloud collisions. Such regions, characterized by high densities and pressures, are often precursors to gravitational collapse and subsequent star formation in molecular clouds.

5.2 Gravitational Response and Clumping Behaviour

The inclusion of gravitational interaction between particles significantly influences the evolution of the system. Although gravitational forces are softened to avoid numerical instability, their cumulative effect becomes noticeable as matter begins to accumulate in dense regions. Over time, these gravitational effects encourage local clumping, especially near the centre of the collision zone

This result is consistent with observational studies and theoretical models suggesting that cloud-cloud collisions can trigger localized gravitational instabilities. In the simulation, such behaviour is evident in the aggregation of particles into compact formations, indicating the initial stages of fragmentation within the shocked gas.

5.3 Pressure Equilibration and Shock Dissipation

The simulation shows that regions undergoing rapid compression are balanced by a proportional increase in pressure, as dictated by the SPH formulation. This pressure buildup resists further inward motion, effectively preventing numerical blow-up and physically modelling the behaviour of real gases under stress. Over time, as the velocity gradients smooth out and particles spread through the pressure gradient, the system tends toward a more stable configuration.

The artificial damping mechanism also plays a role here, dissipating excess kinetic energy generated during high-speed collisions. This simulates the effect of internal friction or turbulence that would naturally occur in a real astrophysical setting, further aiding in stabilizing the post-collision flow.

5.4 Influence of Geometry and Initial Velocities

The geometric arrangement of the clouds significantly affects the collision dynamics. The V-shaped cloud, with its two arms inclined toward the point of impact, helps guide and funnel the incoming material, focusing the collision energy at a central location. This geometrical focusing amplifies the density enhancement in the central region, a behaviour frequently suggested in observational studies of filamentary and hub-like gas structures in star-forming regions.

The rod cloud's trajectory and velocity also determine how the interaction unfolds. Faster approach speeds lead to more violent compression and sharper pressure gradients, while slower velocities allow for more gradual mixing and smoother transitions. Even without specifying precise values, the simulation clearly shows how variations in velocity and orientation lead to distinctly different outcomes.

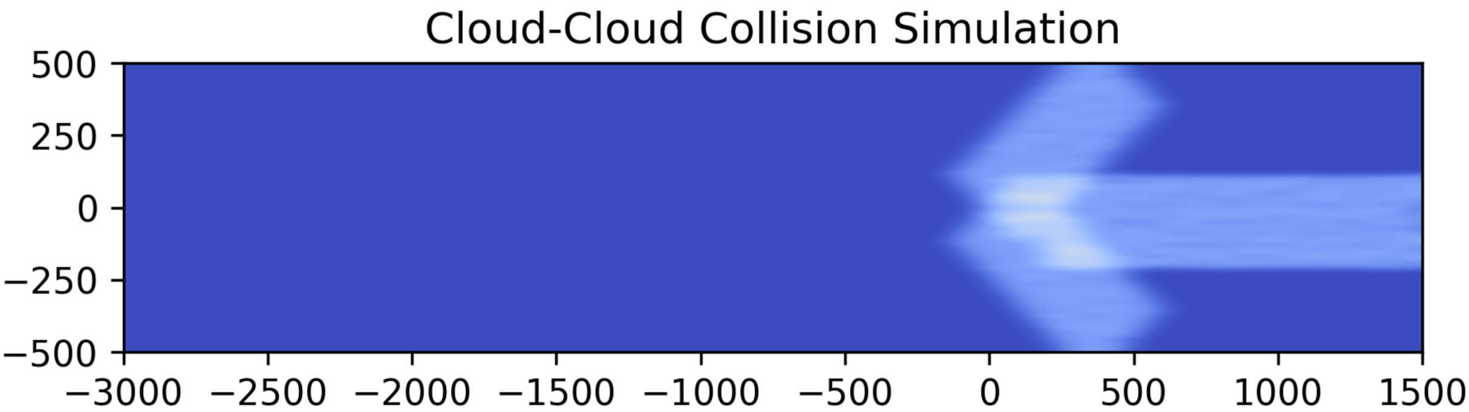
5.5 Long-Term Evolution and System Behaviour

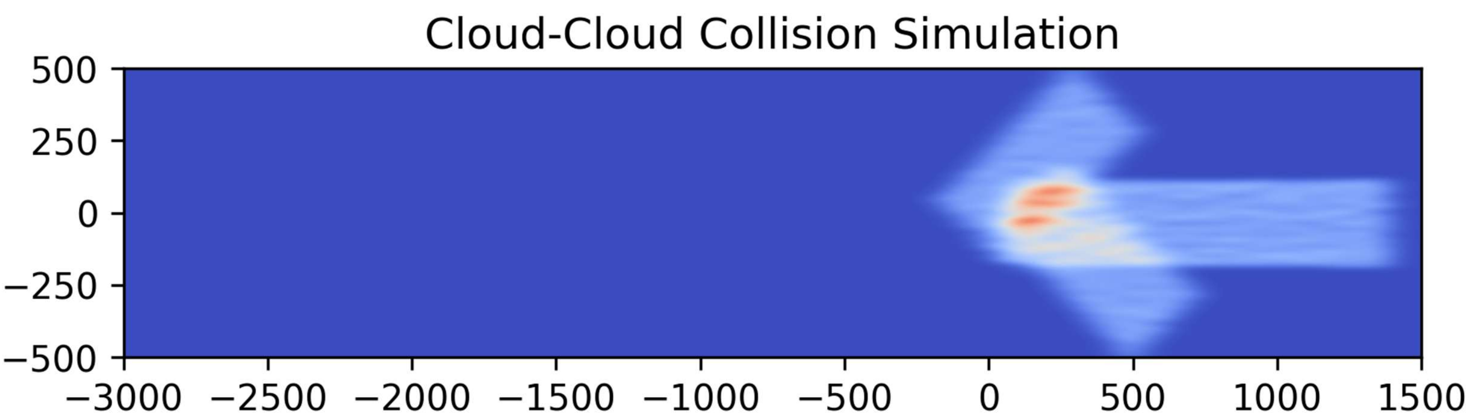
Although the simulation spans a limited number of timesteps, it demonstrates the long-term tendencies of a colliding cloud system. The early phase is dominated by kinetic motion and pressure response. As time progresses, the dominant forces shift toward gravitational aggregation and pressure equilibration. In the absence of additional feedback mechanisms—such as stellar winds, radiation pressure, or magnetic fields—the system continues to evolve toward a state where gravitational and pressure forces reach a quasi-equilibrium.

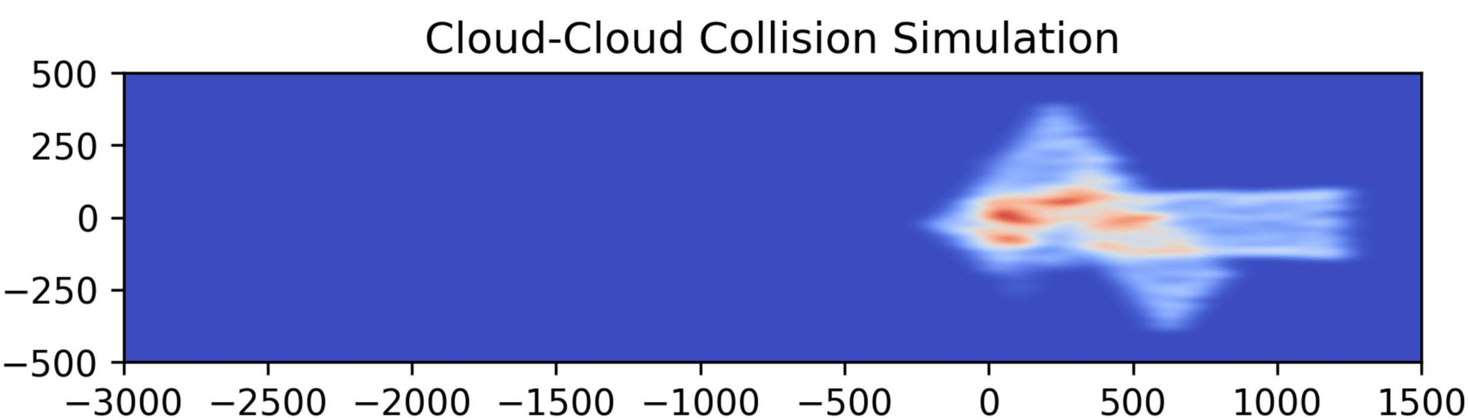
In a real astrophysical context, such systems would eventually experience further fragmentation, cooling, or collapse, possibly leading to the formation of star clusters or filamentary structures. The simulation, therefore, provides a simplified but insightful window into the early stages of such processes.

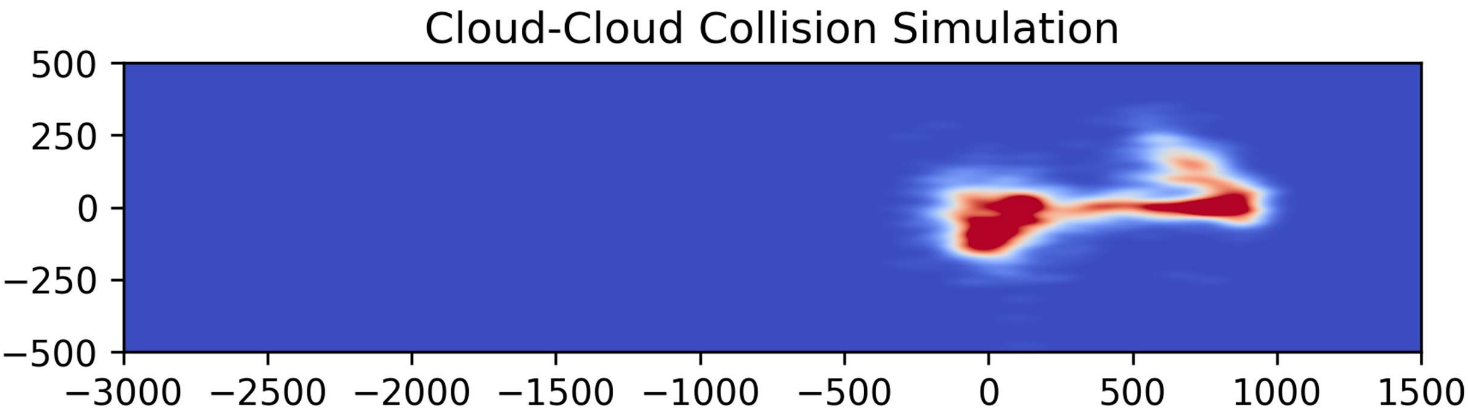
Summary of Inferences

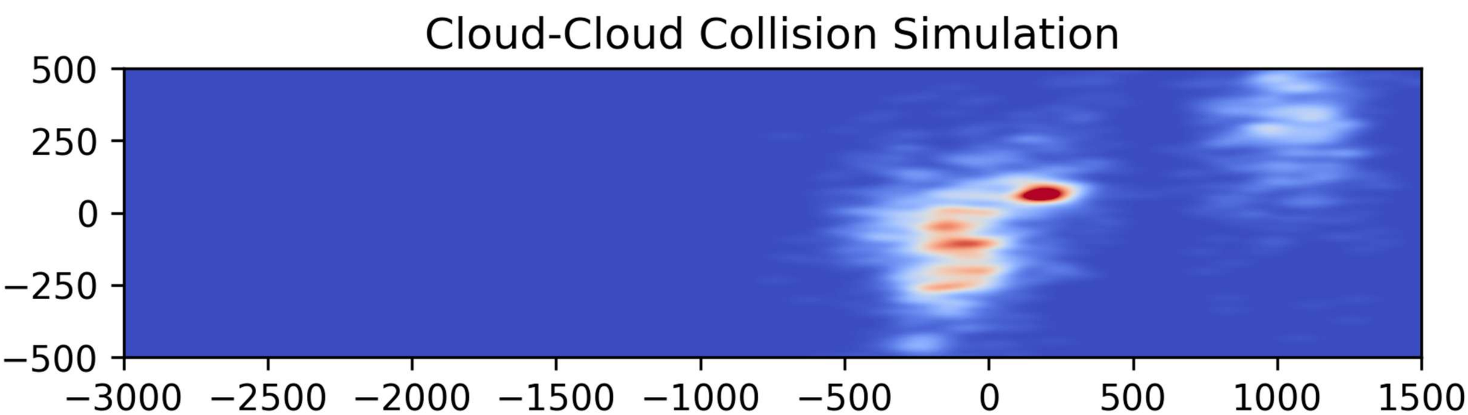
- **Compression fronts** form naturally where the clouds meet, leading to enhanced pressure and density.
- **Gravitational interactions** promote clumping, especially in post-collision high-density zones.
- **Pressure forces** resist unphysical overlap and maintain structural integrity of the flow.
- **Damping effects** help mimic dissipation of turbulent motion during the collision.
- **Initial geometry** significantly affects where and how dense structures emerge.

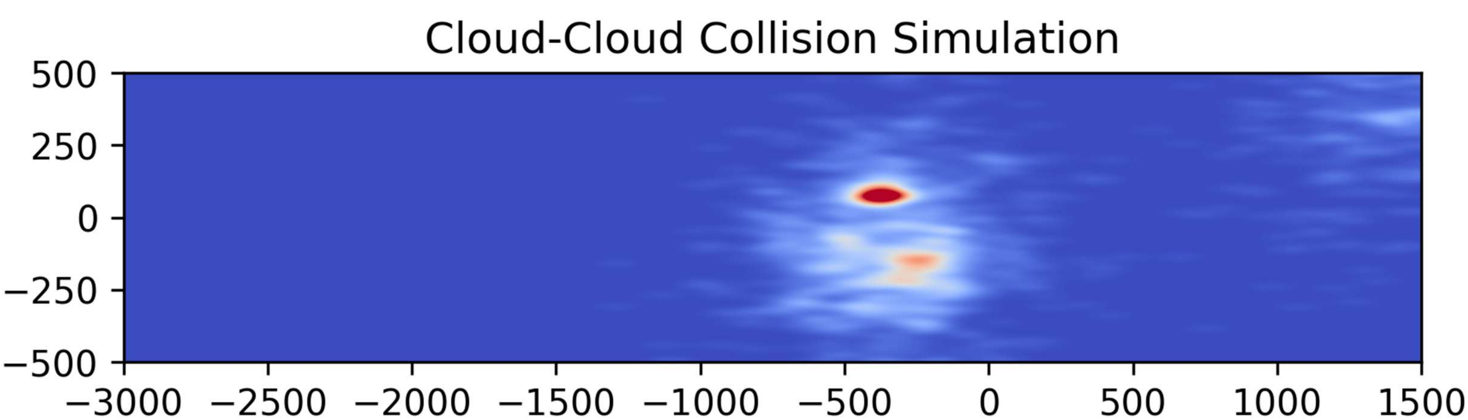

Cloud-Cloud Collision Simulation
500
250
0
−250
−500
−3000 −2500 −2000 −1500 −1000 −500 0 500 1000 1500

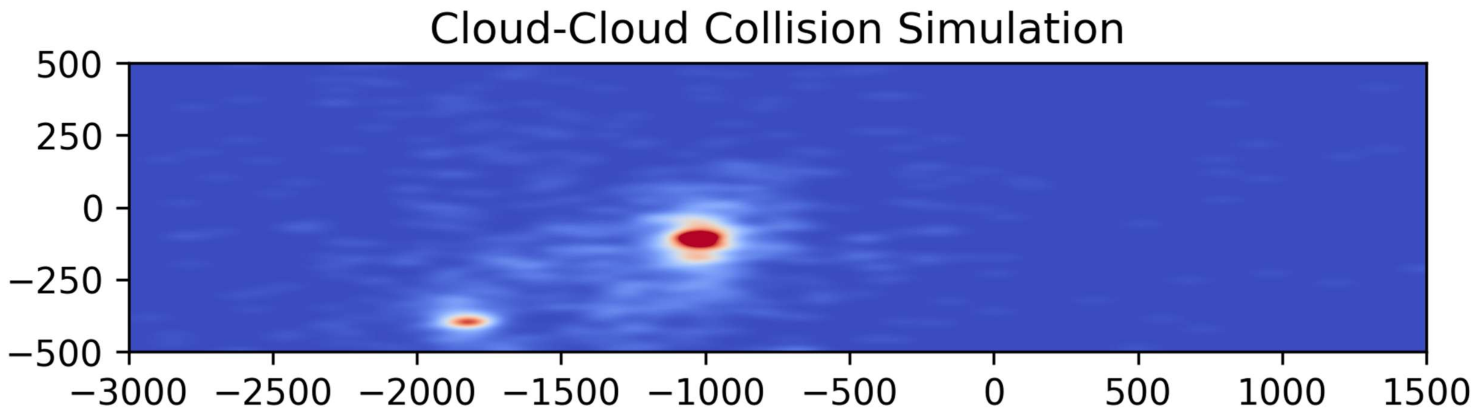

Cloud-Cloud Collision Simulation
500
250
0
−250
−500
−3000 −2500 −2000 −1500 −1000 −500 0 500 1000 1500


Cloud-Cloud Collision Simulation
500
250
0
−250
−500
−3000 −2500 −2000 −1500 −1000 −500 0 500 1000 1500


Cloud-Cloud Collision Simulation
500
250
0
−250
−500
−3000
−2500
−2000
−1500
−1000
−500
0
500
1000
1500


Cloud-Cloud Collision Simulation
500
250
0
−250
−500
−3000
−2500
−2000
−1500
−1000
−500
0
500
1000
1500


Cloud-Cloud Collision Simulation
500
250
0
−250
−500
−3000
−2500
−2000
−1500
−1000
−500
0
500
1000
1500



In conclusion, this SPH simulation provides a physically grounded and computationally tractable method to study the collision of interstellar gas clouds. While simplified in terms of physics and boundary conditions, the model captures the essential dynamics that govern such phenomena in nature. The observed behaviours—compression, clumping, pressure buildup, and eventual stabilization—align well with theoretical expectations and observational evidence, validating the core implementation and setting the stage for more sophisticated future work.

Github Repository link:-

https://github.com/SALMONPreet/SPH-Simulations-Fluid-and-Astrophysics.git

6. References

1. Lucy, Leon B. "A numerical approach to the testing of the fission hypothesis." *Astronomical Journal, vol. 82, Dec. 1977, p. 1013-1024.* 82 (1977): 1013-1024.

2. Gingold, Robert A., and Joseph J. Monaghan. "Smoothed particle hydrodynamics: theory and application to non-spherical stars." *Monthly notices of the royal astronomical society* 181, no. 3 (1977): 375-389.

3. Liu, Gui-Rong, and Moubin B. Liu. *Smoothed particle hydrodynamics: a meshfree particle method*. World scientific, 2003

4. Price, Daniel James, and Joseph John Monaghan. "An energy-conserving formalism for adaptive gravitational force softening in smoothed particle hydrodynamics and N-body codes." *Monthly Notices of the Royal Astronomical Society* 374, no. 4 (2007): 1347-1358.

5. Fraga Filho, Carlos Alberto Dutra. "Development of a computer code using the Lagrangian smoothed particle hydrodynamics (SPH) method for solution of problems in fluid dynamics and heat transfer." In *Proceedings of the XXXVII Ibero-Latin American Congress of Computational Method in Engineering–CILAMCE*, pp. 6-9. 2016.

6. Dutra, C. "Smoothed particle hydrodynamics: Fundamentals and basic applications in continuum mechanics." (2018).

7. Issac, Namitha, Anindya Saha, Saanika Choudhary, Aakash Chaudhary, Anandmayee Tej, Hong-Li Liu, Tie Liu, and Maheswar Gopinathan. "Cloud–

cloud collision and cluster formation in the W5-NW complex." *The Astronomical Journal* 167, no. 4 (2024): 158.

8. Sano, Hidetoshi, Kisetsu Tsuge, Kazuki Tokuda, Kazuyuki Muraoka, Kengo Tachihara, Yumiko Yamane, Mikito Kohno et al. "ALMA CO observations of a giant molecular cloud in M 33: Evidence for high-mass star formation triggered by cloud–cloud collisions." *Publications of the Astronomical Society of Japan* 73, no. Supplement_1 (2021): S62-S74.

9. Hubber, David A., Richard James Allison, R. Smith, and S. P. Goodwin. "A hybrid SPH/N-body method for star cluster simulations." *Monthly Notices of the Royal Astronomical Society* 430, no. 3 (2013): 1599-1616.

10. Press, William H. *Numerical recipes 3rd edition: The art of scientific computing*. Cambridge university press, 2007.