

Fluid Simulation Using Smoothed Particle Hydrodynamics Solver

Jakub Profota

January 19, 2025

Abstract

This work implements a basic smoothed particle hydrodynamics solver, a numerical computational method for simulating fluid flow. Brief overview of the underlying theory is presented, the report then focuses on the practical side. Detailed explanations of the use of OpenMP, MPI, and CUDA for computation are provided and extensively benchmarked on various fluid configurations and systems, ranging from mobile discrete low-power Nvidia GPUs to state-of-the-art many-core RISC-V processors.

Introduction

Fluid simulation can be approached in various ways, depending on the purpose of the simulation. Some implementations aim to study phenomena such as turbulence and wave propagation in engineering, weather forecasting, or environmental science. These applications require rigorously accurate physical results, making computations complex and time-consuming. In contrast, other areas, such as computer games, prioritize visual effects over scientifically accurate fluid behavior, opting for faster computation speeds at the expense of solution accuracy. Whether the chosen simulation method prioritizes real-time performance and visual appeal or is it expensive but physically accurate, it usually depends on approximate solutions to the Navier-Stokes equations that describe fundamental fluid dynamics.

Navier-Stokes

The Navier-Stokes equations are a set of nonlinear partial differential equations describing fluid substances' motion. These equations are fundamental to fluid mechanics and are derived

from the principles of conservation of mass, momentum, and energy. They mathematically express how the velocity field of a fluid evolves under the influence of various forces. There are multiple mathematical forms of the Navier-Stokes equations, and one such is presented below:

$$\nabla \cdot \mathbf{u} = 0$$
$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = -\nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f}$$

The first, so called continuity equation ensures that the fluid is incompressible and its density remains constant by constraining the divergence of the velocity field $\nabla \cdot \mathbf{u}$ to zero.

The second equation is the momentum equation, where ρ is density, \mathbf{u} is the velocity field, t is time, ∇p is gradient of pressure, ν is viscosity, $\nabla^2 \mathbf{u}$ is viscous term computed as Laplacian of the velocity field, and \mathbf{f} is the external force.

The Navier-Stokes equations are challenging to solve due to their nonlinearity. While analytical solutions may be found for simple cases, solving these equations for general cases proves difficult. In fact, one of the most famous unsolved problems in mathematics is proving whether solutions to the three-dimensional Navier-Stokes equations always exist and remain smooth. Since this report focuses on real-time fluid simulation in 3D space, some numerical method is needed to obtain an approximate solution to the equations.

Grid-based and particle-based numerical fluid solvers are two primary approaches to simulating fluid dynamics. Grid-based methods represent the fluid's spatial domain as a fixed lattice or grid. The fluid properties, such as velocity and pressure, are computed at discrete grid points. The approach

proposed by Jos Stam pioneered this technique and usually pops up first when searching for fluid solvers on search engines. Another well-known and well-performing solver is the Lattice Boltzmann Method.

Particle-based fluid solvers represent the fluid medium as a collection of discrete particles. Each particle carries mass, velocity, and density properties and interacts with neighboring particles to approximate fluid behavior. One of the primary advantages of particle-based solvers is their ability to naturally handle complex free-surface flows and interactions between fluids and solid boundaries. Such property makes them especially suitable for applications like splash effects in computer graphics. However, such methods also face challenges, such as maintaining particle coherence and minimizing numerical artifacts. Additionally, they may require careful tuning of parameters to achieve stable and realistic simulations, with one setting of parameters working correctly in one implementation and behaving unexpectedly in another.

Smoothed Particle Hydrodynamics

The SPH solver is a prime example of a particle-based numerical approach. The solver operates by using a smoothing kernel to interpolate the properties of particles within a defined radius, thus providing a way to evaluate fluid properties at any point in space. Since the SPH is mesh-free and does not rely on any topology, the computation interpolation of particle properties can be processed in parallel, proving the method adequate for multi-core and many-core systems.

The basic SPH solver presented in this report assigns the following properties to each particle: position \mathbf{r}_i , velocity \mathbf{v}_i , viscosity force \mathbf{f}_i^ν , density ρ_i , pressure force \mathbf{f}_i^p , and mass m_i . Additionally, some additional global parameters govern the computation: gravity external force \mathbf{g} , mass of a fluid particle m_0 , rest density ρ_0 , viscosity coefficient ν , stiffness coefficient k , and the smoothing radius h . Properties of particles are set to initial state, and the simulation proceeds by iteratively updating the particles at each simulation time step.

The SPH solver does this by computing three forces acting on each particle: gravity, viscosity, and pressure, which also requires computing density. Once the forces are obtained, they are integrated to compute the velocity of each particle by which the positions are updated.

Gravity is an external force and is the only force that does not rely on the neighboring particle properties:

$$\mathbf{f}_i^g = m_i \mathbf{g}$$

Viscosity is a measure of a fluid's resistance to flow or deformation. It describes how thick or sticky a fluid is. In more technical terms, viscosity represents the internal friction within the fluid, the resistance to the shear. The viscosity force is computed as follows:

$$\mathbf{f}_i^\nu = \nu \sum_j m_j \frac{\mathbf{v}_j - \mathbf{v}_i}{\rho_0} \nabla^2 W(|\mathbf{r}_i - \mathbf{r}_j|, h)$$

It is the sum of the viscosity forces acting on particle i due to all neighboring particles j . $\nabla^2 W$ is the Laplacian of smoothing kernel, and $|\mathbf{r}_i - \mathbf{r}_j|$ is the distance between particles i and j .

The density needed to compute the pressure force is computed as follows:

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

Finally, the pressure force:

$$p_i = k \left(\frac{\rho_i}{\rho_0}^7 - 1 \right)$$

$$\mathbf{f}_i^p = - \sum_j m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} \right) \nabla W(|\mathbf{r}_i - \mathbf{r}_j|, h)$$

First, the pressure p_i is computed based on the density of particle i . Then, the pressure force acting on particle i due to all neighboring particles j is computed.

Once all the forces are computed, they are integrated as an acceleration to update the velocity and thus position of each particle:

$$\mathbf{v}_i = \mathbf{v}_i + t (\mathbf{f}_i^g + \mathbf{f}_i^\nu + \mathbf{f}_i^p)$$

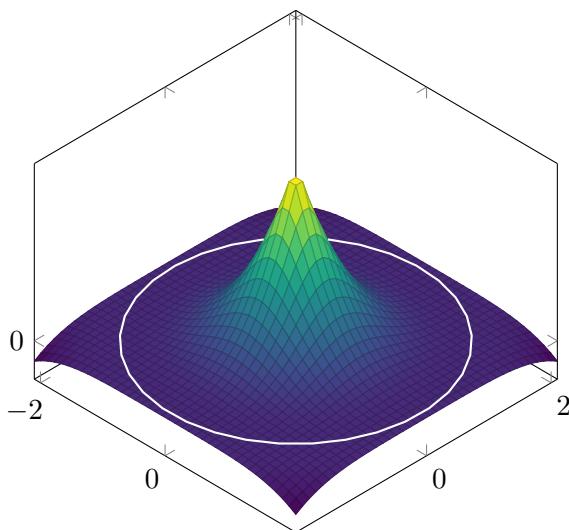
$$\mathbf{r}_i = \mathbf{r}_i + t \cdot \mathbf{v}_i$$

Analysis

Attentive readers may have spotted the streamlined approach to translating all those equations into code. Since the SPH solver sums the contributions of all other particles to determine the properties of each particle, it can be directly implemented using loops. However, iterating over all the particles for each is infeasible for a non-trivial amount of particles. Fortunately, SPH equations utilize specialized functions known as smoothing kernels, which weigh the influence of neighboring particles based on their distance from the particle currently being processed. The farther away a particle is, the lower its contribution will be, and contributions from particles located beyond the smoothing radius are effectively negligible. Thus, such particles can be omitted, and the total amount to be processed is significantly reduced.

Kernel Function

The Gaussian or cubic spline kernel is a common choice for the kernel function. The specific shape of the kernel function significantly affects the simulation results, including the stability and accuracy of the fluid dynamics being modeled. An example of one of the kernel functions used in the implementation presented in this report is depicted below:



The graph shows a mass kernel function used to compute boundary particle masses (more on that later) and densities. The two horizontal axes represent the relative position of neighbor-

boring particles in multiples of the smoothing radius, which is obtained as follows:

$$\mathbf{r}'_j = \frac{|\mathbf{r}_i - \mathbf{r}_j|}{h}$$

Then, the relative distance is computed as:

$$d_j = \sqrt{\mathbf{r}'_j \cdot \mathbf{r}'_j}$$

The SPH solver checks this distance and skips the neighboring particle if its relative distance exceeds number two. The kernel graph depicts the border between the points with valid distance and the points exceeding this threshold as a white circle. In other words, if the distance between a particle and its neighbor is greater than twice the smoothing radius, the neighbor is skipped and not considered. The kernel functions have a characteristic shape. There is the peak in the middle, with the function falling off to zero values at distances equal to two. The equation of the mass kernel function looks like this:

$$W_m(d_j, h) = \frac{0.25}{\pi h^3} \cdot \begin{cases} (2 - d_j)^3, & d_j > 1, \\ d_j^2 \cdot (3d_j - 6) + 4, & d_j \leq 1 \end{cases}$$

Uniform Grid Neighbor Search

The SPH solver can sort particles to a uniform grid to speed up neighbor search. The fundamental idea behind this approach is to divide the simulation space into a grid of uniform cells with size equal to the smoothing radius, where each cell contains a list of the particles located within its boundaries. When a particle evaluates its neighbors, it no longer needs to perform a naive search in the entire space. Instead, the algorithm only considers those particles within the same cell and adjacent cells, significantly reducing the number of comparisons needed. The maximum distance of two particles in two adjacent cells cannot exceed twice the size of the cell, i.e., twice the smoothing radius. This directly corresponds to the cut-off of the kernel functions. The uniform grid is rebuilt from scratch on each simulation time step as follows:

```
// Map particles to cells
for p in particles
    cells_map[p] = CellIndex(p.position)
```

```

// Initialize cells lookup
fill(cells_lookup, 0)
for c in cells_map
    cells_lookup[c]++
exclusive_scan(cells_lookup)

// Sort particles by cell index
copy(cells_lookup, cells_copy)
for p in particles
    c = cells_map[p]
    i = cells_copy[c + 1]--
    i--
    sorted[i] = p

```

First, each particle is mapped to a cell index ranging from zero to the total cell count. Then, the cell lookup is initialized. Its size equals the total cell count plus one and is initially set to zero. All the cell indices of particles are iterated, and each cell index in the cell lookup array is incremented by one. After the exclusive scan, the cell lookup contains a growing sequence of numbers starting from zero and ending with the total cell count. These numbers will later be used as offsets to the array of particles. The cell lookup is copied to a temporary array, which is used to sort particles. When sorting, the cell index is obtained. Then, this cell index is used to look up the offset of the following cell index. The offset of the following cell index is decremented by one, meaning it no longer points to the first particle of the following cell but to the last particle of the current cell. This offset is then used to determine the new position of a particle in a sorted array. Iterating over the following particles decreases the original offset further, pointing to new distinct locations where the corresponding particles should be placed. At the end of the sorting process, the sorted array contains particles with the same cell index group together, and the cell lookup array contains offsets to the first particle in each cell list.

```

for p in particles
    cell = CellPosition(p.position)

    // Check all adjacent cells
    for x in -1..1
        for y in -1..1
            for z in -1..1
                c = CellIndex(cell + (x, y, z))
                if c < 0 or c >= cell_count
                    continue

```

```

    // Index of the first particle
    // of the current cell
    j = cells_lookup[c]

    // Check against index
    // of the first particle
    // of the following cell
    while j < cells_lookup[c + 1]
        // Do stuff
        j++

```

When searching for neighbors, the SPH checks all the adjacent cells and extracts the index of the first particle in that cell from the lookup array. Then, it iterates through the particles until it hits the first particle in the following cell.

Boundaries

Fluid particles must be contained within the simulation space. Additionally, a solid object within this space may interact with the fluid. A straightforward approach to addressing the boundary problem is to reflect particles. This involves adjusting the velocity and position of fluid particles when they come into contact with the boundary, effectively bouncing them back into the fluid domain. Another approach is to use static boundary particles that simulate the behavior of solid surfaces. These particles do not represent the fluid, but they are included in the interpolation in the same manner as fluid particles, thereby enforcing boundary conditions through their interaction with the fluid. Most importantly, utilizing boundary particles increases the visual appeal of the simulation.

The placement of boundary particles can be challenging for complex shapes. However, the simulation space described in this report is a simple cube box. In this case, the static boundary particles are distributed uniformly along the box's walls. The distance between these particles is crucial. While shorter distances enhance the numerical stability of the simulation and help prevent artifacts, they also increase computation time, as more particles need to be processed. Therefore, the implementation positions the boundary particles at a distance equal to the smoothing radius, which is a good compromise.

Build and Run

The code is written in C++17 and uses OpenMP and CUDA for multi-core and many-core computation. The results of the deprecated MPI support for distributed processing are presented in the results section, but the MPI code is not included as it proved to be very buggy. On UNIX platforms, the meson build system is used to compile the project, while on Windows, one needs to compile the project in Visual Studio. The window system used is the SDL2 library, and the rendering is done via OpenGL.

Compilation

The user can compile the meson project using the included Makefile on UNIX platforms by simply running the `make` command. The meson will download and compile the necessary libraries, including the SDL2. However, the compilation may fail on some systems, including NixOS, so the user is advised to install the SDL2 package using his package manager of choice. The compilation outputs the `fluid` binary executable. If the computer has a discrete Nvidia GPU on UNIX system enabled by Nvidia Prime offloader, setting some environment variables before launching the executable is necessary. The user can run the `make run` command that sets the variables and executes the binary for ease of use.

On Windows systems, the compilation is more complicated. The attached Visual Studio solution includes the project configuration, but it relies on the location of header files in the `subprojects` folder used by the meson build system. The user is thus advised first to run the meson build using the commands `meson setup build` and `meson compile -C build`, which is expected to fail, but it downloads the corresponding headers. Then, the user can compile the project from within the Visual Studio.

If the computer does not have a CUDA-enabled GPU, the user has to delete the `USE_CUDA` definition, which serves as a compile-time configuration. To do so, the user must comment out the corresponding

lines in the `meson.build` file. The comments in the file should aid the user. In Visual Studio, the user must delete the definition from the project configuration by right-clicking on the solution in the solution explorer.

Runtime Configuration

The behavior of the simulation can be altered through the following configuration options:

```
{  
    "window": {  
        "title": "Fluid",  
        "width": 800,  
        "height": 800,  
        "point_size": 1.5  
    },  
    "compute": {  
        "cli": false,  
        "gpu": true,  
        "gpu_threads": 256,  
        "omp_threads": 8,  
        "steps": -1  
    },  
    "simulation": {  
        "delta": 0.0025,  
        "space_size": 1.2,  
        "smooth_radius": 0.04,  
        "fluid_grid": [24, 32, 24],  
        "box_boundary": false  
    },  
    "fluid": {  
        "gravity": -9.8,  
        "mass": 0.0000765,  
        "density": 1.0,  
        "viscosity": 0.05,  
        "stiffness": 10.0  
    }  
}
```

The `title`, `width`, and `height` options are self-evident. The `point_size` option sets the size of the fluid particle in screen space. `cli` toggles between graphical and terminal-only execution, but `steps` must be positive to make it work. `gpu` toggles between CPU and GPU computation. `gpu_threads` sets the number of threads per block. It is advised to use multiple of the size of the CUDA warp, i.e., 32. Other values may result in undefined behavior. `omp_threads` sets the number of threads to use on the CPU. The `steps` represents a number of simulation steps after which the execution should stop and print out the computation time. If the value is set

to -1 , the simulation proceeds indefinitely unless paused or shut down. `delta` is a fixed simulation time step. Large values result in unstable simulation. `space_size` defines the length of a side of the cube simulation space, while `smooth_radius` defines the cut-off distance of smoothing kernel functions and the length of a single cell in the cell grid. Increasing this value results in a more correct approximation but significantly slower computation. `fluid_grid` defines the initial configuration of fluid particles. `textttbox_boundary` toggles between boundaries of the box using static boundary particles or not. Then, five constants `gravity`, `mass`, `density`, `viscosity`, and `stiffness` defining the properties of the fluid follow.

The `config.json` file is open by default if the executable is run without an argument. However, `configs` directory contains three configurations used in the benchmark results section. Such a configuration can be selected by running the executable as `./fluid configs/<1-3>.json`.

Controls

Hold the left mouse button and move with the cursor to rotate the camera. Scroll the mouse wheel to zoom in and out. The space bar commences the simulation while another press pauses it. The R key resets the simulation to its initial state. The B key shows or hides static boundary particles if such are configured. Keys 1 to 4 change the colorization of the fluid particles to all blue, velocity-based, pressure-based, and density-based, respectively. The F key toggles between fullscreen and window. Finally, the escape closes the application.

Implementation

All of the codebase is located inside the `source` folder. The `main.cpp`, `window.hpp`, `window.cpp`, `shader.hpp`, and `shader.cpp` are not particularly important in discussing the SPH solver. A quick reference in the form of Doxygen documentation is available.

The `particles.hpp` file defines a structure of particles. The structure points to arrays of various fluid properties like position, viscos-

ity, or mass. All these arrays are the same length as the number of particles. However, the cells lookup array is the size of the total cell count plus one. The array stores offsets to the first particle in each cell and is used in neighbor search, as previously discussed in the analysis section. Note that there are two arrays for positions and velocities. Whenever particles are sorted to cells, and the cells lookup array is initialized, the sorting of positions and velocities does not happen in place but rather to the other two arrays. Those arrays are then marked as the current, and when the particles are sorted again, the positions and velocities are sorted from those arrays back to the former. These swap buffers are utilized to enable parallel sorting.

The `fluid.hpp` defines the actual fluid simulation structure. The data type holds many parameters required for the computation of the simulation. The curious reader is advised to check Doxygen, but the report will discuss the inner workings in the upcoming text.

The `fluid.cpp` file forwards function calls to the CPU or the GPU implementation and is a great place to understand what is happening. The `Create` and `Destroy` functions create and destroy the simulation, respectively. The JSON file is parsed, and the configuration is loaded into the structure. Then, the `Reset` function is called, which resets the simulation to the initial state by placing the particles into the simulation space. Additionally, if static boundary particles are to be used, they are placed in the space, sorted into cells, and their masses are set. When the simulation commences, the `Update` function is called, and it calculates one SPH solver simulation step. The actual implementation of these functions is found in `cpu.cpp` and `gpu.cpp` files.

OpenMP

When the compute on the CPU is requested, and the `PlaceParticles` function is called, the implementation dynamically allocates all the arrays and initializes values. Every property of each particle is set to zero, except the mass, which is set to the mass constant. The `fluid_grid` configuration parameter dictates the initial fluid shape in the simulation space's center. Static boundary particles are

optionally placed along the box's walls but inside the simulation space, so the particle sorting into cells works on them. The `omp_chunk` variable is set to the number of fluid particles divided by the number of OpenMP threads `omp_threads`.

Sorting the particles in the `SortParticles` function follows the pseudocode presented in the analysis section of the report. Practically any loop inside the `cpu.cpp` file is prepended with the following OpenMP directives. The scheduling policy used for iterations is static. Each OpenMP thread is assigned a chunk of data of size `omp_chunk`:

```
#pragma omp parallel for \
    num_threads(omp_threads) \
    schedule(static, omp_chunk)
```

Incrementing the corresponding cell in the cells lookup array must be handled securely. The OpenMP directive ensures that the following statement updates the shared variable atomically, preventing race conditions:

```
for (int i = 0; i < p.size; i++) {
    #pragma omp atomic update
    p.cells_lookup[cells_map[i]]++;
}
```

When sorting the particles into the second swap buffer, the copy of the cells lookup is decremented. Again, such an operation must be safe. This time, a directive with `capture` instead of `update` is used to capture the decremented value into a variable correctly. The variable has to be declared before the OpenMP directive to make it work:

```
for (int i = 0; i < p.size; i++) {
    int c = cells_map[i];
    int j;
    #pragma omp atomic capture
    j = p.cells_copy[c + 1]--;
    j--;

    // Place particle from index i to j
}
```

The `Update` function is split into five additional function calls: `ApplyGravity`, `ApplyViscosity`, `ApplyDensity`, `ApplyPressure`, and `Integrate`. These methods update the particles' properties and then update each particle's velocity vector.

At the end of the `Update` function, all particles move to their new position.

MPI

In a distributed multi-process environment, the `SortParticles` function is challenging to parallelize. The deprecated MPI implementation chose a single driving process that handled the sorting and distributed the sorted arrays to other processes. The driving process was chosen as the one with the MPI rank equal to zero:

```
int mpi_rank;
int mpi_size;
MPI_Init(NULL, NULL);
MPI_Comm_rank(MPI_COMM_WORLD, &mpi_rank);
MPI_Comm_size(MPI_COMM_WORLD, &mpi_size);
```

Now, during the computation, each process checks against its rank. If it is not the driving process, it waits for the one at the next `MPI_BARRIER` and then collects the data broadcasted from the driving process to all other processes:

```
if (mpi_rank == 0)
    SortParticles(fluid);

MPI_Barrier(MPI_COMM_WORLD);
MPI_Bcast(fluid.cells_lookup,
          cell_count + 1, MPI_INT,
          0, MPI_COMM_WORLD);
MPI_Bcast(fluid.positions,
          fluid.size * sizeof(float3), MPI_BYTE,
          0, MPI_COMM_WORLD);
MPI_Scatter(fluid.velocities,
            chunk_size * sizeof(float3), MPI_BYTE,
            fluid.velocities + chunk_index,
            chunk_size * sizeof(float3), MPI_BYTE,
            0, MPI_COMM_WORLD);
```

The positions of all particles are broadcasted to all the processes since the SPH solver computation requires all that information. However, each velocity is only updated by a single process. In that case, the driving process does not need to send out the whole velocities array to all processes but instead distributes corresponding chunks of data to each using the `MPI_Scatter` function. Some functions, like `ApplyPressure`, need all the particles' values of some property, so the `MPI_Allgather` function is used to collect the corresponding chunk of data from each of the processes on all

of the processes. Upon doing so, each process has a complete copy of the corresponding property array:

```
ApplyGravity();

MPI_Barrier(MPI_COMM_WORLD);
MPI_Allgather(/* velocities */);
ApplyViscosity();

MPI_Barrier(MPI_COMM_WORLD);
ApplyDensity();

MPI_Barrier(MPI_COMM_WORLD);
MPI_Allgather(fluid.densities +
    chunk_index, chunk_size,
    MPI_FLOAT, fluid.densities,
    chunk_size, MPI_FLOAT,
    MPI_COMM_WORLD);
ApplyPressure();

MPI_Barrier(MPI_COMM_WORLD);
MPI_Allgather(/* velocities */);
Integrate();
```

CUDA

The code related to Nvidia's CUDA can be easily spotted since it is always between USE_CUDA define guards. Most of the relevant code is in the `gpu.cpp` file, but two other notable differences exist between the CPU and GPU versions. First, declaring the fluid simulation structure in the `fluid.hpp` file introduces new variables related to the CUDA execution graphs. Second, the `Reset` function in the `fluid.cpp` also introduces a new function call, `Setup`, which utilizes the newly created variables and is implemented in the `gpu.cpp` file. The `Setup` function is only called once to setup the execution graphs before the simulation:

```
int blocks = (fluid.size + gpu_threads -
    1) / gpu_threads;
cudaStreamCreate(&stream);

// Setup for each of the swap buffers
for (int i = 0; i < 2; i++) {
    cudaStreamBeginCapture(stream,
        cudaStreamCaptureModeGlobal);

    ApplyGravity_CUDA<<<blocks,
        gpu_threads, 0, stream>>>(
        /* args */);
```

```
ApplyViscosity_CUDA<<<blocks,
    gpu_threads, 0, stream>>>(
    /* args */);

AddViscosity_CUDA<<<blocks,
    gpu_threads, 0, stream>>>(
    /* args */);

ApplyDensity_CUDA<<<blocks,
    gpu_threads, 0, stream>>>(
    /* args */);

SetPressure_CUDA<<<blocks,
    gpu_threads, 0, stream>>>(
    /* args */);

ApplyPressure_CUDA<<<blocks,
    gpu_threads, 0, stream>>>(
    /* args */);

Integrate_CUDA<<<blocks,
    gpu_threads, 0, stream>>>(
    /* args */);

cudaStreamEndCapture(stream, graph[i])
cudaGraphInstantiate(&exec[i],
    graph[i], NULL, NULL, 0);
}
```

The `Setup` function utilizes a CUDA stream to capture CUDA kernel calls within the CUDA execution graph. Once the capture process concludes, the graph is instantiated and ready for future launches. This procedure is repeated twice because the implementation employs two swap buffers for storing positions and velocities. The stream capture records the addresses of the pointers; changing the swap index is not enough. As a result, two execution graphs are created and instantiated for one of the two sets of swap buffers.

Additionally, there are new kernel functions called `AddViscosity` and `SetPressure`. These functions are necessary because the `_syncthreads()` directive, used within a CUDA kernel, only synchronizes threads within the same block. The solver, however, requires synchronization across all particles. Unfortunately, using stream capturing with `cudaMemsetAsync` did not yield the desired results, since `cudaMemset` only sets bytes, while larger data types, such as floats, are needed.

With the instantiated CUDA execution graphs, the captured sequence of kernel calls

is executed in the `Update` function as follows:

```
cudaGraphLaunch(exec[swap], stream);
cudaStreamSynchronize(stream);
```

Another function that is exclusive to the GPU implementation is the `Draw` function. Here, CUDA interoperability with OpenGL is utilized to directly copy the data from the CUDA buffer to the OpenGL's vertex buffer object:

```
struct cudaGraphicsResource *resource;

glBindBuffer(GL_ARRAY_BUFFER, vbo);
glBufferData(GL_ARRAY_BUFFER,
             fluid.size * sizeof(float3),
             NULL, GL_DYNAMIC_DRAW);

cudaGraphicsGLRegisterBuffer(&resource,
                            vbo, /* just too long flag */);
cudaGraphicsMapResources(1, &resource);

void *ptr;
size_t size;
cudaGraphicsResourceGetMappedPointer(
    &ptr, &size, resource);

cudaMemcpy(ptr, fluid.positions,
           fluid.size * sizeof(float3),
           cudaMemcpyDeviceToDevice);

glDrawArrays(GL_POINTS, 0, fluid.size);

cudaGraphicsUnmapResources(1, &resource);
```

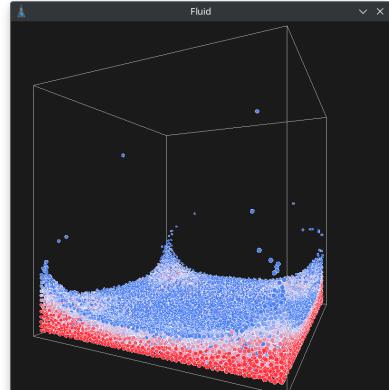
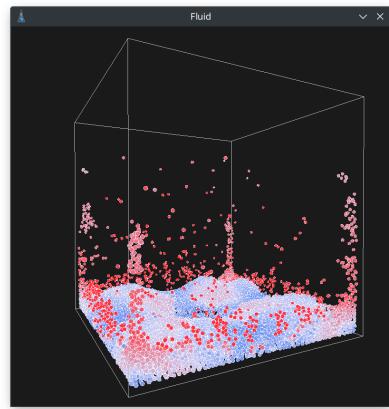
The CUDA registers a graphics resource that encapsulates the OpenGL vertex buffer object. Then, it maps the resource into the memory, and the pointer is obtained. `cudaMemcpy` copies data to the vertex buffer, and the contents can be drawn. Finally, the CUDA graphics resource is unmapped.

The OpenMP CPU version of `SortParticles` uses atomic directives to ensure correct and safe cell lookup and sorting initialization. The CUDA GPU function uses `atomicAdd` and `atomicSub` calls, achieving the same. Furthermore, since they operate on the global memory, the atomicity works across all the particles, not only in the context of the current block.

Results

The implementation is benchmarked on three different CPUs: Intel Core i7-8565U mobile CPU with four cores, 1.8 GHz base, and 4.6 GHz maximum clock speed. It boasts 8 megabytes of cache and consumes 25 Watts. The following CPU is an AMD Ryzen 5 5600X desktop-class CPU. Its six cores run on a 3.7 GHz base and 4.6 GHz maximum clock speed. It requires 65 Watts and has 3 megabytes of L2 and 32 megabytes of L3 cache. They both support hyperthreading. The last CPU is the SOPHON SG2042, a RISC-V processor with a massive 64 cores in 16 clusters. Each cluster shares a 1-megabyte L2 cache, and the whole CPU uses a 64-megabyte system cache. Its base frequency stands at 2 GHz. The CPU typically consumes 120 Watts.

The discrete mobile GPU tested is Nvidia's GeForce MX250. It has 384 shading units in 3 streaming multiprocessors. It operates at around 1520 MHz and drains 25 Watts. On the contrary, the powerful desktop-class Nvidia RTX 3070, with clock speeds of 1815 MHz, boasts 5888 CUDA cores. It requires a 650-watt power supply.

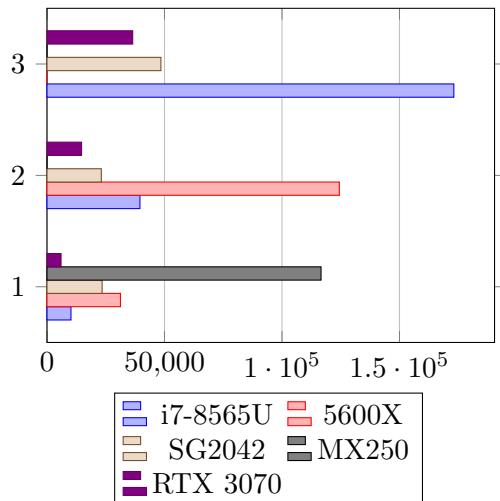


The RISC-V CPU in the Milk-V Pioneer desktop may be the first 64-core RISC-V CPU that got to run a fluid simulation:



The configs directory contains three configuration files, named 1, 2, 3 in the table below, that were used in the measuring for the following table. The first configuration simulates 5,632 fluid particles in a uniform grid of 15,625 cells. The second increases the number of particles to i18,432 and number of cells to i27,000. The last configuration is the most demanding, with particles and cells equal to 49,152 and 52,735, respectively. Benchmark uses 1,500 simulation steps. The reported times are in milliseconds.

CPU	1	2	3
i7-8565U	10,211	39,555	173,139
5600X	31,260	124,411	-
SG2042	23,459	23,115	48,472
MX250	116,623	-	-
RTX 3070	5,974	14,725	36,475



The configurations used the best settings for each of the systems. Intel CPU utilized 8 OpenMP threads, the AMD CPU used 12, and the SOPHON processor used 64. Both GPUs ran on 256 threads per block. It is interesting to see that the mobile CPU on Linux outperformed the desktop CPU on Windows machine by such a large margin. The Nvidia MX250 GPU ran quite well in the first hundreds of iterations, but due to the power throttling, the performance dropped significantly, and the additional tests were not run. Such a drop is suspected to be the result of the underlying operating system, NixOS, driver support and power management. Additionally, the MX250 GPU supports offloading via Nvidia Optimus, a feature which rarely works well on Linux. The following table shows the results of the same configuration but instead of 1,500 steps, the simulation prints out the computation time after 500 steps before the power throttling occurs. Again, the times are in milliseconds.

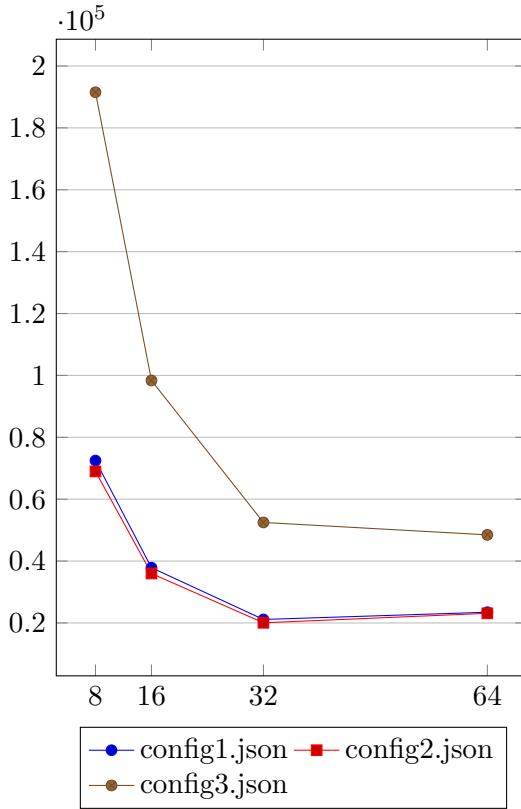
	MX250	500 steps	1,500 steps
Total time:	6,656.000	116,623.000	
Time per step:	13.312	77.748	

Various runtime configuration options greatly influence the simulation's performance, and the following text discusses the most important ones. The `omp_threads` parameter is crucial for the OpenMP implementation, translating directly to the speedup. The following table shows the speedup of the RISC-V CPU when the number of threads is increased. The speedup is proportional to the number of threads as expected on 1,500 simulation steps:

omp_threads	config1.json
8	72,479
16	37,860
32	21,131
64	23,459

omp_threads	config2.json
8	68,968
16	35,981
32	20,054
64	23,115

omp_threads	config3.json
8	191,508
16	98,360
32	52,480
64	48,472

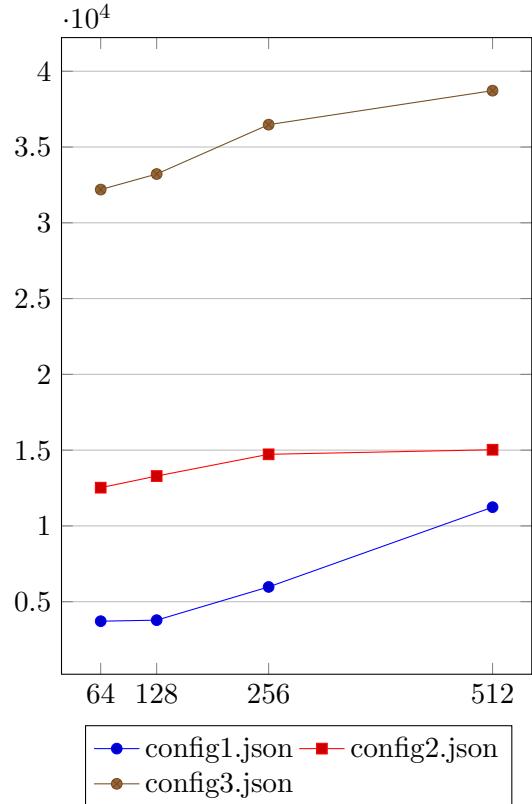


For the GPUs, the `gpu_threads` parameter does not play such a significant role. The following table shows the change in speeds of the Nvidia RTX 3070 GPU when the number of threads per block is changed on 1,500 simulation steps:

gpu_threads	config1.json
64	3,714
128	3,781
256	5,974
512	11,237

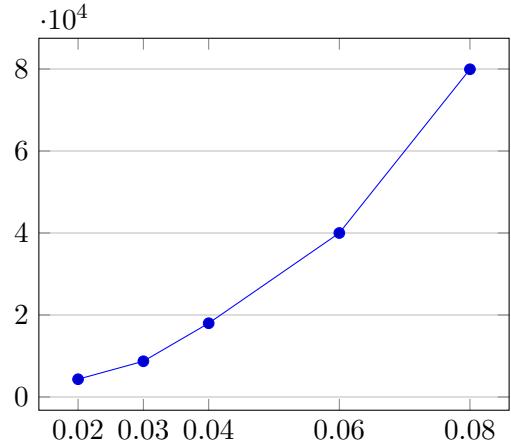
gpu_threads	config2.json
64	12,523
128	13,290
256	14,725
512	15,026

gpu_threads	config3.json
64	32,193
128	33,218
256	36,475
512	38,719

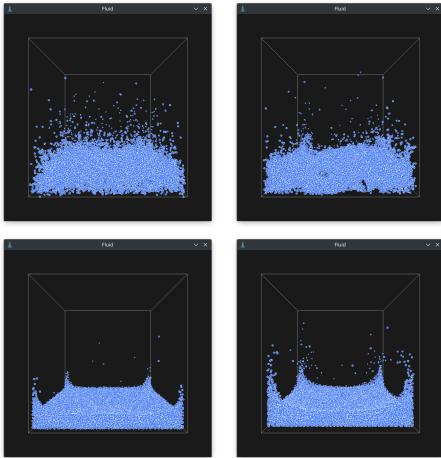


Another important parameter is the `smooth_radius`. Increasing the value results in a more accurate simulation but significantly slower computation. Decreasing the value speeds up the simulation but may render the simulation unstable. The following table presents the impact of various smoothing radii on the i7-8656U CPU on the second configuration on 750 simulation steps:

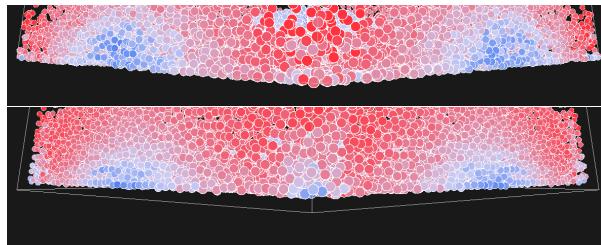
smooth_radius	config2.json
0.02	4,333
0.03	8,716
0.04	17,990
0.06	39,899
0.08	79,939



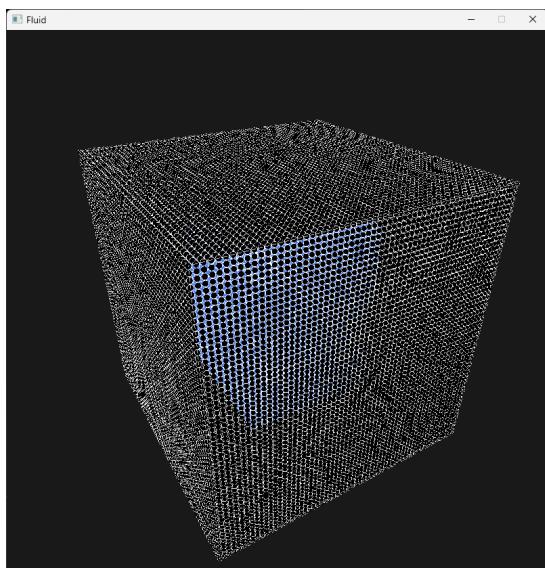
The following images show the fluid simulation state after the 750 simulation steps with the smoothing radii of 0.02, 0.03, 0.04, and 0.08, respectively:



The `box_boundary` does not influence the performance but alters the simulation's visual appearance. The first image shows the simulation without the static boundary particles, while the second image shows the simulation with them:

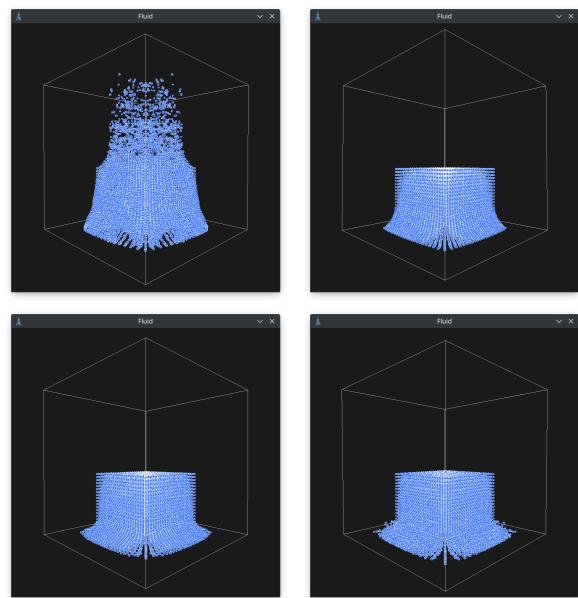


The invisible static boundary particles would look like this:



When the simulation is kept running until the fluid settles, the visual impact of the static boundary particles is even more apparent. The fluid without the static boundary particles settled much faster and did not exhibit as many eye-catching splashes as the one with the static boundary particles.

Lastly, the fluid simulation constants alter the fluid's behavior. Here are the states of the fluid after 125 simulation steps with the viscosity set to 0.5, 0.4, 0.2, and 0.05, respectively. The 0.5 viscosity results in an almost solid-like behavior, causing such pressures that the fluid explodes:



Conclusion

This report presented a fluid simulation implementation using the Smoothed Particle Hydrodynamics solver method both on the CPU and GPU. The resulting code was benchmarked on various hardware configurations. Real-time fluid simulation performance was achieved on all tested systems with pleasing visual results.