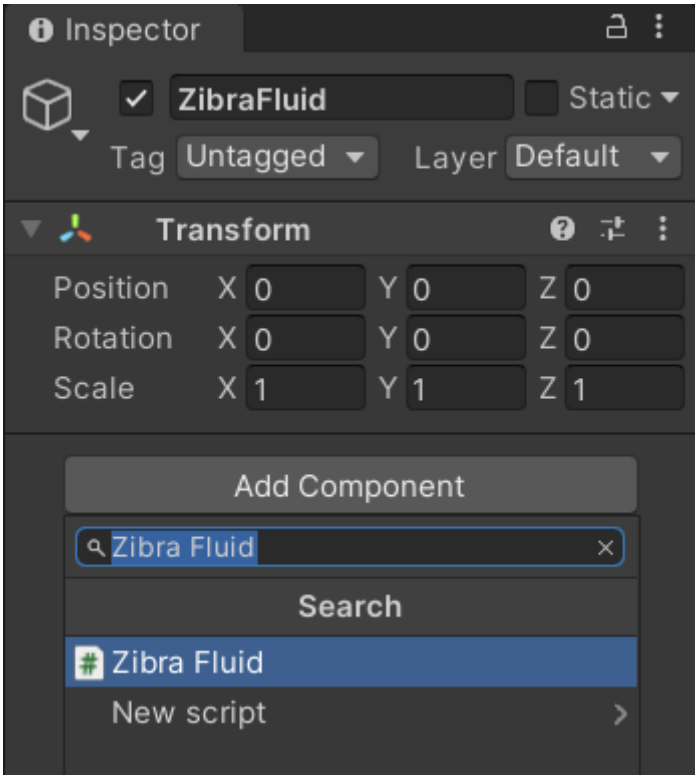
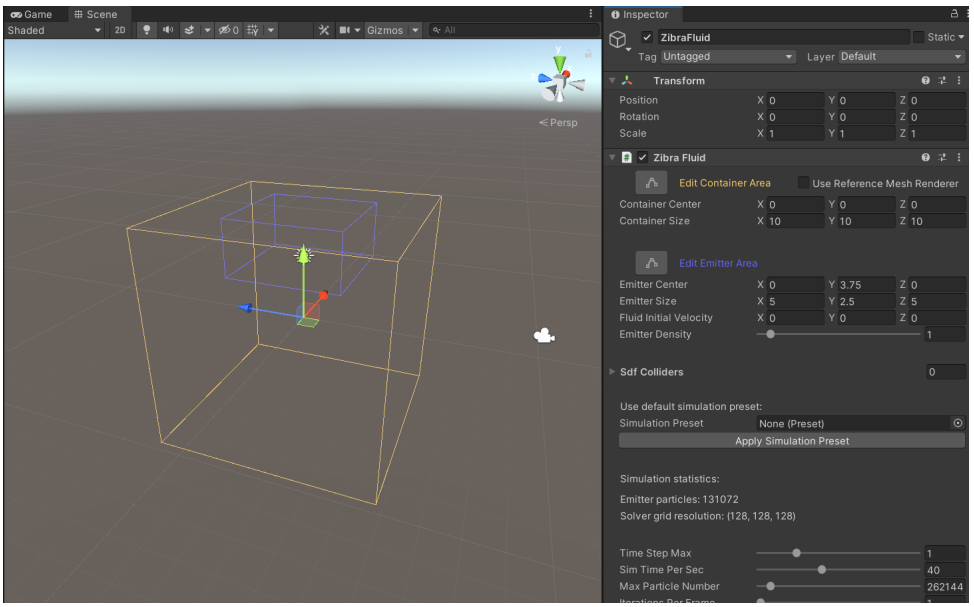
Table of contents:

1. Creation of ZibraFluid simulation

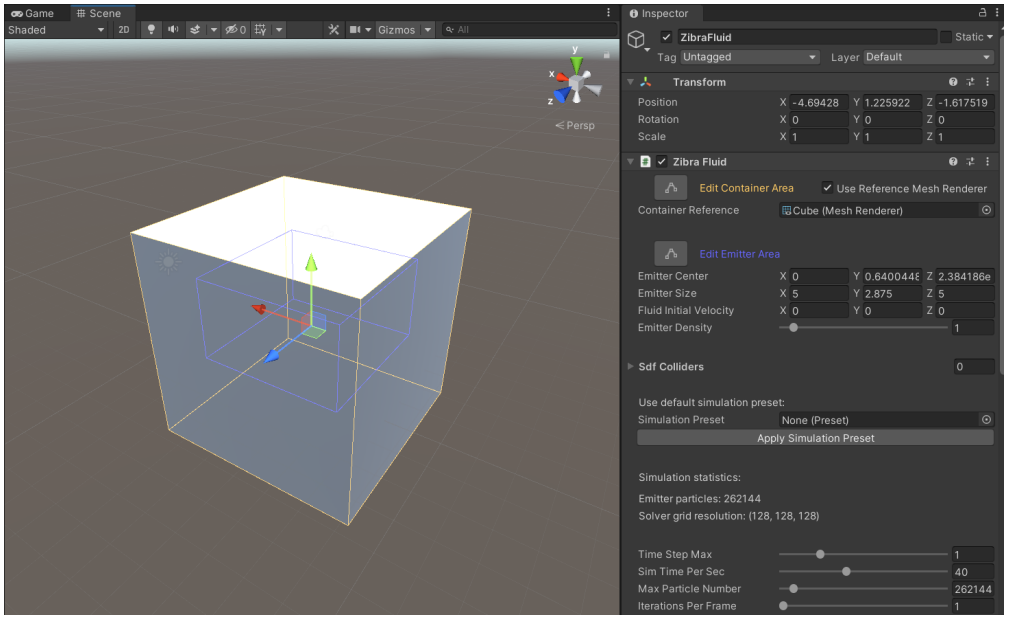
To create a ZibraFluid simulation you can create an empty gameobject and add a ZibraFluid component



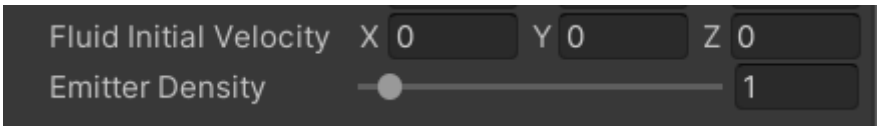
After that a simulation with default parameters and container size will be created. The simulation domain will be located in the container area, and the emitter area will define the initial position, size, speed, and density of the fluid.



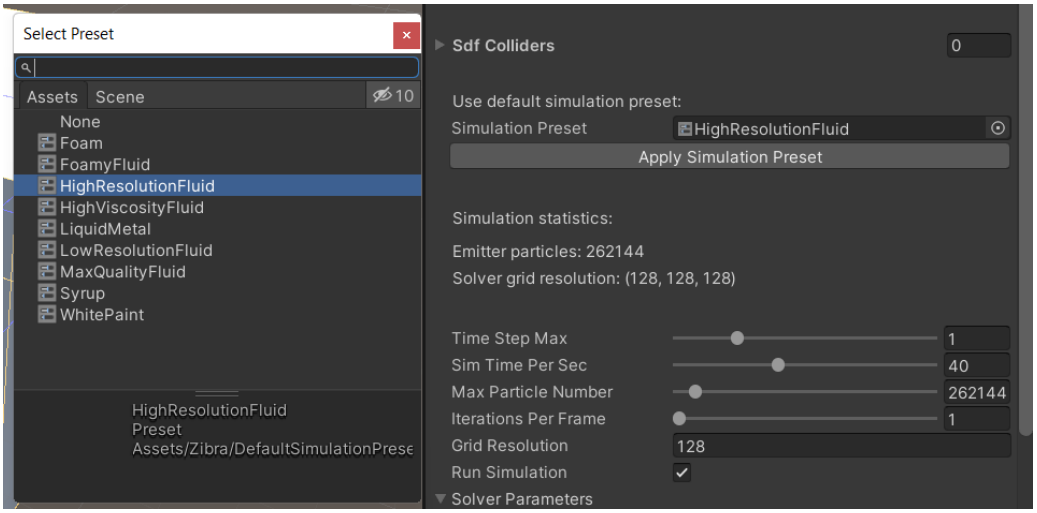
By clicking on the “Edit Container Area” or “Edit Emitter Area” button you can manually change their position and size in the scene editor. Also, you can use a reference object bounding box by checking the “Use reference mesh” checkbox, after which you should add the object of interest as the reference.



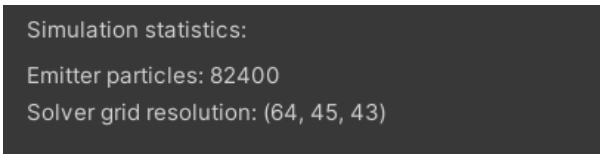
The emitter parameters define how the simulation is initialized.



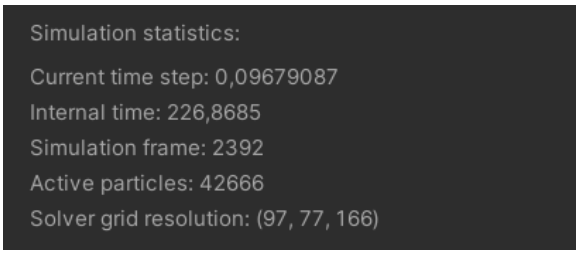
For example, if you wish for the fluid to start by moving in the X direction you can set the X component of the “Fluid Initial Velocity” to 1.0. The “Emitter Density” parameter defines how dense the initial fluid is compared to its rest state (rest density), if the density is higher than 1 than because of the pressure the fluid will explode in all directions.



By using a default simulation preset you can simplify the setup process to create a specific type of simulation. Then you can finetune the parameters to your liking.

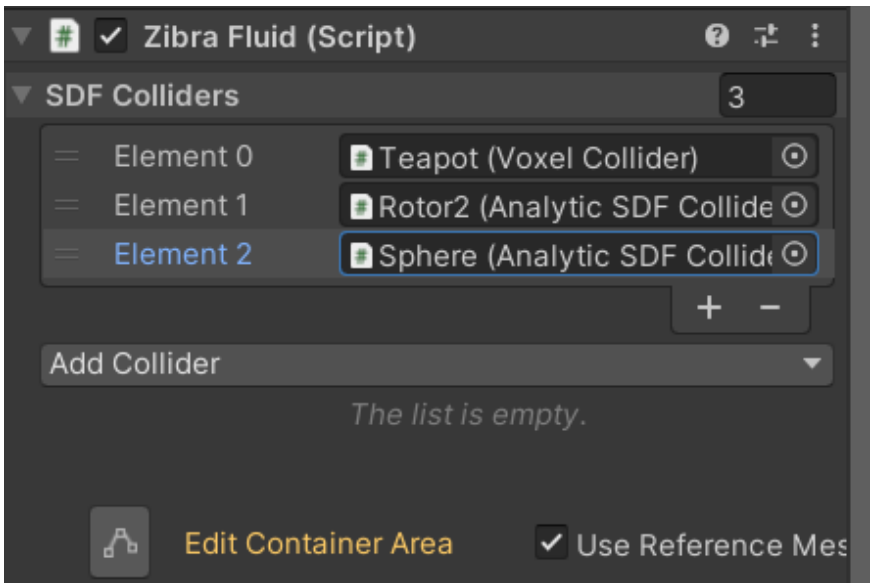


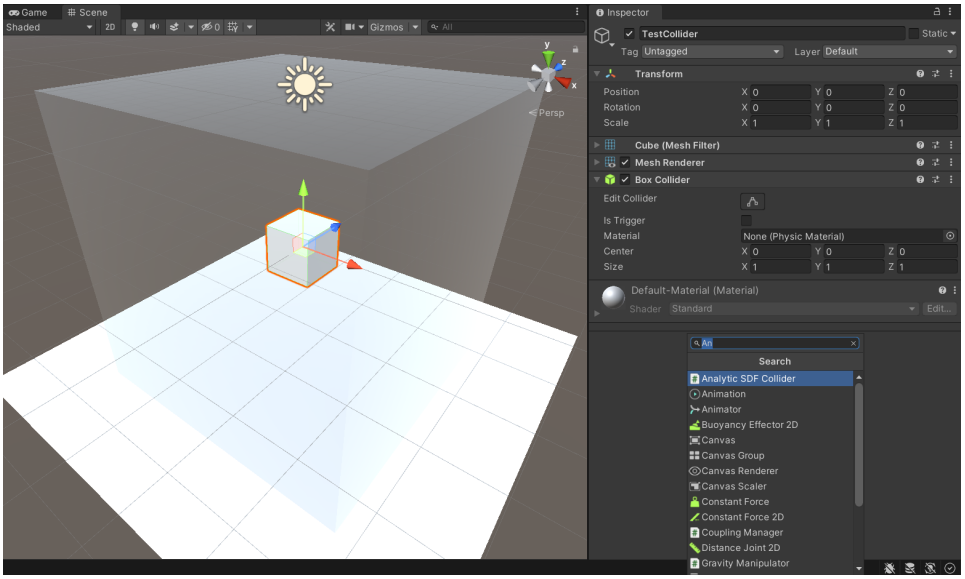
In editor mode

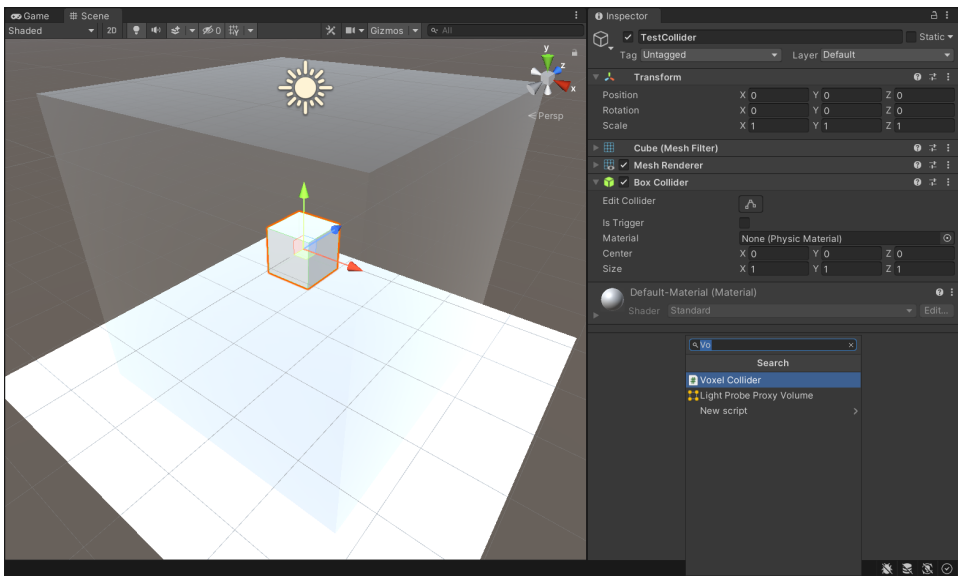


In game mode

The simulation statistics gives general information about the simulation. Based on the emitter size and density the number of particles is calculated and shown here. The solver grid resolution is calculated based on the cell size and the container size, to get a higher quality simulation you need to reduce the grid cell size, but it is recommended to keep the grid resolution smaller than (256,256,256) due to memory limitations of the current solver implementation.


Zibra Fluid (Script)
SDF Colliders
3
Element 0
Teapot (Voxel Collider)
Element 1
Rotor2 (Analytic SDF Collide
Element 2
Sphere (Analytic SDF Collid
Add Collider
The list is empty.
Edit Container Area
Use Reference Mes

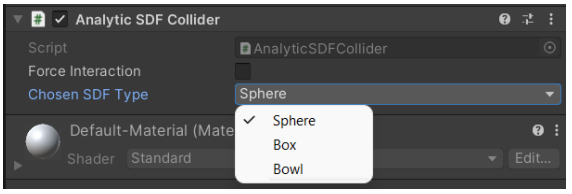

Game
Scene
Inspector
TestCollider
Transform
Cube (Mesh Filter)
Mesh Renderer
Box Collider
Edit Collider
Is Trigger
Material
None (Physic Material)
Center
Size
Default-Material (Material)
Shader
Standard
Search
Analytic SDF Collider
Animation
Animator
Buoyancy Effector 2D
Canvas
Canvas Group
Canvas Renderer
Canvas Scaler
Constant Force
Constant Force 2D
Coupling Manager
Distance Joint 2D
Gravity Manipulator


Game
Scene
Inspector
TestCollider
Transform
Cube (Mesh Filter)
Mesh Renderer
Box Collider
Edit Collider
Is Trigger
Material
None (Physic Material)
Center
Size
Default-Material (Material)
Shader
Standard
Search
Voxel Collider
Light Probe Proxy Volume
New script

2. Adding colliders

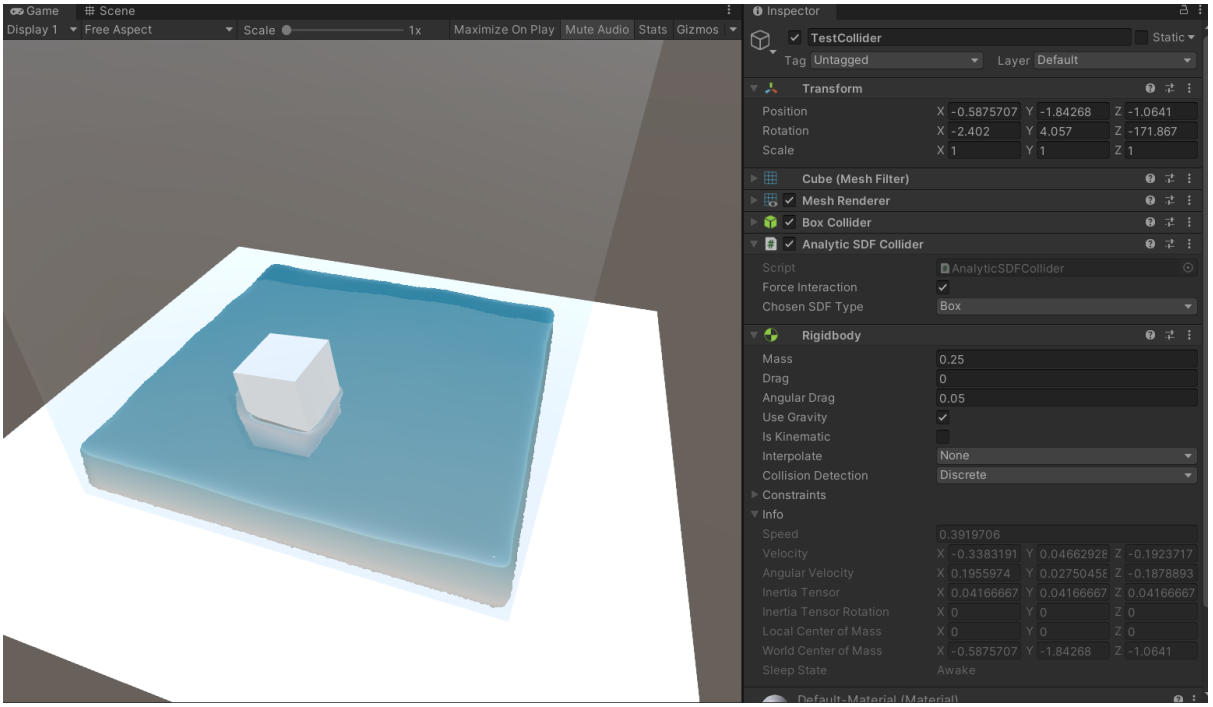
To add objects the solver can interact with, you first need to first add a Voxel Collider/Analytic SDF Collider component to the object of interest. After that depending on the type of component, you can either generate a Neural SDF representation of the object, for the Voxel Collider component, or select one of analytical SDF types.

Voxel Collider Component (it is not available in the free version of the plugin, since it requires sending the model to the backend server to generate the representation)

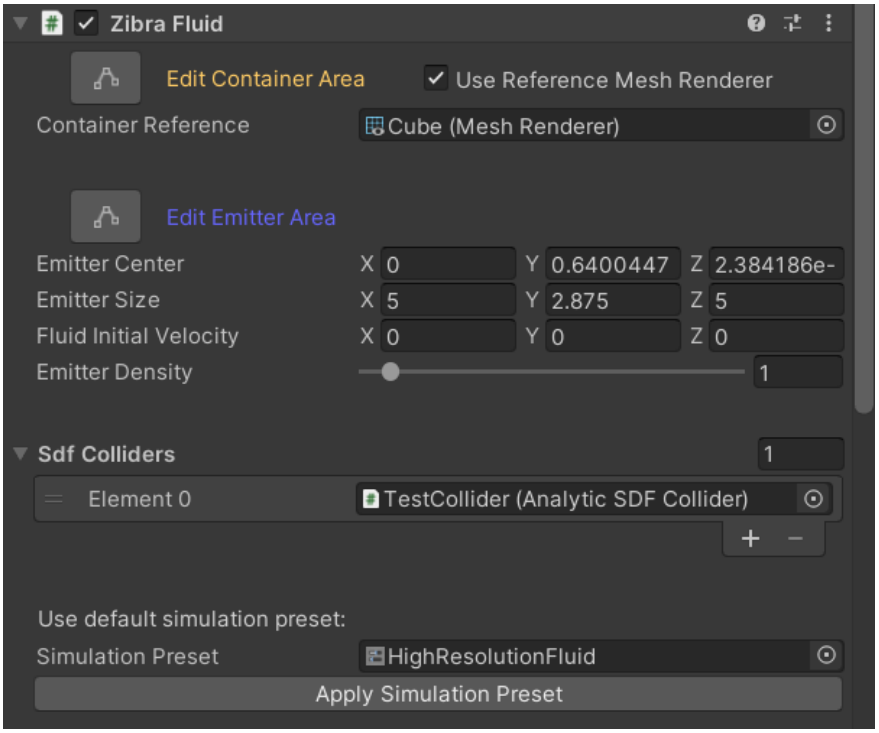


Analytical SDF Collider

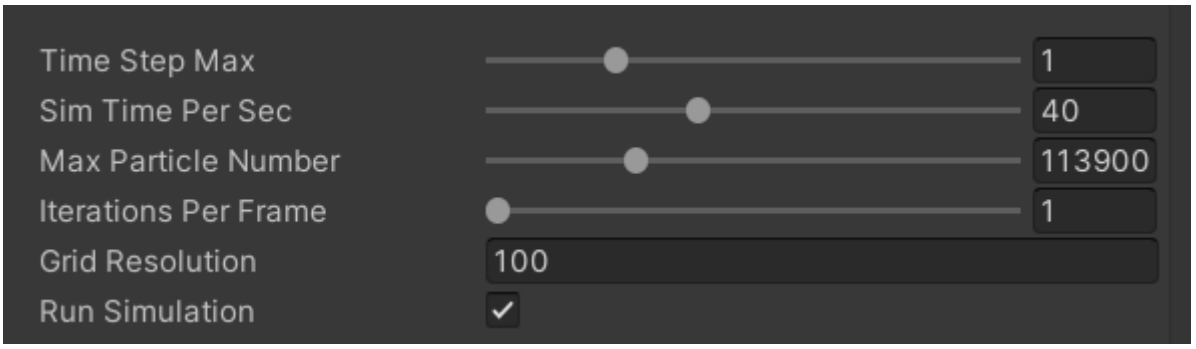
You can also turn on “Force interaction”, this allows the fluid to interact with rigid bodies. For this to work the object needs to also have a rigid body component.



Then for a ZibraFluid instance to interact with a collider it needs to be added to the SDF Colliders list in the ZibraFluid component settings.



3. Main simulation parameters

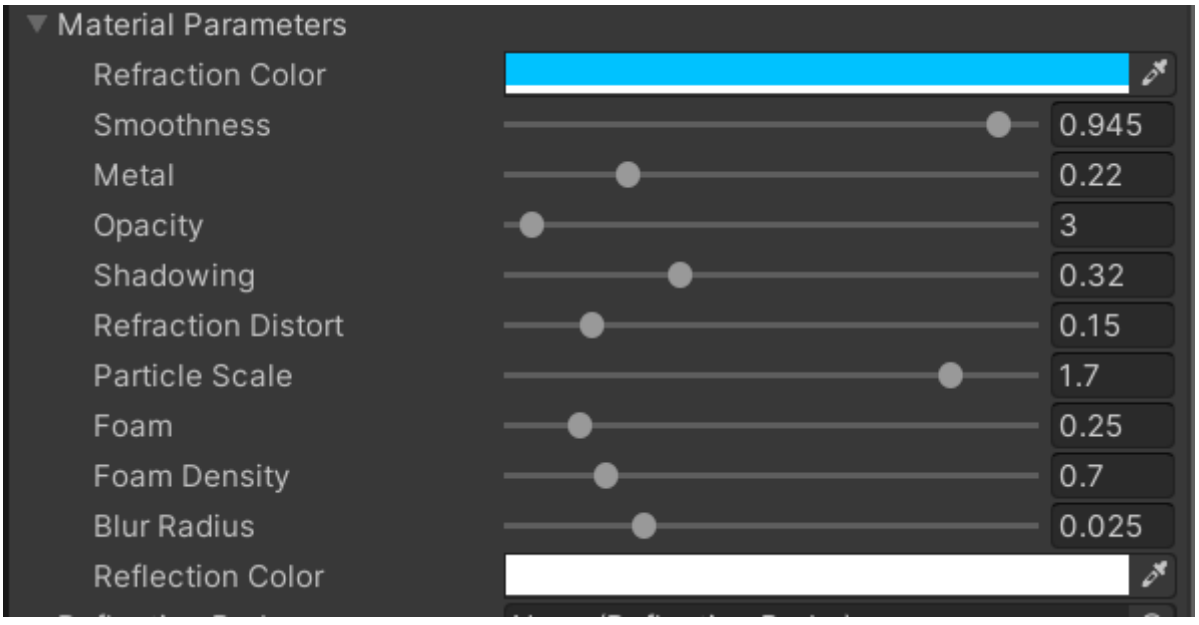


- TimeStepMax - The maximum allowed simulation timestep. The higher the max timestep is the less stable the simulation might get.
- SimTimePerSec - The speed of the simulation, how many simulation time units per second
- MaxParticleNumber - The maximum allowed particle number
- IterationsPerFrame - The number of solver iterations per frame, in most cases one iteration is sufficient
- Grid Resolution - Main parameter that regulates the resolution of the simulation. Defines the size of the simulation grid based on the largest side length of the simulation container. The actual grid resolution is given in the simulation statistics section.

4. Solver parameters

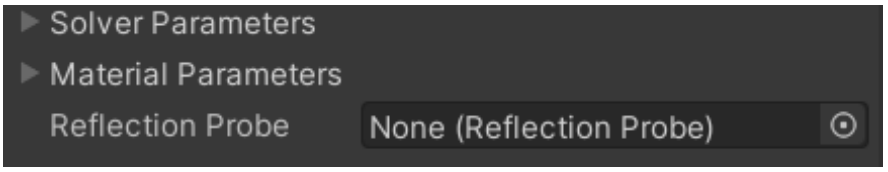
- Gravity - The strength and direction of the gravity, given as a Vector3
- Fluid Stiffness - The stiffness of the fluid, recommended 0.1f. The higher the value the more incompressible the fluid becomes.
- Particles Per Cell - The average number of particles per grid cell, the higher the value the more accurate the simulation becomes. But for the same volume of fluid you will need a higher number of particles.
- Velocity Limit - The velocity limit of the particles, in grid cells per simulation time unit.
- Viscosity - Defines how well the simulation preserves the energy of the fluid.
- Boundary Force - The strength of the force acting on a fluid while its touching objects.

5. Material parameters



- RefractionColor - The color of the fluid volume
- Smoothness - The smoothness of the fluid surface
- Metal - The metalness(reflectivity) of the surface
- Opacity - The opacity of the fluid volume
- Shadowing - Fluid depth dependent darkening of the color

- Refraction Distort - The amount of refraction distortion
- Particle Scale - Particle rendering scale compared to the cell size
- Foam - intensity of a foam effect.
- Foam Density - the density threshold of the foam, sets a maximal density below which the foam can be seen.
- Blur Radius - the relative radius of a screen space bilateral blur of the particle normals and depth.



You can also specify a custom reflection probe which will be used in the fluid rendering instead of the default Unity one.