

SPH Fluids - Documentation

Implementation notes

This SPH solver is based on the paper Particle-based Viscoelastic Fluid Simulation. In the paper, they introduce a Double Density Relaxation algorithm. In SPH, when you increase the rest density, the particles will "attract" each other more. In the traditional SPH implementations, this caused particles to "clump." (You can see this effect by turning "Stiffness near" to zero in the node). For example if a particle had a few very close particles around it, its density would be equal to if it had many moderately close particles around it. Thus, in the paper, they proposed that we use a second kernel to keep particles from clumping.

Now, I'll try to explain which parameters go with which variables. "Surface tension" in the node is actually "rest density" in the paper. The higher it is, the more the particles will attract each other. "Stiffness" determines how hard the fluid will try to reach the rest density. "Stiffness near" will keep the particles from going too close to each other and it's what gives us that nice surface tension effect instead of a bunch of particle clumps.

How to start

Emitting fluids from objects

With the SPH Addon you will get a new toolbar under View>Others>SPH_Tools.

To start a fluid simulation just select the object you want and click: "Emit Fluids".

The script

will automatically create a point cloud and connect all the nodes necessary in the ICEtree.

Caching

Like all fluids simulators, large number of particles or higher accuracy (more substeps) will take longer to simulate.

You will find in the SPH_Tools toolbar a "Cache" button, which will quickly cache the selected point cloud

to a temporary location on the simulate folder (by default

XSIPROJECTPATH\Simulation\SPH_Fluids_Cache\Name of the point cloud).

To see the cached simulation, select the point cloud and "Load cache Cloud". This will load the last cached simulation and hide the simulated cloud (to prevent that XSI recalculates the simulation cloud).

After you do changes to the simulation you must click "Cache" in order to see the updated cached simulation.

Nodes reference

SPH Emit Fluids from Volume

Node Description

Parameters/Ports Reference

Enable	turn the node on or off.
Emitter (port)	Emitter geometry (or geometries). Plug the

	emitter geometry on this port or a group with the emitter. Total number of particles/Rate per second Amount of particles emitted. If rate type is set to "Total" particles will be emitted at once on the first frame of the simulation. If set to "rate per Second", the number of particles will be equal to the amount emitted per second of simulation. Tip: This parameter is animatable, if you want a "jet" of fluid coming from the emitter you can set this on/off as desired or you can reduce the number of particles to have the emitter cease the flow.
Select Rate Type	
Seed	Variance of particle at creation. Tip: If you use more than one Emit Fluids from Volume node, each one has to have it own "seed" parameter.
Initial Values Group	all parameters on this section are just used once the particle is created. If you want to change some parameter interactive throughout the simulation you must set it after the Emit Fluids node. (for example you can plug a "Set Particle size" node directly in the ICEtree)
Color	Initial color of the particles
Mass	the initial mass of the particle
Size	the initial size of the particle
Shape	The initial shape of the particle. Blobs must be set in order to get the particles to produce a continuous surface at rendertime.
Orientation	the initial orientation of the particles
State ID	the state the particles are born
Direction and Speed group	
Fast Moving Emitter	turn this option on will "jitter" the position of the particles in space at emission. This would help smooth out the placement of particles at birth in case the emitter is making too large steps on each frame.
Execute on Emit	Set data per particle at creation

SPH Fluid types nodes

All nodes on the "Fluid Type" section are identical. They work as presets, setting data to the SPH simulator of different types of liquids.

They work as a start point to the desired fluid effect.

Smooth Length	Search Radius
Surface Tension	Rest Density
Stiffness	The stiffness of the liquid. It's basically how much force the pressure vectors are going to make when the particle get too close to a neighboring particle. Stiffness works together with Surface tension, if you have high stiffness liquid (like water), the surface tension needs to be decreased in order to maintain the stability of the flow.
Stiffness Near	Same as stiffness but used on near particles. This parameter is useful to make the particles spread apart a little more without influence their surface tension.
Enable Viscosity	Turn on/off the viscosity of the fluid. Set the amount of viscosity for the fluid. Water would have between 1 and 20. Oil could be modeled with values of 50 to 100.
Viscosity Multiplier	A very viscous material could have a value of 250 or more. If you increase the viscosity to something more than

SPH Soft Contact force

Geometry (port)	Port to Plug the contact geometry
Attraction Radius	The radius of attraction of the contact Geometry
Multiplier	The force multiplier
Rest distance	The rest distance, particles will try to reach this distance when they come inside of the attraction radius
Distance to Surface	A multiplier of the distance from the surface to the until the rest density diverges
Display Force	Option to display the force vectors, so you can see how they work with the contact surfaces

Simulate SPH node

Simulation group	
Substeps	If you notice instability in the flow, increase substeps. For example, 2 substeps is the same as break one frame in two, therefore the solver can reach greater stability because it will actually process the same frame two times, doing these smaller steps that are less than a whole frame will improve stability and also precision of the flow as a whole (pressure, collisions, viscosity...). Becareful, more substeps means more precision. Too many substeps means waste of processing power. Tips: Fluids pilling up like in a bowl or in a can will need more substeps to get stable. This is because the pressure of the particles that pile up will push the fluid down (with gravity that is), and eventually this gravity force can be greater than the pressure force, making particles too close of each other. In this situation the pressure of the fluid will increase in a fraction of time and overshoot the particle trying to decompress the flow. This will cause a chain reaction inside of the flow making it explode, pop, or even pass through the collision objects. The substeps have sort of a relation with the smooth length. If smooth lenght is too small the particles will get closer to each other creating smaller features. Therefore more substpes will be necessary to improve the stability of this close particles.
	Show substeps Display the substeps passed between one frame and the next.
	Collision Method Option to collide with front face or back face of the colliding geometry.
Collision objects	
Geometry (port)	The input collision geometries to be considered in the simulation. You can plug either a group of objects or individual objects
	Distance The distance field created around the object

Collision Side	to detect collision. If you notice objects passing through you can 1-increase this distance field (if the particles are moving at a normal speed) 2-increase substeps (if the particles are moving really fast) The side you want the Solver to consider upon a collision. Backface or Frontface. If you want a "container" you will want to select "backface"
Friction group	
Interaction range	The radius that friction will start to take place. A small value will just apply friction to objects that are in contact with the surface. (this is the natural behavior for most viscous fluids). If you use a high value the particles will "slow down" the upon a collision detection. Tip: this value needs to be equal or smaller than the Distance.
Enable Static Friction	Enable Static friction (off by default) The friction that takes place when objects are in contact with a surface, it's pretty much how much "force" it will take to make the particles move when in contact with a surface or inside of the interaction range.
Static Friction multiplier	
Enable Dynamic Friction	Enable Dynamic Friction (off by default) Dynamic friction occurs when two objects are moving relative to each other.
Dynamic Friction multiplier	The dynamic friction will appear only when objects are moving (both particle and interaction surface). A value of 0 to 1 is recommended but you can extrapolate to get different effects. For example if you want the fluid to accelerate in the direction the collision object is moving.
Enable No Slip	Enable the No Slip condition
No Slip condition	The No Slip condition states that a fluid in contact with a surface would have zero velocity relative to that surface. A value of 1 will try to stick the fluid on the

	surface once the surface is moving. Let's say you drop blood in a table, if you move the table you want the blood to stay in the table, the NoSlip set to 1 would make the blood stick to that surface. (You can combine this with friction) A value between 0 and 1 is recommended but you can extrapolate to get different effects.
Soft Repulsion group	Soft Repulsion is a force applied when the fluid get's inside the distance field of collision objects. Soft repulsion is useful if you want to make the fluid to be "repealed" from a surface, if you want it to "stick" to a surface (making the opposite force), or even if you want the fluid to "float" from a surface. The force will resize (based on rest distance and Distance to Surface parameters) to the distance the particles have to reach the surface of the collision surface. Soft repulsion also solves the problem of particles laying out flat on the collision surface (this is a problem on ALL SPH solvers), applying a bit of force between the interacting surface and the particles.
Enable Soft Repulsion	Enable Soft Repulsion
Soft Repulsion multiplier	Multiplier of the soft repulsion force
Rest Distance	The Rest Distance particles will try to reach from the surface.
Distance to Surface	The distance particles will stay once they reaches Rest Distance.
Display Soft Repulsion	Display the Soft Repulsion force vectors

Tips and tricks

Rendering

Particle blobs with different color require the blob's algorithm to produce the color blending between the particles, this can take a long time to render if the number of particles is really high. So use it carefully. 

