Using Blender 2.7 for Animation - Part 4 - Booleans & Fluid Simulation: Fluids

Each fluid simulation usually has 3 parts: **Domain**, **Obstacle**, and **Flow**. You make an object into one of these by selecting the object, clicking Physics  button tab, fluid, then choose Domain, Obstacle, or Flow.

The domain is just a large cube that is around your whole simulation, and there can only be one. You could think of it as the walls to the room. Use Wireframe view so that you can see into it (hit ‘z’ on the keyboard).

* Select that cube. Make sure it is big enough so fluid wouldn’t touch it (except sides of room) but not too big to make simulation lose detail.
* Make the cube transparent in the 3d view, by clicking object button, then change Maximum Draw to Wire
* Click Physics button, Fluid, Domain. Your fluid domain has to be a cube, and even if you stretch it into a box shape, it will act like a cube. It is the boundary for the whole simulation. Make it big enough to contain the fluid and the container (obstacle). Make the Resolution for Final and Preview each about 80
* If you are using a network drive, choose a local folder such as Documents, then in the second box name the cache 3da5fluid (make a different name for each scene you make to prevent blender from using the same simulation for multiple scenes)

(click the yellow folder button and choose a local folder such as my documents)

* File, Save As, on the left click your home folder such as H: (or T: then username), and in the second box name it 3da6fluid

Detailed Instructions (for Bonus, first look at requirements at the end of this part to see how to setup scene for that):

* Add Fluid physics using the Physics  button tab while an object is selected. Choose the fluid object type from the list by “Type:”. For a fluid simulation to work, you must have at least an **Inflow or Fluid**, and a **Domain.**
* **Obstacle** is the fluid object type that should be set for all objects that should block the fluid (such as your container if you made a container).
* If fluid goes through an object, fluid resolution may need to be higher. Making domain smaller also makes grid more dense making simulation more accurate. Another way to help fluid collision objects work is to remove negative scale, make sure there is no self-intersecting geometry, and go to Edit Mode, Mesh, Normals, Recalculate Outside (Ctrl N will only recalculate **N**ormals in **edit mode** otherwise blender will ask to load “**N**ew” scene!)
* The real-world size of the cube is in meters, and should be changed to whatever size
* It is a good idea to name your domain object so you can find it later.
* Before you press Bake, it is good to choose a **location** for your bake, (so that it won’t end up in your “Temp” folder, a temporary folder which is periodically **erased** by windows disk cleanup & other cleanup software)—if you name it, it will be in a folder with that name next to your blend file.

**Part 4 Requirements**

Booleans & Fluid Simulation

* Make a container using Booleans and make it a fluid that is an “Obstacle” (you can use other scene you made)
* Have to have at least one fluid object that is “Fluid” or “Inflow”
* To limit the animation length to the simulation length, multiply the number of seconds of the domain by 24 and **set your animation's end frame to the result**
* **Click Render button, for preset choose HD 720p, then below choose H.264, then under save it in**

**//3da6fluid video**

**BONUS: Have one container pour into another without spilling (not counting physics glitches as spilling)**