

Getting Started with Chaos Phoenix

This page contains information on getting started with Chaos Phoenix for 3ds Max.

Overview

This is an Entry Level tutorial which requires no previous knowledge of Phoenix. A basic understanding of 3ds Max would be helpful but is not a prerequisite for being able to follow along.

You can download official Phoenix and V-Ray from <https://download.chaos.com>.

To create fire and liquid effects, Phoenix creates objects and helpers in the scene. Phoenix provides several **Quick Simulation presets** that create these scene elements for you, with settings for common scenarios. You can also create these objects and helpers manually.

If you are just getting started with Phoenix, it is recommended that you use the Quick Simulation presets and become familiar with the scene elements they create. As your experience with Phoenix increases, you will be in a better position to create these scene elements manually and set parameters appropriately.

Preparing a Simulation

To create a simulation, you'll need an object or objects to emit the fire or liquid, called **emitters**.

Emitters can be ordinary 3ds Max objects such as a Box, Cylinder, Editable Poly, or Editable Mesh. It is not necessary to collapse the objects to an Editable Mesh or Editable Poly. Object(s) with modifiers on the Modifier Stack will still emit correctly.

The following example shows some log objects ready for burning. For the fire setup, the logs themselves will be the emitters.



For a liquid simulation of pouring liquid, you can also include a vessel to catch or deflect the liquid. For example, for a simulation of beer pouring into a glass, you'll need an object to represent the beer glass that will deflect and contain the falling liquid.

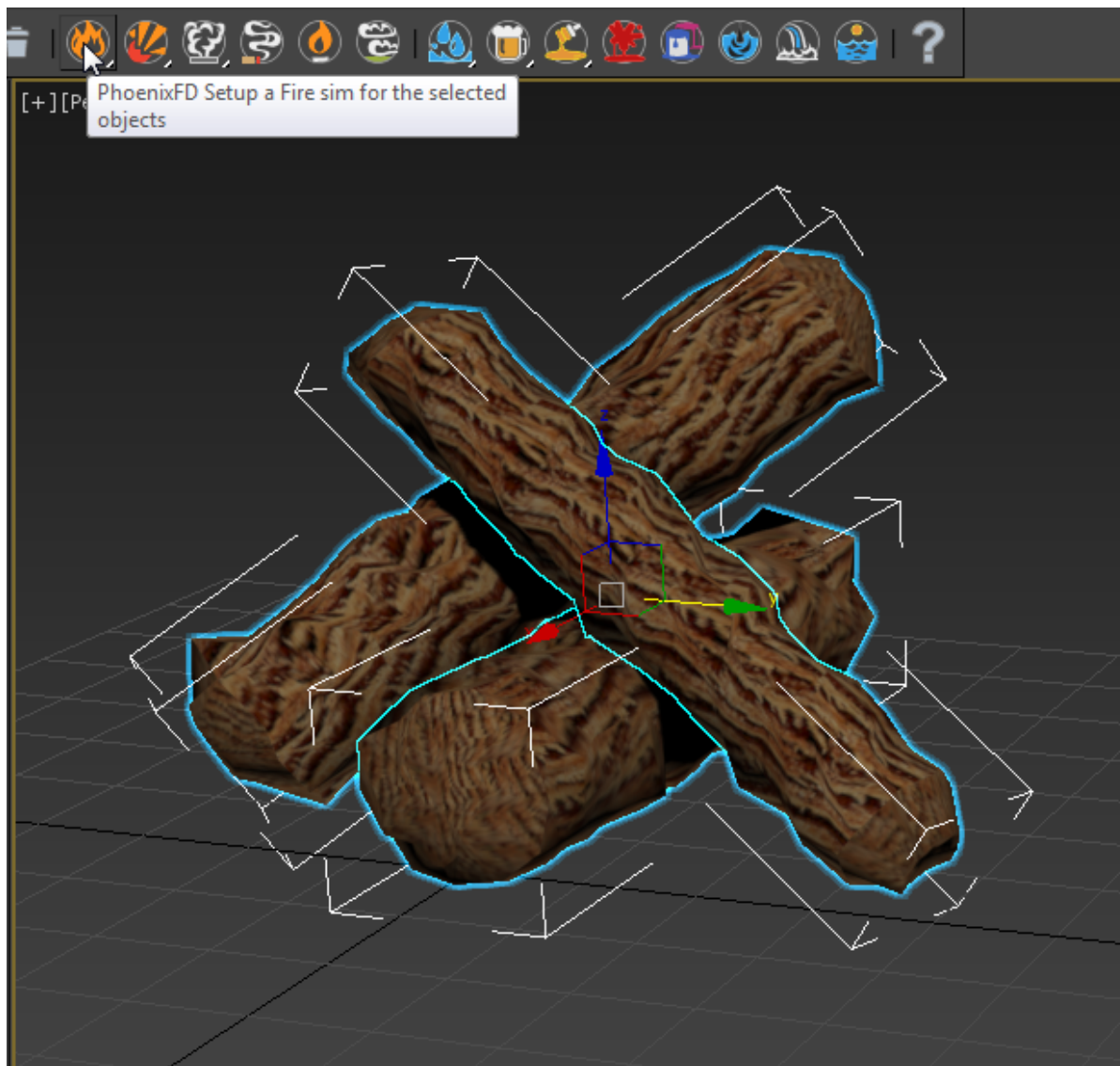
The following example shows a simple setup for water coming out of a spigot and into a trough. For the water setup, a small planar object inside the spigot (not visible in the image) will emit the water, which will fall into the trough. The trough is slightly angled away from the spigot end so the water will flow away from it.



When possible, build emitters and vessels to real-world scale. Since Phoenix uses real-world calculations to create its effects, working with real-world units will make your work much easier.

Fire Simulation

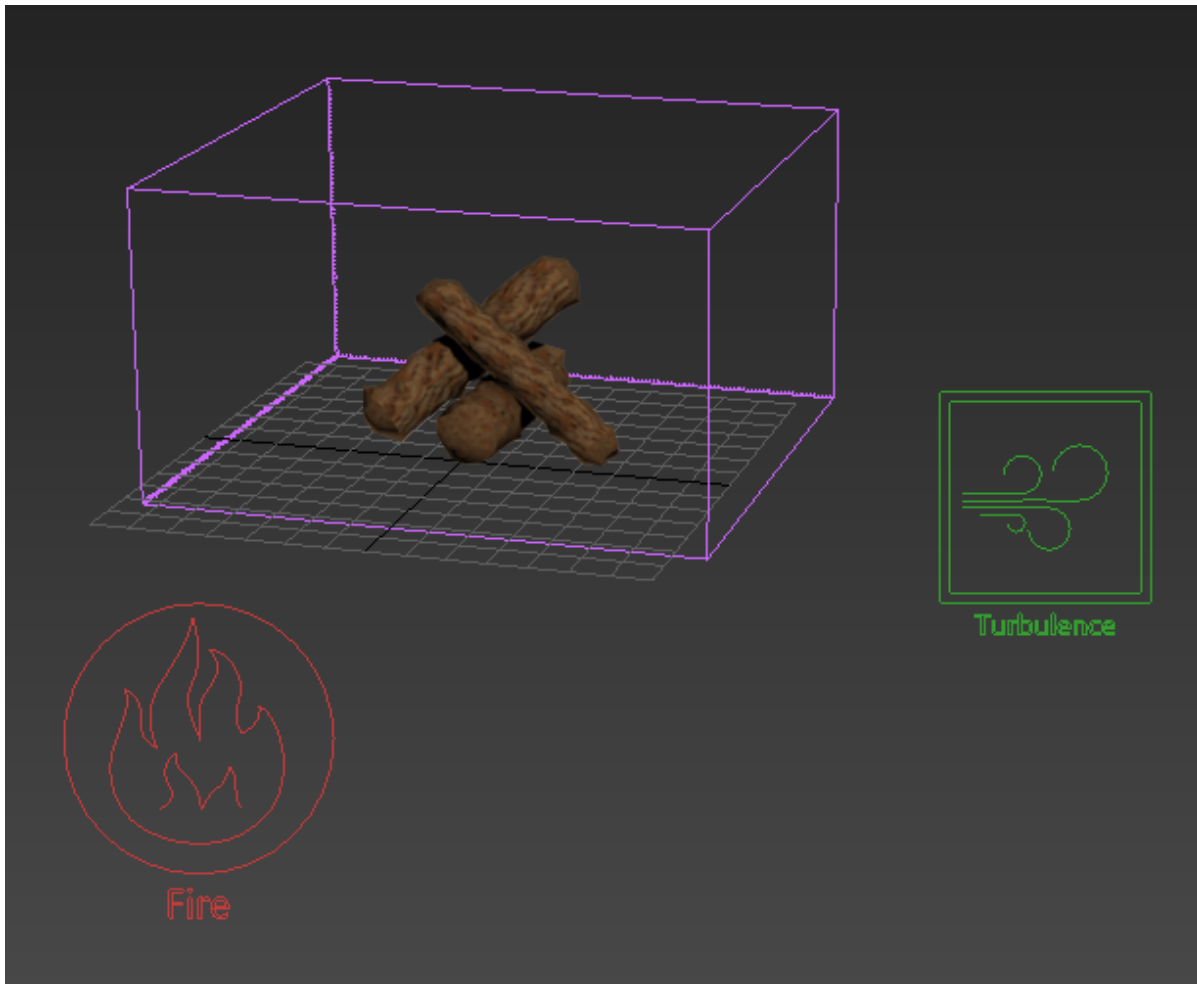
To create a fire simulation, select the object(s) you wish to burn and click the Fire Quick Simulation button on the [Phoenix toolbar](#).



The Fire Quick Simulation button creates three objects in the scene and sets up their parameters for a fire simulation:

- PhoenixFDFire object - A box-shaped object that represents the grid within which the simulation takes place. The grid shape encompasses the emitter objects and also leaves space above the objects for the fire to flow. This non-rendering object is required for the simulation.
- PHXSource helper - A helper (labeled **Fire** in the viewport) that associates the emitter(s) with the simulation. This helper is required for the simulation.
- PHXTurbulence helper - A helper (labeled **Turbulence** in the viewport) that creates turbulence in the simulation. This helper is optional, and is added to the Quick Simulation setup to improve the results of the simulation.

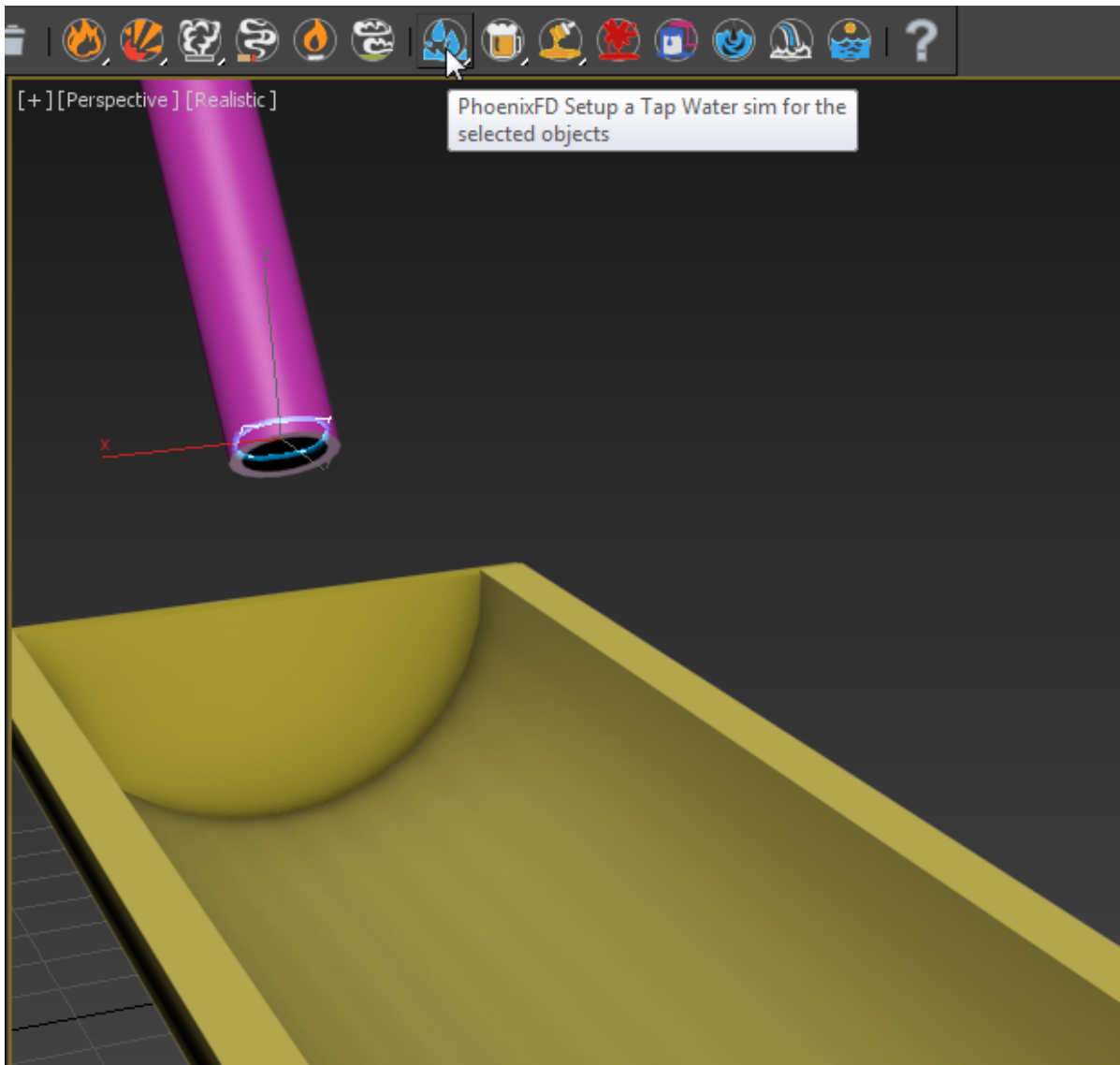
With all the scene elements in place, you can now run the simulation. See [Running the Simulation](#) below for the next steps.



Liquid Simulation

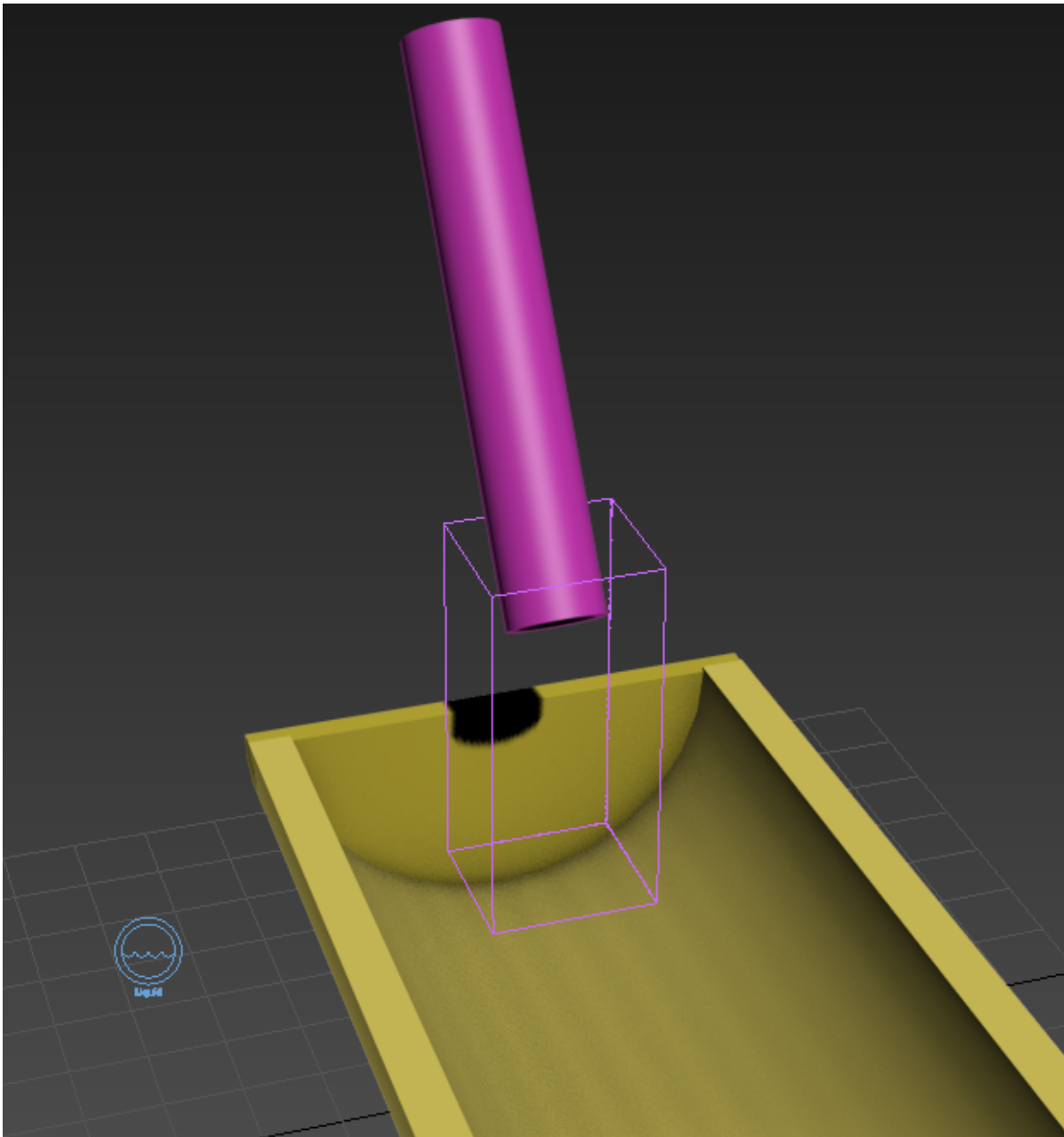
To create a liquid simulation, select the emitter object(s) and click the Tap Water Quick Simulation button on the Phoenix Toolbar.

Here, the small planar object inside the spigot is selected as the emitter.



The Liquid Quick Simulation button creates two objects in the scene and sets up their parameters for a liquid simulation:

- PhoenixFDLiquid object - A box-shaped object that represents the grid within which the simulation takes place. The grid shape encompasses the emitter objects and also leaves space below the objects for the liquid to flow. This non-rendering object is required for the simulation.
- PHXSource helper - A helper (labeled **Liquid** in the viewport) that associates the emitter(s) with the simulation. This helper is required for the simulation.



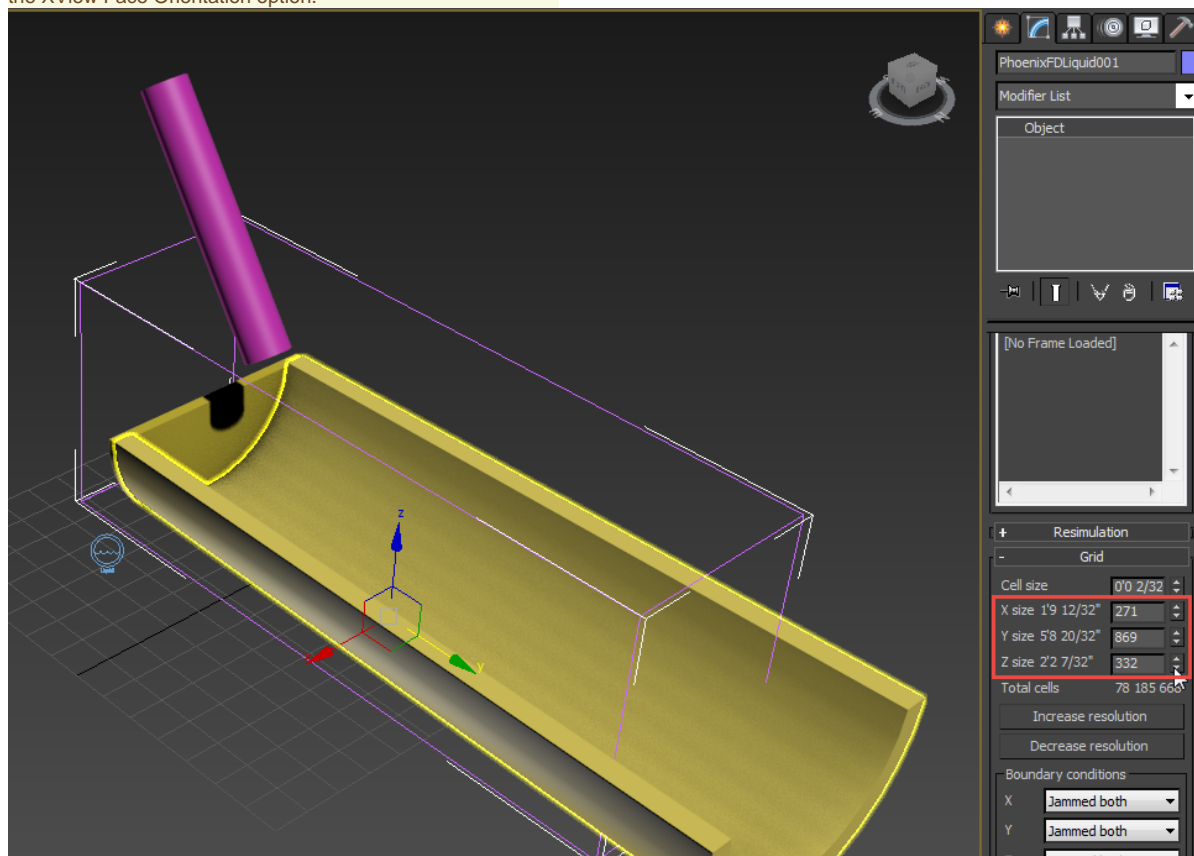
In this case, the Grid that was automatically created is not large enough to encompass a simulation of water flowing down the trough.

To increase the **Grid Size**, select the Simulator and change the **XYZ** size parameters in the **Grid** rollout.

For objects that will contain or deflect the liquid, there is no need to select them as deflectors or collision objects. Phoenix will automatically detect them when the simulation is run, and will cause the liquid to react appropriately when it hits them.



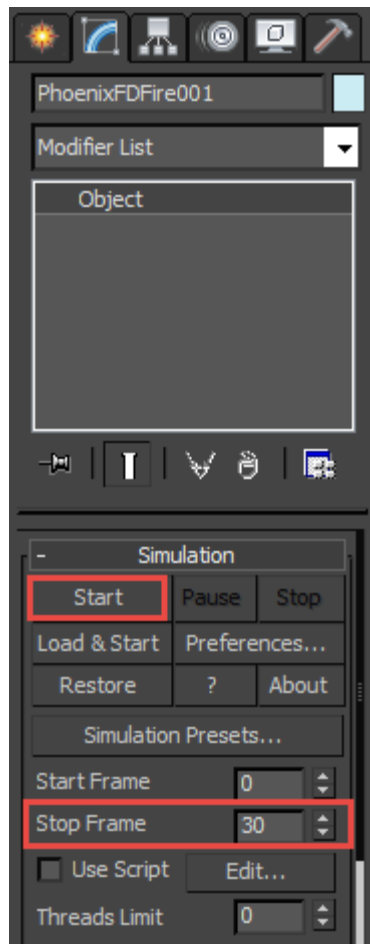
A liquid simulation uses geometry face normals to deflect and contain liquid, so any geometry that the liquid will hit needs to have its normals oriented to point away from the deflecting surface. With an object created with certain 3ds Max tools such as the Lathe modifier, it is not uncommon to find that the default normals point inside the vessel rather than outward from it. With such an object, the flow of the liquid won't be contained; the liquid will flow right through it. To check the normals, you can use the Edit Normals modifier or the XView Face Orientation option.



Running the Simulation

Once all the necessary scene elements are in place, you can run the simulation to see the fire or liquid effects in action.

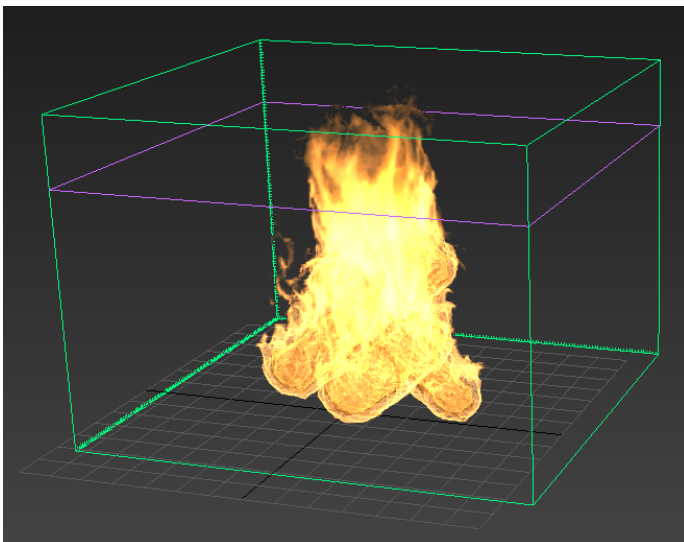
1. Select the PhoenixFDFire object (the box around the emitters).
2. On the Modify panel, locate the **Stop Frame** parameter in the **Simulation** rollout. Increase this value from 0 to a larger frame number such as 30.
3. In the **Simulation** rollout, click the **Start** button. The simulation will start running.

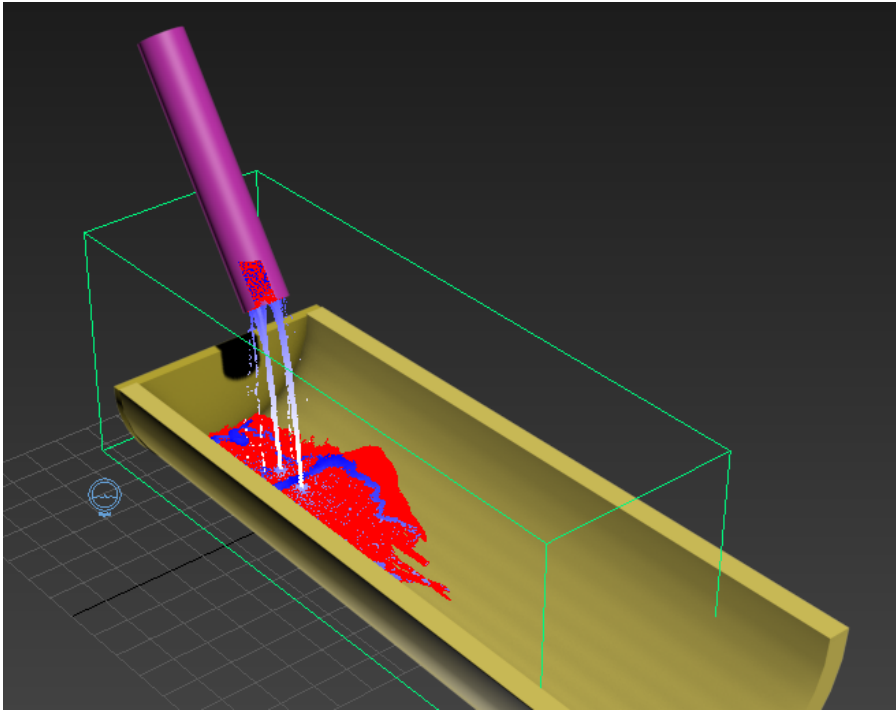


While running the simulation, you will see the fire or liquid flowing in the active viewport.

In addition, the Grid might expand beyond its original size to allow the effect to flow into a larger area.

You can also select other objects and continue working on the scene while the simulation runs.





In a liquid simulation, you will see the liquid particles react to gravity and also get deflected or contained by any scene geometry they encounter, such as the trough. This collision detection occurs by default, even though you have not specifically designated the trough geometry as a deflector or collision object.

If you wish to **Stop** the simulation before it reaches the Stop Frame, select the grid and click the **Stop button** in the **Simulation** rollout.

When the simulation is complete, render the scene as usual. The simulated effects will appear in the rendering.

