

# Force Field

This page provides some details on the settings available for the Force Field (PhoenixFDField) object.

## Overview

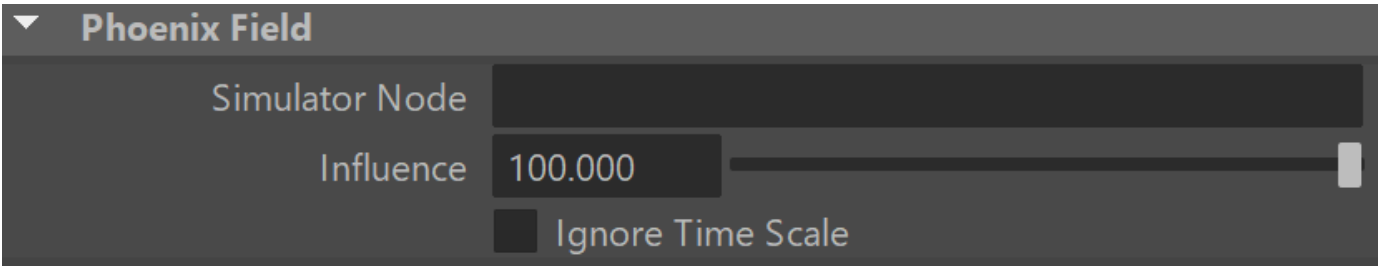
This field enables the involvement of **Maya particles** and other dynamic objects in the fluid's motion. It requires the simulation to be performed with **Velocity** output channel switched on. Note that you don't need this field for particles created by the simulator itself.

When doing Liquid simulations, Phoenix populates the grid **Velocity** channel only for cells / voxels inside the liquid volume - empty (air) voxels contain no velocity information. To generate velocity information for the empty voxels, enable **Simulate Air Effects** under the Dynamics rollout of the Liquid Simulator and re-run the simulation.

**See Also:** [Output Channels](#) | [Frame Index Transform](#)

||Phoenix FD menu|| > **Create > Force Field**

## Attributes



**Simulator Node** | *simNode* – The name of the Phoenix Simulator node as a string.

**Influence** | *influence* – Determines how strong the fluid holds the particle. 100% means that the particle will follow the fluid exactly. Influence less than 100% causes an approaching of the particle's speed to the fluid's speed. Influence of 10% means that at every integration step the velocity of the particle will overcome 10% of the difference between its velocity and fluid's velocity.

**Ignore play speed** | *ignoreTimeScale* – If checked, the **Play speed** attribute of the node and the frame duration during the simulation are not taken into consideration.