

Procedural Ground Dust

This page provides a tutorial on creating a Ground Dust simulation with Voxel Tuner in 3ds Max.

Overview

This is an Intermediate Level tutorial. Even though no previous knowledge of Phoenix is required to follow along, re-purposing the setup shown here to another shot may require a deeper understanding of the host platform's tools, and some modifications of the simulation settings.

In this tutorial we explore how to create dust-like simulation using Chaos Phoenix smoke. We go through setting the right simulation units, appropriate geometry dimensions and scene setup suggestions. Most importantly, we guide you through how to make a realistic sand smoke coming up as the horses step on the ground using Voxel Tuner. We compare how different values of some crucial parameters change the whole simulation's look.

This simulation requires **Phoenix 4.10 Official Release** and **V-Ray Next Official Release** for **3ds Max 2017** at least. You can download official Phoenix and V-Ray from <https://download.chaos.com>. If you notice a major difference between the results shown here and the behavior of your setup, please reach us using the [Support Form](#).

[Download Project Files](#)

Units Setup

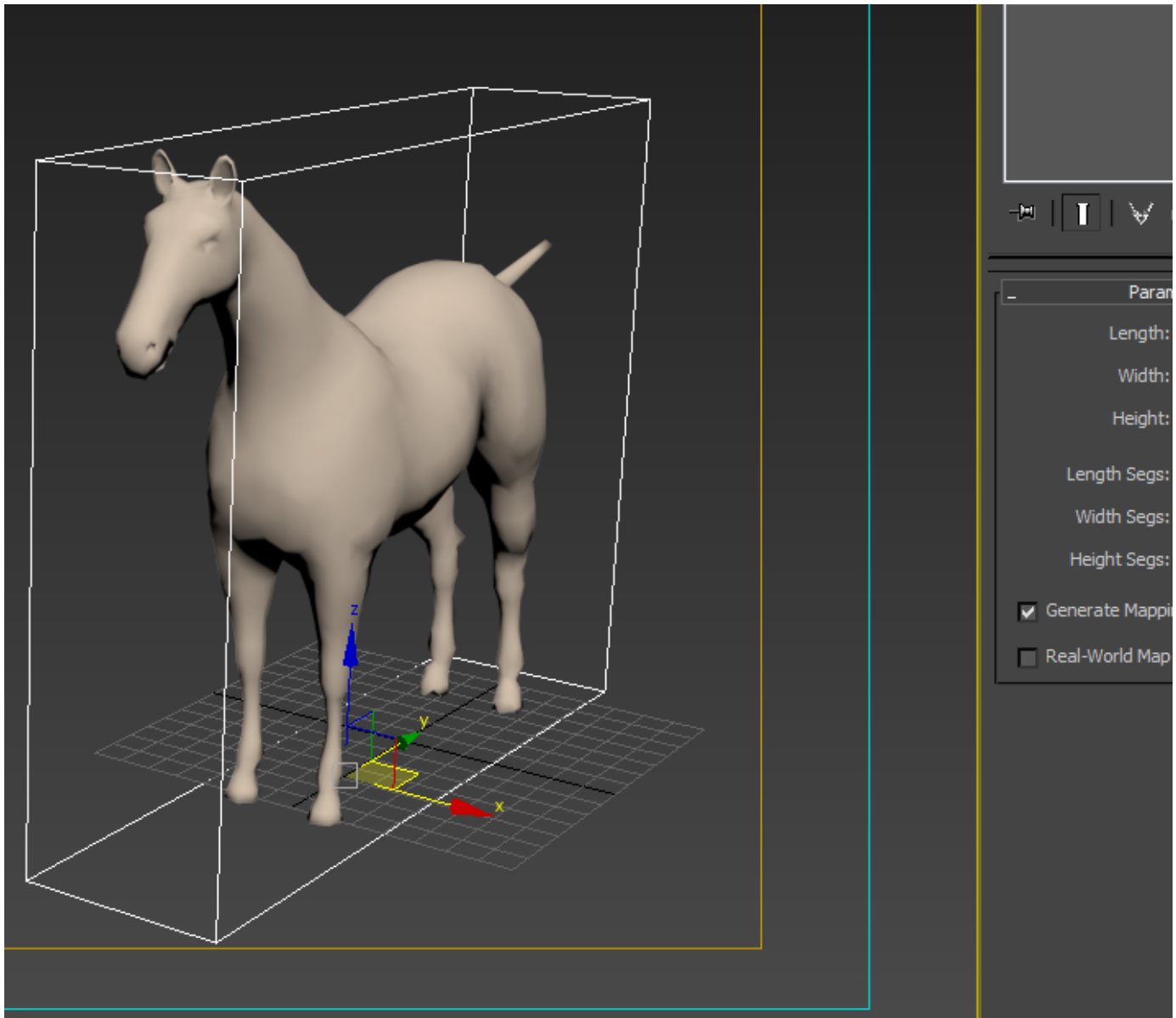
Scale is crucial for the behavior of any simulation. The real-world **size of the Simulator** in **units** is important for the simulation dynamics. Large-scale simulations appear to move more slowly, while mid-to-small scale simulations have lots of vigorous movement. When you create your Simulator, you must check the **Grid** rollout where the real-world extents of the Simulator are shown. If the size of the Simulator in the scene cannot be changed, you can cheat the solver into working as if the scale is larger or smaller by changing the **Scene Scale** option in the **Grid** rollout.

The Phoenix solver is not affected by how you choose to view the Display Unit Scale - it is just a matter of convenience.

The horse used in the simulation is modeled in centimeters and throughout the fluid simulation we keep using the same unit. If you want to simulate a massive scene, then you might consider using meters.

Go to **Customize Units Setup** and set Display Unit Scale to **Metric Centimeters**.

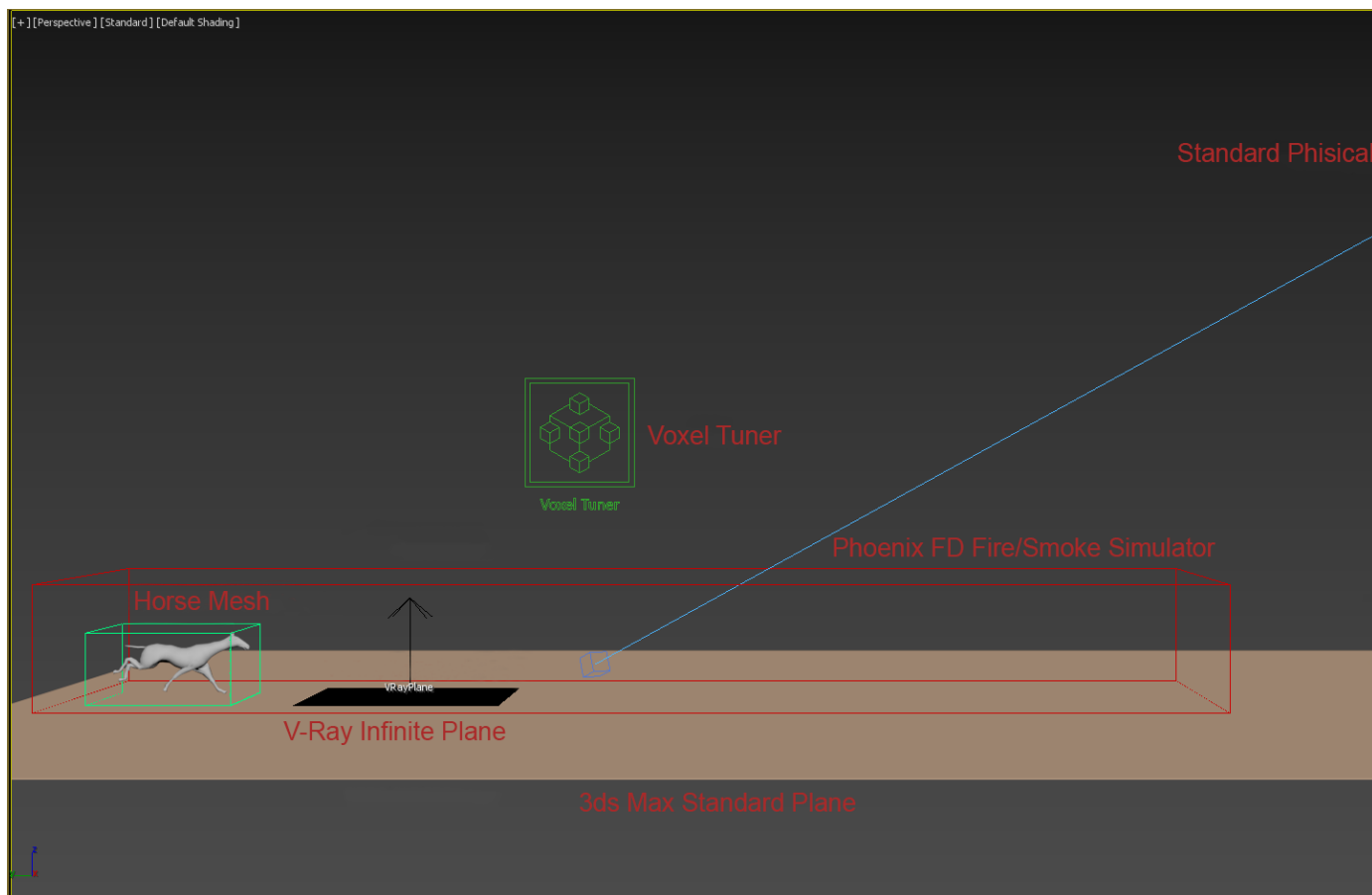
Also, set the **System Units** such that **1 Unit** equals **1 Centimeter**.



Scene Layout

The final scene consists of the following elements:

1. An **animated horse** exported to an alembic file provided with the **single_horse_01.ABC** file.
2. A dust emitting ground is set by a **3ds Max Standard Plane** with Shell modifier applied.
3. A **V-Ray Infinite Plane** is used for easily setting the ground.
4. A **Phoenix Fire/Smoke Simulator** with some tweaks in the Grid, Dynamics, and Rendering rollouts.
5. A **Voxel Tuner** is used for a realistic sand smoke coming up as the horse steps on the ground.
6. A **Standard Physical Camera** with minor tweaks for final rendering.
7. A **V-Ray Sun Light**.



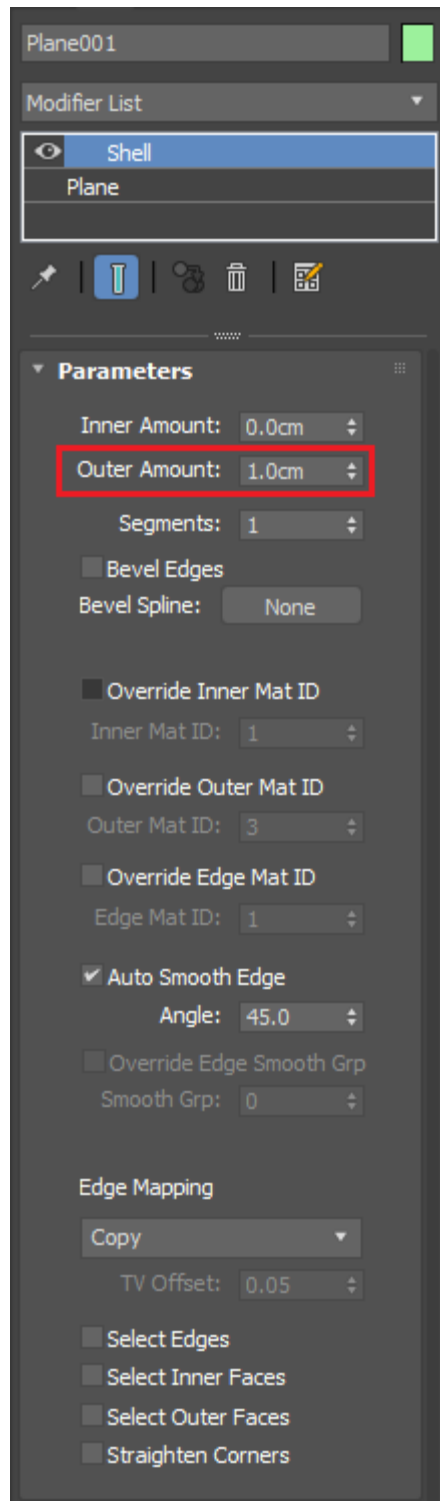
Scene Setup

Create a **3ds Max Standard Plane**. The exact dimensions of the plane are **6225** cm by **2793** cm.

Since Phoenix works best with closed geometry, add a **Shell** modifier to the created plane.

Set the **Outer Amount** to **1.0** cm. The actual value does not have weight in the final render, as the plane will not be visible at all there.

The exact **Position** of the **plane** in the scene is [**X:0cm**, **Y:-4320cm**, **Z:-20cm**].

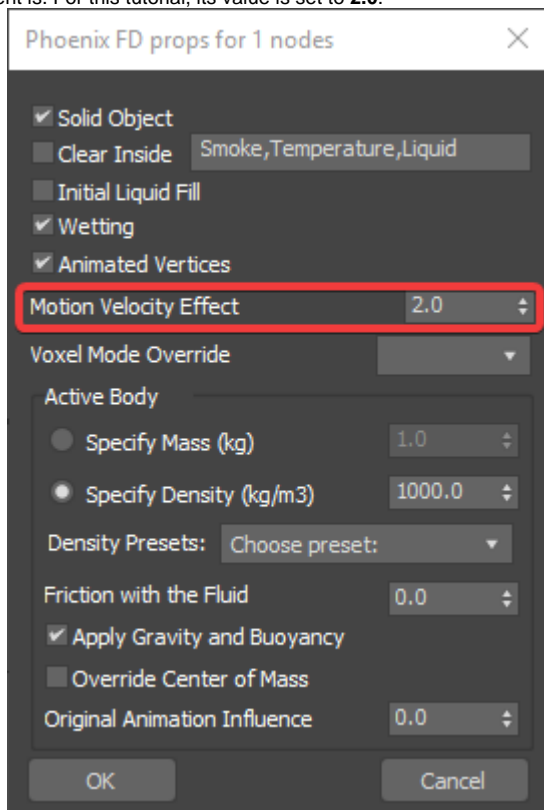


Import the **single_horse_01.ABC** geometry by going to **File Import Import...**

This tutorial starts off with pre-built geometry to save time setting up the scene. Feel free to use your own personal models.

The exact start **Position** of the **alembic mesh** in the scene is [X:0cm, Y:0cm, Z:0cm].

You can control the interaction strength between the horse mesh and the ground plane by adjusting the **Motion Velocity Effect**, located in the [Phoenix Per-Node Properties](#). The higher the value is, the stronger the fluid reaction to the body's movement is. For this tutorial, its value is set to **2.0**.

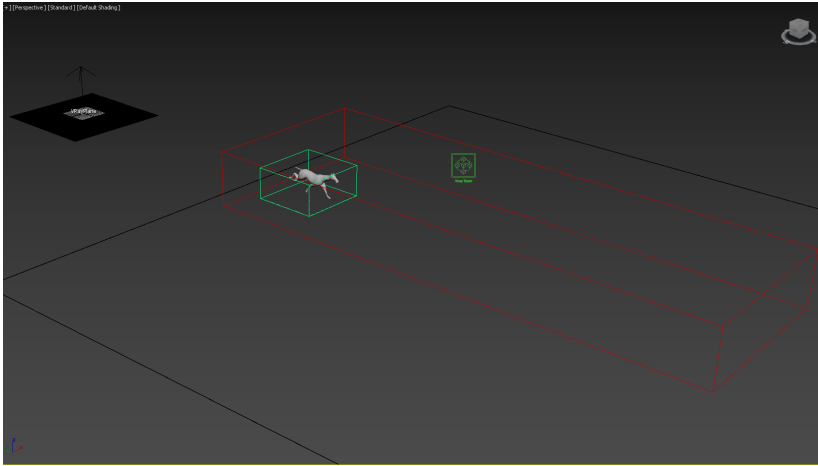


Create a **V-Ray Infinite Plane** used during rendering.

The exact **Position** of the **V-Ray plane** in the scene is [X:0cm, Y:0cm, Z:-15cm].

Create a [Fire Smoke Simulator](#).

The exact **Position** of the **simulator** in the scene is [X:0cm, Y:-1500cm, Z:-20cm].



Voxel Tuner

We are going to use a [Voxel Tuner](#) to emit Smoke where the horse model touches the ground. We will do it in such a way, that we don't even need a Phoenix Source.

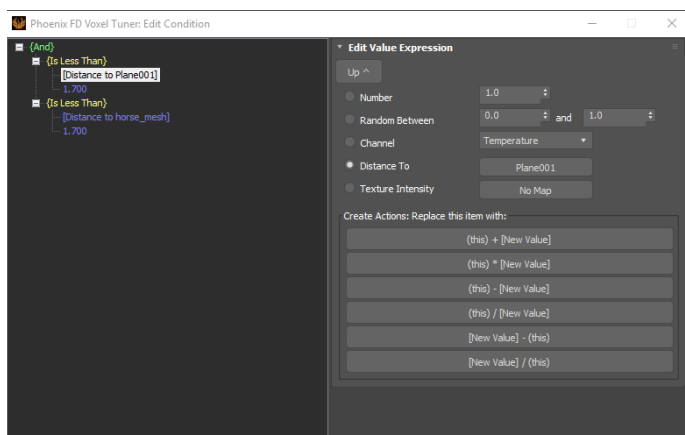
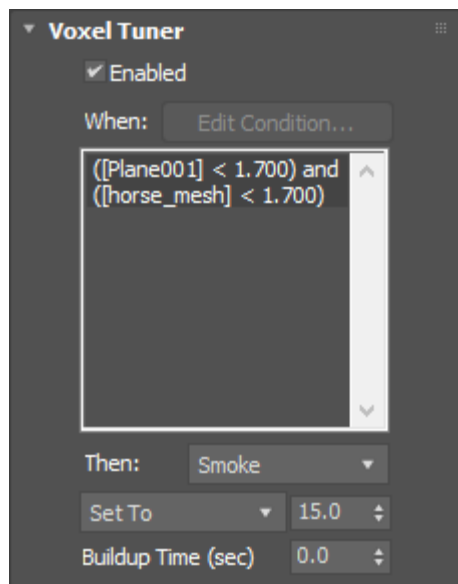
In this case, the Voxel Tuner will go through all the voxels of the Simulator, and check if each voxel is close to the ground plane and simultaneously close to the horse mesh. If it passes this condition, then the Tuner will create Smoke in this voxel.

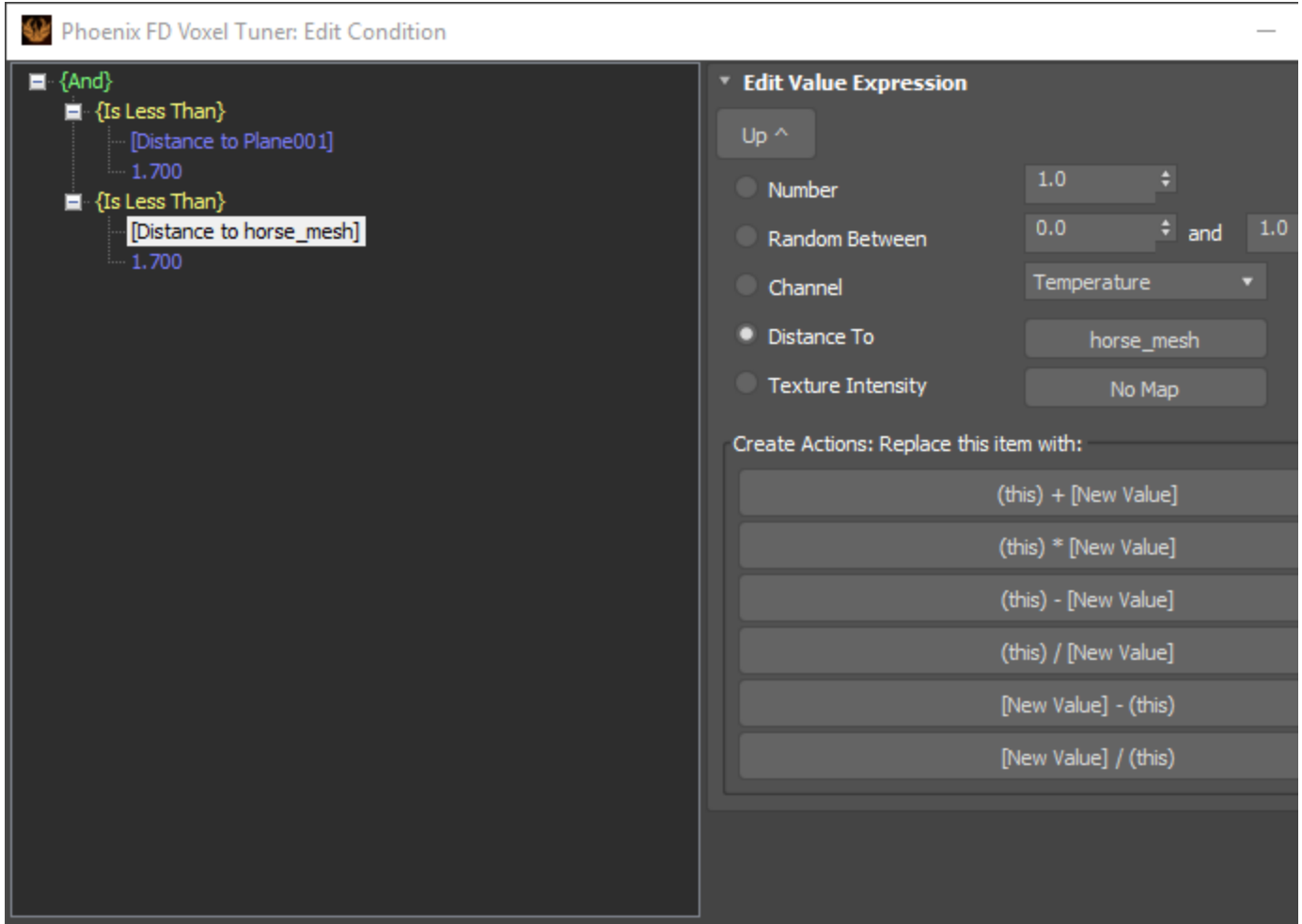
The **Voxel Tuner expression** is created as follows:

- If a voxel's **Distance to Plane001** is **Less Than 1.7** voxels and If the **Distance to horse mesh** is **Less Than 1.7** voxels.
- Then the **Smoke** will be **Set to 15.0**.

We picked very low values for both distance conditions - if you lower them even more, less voxels will pass the condition and Smoke will be emitted in fewer places. On the contrary, if you increase the distances, Smoke will start getting created in the air around the contact points between the horse and the ground.

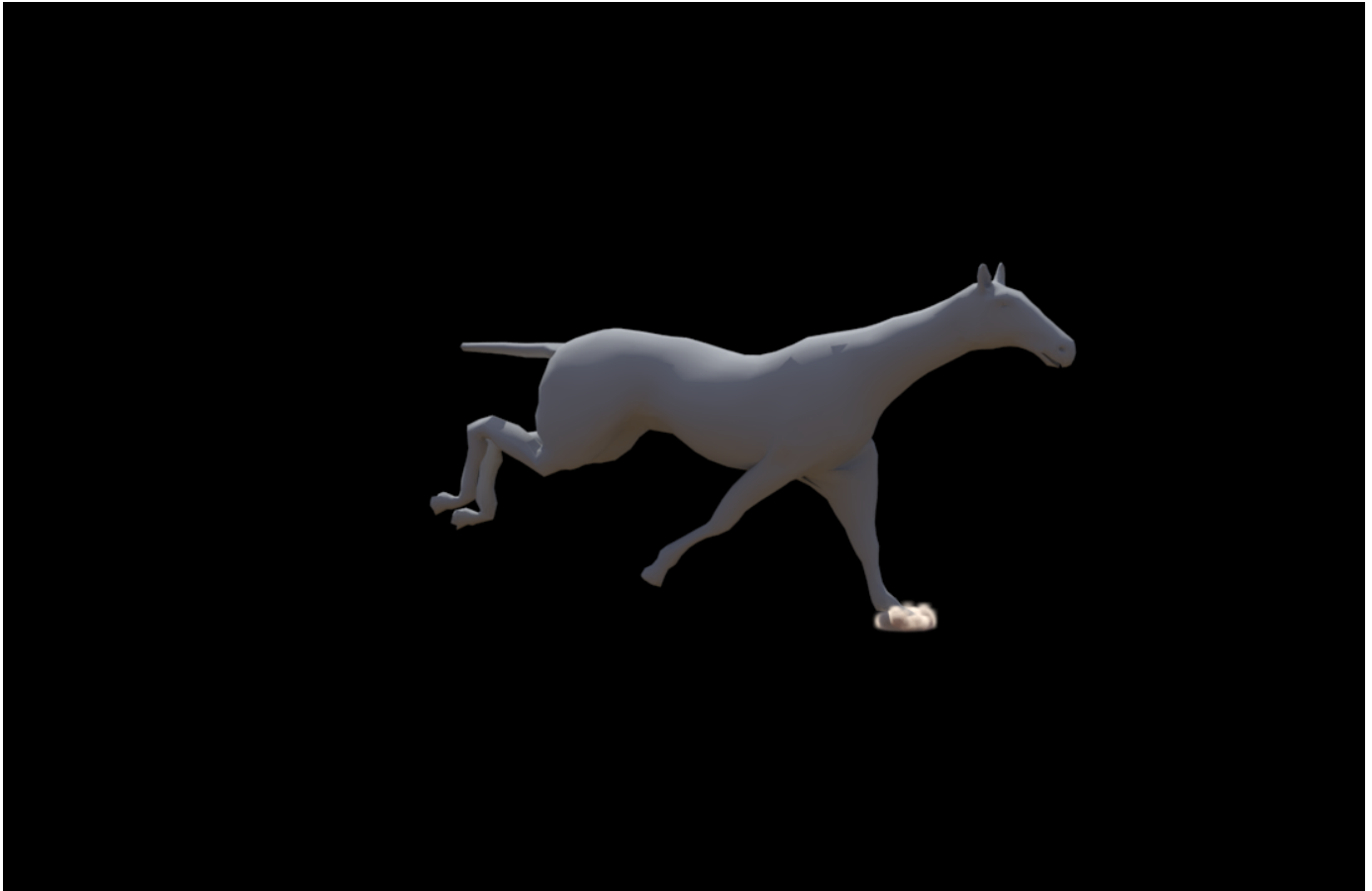
We also create very dense Smoke - usually we don't need smoke above 1.0, but in this case we emit 15 units of smoke, which would make the smoke render very dense with the default render settings. Below we will enable **Smoke Dissipation** so the smoke will be thick at first, but then will quickly fade out.





Run the simulation for a few frames and render out one frame with the running horse.

Notice how the **Voxel Tuner** generates smoke around the horse's hoof when it touches the ground.



Simulation

The **Simulator** **Grid** parameters are tweaked as follows:

The **Scene Scale** is set to **5.0**.

The **Cell Size** is set to **2.0cm**.

The **Size** of the Simulator is set to [**X:150cm, Y:140cm, Z: 70cm**] as a starting point.

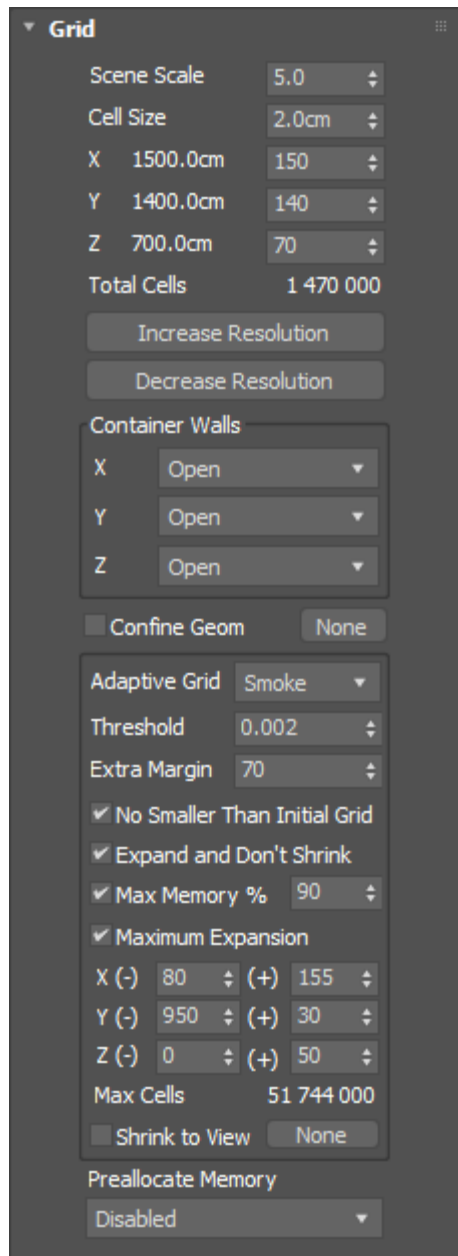
The **Adaptive Grid** is set to **Smoke**. The **Adaptive Grid** allows the container to resize on-demand as the simulation progresses, saving you a lot of time. Instead of calculating a giant grid from the start, the specified channel is tracked (Smoke in this case) and the grid is automatically resized around it.

The **Threshold** is set to **0.002**. The grid will expand when the content of a cell near one of the container walls crosses this value. In this case any Smoke value above 0.002 will cause the grid to expand.

The **Extra Margin** is set to **70**. This is important, because if left at 0, the Adaptive Grid will not expand fast enough and the horse will be running without emitting dust.

The **Expand and Don't Shrink** is **enabled**.

The **Maximum Expansion** is **enabled**. The limitations are set to [**-X/+X: 80/155, -Y/+Y: 950/30, -Z/+Z: 0/50**].



The three images here compare the differences between **Scene Scale** values [set from top to bottom to 1, 5 and 10].

The higher the value of the **Scene Scale**, the slower the movement of the fluid would be. This will keep the smoke close to the ground.

The goal here is to make the smoke look like sand, so there is no need to be too diffused or raising high from the ground. At this tutorial the **Scene Scale** is set to **5**, but feel free to experiment with other values as well.



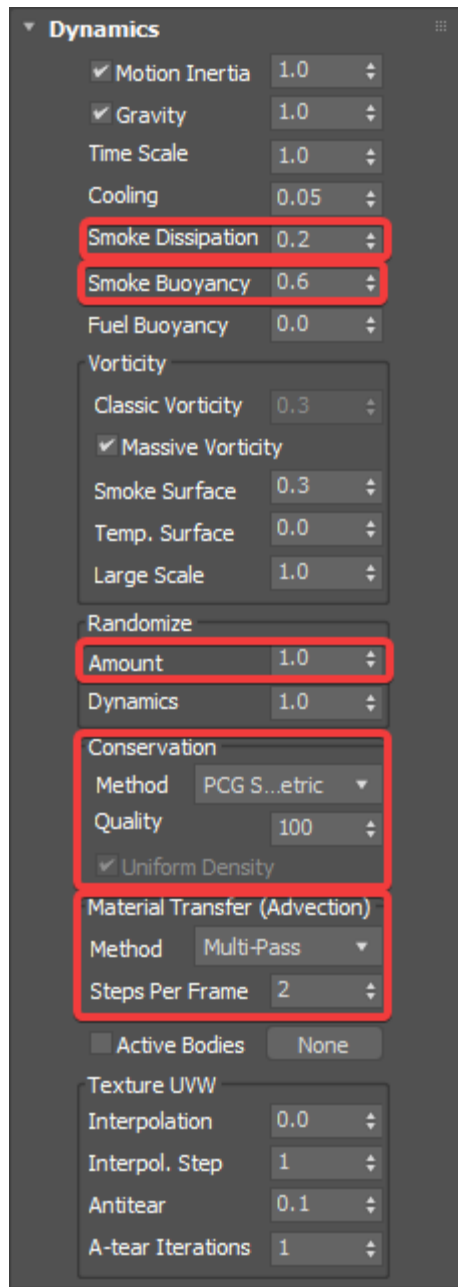
In the [Dynamics](#) rollout menu, switch the **Smoke Dissipation** to **0.2** and the **Smoke Buoyancy** to **0.6**.

The **Randomize Amount** is given a value of **1.0**.

The **Conservation Method** is set to **PCG Symmetric**, with a **Quality** of **100**. The PCG Symmetric method, in general, produces the most interesting smoke simulations, preserving both detail and symmetry. The high Conservation Quality will allow the dust to produce swirling motion. [For in-depth information, please check the Conservation documentation.](#)

The **Material Transfer (Advection) Method** is set to **Multi-Pass**.

The **Steps per frame** are set to **2**.



Compare the difference in the **Smoke Buoyancy** values displayed in these three images.

The values are set as follows from top to bottom to [0, 0.6 and 1].

The **Smoke Buoyancy** could be even set to a negative value so the dust would subside with time.



The **Smoke Dissipation** parameter controls how fast the smoke will disappear. The maximum value of 1 will make the smoke disappear immediately after it was emitted.

The three images here compare the differences between **Smoke Dissipation** values [set from top to bottom to 0, 0.2 and 0.7].

You might notice that with values of 0.2 and 0.7 there is a lot less detail in the smoke simulation because as it becomes more transparent, sharp edges and sharp shadows are diminished.

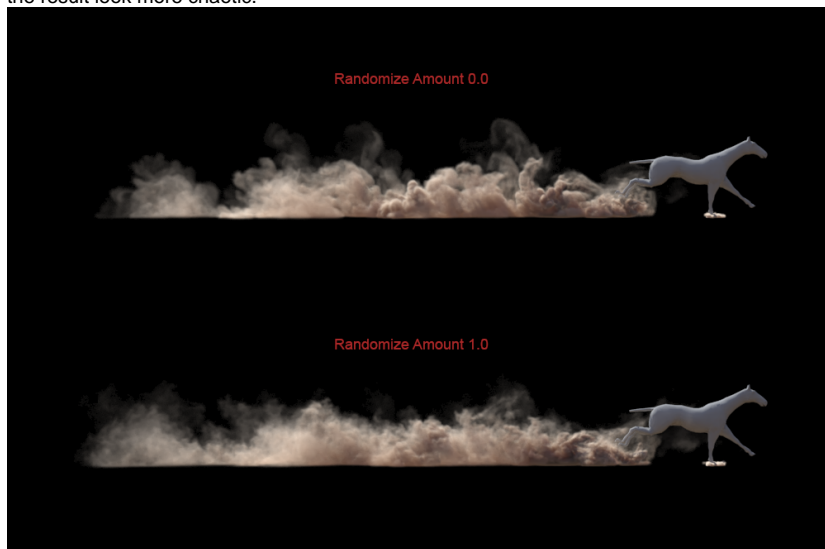


The **Randomize** parameter is responsible for the random fluctuations of the fluid's velocity for each grid voxel.

Top image: value of 0 (default value).

Bottom image: value of 1.

The value of 1.0 is chosen for this parameter in order to break the natural fluid mushroom shapes and puffs and make the result look more chaotic.



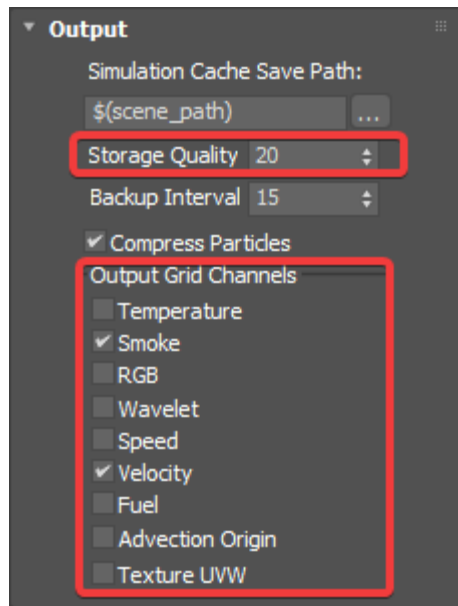
Open the [Output](#) rollout menu, which holds the controls for the simulation result.

The **Storage Quality** is set to **20** to prevent artifacts in the simulation due to the cache compression.

Since the Temperature is not used in the scene, the **Temperature** Output Grid Channel is **disabled**.

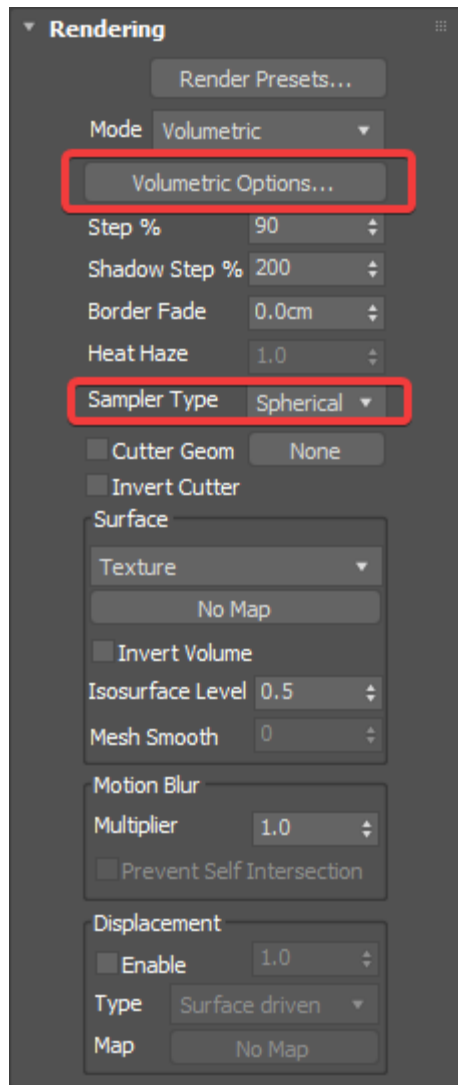
The **Smoke** Output Channel is **enabled**.

Enable the **Velocity** Channel, if you'd like to render the simulation with **Motion Blur** or start a [Resimulation](#).



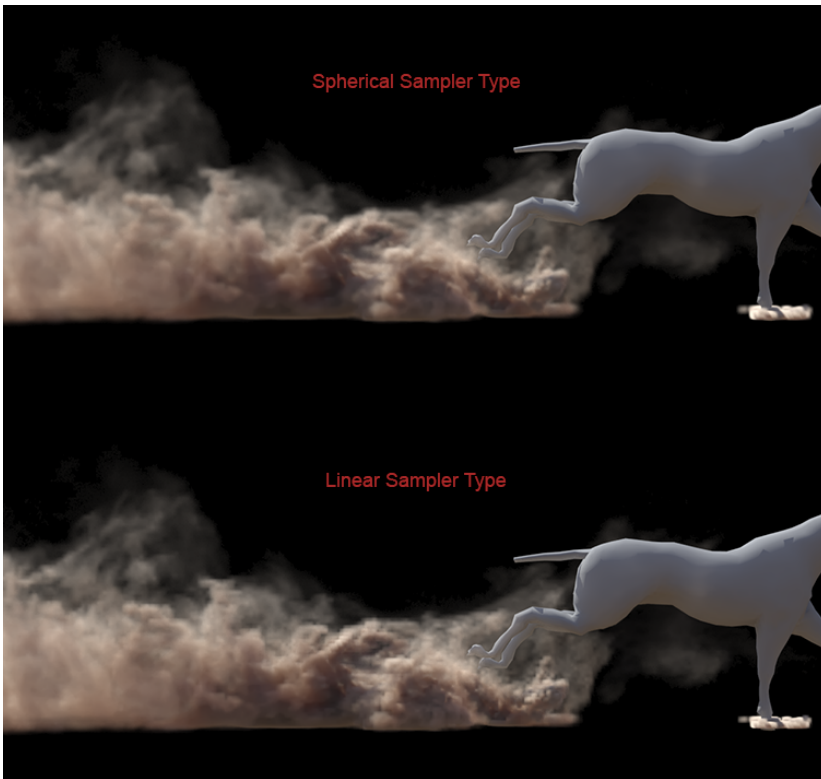
Open the [Rendering](#) rollout menu to prepare the simulation for rendering.

The **Sampler type** parameter determines the blending method between adjacent grid cells. The sampler is set to **Spherical**.



In those two images are compared two different **Sampler types** - **Spherical** (top) and **Linear** (bottom).

With **Linear** Sampler Type the produced dust looks grainy, so for this tutorial the **Spherical** sampling is used instead. That effectively reduces this issue.



Open the **Volumetric Render Settings** found by pressing the **Volumetric Options** button in the **Rendering** rollout menu.

The **Volumetric Render Settings** are tweaked as follows:

The **Fire's Based on** is set to **Disabled**.

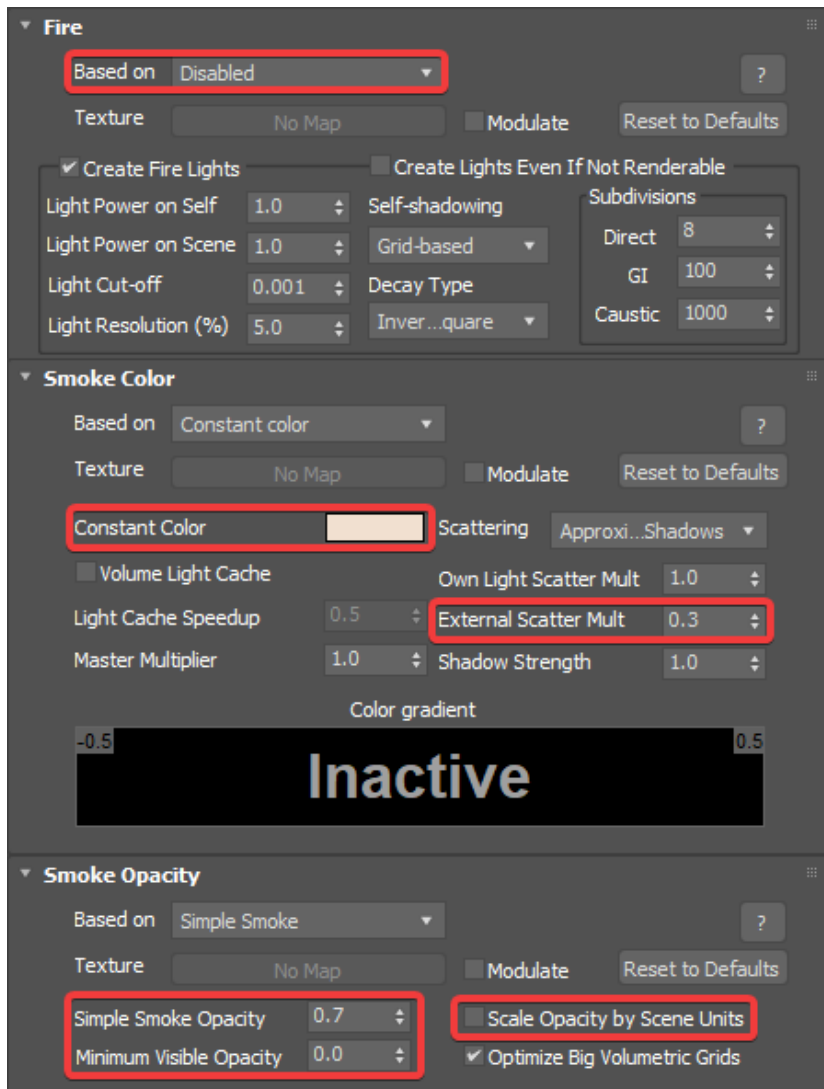
The **Smoke Color** is Based on **Constant color**. We aim at a sand-looking smoke, so we use **RGB [225, 191, 163]** for **Constant color**, but you can tweak it further.

The **External Scatter Mult.** is set to **0.3**.

The **Simple Smoke Opacity** is set to **0.7**.

The **Minimum Visible Opacity** is set to **0.0**.

The **Scale Opacity by Scene Units** is **disabled**.

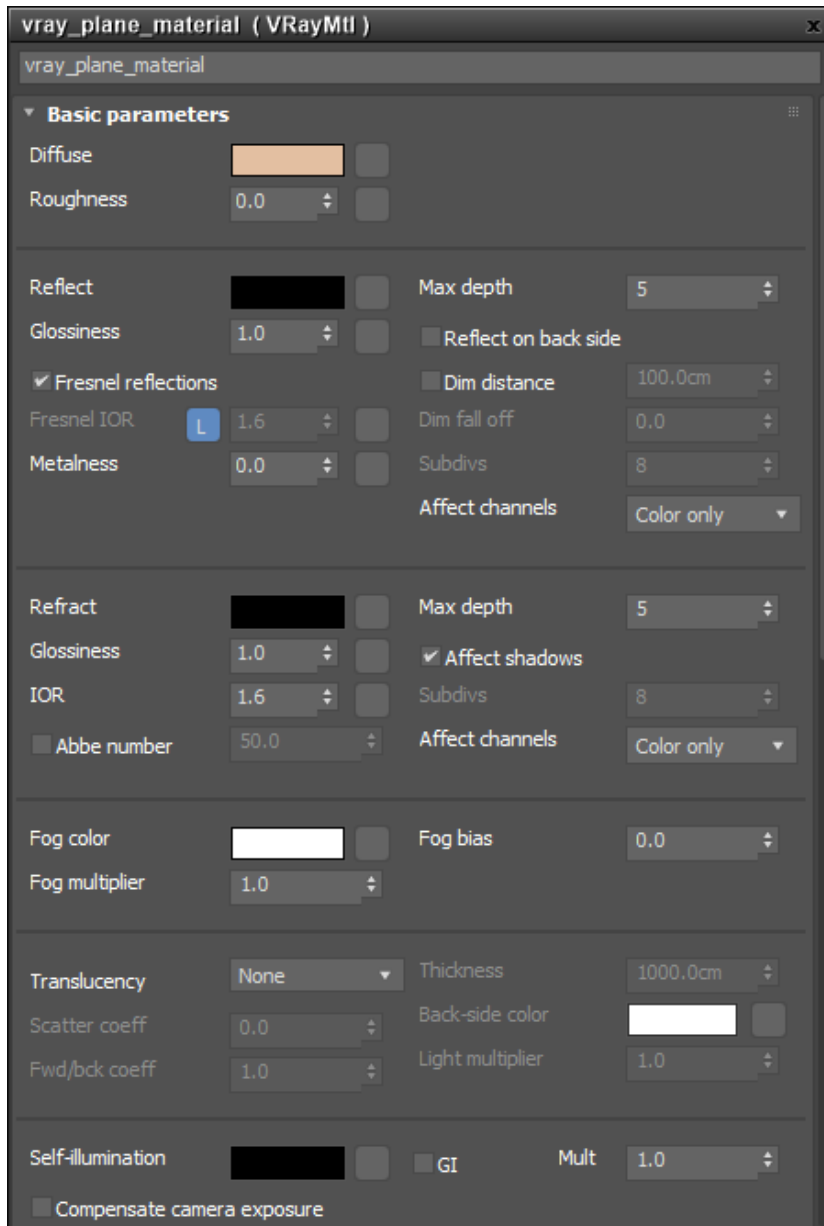


Materials

The main focus in this tutorial is on the simulation, so the materials used for the **Infinite Plane** (ground) and the **Horse mesh** are standard **V-Ray Materials**.

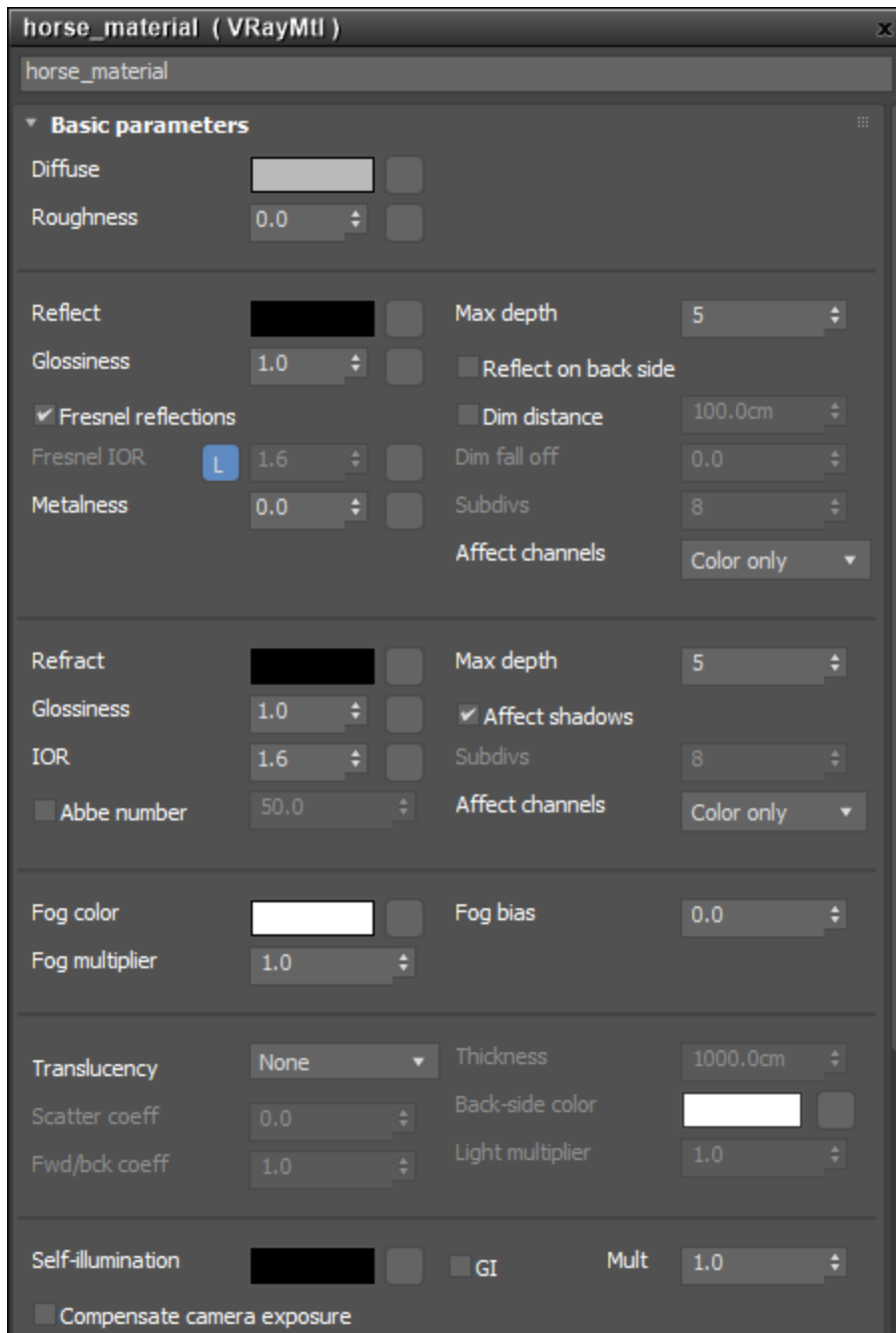
The **Diffuse** color of the "**vray_plane_material**" is **RGB [197, 135, 93]**.

All other parameters are left with their default values.



The **Diffuse** color of the "horse_material" is **RGB [128, 128, 128]**.

All other parameters are left with their default values.

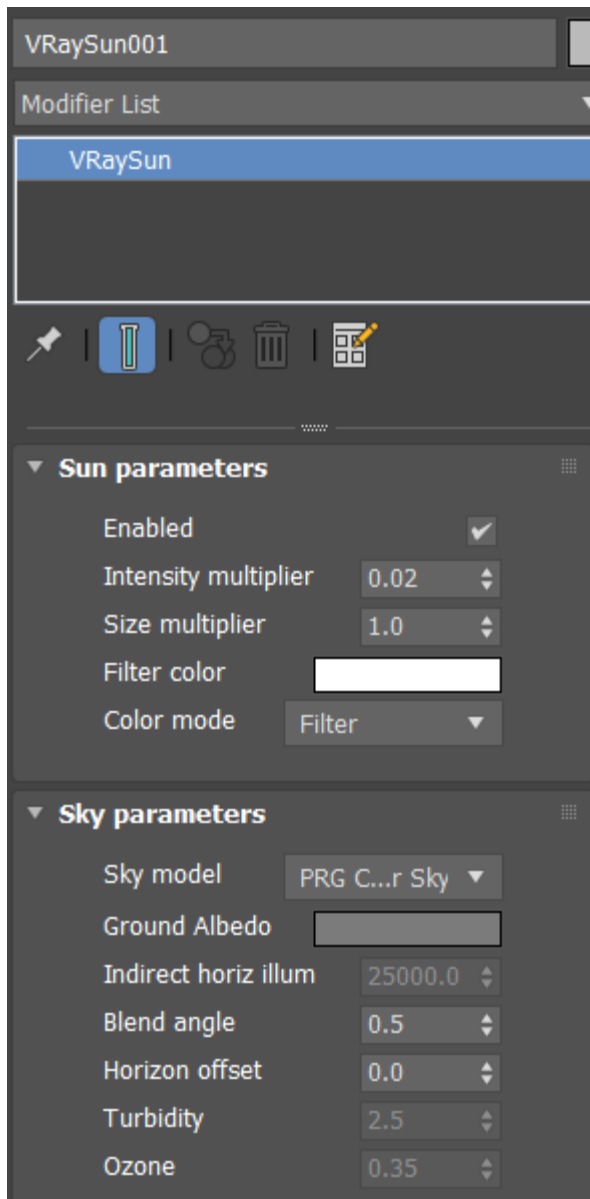


Lighting and Camera

The source of illumination in the scene is a single V-Ray Sun.

To add one, go to **Create Lights V-RaySun**.

Set the **Intensity multiplier** to **0.02**.



For this setup a Standard Physical Camera is used.

The **Film / Sensor Preset** is set to **35mm (Full Frame)**.

The **Film / Sensor Width** is set to **36.0** mm.

The **Focal Length** is set to **40.0**.

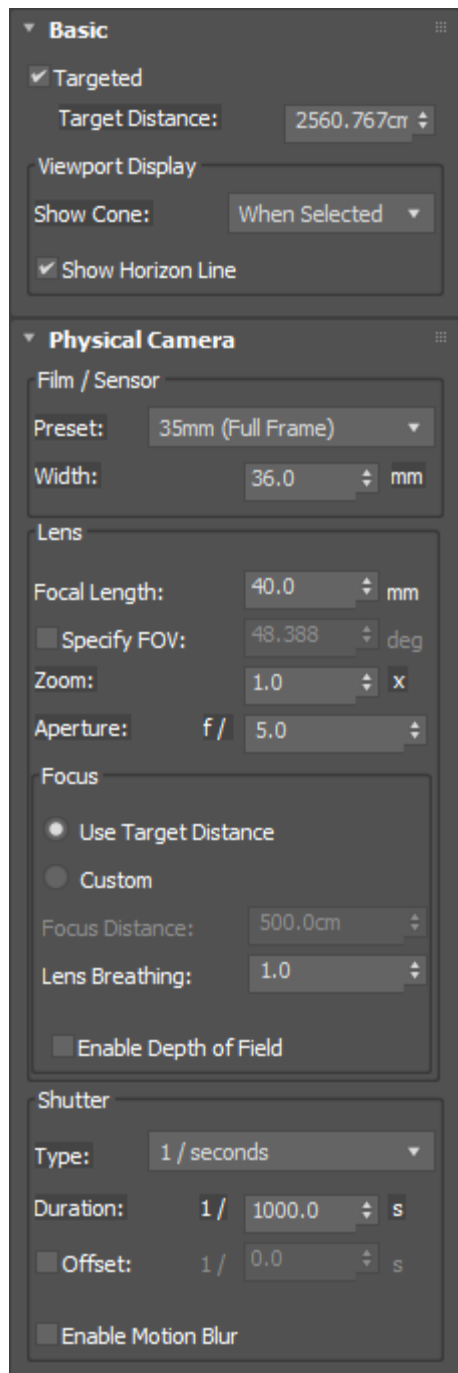
The **Aperture** is set to **5.0**.

The **Film Speed (ISO)** is set to **100**.

The **Shutter Duration** is set to **1000.0**.

The exact position of the **Camera** is [**-1730cm, -3785cm, 907cm**]

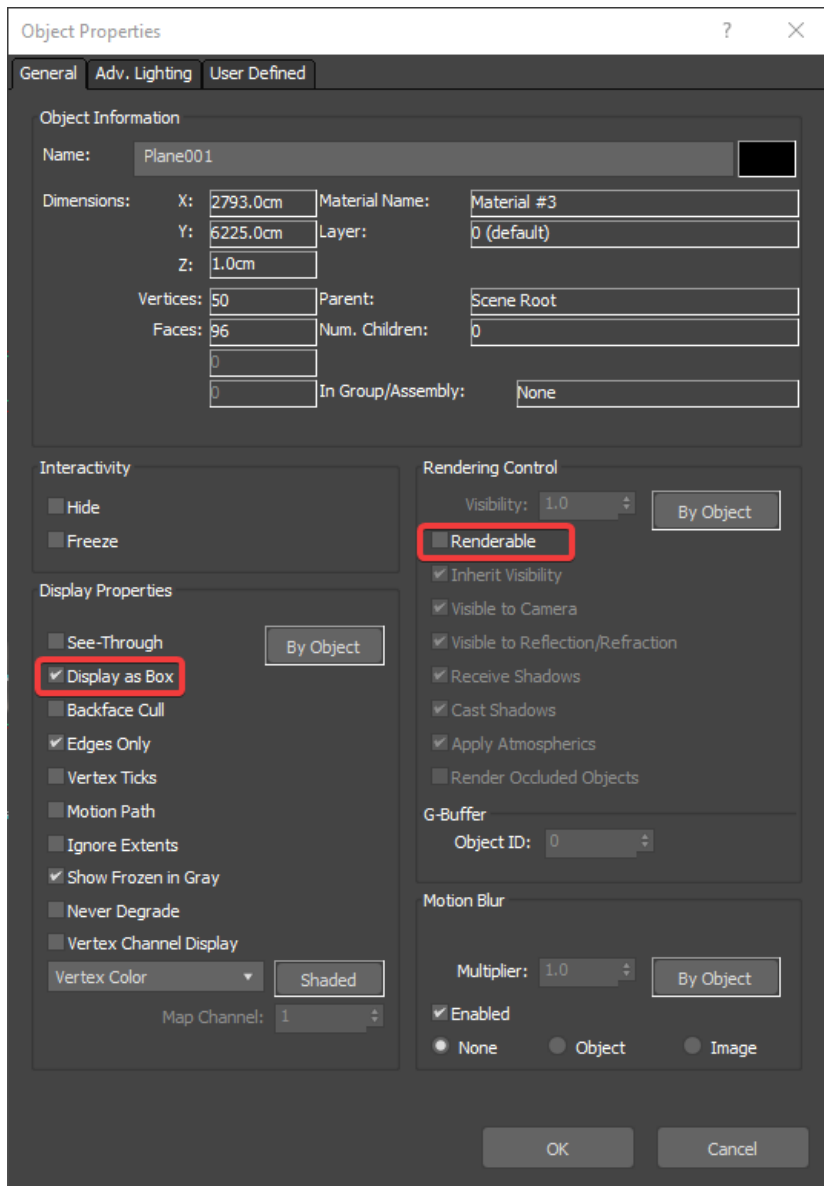
and of the **Camera Target** is [179.267cm, -2317.455cm, 36.06cm].



Render Settings

In the previous steps, when setting the dust/smoke simulation we used the shelled plane.

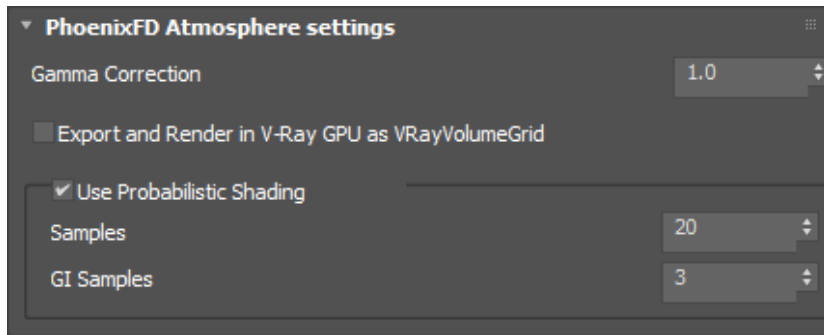
As we don't need this plane to be rendered in the final image - select the plane, right-click it and go to its **Object Properties** menu. Disable the **Renderable** and enable **Display as Box** options.



In the final render a **Probabilistic Shading** is used. The option is located in Environment Atmosphere **PhoenixFD Atmosphere Settings**. When **Use Probabilistic Shading** is enabled, the volumetrics will select a only few samples (based on the smoke density) along each camera ray, and evaluate the volume lighting at those points. In some situations this will make the render time shorter.

The **Use Probabilistic Shading** is enabled.

All parameters are left with their default values.



In the Render Setup Tab:

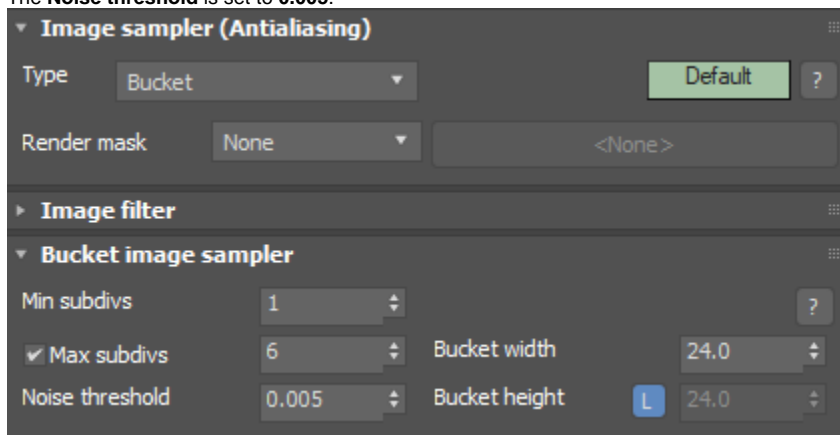
The **Image Sampler Type** is set to **Bucket**.

The **Min Subdivs** are set to **1**.

The **Max Subdivs** are set to **6**.

The **Bucket width** is set to **24.0**.

The **Noise threshold** is set to **0.005**.



A V-Ray Denoiser Render Element is added to the final image. The Denoiser takes an existing render and applies a denoising operation to it after the image is completely rendered in order to remove the noise in the image.

For this tutorial the **Custom** preset is used.

The **strength** is set to **0.5**.

The **radius** is set to **10.0**.

▼ **VRayDenoiser Parameters**

vrayVFB..... ☒

deep output..... ☒

denoising engine..... Default V-Ray denoiser ▼

preset..... Custom ▼

strength..... 0.5 ▼

radius..... 10.0 ▼

Update...

Final Results

