Procedural Ground Dust

This page provides a tutorial on creating a Procedural Ground Dust simulation in Maya.

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 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 FD 4 Official Release** and **V-Ray Next Official Release** for **Maya 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.

Go to **Windows Settings and Preferences Preferences Settings** and set the Working Units to **centimeters**.

Set the **Time** to **29.97 fps**. We do this to ensure the setup is identical to the Procedural Ground Dust tutorial for 3ds Max. You may skip this step if you find it unnecessary. Keep in mind the final result of your simulation may turn out slightly different.
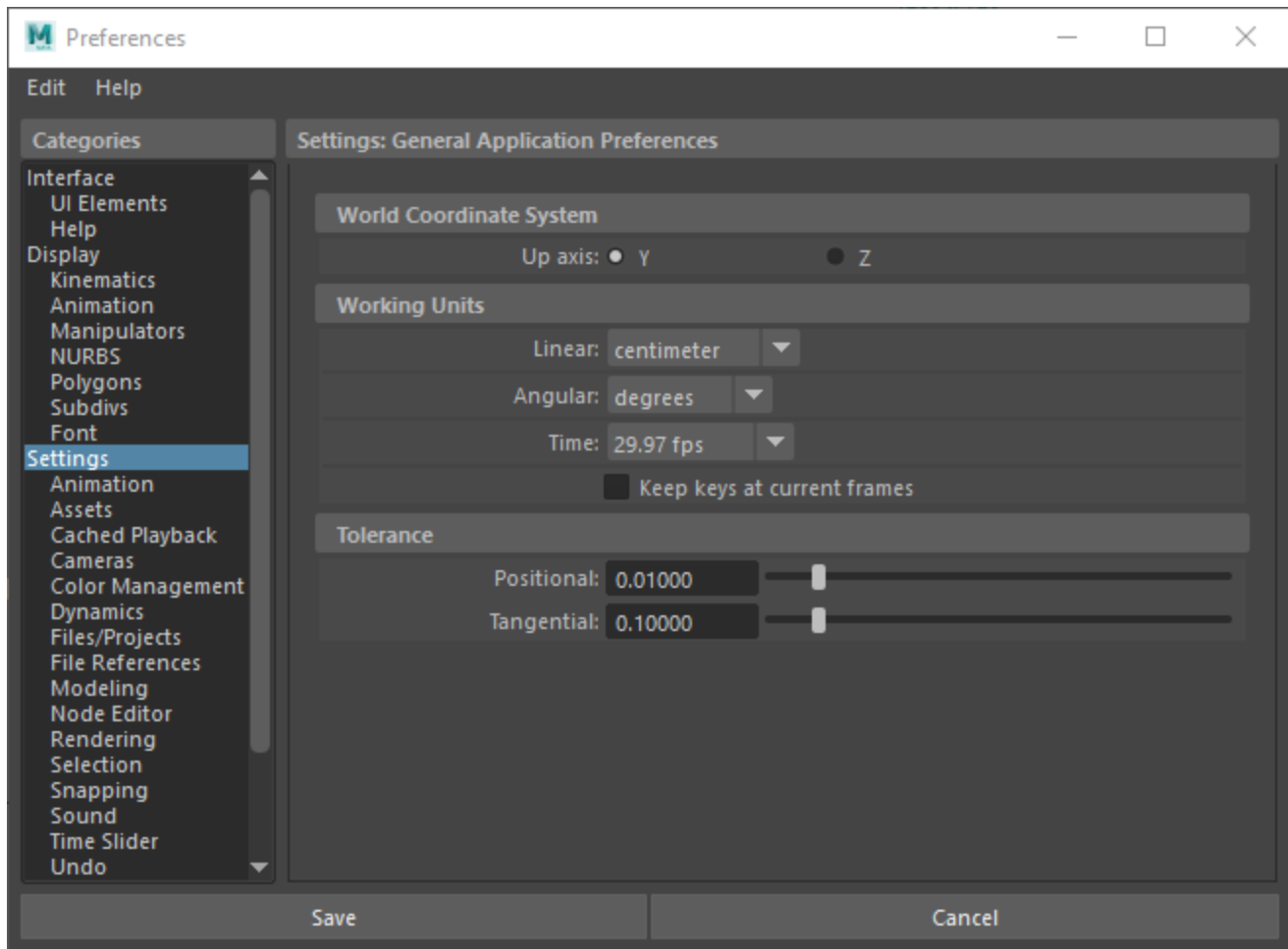
Scene Layout

The final scene consists of the following elements:

1. An **Alembic** file containing the **baked horse animation**.
2. A **Poly Plane** extruded to give it some **thickness**. The plane is used as **emission geometry** for the Phoenix Source.
3. A **V-Ray Infinite Plane** is used for easily setting the ground.
4. A **Phoenix Simulator** with some tweaks to the **Dynamics** and **Rendering** parameters.
5. A **Voxel Tuner** used for a realistic sand smoke coming up as the horse step on the ground.
6. A **V-Ray Physical Camera** with minor tweaks for final rendering.
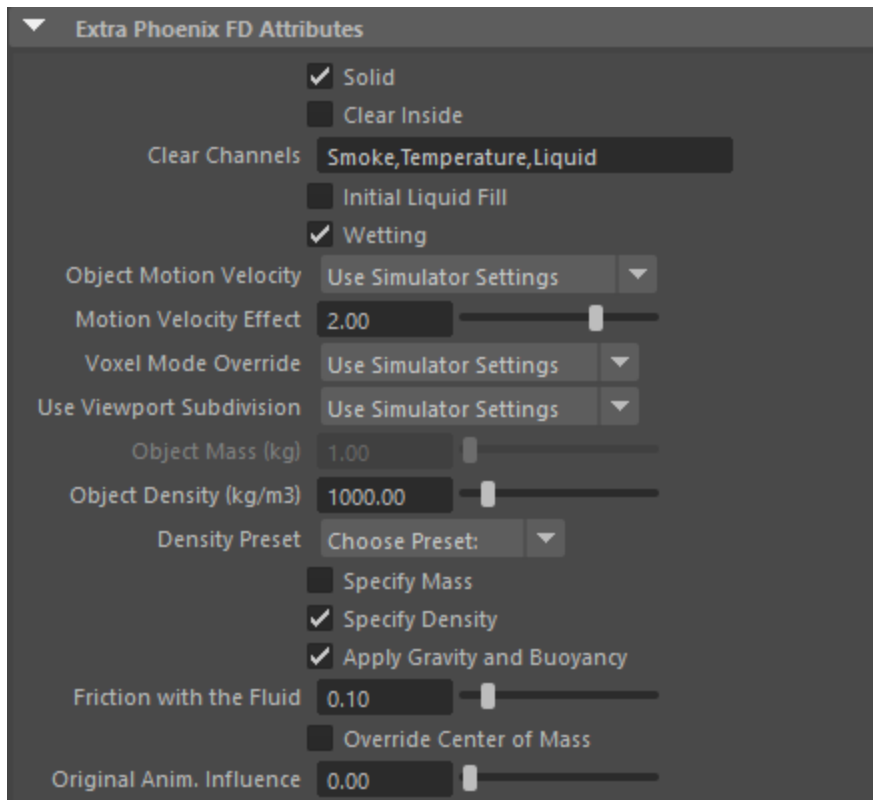7. A **V-Ray Dome Light**.
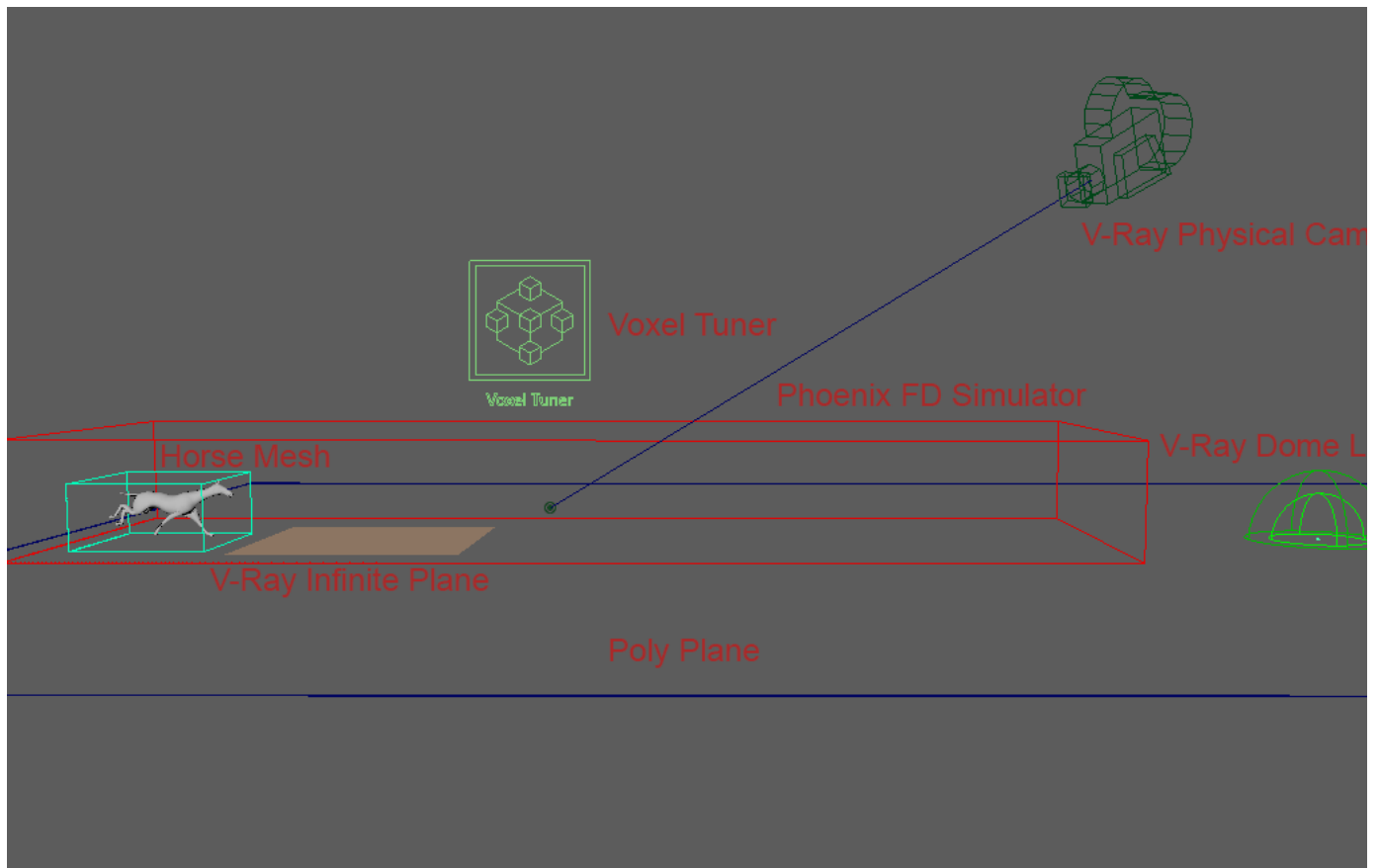
Scene Setup

Let's start by importing the **single_horse_01.abc** file.

Go to **Cache -> Alembic Cache -> Import Alembic...** and select the file. Hit **F** to **frame** it in the Viewport.

The file will come with a parent group called All_trans. You may safely **unparent the deer_trans** node (using Shift + P), **r ename** it to **single_horse** and delete the empty groups.

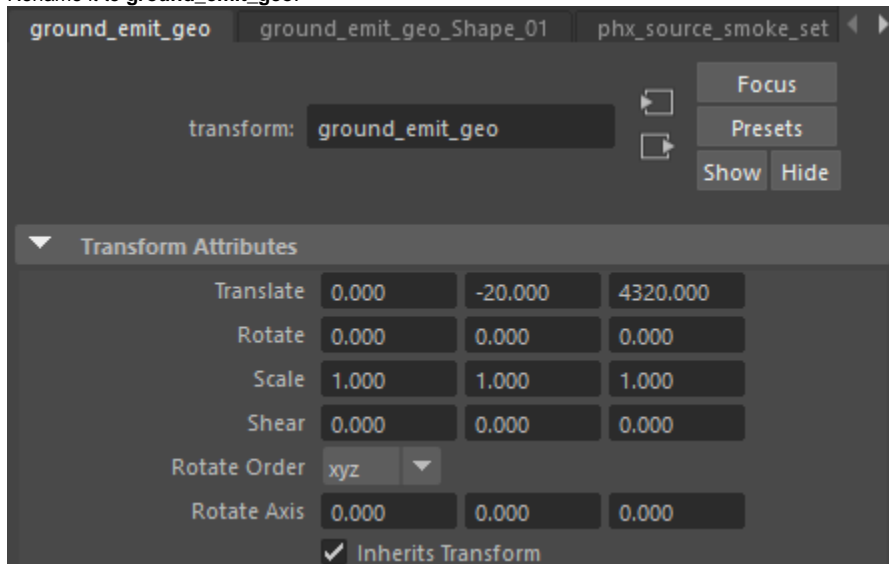The exact start **Position** of the **alembic mesh** in the scene is **[X:0cm, Y:0cm, Z:0cm]**.

You can control the interaction strength between the horse mesh and the polygon plane by adjusting the **Motion Velocity Effect**, located in the Phoenix FD 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**.

Extra Phoenix FD Attributes

☑ Solid
☐ Clear Inside
Clear Channels   Smoke,Temperature,Liquid
☐ Initial Liquid Fill
☑ Wetting
Object Motion Velocity   Use Simulator Settings ▼
Motion Velocity Effect   2.00
Voxel Mode Override   Use Simulator Settings ▼
Use Viewport Subdivision   Use Simulator Settings ▼
Object Mass (kg)   1.00
Object Density (kg/m3)   1000.00
Density Preset   Choose Preset: ▼
☐ Specify Mass
☑ Specify Density
☑ Apply Gravity and Buoyancy
Friction with the Fluid   0.10
☐ Override Center of Mass
Original Anim. Influence   0.00

Add a **Polygon Plane** to the scene and set it's **Width/Height** to **2800/6200**. **The plane is used to emit smoke into the simulation.**

**Possition** the plane to **[X:0cm, Y:-20cm, Z:4320cm]**.

Rename it to **ground_emit_geo**.

| ground_emit_geo | ground_emit_geo_Shape_01 | phx_source_smoke_set | ◀ ▶ |

Focus

transform: ground_emit_geo

Presets
Show Hide

Transform Attributes

Translate	0.000	-20.000	4320.000
Rotate	0.000	0.000	0.000
Scale	1.000	1.000	1.000
Shear	0.000	0.000	0.000
Rotate Order	xyz ▼		
Rotate Axis	0.000	0.000	0.000

☑ Inherits Transform

With the plane selected, go to **Edit Mesh Extrude** and give the **Thickness** a value of **1**.

Using open geometry or geometry with no thickness (such as the plane in this example) can give you unpredictable simulation results. Making sure that your geometry is clean is crucial for a smooth workflow. Phoenix (and many simulation packages in general) use a volumetric representation of the emission geometry for the simulation. The process of creating this volumetric representation is called *voxelization*. The algorithms responsible for *voxelizing* the geometry can fail when using open (with holes) or planar (no thickness) geometry.

The **Extrusion** is applied here to **turn the plane into** what is essentially a very thin **box**. This is enough to sort out any possible problems with the voxelization at simulation time.
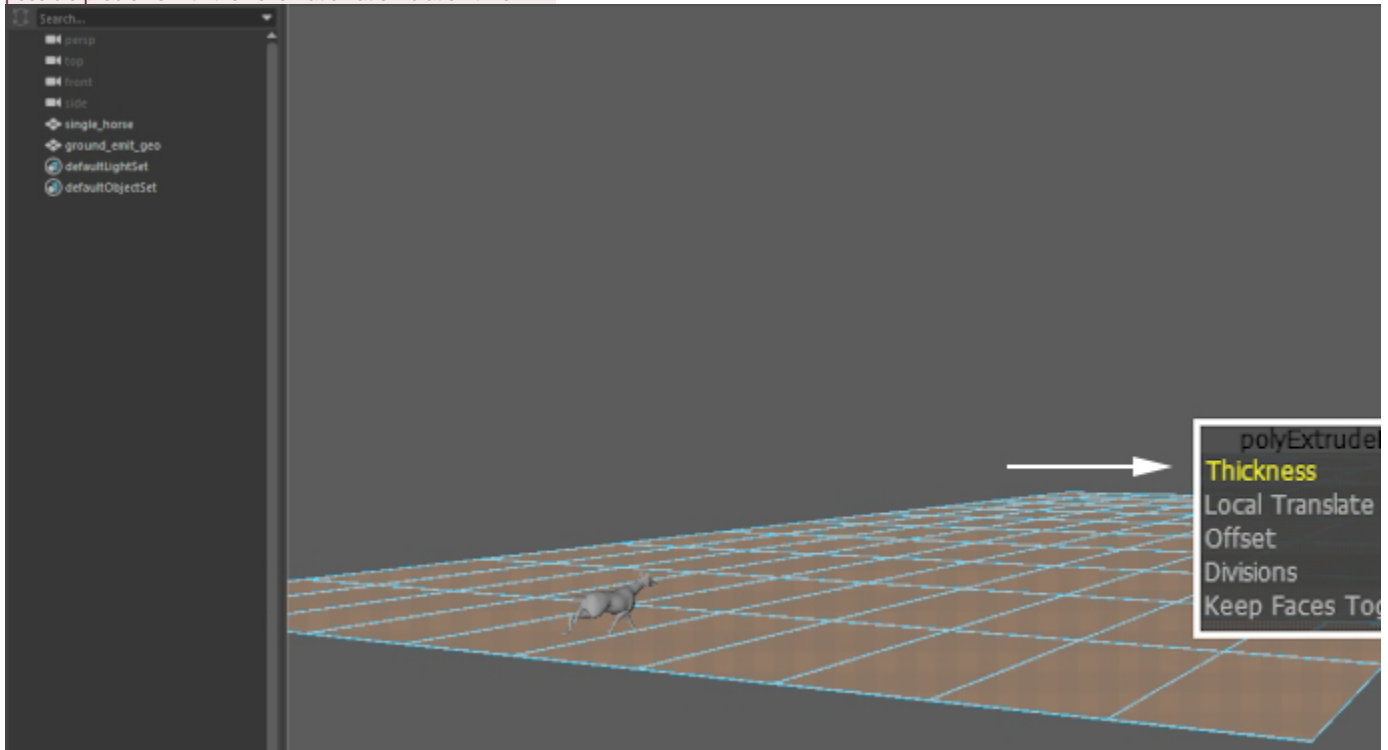
Create a **V-Ray Infinite Plane** used during rendering.

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

Create a new **Phoenix Simulator** and draw it such that it envelops the horse geometry.

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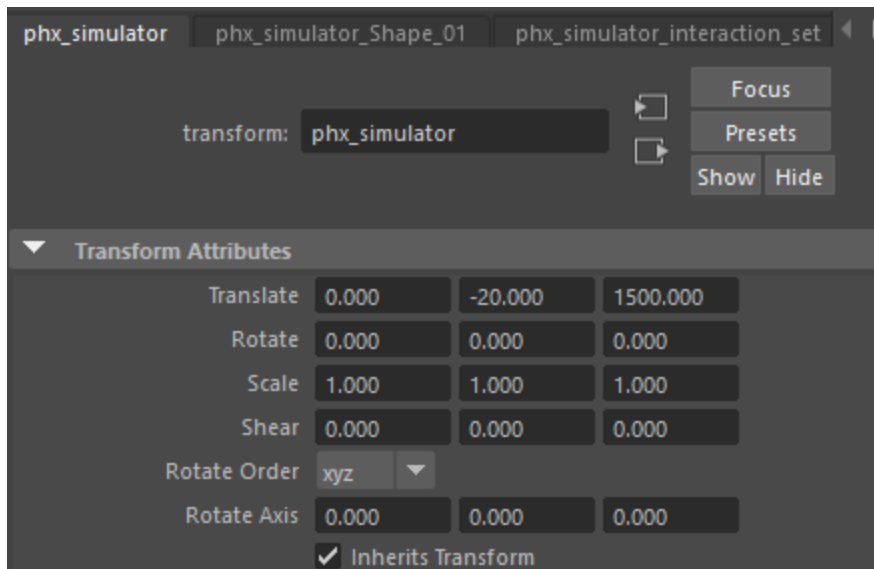
Voxel Tuner

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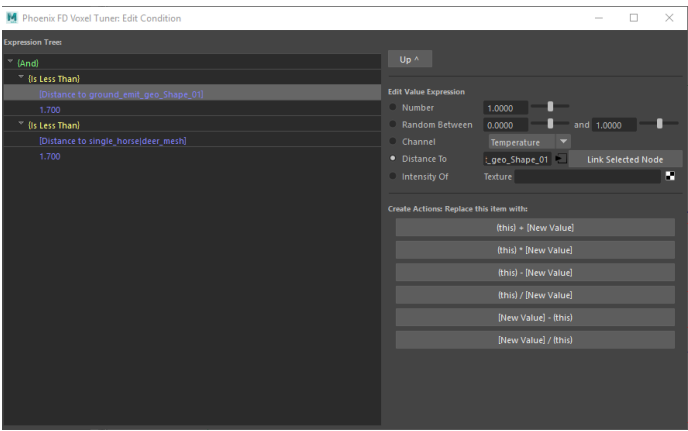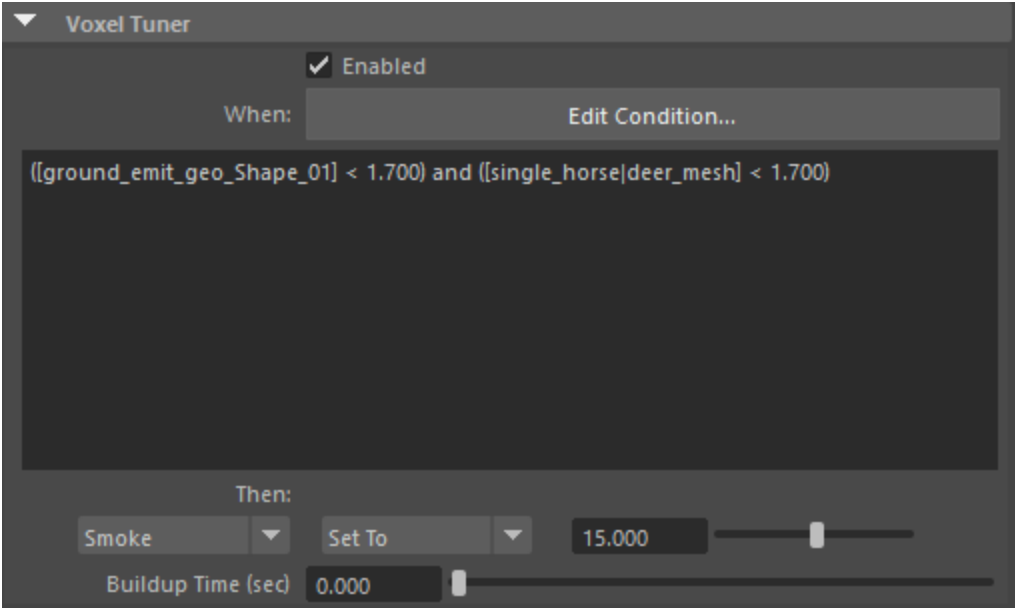
In this case, the Voxel Tuner will go through all the voxels of the Simulator, and check if each voxel is close to the ground plane and simultaneously close to the horse mesh. If it passes this condition, then the Tuner will create Smoke in this voxel.
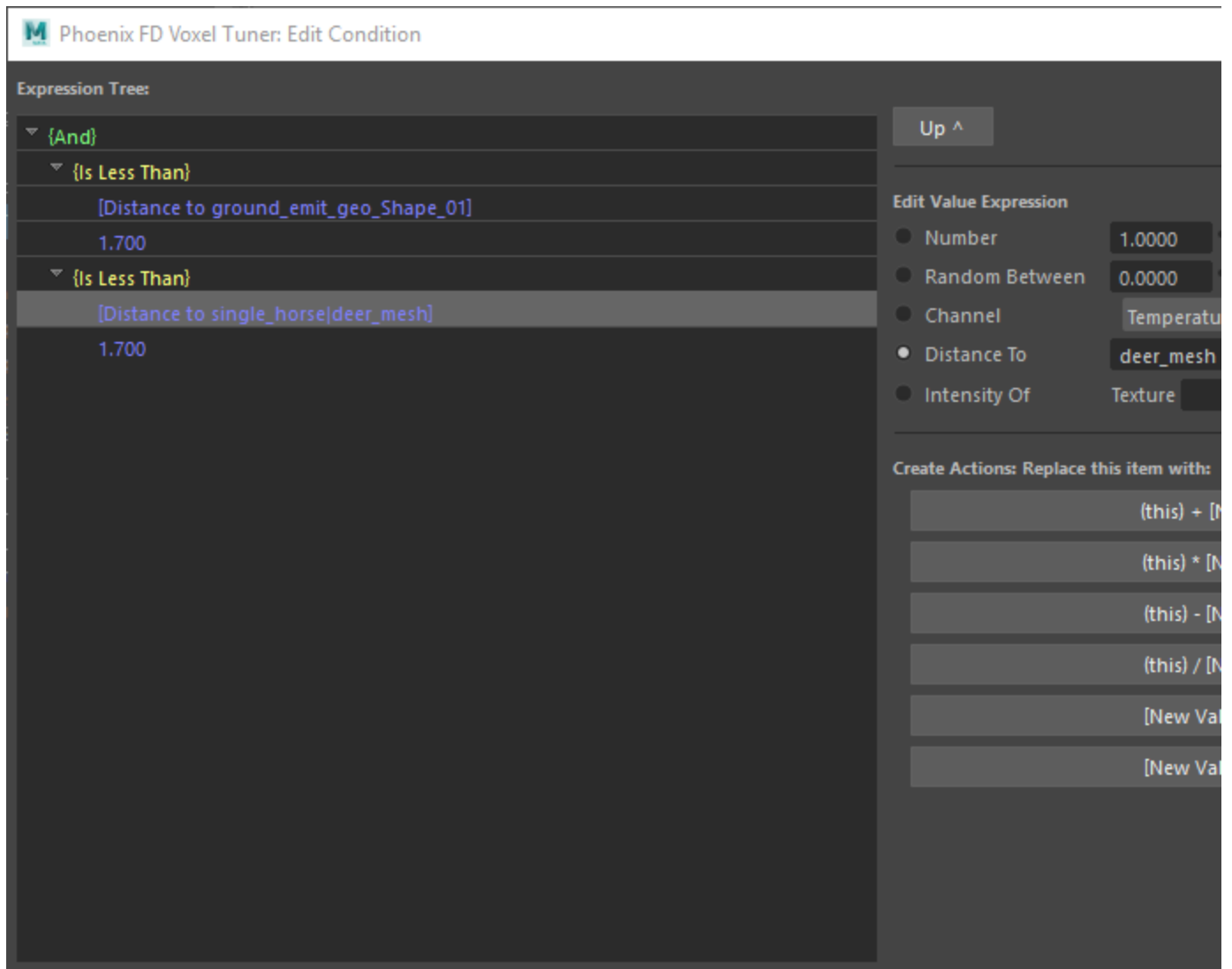
The **Voxel Tuner expression** is created as follows:

- 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.

Phoenix FD Voxel Tuner: Edit Condition

Expression Tree:

▼ {And}
  ▼ {Is Less Than}
    [Distance to ground_emit_geo_Shape_01]
    1.700
  ▼ {Is Less Than}
    [Distance to single_horse|deer_mesh]
    1.700

Up ^

Edit Value Expression

- Number   1.0000
- Random Between   0.0000
- Channel   Temperatu
- Distance To   deer_mesh
- Intensity Of   Texture

Create Actions: Replace this item with:

(this) + [N
(this) * [N
(this) - [N
(this) / [N
[New Val
[New Val

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.
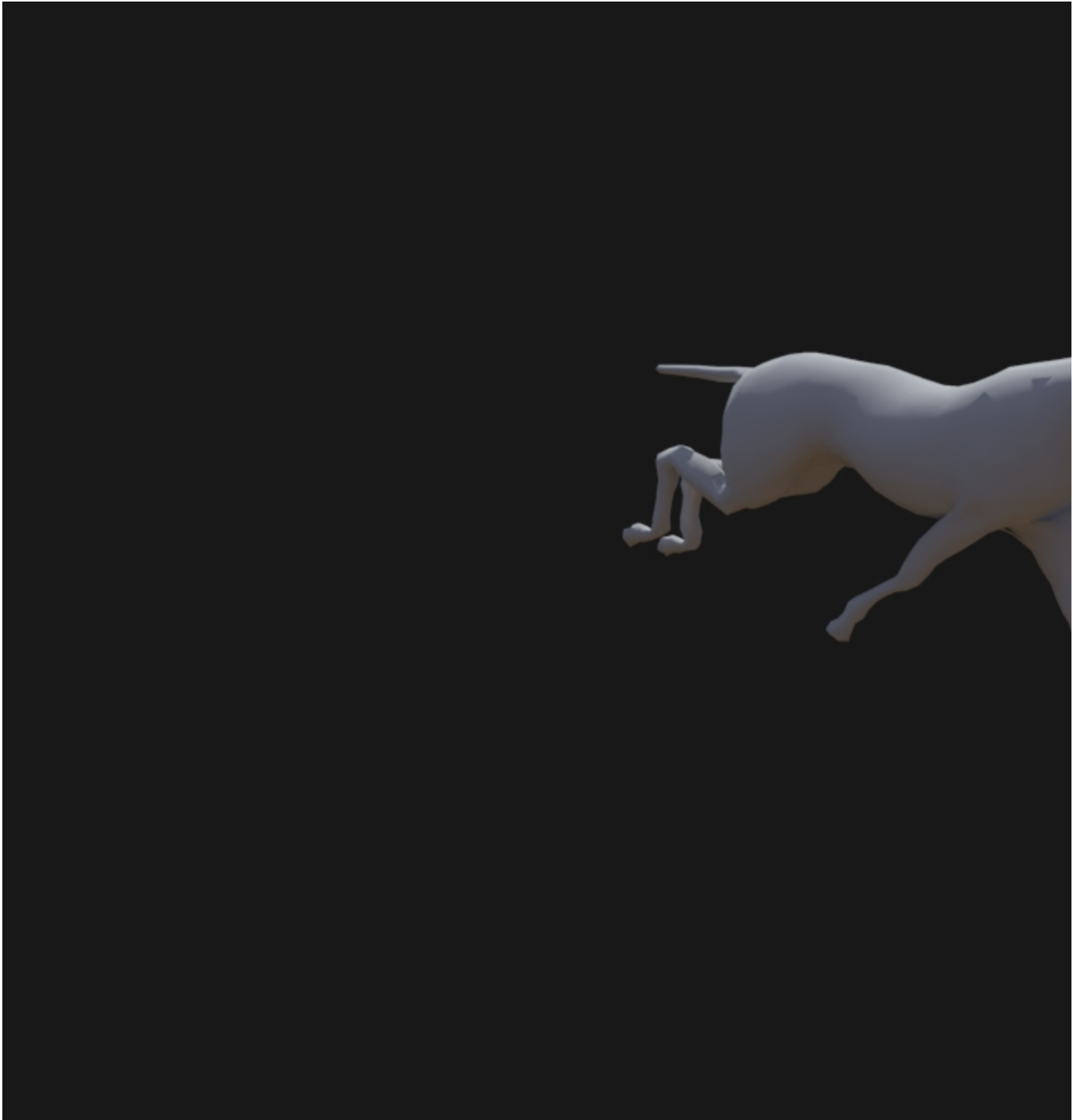
Simulation

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

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

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

The **Size** of the Simulator is set to **[X:150cm, Y:70cm, Z: 140cm]** 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. Since the Smoke is emitting Smoke 1.0, this means that the grid will be quite sensitive to smoke coming near the grid walls.

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: 155/80, +Y/-Y: 50/0, +Z/-Z: 950/30]**.
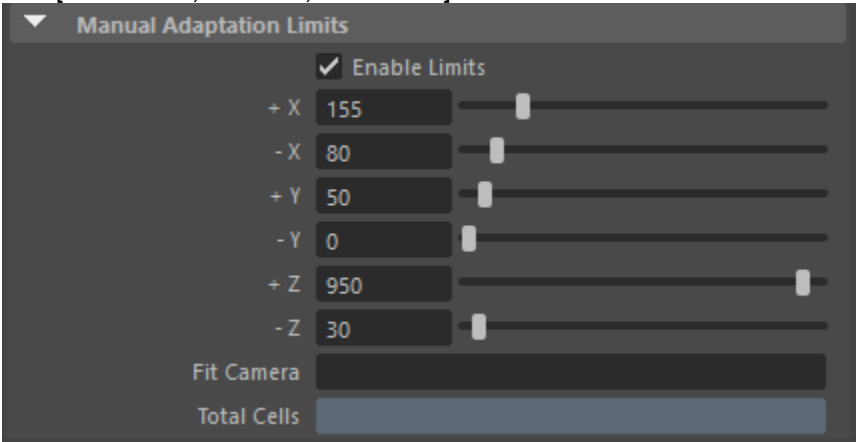
The three images here compare the differences between **Sce
ne Scale** values [set from top to bottom to 1, 5 and 10].

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 diffuse 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.
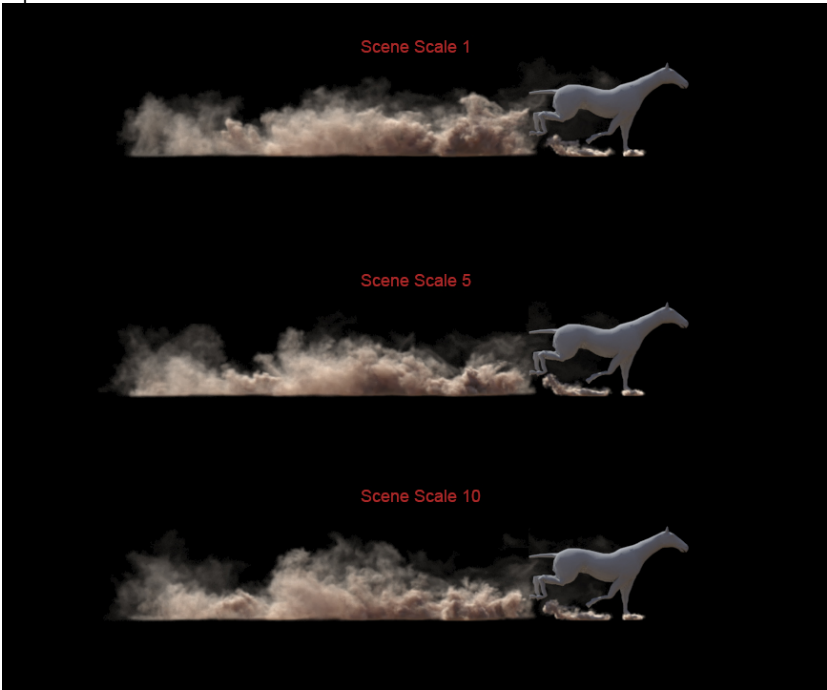
In the Dynamics rollout menu, switch the **Smoke Dissipation** to **0.2** and the **Smoke Buoyancy** to **0.6**.

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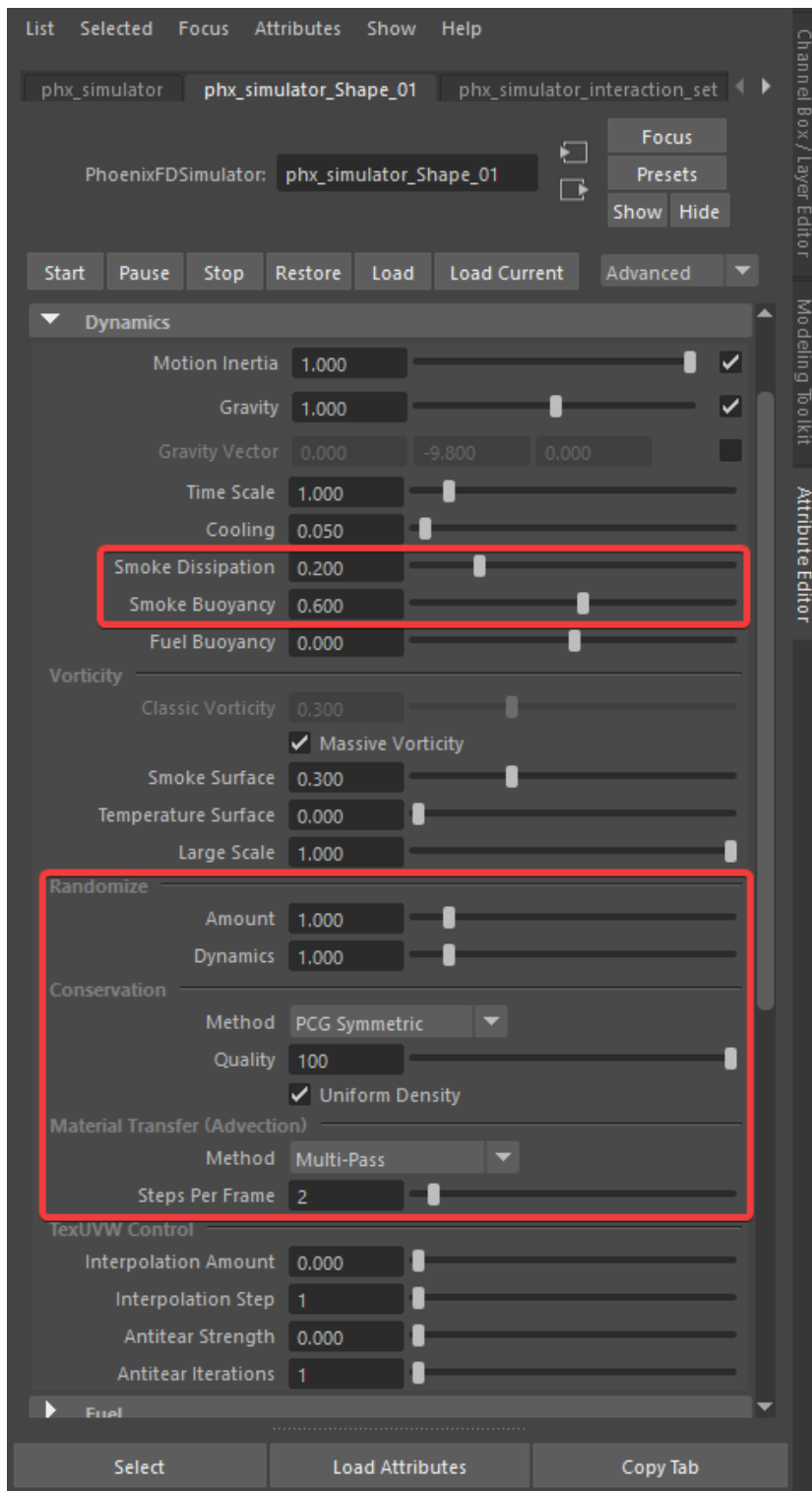
Compare the difference in the **Smoke Buoyancy** values displayed in these three images.

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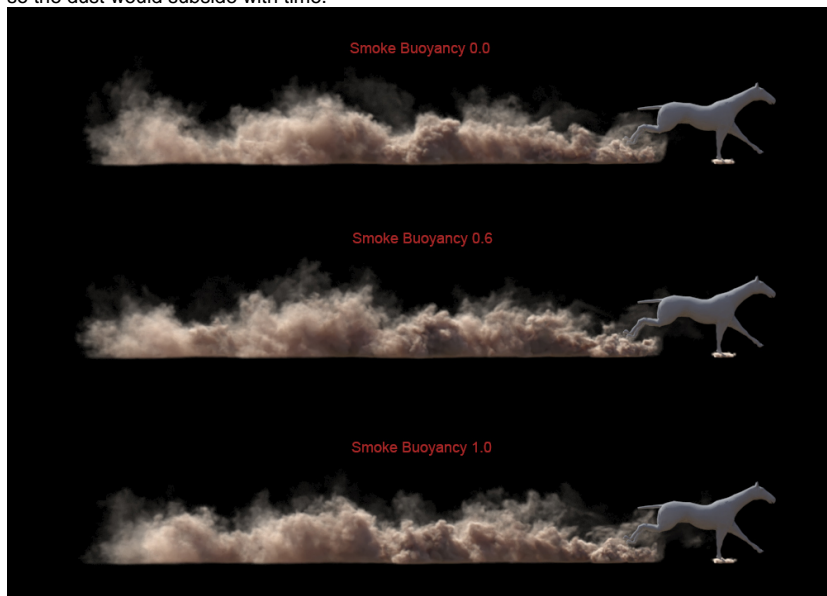
The **Smoke Dissipation** parameter controls how fast the
smoke will disappear. The maximum value of 1 will make the
smoke disappear immediately after it was emitted.

The three images here compare the differences between **Sm
oke 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.
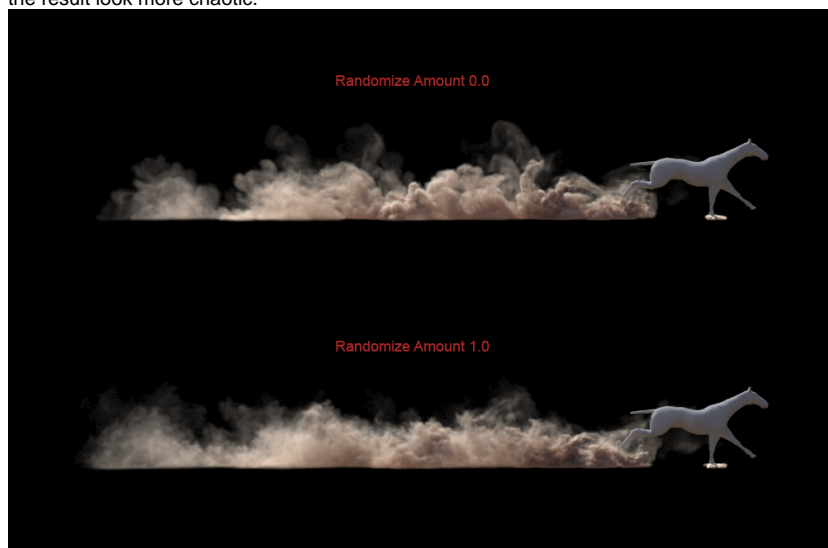

Open the Output rollout menu, which holds the controls for the simulation result.

The **Storage Quality** is set to **20** to avoid appearing artifacts in the simulation.

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**.
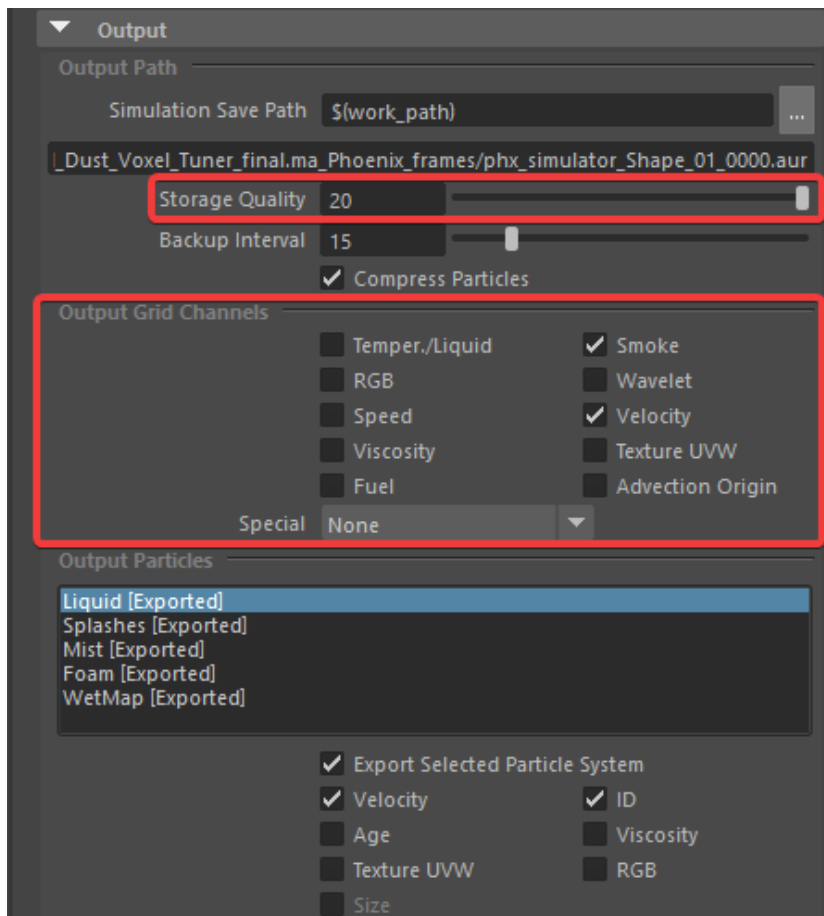
Open the Rendering rollout menu to prepare the simulation for rendering.

The **Sampler type** parameter determines the blending method between adjacent grid cells. The sampler is set to **Sp herical**.
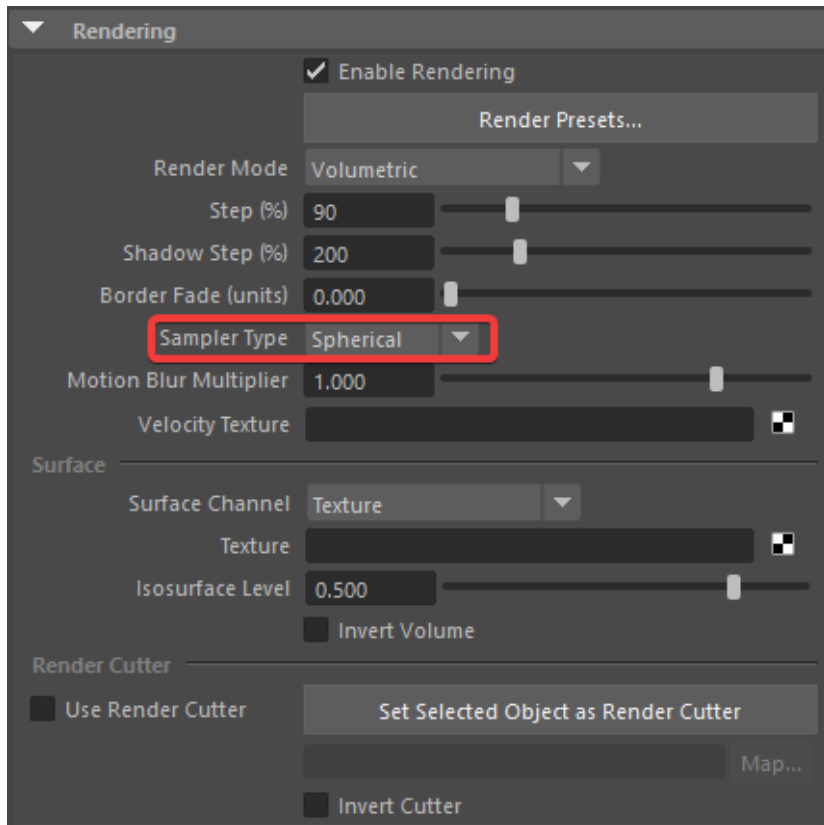
In those two images are compared two different **Sampler types** - **Spherical** (top) and **Linear** (bottom).

With **Linear** Sampler Type the produced dust looks grainy, so for this tutorial is used the **Spherical** sampling, that effectively reduces this issue.
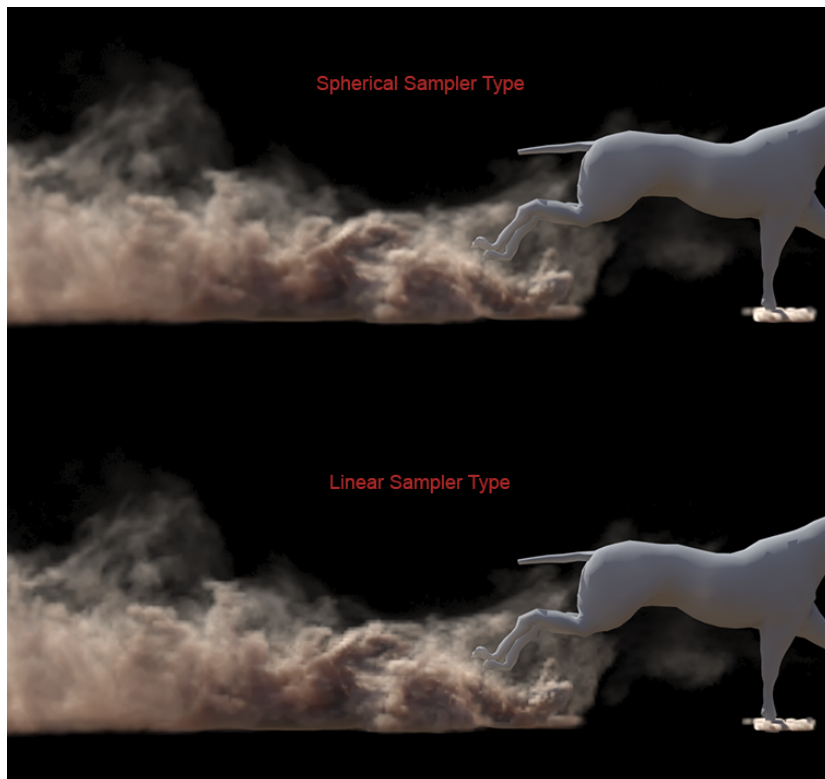
The **Fire**'s Based on is set to **Disabled**.

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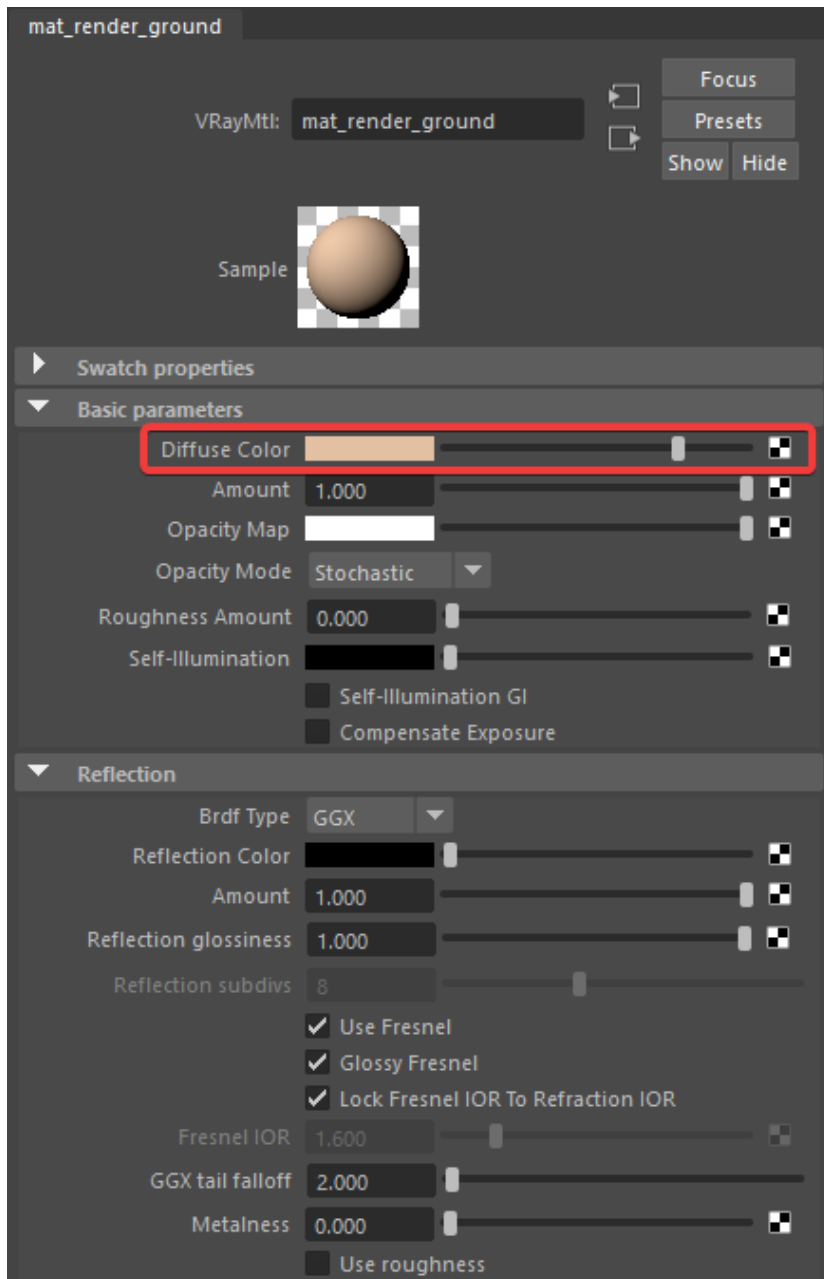
The **Scale Opacity by Scene Units** is **disabled**.
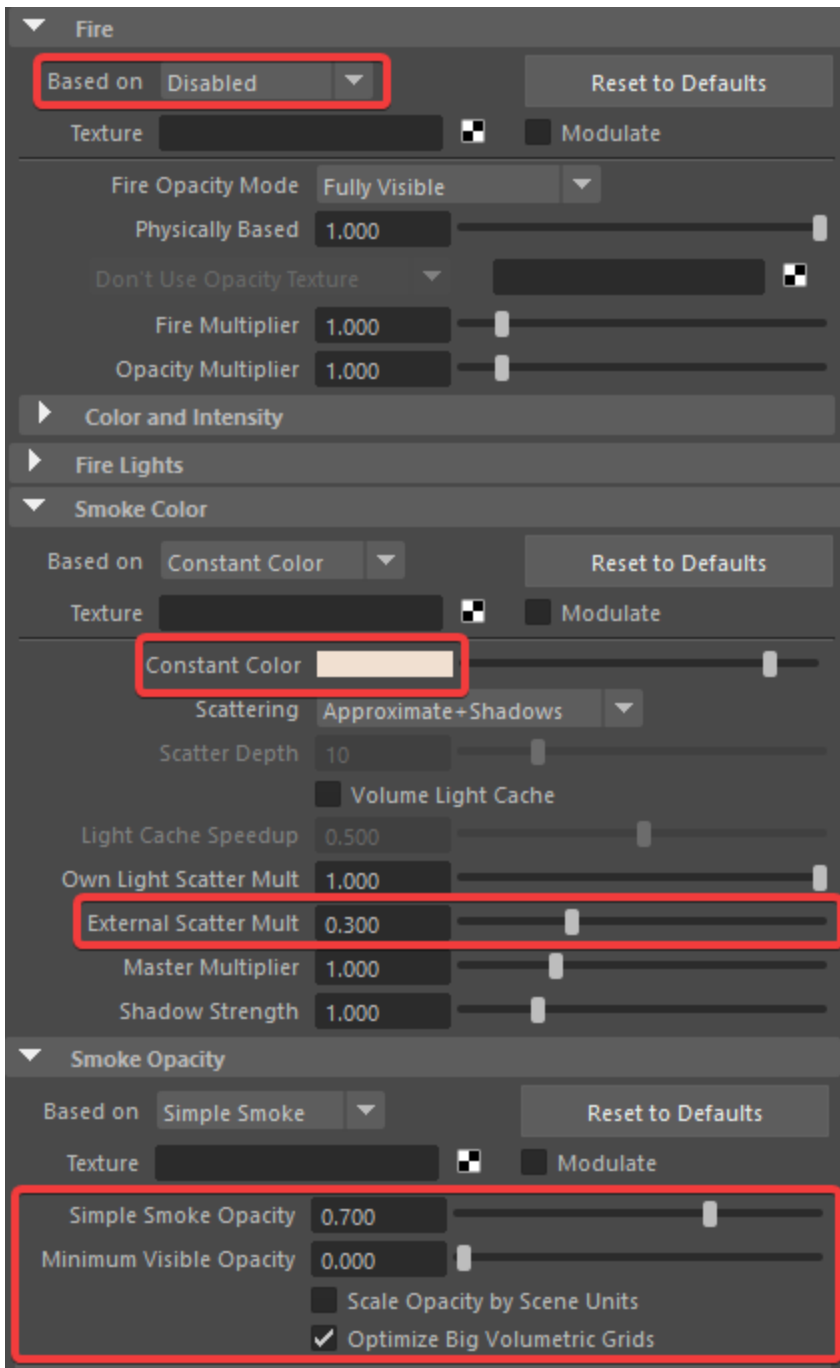
Materials

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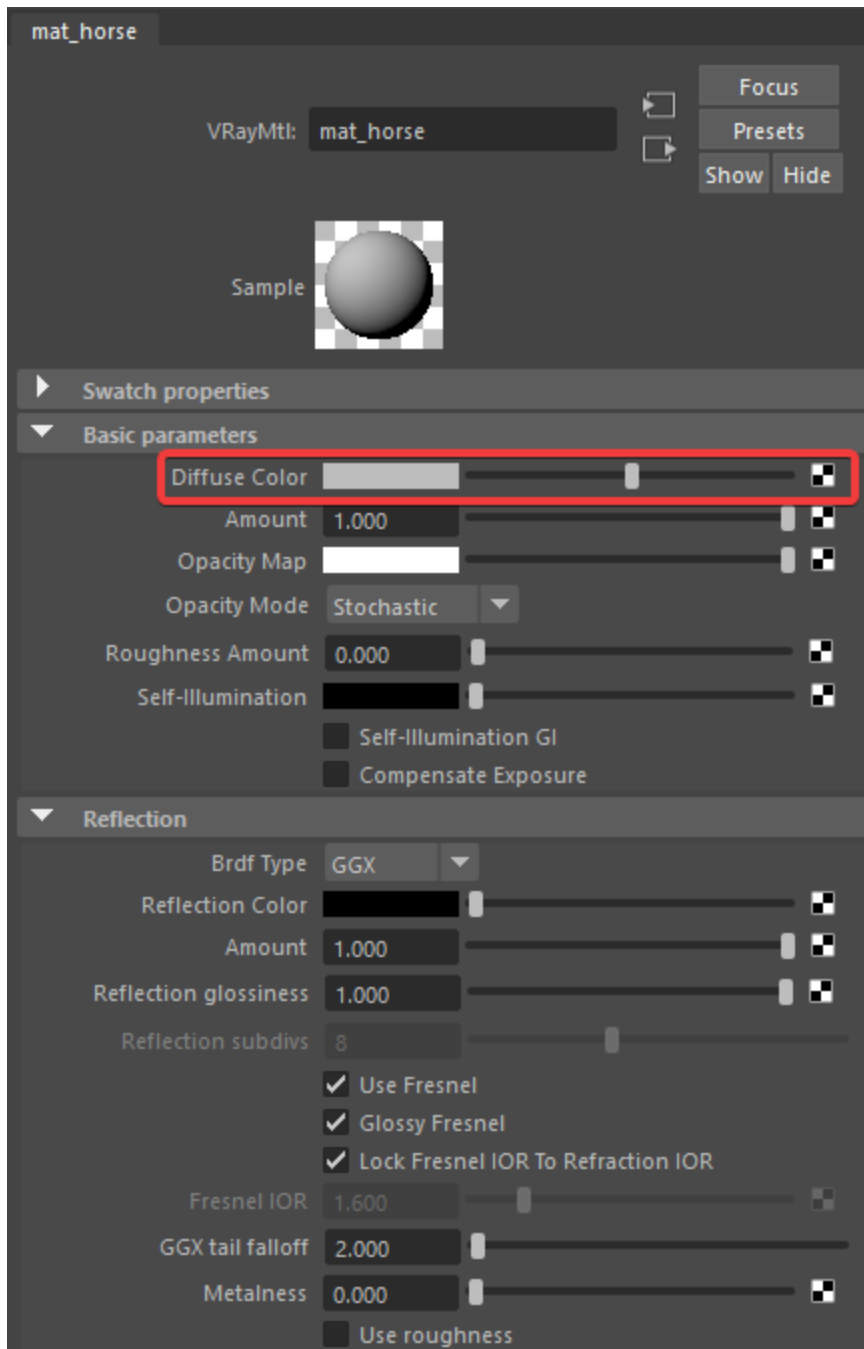
The **Diffuse** color of the **"mat_render_ground"** is **RGB [197, 135, 93]**.

All other parameters are left with their default values.

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

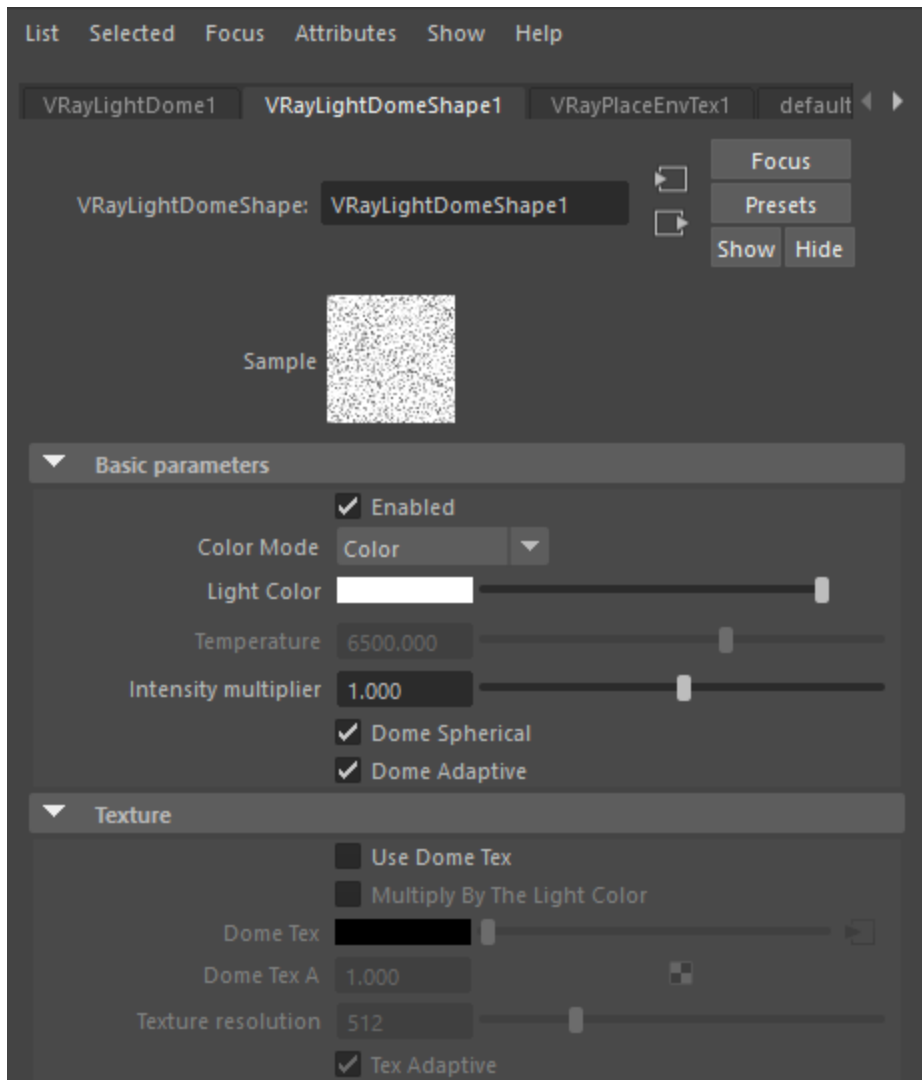All other parameters are left with their default values.

Lighting and Camera

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

To add one, go to **Create ▸ Lights ▸ V-Ray Dome Light**.

Set the **Multiplier** to **1.0**.

You can also use **V-Ray Sky and Sun** system. Remember to
set your camera exposure to be suitable for outdoor lighting.

For this setup a VRayPhysicalCamera is used.

To add **Physical Camera** parameters, select the Maya camera used for rendering.

Under its **Shape node** in the Attribute editor go to **Attributes V-Ray Physical Camera**.
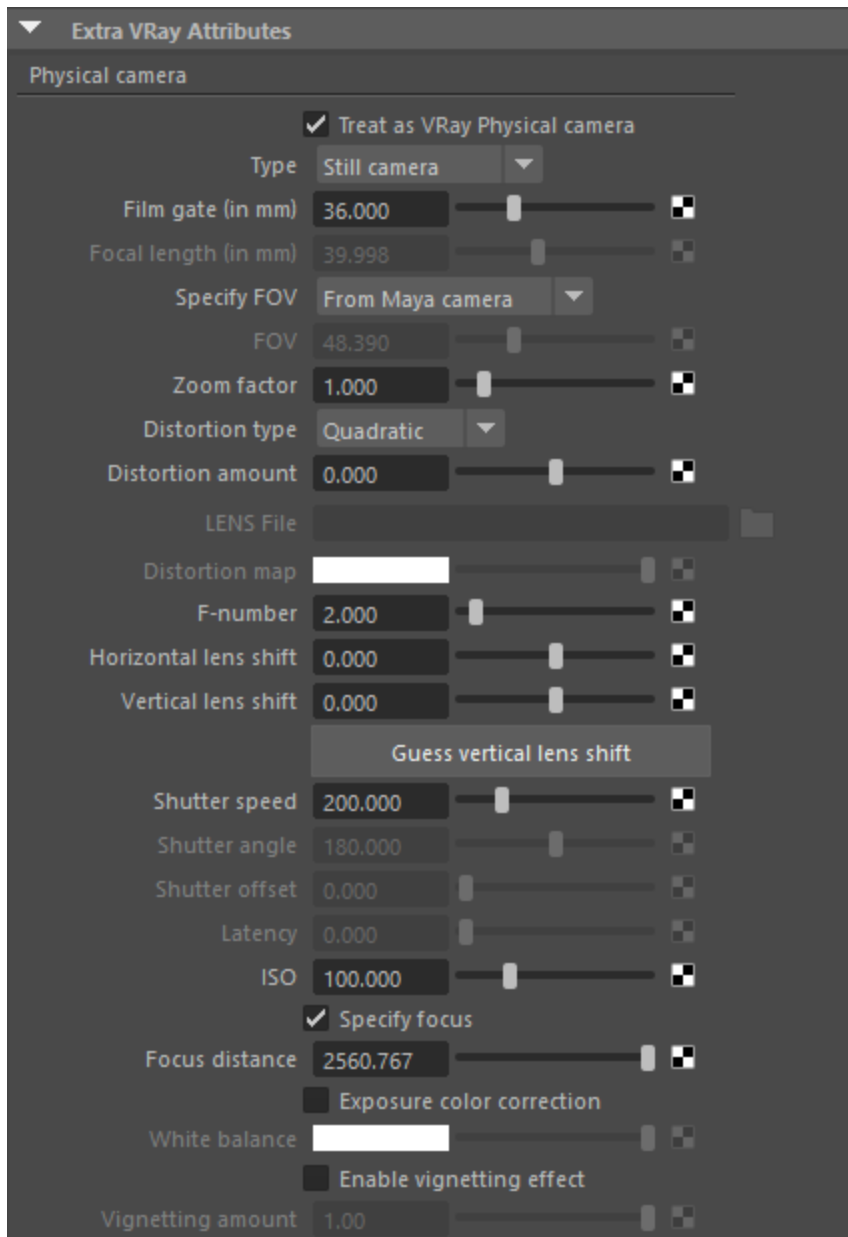
The **Film Gate** is set to **36.0**.

The **F-Number** is set to **2.0**.

The **Shutter Speed** is set to **200**.

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

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

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

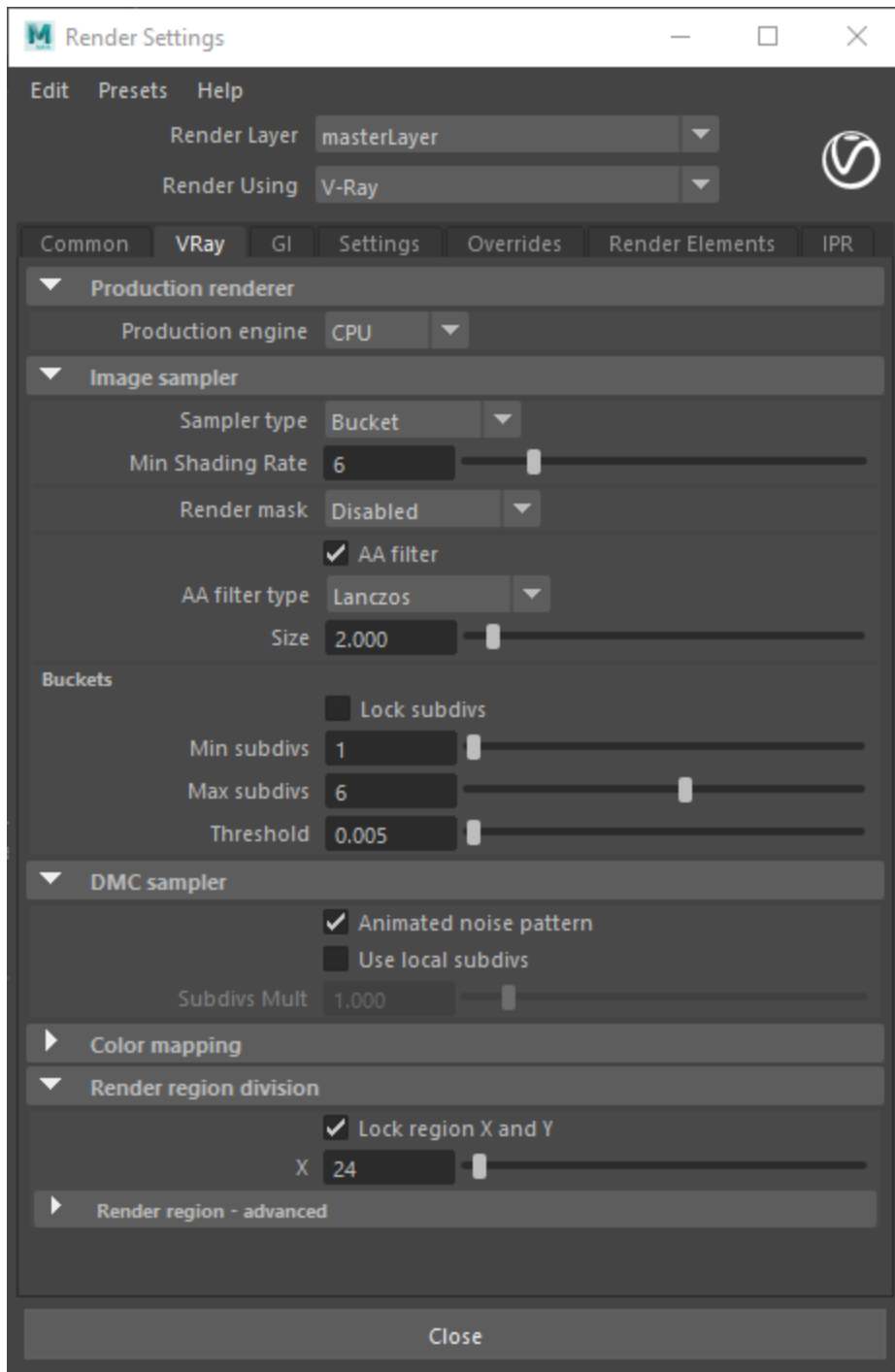## Render Settings

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 **Threshold** is set to **0.005**.

The **Render Region Division** is set to **24.0**.

In the final render a Volumetrics Overrides is used. The option is located in **Render Setup -> Overrides -> Volumetri cs**. When **Use Probabilistic Volumetrics** 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 **Probabilistic volumetrics** is **enabled**.

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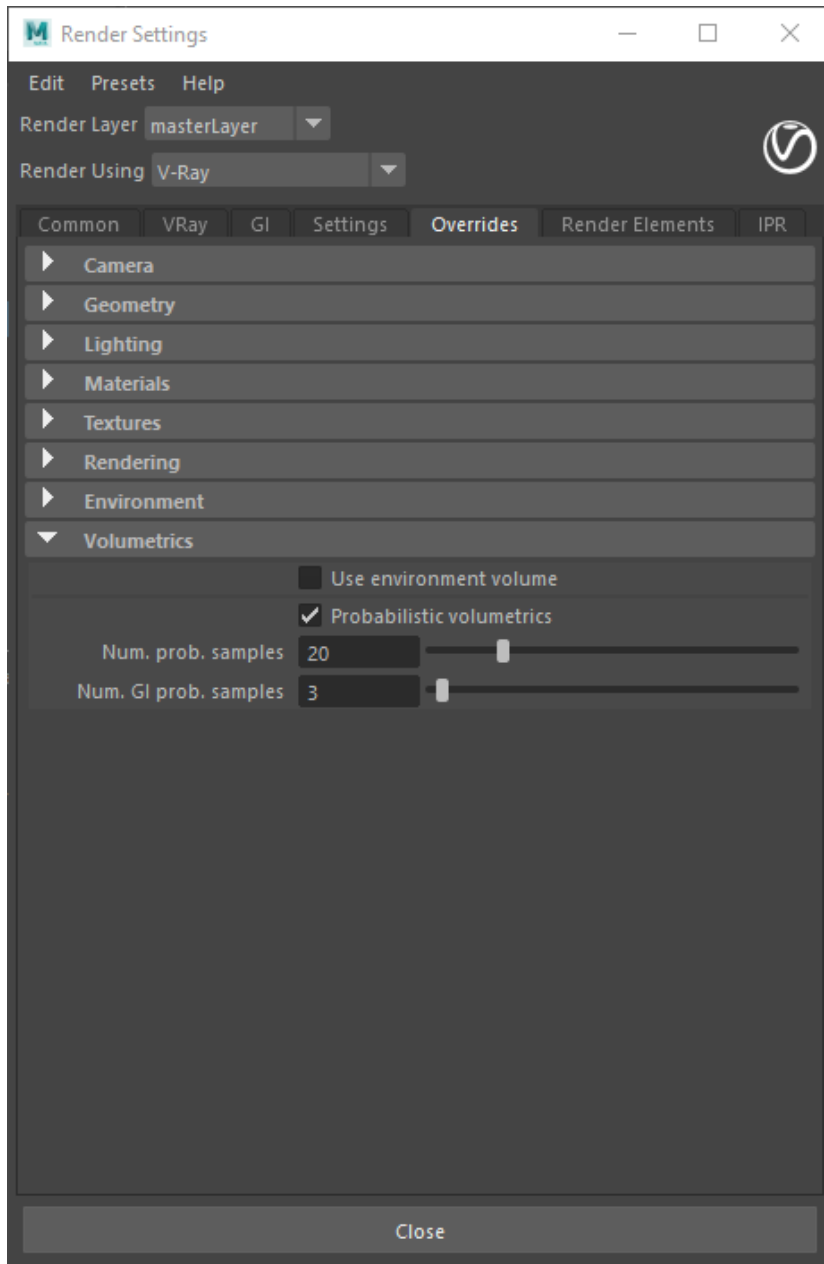
A V-Ray Denoiser Render Element is added to the final image. The Denoiser takes an existing render and applies a denoising operation to it after the image is completely rendered in order to remove the noise in the image.

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

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

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

Final Results