



***SpeedSix Fluidz Raptor V1.2 for Autodesk (Discreet)
Advanced Products on Linux Red Hat EL WS 3 and 4***

32 bit versions:

Flame 9.0 – 9.4

Flint 9.0 – 9.4

Smoke 6.5 – 6.9

64 bit versions:

Flame 9.5+ (e.g. Flame 2008)

Flint 9.5+ (e.g. Flint 2008)

Smoke 7.0+ (e.g. Smoke 2008)

30–September–2008

Table of Contents

1 SpeedSix Fluidz Raptor V1.2 for Autodesk (Discreet) Advanced Systems on Red Hat Linux.....	1
1.1 Introduction.....	1
1.2 Pre-requisites.....	1
1.3 Downloading the Fluidz Raptor.....	2
1.4 Installation.....	2
1.5 Licensing.....	4
1.6 Support.....	5
1.7 Sales.....	5
1.8 Thanks!.....	5
2 SpeedSix.Fire (Discreet Fluidz Raptor).....	6
3 SpeedSix.ImageFlow (Discreet Fluidz Raptor).....	12
4 SpeedSix.ClipFlow (Discreet Fluidz Raptor).....	19
5 SpeedSix.TransFlow (Discreet Fluidz Raptor).....	25

1 SpeedSix Fluidz Raptor V1.2 for Autodesk (Discreet) Advanced Systems on Red Hat Linux

1.1 Introduction

This manual describes the SpeedSix Fluidz Raptor for Autodesk (Discreet) "Advanced Systems" on Red Hat Linux. Fluidz is a set of four plugins that use 2D fluid flow algorithms to generate a number of effects:

- ◆ **Fire:** Generates a wide variety of flame and fire effects.
- ◆ **ImageFlow:** Makes the colours in a single image flow over time in a very natural way. It can be used to create sequences displaying a great range of liquid and gas flow effects.
- ◆ **ClipFlow:** Distorts a moving clip by flowing texture coordinates and using the flowed texture coordinates to distort each frame of an input sequence. Useful primarily for transitions.
- ◆ **TransFlow:** Creates a variety of novel transitions based on fluid flow effects.

Below we tell you how to install the Fluidz Raptor and how to license it.

Please note that the Fluidz Raptor is fully functional without a license – a SpeedSix logo will be superimposed on all output frames, however.

1.2 Pre-requisites

Please ensure that your Autodesk (Discreet) software is an appropriate version. The 32 bit version of the Fluidz Raptor is intended for use with these 32 bit programs:

- Flint V9.0 – V9.4.
- Flame V9.0 – V9.4.
- Smoke V6.5 – V6.9.

The following Autodesk (Discreet) products are 64 bit programs and need the 64 bit version of the Fluidz Raptor:

- Flint V9.5 and higher (e.g. Flint 2007).
- Flame V9.5 and higher (e.g. Flame 2007).
- Smoke V7.0 and higher (e.g. Smoke 2007).
- Inferno V6.5 and higher (e.g. Inferno 2007).

1.3 Downloading the Fluidz Raptor

Download the latest version of the Fluidz Raptor from:

<ftp://ftp.speedsix.com/Raptors/discreet/Fluidz>

You will find separate installation packages for IRIX 32 bit and 64 bit, and Linux 32 bit and 64 bit (four packages in total). There are no Burn packages, as all the Fluidz effects are cumulative and cannot be usefully run in Burn. Choose the Linux package you need for compatibility with your Autodesk (Discreet) product.

The package names are of the form:

Fluidz{32|64}_V1.2.{release}_linux.tar.gz

E.g.

Fluidz64_V1.2.706_linux.tar.gz

1.4 Installation

To install the Fluidz Raptor proceed as follows.

1. Obtain the appropriate installer package (see above). You will usually obtain this from the SpeedSix FTP site, but it may occasionally be provided on CD by your reseller.
2. Create a temporary directory in which to unpack the distribution. E.g.

```
your_prompt> mkdir /var/tmp/s6
```

3. Copy the distribution file from its current location to the temporary directory. E.g.

```
your_prompt> cp /somedisk/mydownloads/Fluidz64_V1.2.706_linux.tar.gz /var/tmp/s6
```

4. Go to the temporary directory:

```
your_prompt> cd /var/tmp/s6
```

5. Unpack the distribution file (replace 64 with 32 if on a 32 bit system):

```
your_prompt> tar xzvf Fluidz64_V1.2.706_linux.tar.gz
```

1 SpeedSix Fluidz Raptor V1.2 for Autodesk (Discreet) Advanced Systems on Red Hat Linux

6. Become super-user if you have not already done so:

```
your_prompt> su  
Password: <enter your root password>
```

7. Start the installation procedure (omit -64 in the cd if on a 32 bit system):

```
root_prompt> cd discreet-linux-fluidz-raptor-64-1.2-nnn-dist (e.g. nnn=706)  
root_prompt> ./install_s6
```

8. You will be asked if you are ready to read the SpeedSix License Agreement. Respond **y[enter]** or **n[enter]**. Installation will end at this point if you choose **n[enter]**.
9. After reading the SpeedSix License Agreement you will be asked if you accept it or not. Choose **y[enter]** or **n[enter]**. Installation will end at this point if you choose **n[enter]**.
10. Everything needed will then be automatically installed. (There are no options to worry about).
11. Finally, the installer checks if you have a valid license for each box and bundle, and copies an appropriate proxy image. This will appear when browsing for Sparks in proxy mode in your Autodesk application and lets you easily tell which plugins you have licenses for.

1.4.1 After Installation

When the Fluidz Raptor has been installed, you will have the following:

- **/usr/discreet/sparks/SpeedSix_V1.2_Raptors** or **SpeedSix_V1.2_Raptors64** is where you will find the new Fluidz Raptor Sparks. This is where you should browse to to load a Spark from within the Autodesk software.
- **/usr/local/SpeedSix/Licenses** is where Fluidz Raptor license lives. When you get a license file from SpeedSix, it should be copied **unaltered** to this directory. See below for more information.
- **/usr/local/SpeedSix/bin** contains SpeedSix utility programs – mainly for licensing. See below for more information. It also contains an un-install script.
- **/usr/local/SpeedSix/dl/help** contains extensive HTML format help for every plugin. This is accessible via the **Help** button in each Fluidz Raptor plugin when running your Autodesk product, or you can browse it "offline" using any Web browser. The master index page is:
/usr/local/SpeedSix/dl/help/raptorsindex1.htm then the **Fluidz** link from there.
- Note that "online" help – obtained by pressing a plugin's **Help** button – will be displayed using the default Web browser. The default browser is **firefox** (if that is installed) or

1 SpeedSix Fluidz Raptor V1.2 for Autodesk (Discreet) Advanced Systems on Red Hat Linux

mozilla (if there is no **firefox**). You can select another browser by setting the environment variable **S6_HELP_BROWSER** to the executable (or equivalent) of your chosen browser. If you start your Autodesk product from a desktop icon, you may have to set **S6_HELP_BROWSER** by editing the shell script associated with the icon. This is **/usr/discreet/<flame...>/bin/startApplication** where **<flame...>** is the name of the specific Autodesk product you are using.

- **/usr/local/SpeedSix/Docs** contains a PDF format manual for the Fluidz Raptor (derived from the HTML help), which you can view with (for example) **xpdf** or print. The manual is located at:
/usr/local/SpeedSix/Docs/Discreet_Linux_Fluidz_Raptor_PDFManual.pdf
- **/usr/local/SpeedSix/bin/CHECK_LICENSE_FLUIDZ{ _LINUX_64 }** checks for a license for the Fluidz Raptor and creates proxy images that indicate whether a Fluidz Raptor plugin is licensed or not when using the proxy image browser to select a plugin in the Autodesk product. After installing new licenses, you can run this to update your proxy images if you wish.

1.5 Licensing

You do not need a license key to run the software for evaluation purposes, but the images it creates will be watermarked.

To remove the watermarks, you will need a SpeedSix license file. This will be provided when you purchase the software.

Your Fluidz Raptor license is tied to the MAC address of the **eth0** Ethernet adaptor of your machine. The easiest way of displaying this number in the right format is to use:

```
your_prompt> /usr/local/SpeedSix/bin/ssid
```

after you have installed the Fluidz Raptor.

Alternatively, you can use this Linux command:

```
your_prompt> /sbin/ifconfig eth0
```

and send us the **HWaddr: xx:yy:zz:aa:bb:cc** part of the output.

SpeedSix (or your reseller) will send you your license as an email attachment. Proceed as follows:

1. Save the attachment as a file. This ***is*** the license – ***please keep a copy in a safe place.***
2. Copy the file to the following directory on the machine to be licensed:

```
/usr/local/SpeedSix/Licenses
```

1 SpeedSix Fluidz Raptor V1.2 for Autodesk (Discreet) Advanced Systems on Red Hat Linux

Your Fluidz Raptor is now licensed and will render without watermarks next time you start the Autodesk application and load a Fluidz Raptor plugin.

You may want to run `/usr/local/SpeedSix/bin/CHECK_LICENSE_FLUIDZ{_LINUX_64}` to update your proxy images after installing a license.

PLEASE NOTE

Do not edit the license files provided in any way. The licenser searches all the files in the License directory to find valid licenses, so more than one license file can be added to the directory without difficulties arising.

1.6 Support

In the event of any difficulties with the installation or the software please contact support@speedsix.com

1.7 Sales

For sales enquiries contact sales@speedsix.com, or see www.speedsix.com. You will need to contact your reseller or SpeedSix sales to obtain a license.

1.8 Thanks!

Thanks for using SpeedSix software!

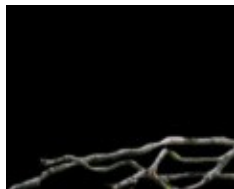
2 SpeedSix.Fire (Discreet Fluidz Raptor)

PURPOSE

Simulates flames in two dimensions using a fluid flow approach (a flame is basically a hot fluid – so hot it glows). The flames are driven primarily by buoyancy when there is gravity around, hot air (which is less dense than cold air) gets pushed up as the cold air sinks. The simulation deals with temperature and density. You can map the different temperatures to colours of your choice.



INPUT CLIPS



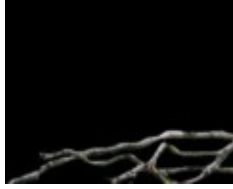
Example : Fuel

1: Fuel : Specifies the location and relative temperature of heat sources Black is cold and white is hot. The heat from the heat sources in each frame of this clip gets added in to the simulation at each frame (so heat builds up around the sources). This heat is the main driving force for the simulation so it is well worth experimenting with the input clip to see how it affects the generated fire. It is highly recommended that you work through the **Fire** tutorials available on the Speedsix web site.



Example : BackGround

2: BackGround : The simulated flames will be composited over this clip.



Example : Barrier

3: Barrier : Barrier clip. Lets you specify where there are solid objects that the flame will flow around. White denotes the presence of an object. Please note that at a given point there is either part of a solid object or there isn't. The resolution at which "objects" are defined is the resolution of the velocity simulation grid. This is independent of the image resolution being used and is set to 128 x 128. This will inevitably give rise to "jagged" edges around solid objects unless they align perfectly with the grid. A way of dealing with these is to composite full resolution versions of the solid objects over the result.

Fire CONTROL PAGE

Restart (Pushbutton)

Restarts the flame simulation. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random. You must render a complete sequence in order from start to end. This makes it unsuitable for use in some background rendering systems.

PreMultiplyForeground (Checkbox **Default:** Off)

Have the colours in the foreground fire premultiplied by the fire density before compositing. The simulated density usually makes a very effective alpha channel.

SmokeFader (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.2)

Reduces density with time. This has a visible effect mostly in the older and colder regions – which can appear like smoke with a suitable choice of colours. So it fades out the smokier regions with time. The higher this value the more quickly things fade away.

Cold Transparency (Number **Min:** 0.0, **Max:** 21.0, **Default:** 0.0)

This normalized gas density in the simulation will be fully transparent when the flame is composited over the background.

Hot Transparency (Number **Min:** 0.0, **Max:** 21.0, **Default:** 1.0)

This normalized gas density in the simulation will be fully opaque when the flame is composited over the background.

Smoothness (Number **Min:** 5, **Max:** 50, **Default:** 5)

Increasing this will reduce the "scaloping" effects that can sometimes be seen when very thin, hot, regions are moving quickly. It will, however, make the simulation slower.

Speed (Number **Min:** 0.1, **Max:** 10.0, **Default:** 1.0)

Relative length of the simulation time step between frames. To speed up the motion make this

bigger. To slow it down, make it smaller. Best to keep it near 1.0 usually.

Flicker (Number Min: 0.0, Max: 10.0, Default: 0.5)

Increasing this will have the visual effect of making the flames flicker and swirl more DLPage
PageOne

SmokeWeight (Number Min: 0.0, Max: 10.0, Default: 0.0)

Relative density of the gas for a fixed temperature. The bigger this is the faster cold gas (i.e. smoke) will fall.

FlameHeight (Number Min: 0.0, Max: 10.0, Default: 1.0)

How quickly the density falls with increasing temperature. The bigger this is, the more quickly the flames will tend to rise. It will also tend to stretch the flames making them thinner.

TempDiss (Number Min: 0.0, Max: 1.0, Default: 0.5)

Temperature dissipation rate. How quickly heat is lost. The bigger this is, the quicker hot gas will cool.

FuelAmount (Number Min: 0.0, Max: 1.0, Default: 1.0)

We feed the fire by adding in the heat from the **Fuel Clip** to add to the temperature and density of the gas. This control sets how hot the **Fuel Clip** is. As this is increased, a given grey level represents a higher temperature and density.

Output Matte (Checkbox Default: Off)

Output the alpha channel rather than the full colour image.

Use Barriers (Checkbox Default: Off)

Use the **Barrier Clip** to define solid objects around which the flames should flow. White denotes the presence of an object. It is strongly recommended that to get anti-aliased flame edges near the solid objects, images of those objects should be composited over the output flame image.

BoundaryLayer (List Box Options: Slippery | Rough | Sticky, Default: Slippery)

Barrier object surface characteristics which determine how the flames flow over a surface.

Demos (List Box Options: Base | More Smoke | Wilder | Calm Smoke | Extreme | Bold Smoke | Smooth Dramatic | Lapping Fast | Lapping Slow | Smokey, Default: Base)

Select a preset to get you going. The simulation is controlled by a large number parameters, many of which interact. Starting from a preset will help to get appropriate results more quickly.

WandV CONTROL PAGE

Restart (Pushbutton)

Restarts the flame simulation. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random. You must render a complete sequence in order from start to end. This makes it unsuitable for use in some background rendering systems.

Wind 1 Local (Checkbox **Default:** On)

Make Wind 1 local. It will die away over the distance from Start to End.

Use Wind 1 (Checkbox **Default:** Off)

Turn on-off the first wind force. Values set will not be lost when turned off, only ignored. Wind forces can be used to blow the flames about.

Wind 1 Start (Position **Default:** 0.0,0.0)

Position of the start of Wind 1.

Wind 1 End (Position **Default:** 0.5,0.5)

Position of the end of Wind 1.

Wind 1 Strength (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.05)

The strength of Wind 1.

Wind 1 Tight (Number **Min:** 0.0, **Max:** 200.0, **Default:** 1.0)

The directional tightness of Wind 1.

Use Vortex 1 (Checkbox **Default:** Off)

Turn on-off the first vortex force. Values set will not be lost when turned off, only ignored. A vortex force will make the gas swirl locally. They are rarely useful for flames.

Vortex 1 Center (Position **Default:** 0.333 0.333)

Position of the center of Vortex 1.

Vortex 1 Radius (Number **Min:** 0.001, **Max:** 1.0, **Default:** 0.1)

The radius of Vortex 1.

Vortex 1 Strength (Number **Min:** -1.0, **Max:** 1.0, **Default:** 0.1)

The strength of Vortex 1.

***Colour* CONTROL PAGE**

Restart (Pushbutton)

Restarts the flame simulation. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random. You must render a complete sequence in order from start to end. This makes it unsuitable for use in some background rendering systems.

AdjustColoursOnly (Checkbox **Default:** Off)

Adjust only colours. Do not advance the timestep. Note: This will be automatically turned off when **Restart** is called. This lets you control the colour of the flame (based on its temperature) without affecting its development and motion.

HotBase (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

See **ColdBase**.

ColdBase (Number **Min:** 0.0, **Max:** 10.0, **Default:** 1.0)

HotBase and **ColdBase** map the range of the internal "image" used to represent the temperature to the range 0.0 to 1.0. You can use this to stretch out a small internal temperature range to a large variation in flame colour or, conversely, compress a large internal temperature range to a small colour range. The value entered for **ColdBase** is transformed to 0.0 for the value used as input to the temperature to colour mapping process. The value entered for **HotBase** is transformed to 1.0. The colour map you specify (see below) is defined in terms of mapping this 0.0 to 1.0 range to specific colours. We will refer to the 0.0 to 1.0 range as the *normalized temperature* below. This can be used to control whether you have flames without smoke or smoke without flames.

Cold Colour (Colour Box **Default:** R: 0, G: 0, B: 0, A: 65535)

The colour that will be used for the normalized temperature set in the **Cold** control.

Warm Colour (Colour Box **Default:** R: 8320, G: 2816, B: 2048, A: 65535)

The colour that will be used for the normalized temperature set in the **Warm** control.

Warmer Colour (Colour Box **Default:** R: 65535, G: 100, B: 100, A: 65535)

The colour that will be used for the normalized temperature set in the **Warmer** control.

Hot Colour (Colour Box **Default:** R: 65535, G: 32214, B: 100, A: 65535)

The colour that will be used for the normalized temperature set in the **Hot** control.

Hotter Colour (Colour Box **Default:** R: 65535, G: 65535, B: 100, A: 65535)

The colour that will be used for the normalized temperature set in the **Hotter** control.

Hottest Colour (Colour Box **Default:** R: 65535, G: 65535, B: 65535, A: 65535)

The colour that will be used for the normalized temperature set in the **Hottest** control.

Cold (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

The normalized temperature corresponding the the **Cold Colour**.

Warm (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.2)

The normalized temperature corresponding the the **Warm Colour**.

Warmer (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.4)

The normalized temperature corresponding the the **Warmer Colour**.

Hot (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.6)

The normalized temperature corresponding the the **Hot Colour**.

Hotter (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.8)

The normalized temperature corresponding the the **Hotter Colour**.

Hottest (Number **Min:** 0.0, **Max:** 1.0, **Default:** 1.0)

The normalized temperature corresponding the the **Hottest Colour**.

3 SpeedSix.ImageFlow (Discreet Fluidz Raptor)

PURPOSE

Treats a single image as though it were made of coloured liquid or gas and outputs a result clip showing that fluid flowing over time. The fluid flow is simulated by physically based two dimensional fluid flow calculations resulting in natural flow effects. Reverse the clip and the single image appears out of the swirling flow.



INPUT CLIPS

1: Flow Clip : Clip to apply the flow effect to. The first image in the clip is used as the one that will flow – or the first encountered after the **Restart** button is pressed. Other frames in the clip may be used as "sources" using the **Self Source** option, but are otherwise ignored.

2: Background : The flowed image will be composited over this clip.

3: Alt Clip : A second input, which can be used for several purposes – e.g. to supply relative density information for buoyancy forces. In this case the luminosity of the Alternative clip will influence the buoyancy of the source clip.

4: Sources Clip : Sources clip. Contents may be added to the flow at each frame. Without a source clip the image clip will flow away. You can use the source clip to replace the fluid so that the image decays away more slowly or remains in the image leaking fluid (for example as a smoking logo). Care has to be taken not to overdo the source on a stationary image or it may go white.

5: Sinks Clip : Sinks clip. Contents may be subtracted from the flow at each frame. This is not a widely useful as the sources clip but can be used to create pseudo plug-holes etc.

6: Barrier Clip : Barrier clip. Lets you specify where there are solid objects in the flow that the fluid will bounce off or flow around. White denotes the presence of an object. Please note that at a given point there is either part of a solid object or there isn't. This will inevitably give "jagged" edges around solid objects. A way to avoid this is to composite full resolution versions of the solid objects over the result images. To increase the resolution of the barrier you can select the BFinerGrid option.

ImageFlow CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Viscosity (Number **Min:** 0.00001, **Max:** 1.0, **Default:** 0.01)

Viscosity of the liquid. The higher the viscosity, the more reluctant the fluid is to flow and the quicker any flow comes to a halt in the absence of applied forces.

Swirl (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

The higher this control, the more pronounced swirls (vortices) will be and the more "detailed" the flow will seem.

Buoyancy Mode (List Box **Options:** None | Input Luma | Alt Luma | Pressure Input | Pressure Alt, **Default:** Input Luma)

Whether to calculate internal buoyancy forces, and, if so, how. **Input Luma** will use the luminance of the input image to calculate the relative density of the fluid. **Alt Luma** uses the luminance of the **Alternative** image instead. For example it is possible to use varying luminosity in the BAlternative clip to create specific motion in the input clip. The Pressure controls flow the image in a different manner.

RiseFall (Number **Min:** -10.0, **Max:** 10.0, **Default:** 1.0)

Tendency to rise (positive) or fall (negative).

Pressure Force (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

Strength of a pressure force derived from the local density gradient. This force makes dense regions expand into surrounding less dense regions.

Restart On Process (Checkbox **Default:** Off)

If available turn **On** immediately before processing to ensure that the first frame is taken from the clip with no flowing affecting it.

Random Force (List Box **Options:** Type 1 | Type 2, **Default:** Type 1)

There are two separate modes provided for generating random forces in the flow. The BType 2 option is a good approximation of real world turbulence that introduces complex "eddies" into the flow.

InitialRandom (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Initial random velocities added to the force field. This is not animatable as it is only used once at restart.

EachRandom (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Random velocities added each frame, which can sustain or grow turbulence.

Eddy Size (Number **Min:** 2.0, **Max:** 30.0, **Default:** 16.0)

Determines the number and size of "eddies" induced in Type 2 turbulent flow. The larger the value, the smaller the "eddies" will be.

Finer Grid (Checkbox **Default:** Off)

Turn on/off the higher grid resolution option. **On** Uses a grid of 256x256. This is mainly useful for increasing the resolution of any barriers used, however processing speed will be slower.

Off Uses the speedier grid resolution of 128x128.

Edge Escape (Checkbox **Default:** On)

Turn on/off the edge confinement force. **On** and the flow will escape into the ether. **Off** and the flow will bounce back off the edges of the image.

Use Dissipation (Checkbox **Default:** Off)

Turn on/off dissipation. With dissipation **On**, the liquid will become "thinner" and more transparent (over a background) as time goes on.

Dissipation Frames (Number **Min:** 2, **Max:** 1000, **Default:** 50)

Frames over which to dissipate to the specified **Percent Left** (see below).

Percent Left (Number **Min:** 0.01, **Max:** 99.99, **Default:** 1.0)

Percent of the intensity and density that is to be left after **Dissipation Frames** frames.

Use Diffusion (Checkbox **Default:** Off)

Turn on-off diffusion. With diffusion **On**, coloured fluids will spread out as time goes on.

Diffusion (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Diffusion constant for the fluid (if **Use Diffusion** is **On**). **Diffusion** controls the rate at which the fluid spreads out as time passes.

Output (List Box **Options:** Result | Result Over Background | Matte | Flowed At Grid Res | Grid | Barriers, **Default:** Result)

What to output.

Result is the flowed image.

Result Over BackGround is the flowed image composited over the background clip.

Matte Matte of the flowed image.

Flowed at Grid Res The output resampled to the grid resolution.

Grid A representaion of the grid.

Matte The barrier image. If the barriers are not active the barrier input clip is displayed.

If the barriers are active and a **Restart** has been asked for the current barrier clip is shown.

PreMultiplyForeground (Checkbox **Default:** Off)

Have the colours in the foreground premultiplied by the alpha before compositing.

Forces1 CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Use Barriers (Checkbox **Default:** Off)

Use the barrier image to define solid objects in the flow around which the image should flow. White denotes the presence of an object. The solid objects are there (or not) at the resolution of the grid. The jagged edges of the grid can be reduced by either aligning the barrier with the grid or selecting the BFinerGrid. It is strongly recommended that to get anti-aliased edges for solid objects, images of those objects are composited over the flow result image.

BoundaryLayer (List Box **Options:** Slippery | Rough | Sticky, **Default:** Slippery)

Barrier object surface characteristics.

Use Gravity (Checkbox **Default:** Off)

Turn on/off gravity.

GravForce (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

Strength of the gravitational force (which acts on density and is pointing in a downwards direction if the **GravAngle** is 0.0).

Gravity Direction (Number **Min:** -360, **Max:** 360, **Default:** 270)

Rotates the direction of the gravity force vector (at 0 rotation, the vector is (1,0)).

Use Velocity A (Checkbox **Default:** Off)

Turn on/off the velocity force. The velocity "force" is a constant velocity, defined to exist within a user defined rectangular region. It is very useful for inducing constant flows and "jets" in the fluid. The velocity is a unit vector in the X direction. The direction can be adjusted by the **Velocity Direction** control. The magnitude can be adjusted by the **VelAdd** control.

Centre A (Position **Default:** 0.02,0.5)

Center of the rectangular region in which the velocity force acts.

Scale A X (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Width of the rectangular region in which the velocity force acts.

Scale A Y (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Height of the rectangular region in which the velocity force acts.

Velocity A Direction (Number Min: -360, Max: 360, Default: 0)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Vel Add A (Number Min: 0.0, Max: 1.0, Default: 0.5)

Scales the magnitude of the velocity force vector.

Use Velocity B (Checkbox Default: Off)

Turn on/off the velocity force. The velocity "force" is a constant velocity, defined to exist within a user defined rectangular region. It is very useful for inducing constant flows and "jets" in the fluid. The velocity is a unit vector in the X direction. The direction can be adjusted by the **Velocity Direction** control. The magnitude can be adjusted by the **VelAdd** control.

Centre B (Position Default: 0.62,0.2)

Center of the rectangular region in which the velocity force acts.

Scale A B (Number Min: 0.01, Max: 2.00, Default: 0.08)

Width of the rectangular region in which the velocity force acts.

Scale B Y (Number Min: 0.01, Max: 2.00, Default: 0.08)

Height of the rectangular region in which the velocity force acts.

Velocity B Direction (Number Min: -360, Max: 360, Default: 270.0)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Vel Add B (Number Min: 0.0, Max: 1.0, Default: 0.5)

Scales the magnitude of the velocity force vector.

Remove Net Force (Checkbox Default: Off)

When **Off** for every frame the active forces are added to the current state of the force field pushing the image about. Removing the "Net Force" only applies the currently calculated forces to the motion so the fluid will tend to stay in the image and not flow off. If random forces you get static turbulent flow.

Forces2 CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Wind 1 Local (Checkbox Default: On)

Make Wind 1 local. It will die away over the distance from Start to End.

Source Mode (List Box **Options:** No Source | Self Source | Source Clip, **Default:** No Source)

Which image to use (if any) to define "sources". Sources are additional fluids that are poured into the flow. They can be very important, because, without wrapping, the fluid will tend to flow out of the image, so after a while it will all be gone unless more fluid is added. **No Source** doesn't use any sources. **Source Clip** uses the **Source** image, and **Self Source** uses the input clip itself as the source.

Source Strength (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

How much of any source image gets poured into the flow at each frame.

Sink Mode (List Box **Options:** No Sink | Self Sink | Sink Clip, **Default:** No Sink)

Which image to use (if any) to define "sinks". Sinks are where the flowing fluid pours out of the image and vanishes from consideration. They are generally less important than sources. A scaled fraction of the sink image is subtracted out of the flow at each frame. If you want stuff to vanish down a plughole, you could use a white blob. Note that the sink doesn't generate any appropriate forces; you would need to add a vortex at the same location to get fluid to swirl down the plughole. **No Sink** doesn't use any sinks. **Sink Clip** uses the **Sink** image, and **Self Sink** uses the input clip itself as the sink – which would usually be pretty weird!

Sink Strength (Number **Min:** -1.0, **Max:** 1.0, **Default:** 0.0)

How much of any sink image gets sucked out of the flow at each frame.

Use Wind 1 (Checkbox **Default:** Off)

Turn on/off the first wind force. Values set will not be lost when turned off, only ignored.

Wind 1 Start (Position **Default:** 0.0,0.0)

Position of the start of Wind 1.

Wind 1 End (Position **Default:** 0.5,0.5)

Position of the end of Wind 1.

Wind 1 Strength (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

The strength of Wind 1.

Wind 1 Tight (Number **Min:** 0.0, **Max:** 200.0, **Default:** 1.0)

The directional tightness of Wind 1.

Use Vortex1 (Checkbox **Default:** Off)

Turn on/off the first vortex force. Values set will not be lost when turned off, only ignored. A vortex force induces a rotational motion in the fluid.

Vortex 1 Center (Position **Default:** 0.333 0.333)

Position of the center of Vortex 1.

Vortex 1 Radius (Number **Min:** 0.001, **Max:** 2.0, **Default:** 0.1)

The radius of Vortex 1.

Vortex 1 Strength (Number **Min:** -1.0, **Max:** 1.0, **Default:** 0.1)

The strength of Vortex 1.

4 SpeedSix.ClipFlow (Discreet Fluidz Raptor)

PURPOSE

Treats a clip as though it were made of coloured liquid or gas and outputs a result clip showing that fluid flowing over time. The fluid flow is simulated by physically based two dimensional fluid flow calculations resulting in natural flow effects. Because a new image from the clip is used each frame the motion and effect created will differ from the results of "ImageFlow".



INPUT CLIPS

1: Flow Clip : Clip to apply flow to.

2: Background : The flowed clip will be composited over this clip.

3: Alt Clip : A second input, which can be used for several purposes – e.g. to supply relative density information for buoyancy forces. In this case the luminosity of the Alternative clip will influence the buoyancy of the source clip.

4: Barrier Clip : Barrier clip. Lets you specify where there are solid objects in the flow that the fluid will bounce off or flow around. White denotes the presence of an object. Please note that at a given point there is either part of a solid object or there isn't. This will inevitably give "jagged" edges around solid objects. A way to avoid this is to composite full resolution versions of the solid objects over the result images. To increase the resolution of the barrier you can select the BFinerGrid option.

ClipFlow CONTROL PAGE

Restart ([Pushbutton](#))

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate frames at different randomly chosen resolutions. No matter how much various compositing systems want to do these things,

Viscosity (Number **Min:** 0.000001, **Max:** 1.0, **Default:** 0.01)

Viscosity of the fluid. The higher the viscosity, the more reluctant the fluid is to flow and the quicker any flow comes to a halt in the absence of applied forces.

Swirl (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

The higher this control, the more pronounced swirls (vortices) will be and the more "detailed" the flow will seem.

Buoyancy Mode (List Box **Options:** None | Input Luma | Aux Luma | Pressure Input | Pressure Aux, **Default:** Input Luma)

Whether to calculate internal buoyancy forces, and, if so, how. **Input Luma** will use the luminance of the input image to calculate the relative density of the fluid. **Alt Luma** uses the luminance of the **Alternative** image instead. For example it is possible to use varying luminosity in the **BA**lternative clip to create specific motion in the input clip. The Pressure controls flow the image in a different manner.

RiseFall (Number **Min:** -10.0, **Max:** 10.0, **Default:** 1.0)

Tendency to rise (positive) or fall (negative).

Pressure Force (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Strength of pressure force derived from the local density gradient. This force makes dense regions expand into surrounding less dense regions.

Restart On Process (Checkbox **Default:** Off)

If available turn **On** immediately before processing to ensure that the first frame is taken from the clip with no flowing affecting it.

Random Force (List Box **Options:** Type 1 | Type 2, **Default:** Type 1)

There are two separate modes provided for generating random forces in the flow. The BType 2 option is a good approximation of real world turbulence that introduces complex "eddies" into the flow.

InitialRandom (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Initial random velocities added to the force field. This is not animatable as it is only used once at restart.

EachRandom (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Random velocities added each frame, which can sustain or grow turbulence.

Eddy Size (Number **Min:** 2.0, **Max:** 30.0, **Default:** 16.0)

Determines the number and size of "eddies" induced in Type 2 turbulent flow. The larger the value, the smaller the "eddies" will be.

Finer Grid (Checkbox **Default:** Off)

Turn on/off the higher grid resolution option. **On** Uses a grid of 256x256. This is mainly useful for

increasing the resolution of any barriers used, however processing speed will be slower.

Off Uses the speedier grid resolution of 128x128.

Edge Escape (Checkbox **Default:** On)

Turn on/off the edge confinement force. **On** and the flow will escape into the ether. **BOff** and the flow will bounce back off the edges of the image.

Output (List Box **Options:** Result | Result Over Background | Matte | Flowed At Grid Res | Grid | Barriers, **Default:** Result)

What to output.

Result is the flowed image.

Result Over BackGround is the flowed image composited over the background clip.

Matte Matte of the flowed image.

Flowed at Grid Res The output resampled to the grid resolution.

Grid A representaion of the grid.

Matte The barrier image. If the barriers are not active the barrier input clip is displayed.

If the barriers are active and a **Restart** has been asked for the current barrier clip is shown.

PreMultiplyForeground (Checkbox **Default:** Off)

Have the colours in the foreground premultiplied by the alpha before compositing.

Forces1 CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Use Barriers (Checkbox **Default:** Off)

Use the barrier image to define solid objects in the flow around which the image should flow. White denotes the presence of an object. The solid objects are there (or not) at the resolution of the grid. The jagged edges of the grid can be reduced by either aligning the barrier with the grid or slecting the BFinerGrid. It is strongly recommended that to get anti-aliased edges for solid objects, images of those objects are composited over the flow result image.

BoundaryLayer (List Box **Options:** Slippery | Rough | Sticky, **Default:** Slippery)

Barrier object surface characteristics.

Use Gravity (Checkbox **Default:** Off)

Turn on/off gravity.

GravForce (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

Strength of the gravitational force (which acts on density and is pointing in a downwards direction if the **GravAngle** is 0.0).

Gravity Direction (Number **Min:** -360, **Max:** 360, **Default:** 270)

Rotates the direction of the gravity force vector (at 0 rotation, the vector is (1,0)).

Use Velocity A (Checkbox **Default:** Off)

Turn on/off the velocity force. The velocity "force" is a constant velocity, defined to exist within a user defined rectangular region. It is very useful for inducing constant flows and "jets" in the fluid. The velocity is a unit vector in the X direction. The direction can be adjusted by the **Velocity Direction** control. The magnitude can be adjusted by the **VelAdd** control.

Centre A (Position **Default:** 0.02,0.5)

Center of the rectangular region in which the velocity force acts.

Scale A X (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Width of the rectangular region in which the velocity force acts.

Scale A Y (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Height of the rectangular region in which the velocity force acts.

Velocity A Direction (Number **Min:** -360, **Max:** 360, **Default:** 0)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Vel Add A (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.5)

Scales the magnitude of the velocity force vector.

Use Velocity B (Checkbox **Default:** Off)

Turn on/off the velocity force. The velocity "force" is a constant velocity, defined to exist within a user defined rectangular region. It is very useful for inducing constant flows and "jets" in the fluid. The velocity is a unit vector in the X direction. The direction can be adjusted by the **Velocity Direction** control. The magnitude can be adjusted by the **VelAdd** control.

Centre B (Position **Default:** 0.62,0.2)

Center of the rectangular region in which the velocity force acts.

Scale A B (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Width of the rectangular region in which the velocity force acts.

Scale B Y (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Height of the rectangular region in which the velocity force acts.

Velocity B Direction (Number **Min:** -360, **Max:** 360, **Default:** 270.0)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Vel Add B (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.5)

Scales the magnitude of the velocity force vector.

Remove Net Force (Checkbox **Default:** Off)

When **Off** for every frame the active forces are added to the current state of the force field pushing the image about. Removing the "Net Force" only applies the currently calculated forces to the motion so the fluid will tend to stay in the image and not flow off. If random forces you get static turbulent flow.

***Forces2* CONTROL PAGE**

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Wind 1 Local (Checkbox **Default:** On)

Make Wind 1 local. It will die away over the distance from Start to End.

Use Wind1 (Checkbox **Default:** Off)

Turn on/off the first wind controls. Values set will not be lost when turned off, only ignored.

Wind 1 Start (Position **Default:** 0.0,0.0)

Position of the start of Wind 1.

Wind 1 End (Position **Default:** 0.5,0.5)

Position of the end of Wind 1.

Wind 1 Strength (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

The strength of Wind 1.

Wind 1 Tight (Number **Min:** 0.0, **Max:** 200.0, **Default:** 1.0)

The directional tightness of Wind 1.

Use Vortex1 (Checkbox **Default:** Off)

Turn on/off the first vortex controls. Values set will not be lost when turned off, only ignored.

Vortex 1 Center (Position **Default:** 0.333 0.333)

Position of the center of Vortex 1.

Vortex 1 Radius (Number **Min:** 0.001, **Max:** 1.0, **Default:** 0.1)

The radius of Vortex 1.

Vortex 1 Strength (Number **Min:** -1.0, **Max:** 1.0, **Default:** 0.1)

The strength of Vortex 1.

5 SpeedSix.TransFlow (Discreet Fluidz Raptor)

PURPOSE

The start clip flows revealing the end clip.



INPUT CLIPS

1: Start Clip : Clip to apply flow to and be replaced by the End Clip.

2: End Clip : This clip will be revealed at the end of the transition.

TransFlow CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Viscosity (Number **Min:** 0.001, **Max:** 100.0, **Default:** 0.01)

Viscosity of the fluid. The higher the viscosity, the more reluctant the fluid is to flow and the quicker any flow comes to a halt in the absence of applied forces.

Swirl (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

The higher this control, the more pronounced swirls (vortices) will be and the more "detailed" the flow will seem.

Buoyancy Mode (List Box **Options:** None | Input Luma | Pressure Input, **Default:** Input Luma)

Whether to calculate internal buoyancy forces, and, if so, how. **Input Luma** will use the luminance of the input image to calculate the relative density of the fluid and the

Rise Fall control governs the rate. **Pressure Input** uses the input luminance as a pressure force controlled by **Pressure Force**.

RiseFall (Number **Min:** -10.0, **Max:** 10.0, **Default:** 5.0)

Tendency to rise (positive) or fall (negative) when **Input Luma** is used as the **Buoyancy Mode**.

Pressure Force (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Strength of pressure force derived from density gradient of the input clip.

Restart On Process (Checkbox **Default:** Off)

If available turn **On** immediately before processing to ensure that the first frame is taken from the clip with no flowing affecting it.

Random Mode (List Box **Options:** Type 1 | Type 2, **Default:** Type 1)

There are two separate modes provided for generating random forces in the flow.
The BType 2 option is a good approximation of real world turbulence that introduces complex "eddies" into the flow.

InitialRandom (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Initial random velocities added at the start only to push the image.

EachRandom (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.0)

Random velocities added each frame to augment the motion.

Eddy Size (Number **Min:** 2.0, **Max:** 30.0, **Default:** 16.0)

Determines the number and size of "eddies" induced in Type 2 turbulent flow. The larger the value, the smaller the "eddies" will be.

Finer Grid (Checkbox **Default:** Off)

Turn on/off the higher grid resolution option. **On** Uses a grid of 256x256. This is mainly useful for increasing the resolution of any barriers used, however processing speed will be slower.

Off Uses the speedier grid resolution of 128x128.

Transition Length (Number **Min:** 2, **Max:** 250, **Default:** 25)

Number of frames over which the transition takes place.

Transition Start (Number **Min:** 0, **Max:** 500, **Default:** 5)

The transition will start from this frame.

Fade (Checkbox **Default:** On)

Turn on/off the fade out function. This ensures the Start clip will disappear by the end of the transition.

Fade Length (Number **Min:** 2, **Max:** 250, **Default:** 10)

How long the fade lasts. The fade will start at the chosen number of frames before the end of the transition.

Slip (Checkbox **Default:** Off)

Turn on/off to display and use the Slip controls.

Slip Start (Number **Min:** -500, **Max:** 500, **Default:** 0)

Slip the start clip to the desired starting frame.

Slip End (Number **Min:** -500, **Max:** 500, **Default:** 0)

Slip the end clip to start the transition from this frame.

Output (List Box **Options:** Result | Result Over Background | Matte | Flowed At Grid Res | Grid, **Default:** Result Over Background)

What to output.

Result is the flowed image.

Result Over BackGround is the flowed image composited over the background clip.

Matte Matte of the flowed image.

Flowed at Grid Res The output resampled to the grid resolution.

Grid A representaion of the grid.

PreMultiplyForeground (Checkbox **Default:** Off)

Have the colours in the foreground clip premultiplied by its alpha (density) before compositing.

Forces1 CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Use Gravity (Checkbox **Default:** Off)

Turn on/off gravity.

GravForce (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

Strength of the gravitational force (which acts on density and is pointing in a downwards direction if the **GravAngle** is 0.0).

Gravity Direction (Number **Min:** -360, **Max:** 360, **Default:** 270)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Use Velocity A (Checkbox **Default:** Off)

Turn on/off the velocity force. The velocity "force" is a constant velocity, defined to exist within a user defined rectangular region. It is very useful for inducing constant flows and "jets" in the fluid. The velocity is a unit vector in the X direction. The direction can be adjusted by the **Velocity**

Direction control. The magnitude can be adjusted by the **VelAdd** control.

Centre A (Position **Default:** 0.02,0.5)

Center of the rectangular region in which the velocity force acts.

Scale A X (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Width of the rectangular region in which the velocity force acts.

Scale A Y (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Height of the rectangular region in which the velocity force acts.

Velocity A Direction (Number **Min:** -360, **Max:** 360, **Default:** 0)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Vel Add A (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.5)

Scales the magnitude of the velocity force vector.

Use Velocity B (Checkbox **Default:** Off)

Turn on/off the velocity force. The velocity "force" is a constant velocity, defined to exist within a user defined rectangular region. It is very useful for inducing constant flows and "jets" in the fluid. The velocity is a unit vector in the X direction. The direction can be adjusted by the **Velocity Direction** control. The magnitude can be adjusted by the **VelAdd** control.

Centre B (Position **Default:** 0.62,0.2)

Center of the rectangular region in which the velocity force acts.

Scale A B (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Width of the rectangular region in which the velocity force acts.

Scale B Y (Number **Min:** 0.01, **Max:** 2.00, **Default:** 0.08)

Height of the rectangular region in which the velocity force acts.

Velocity B Direction (Number **Min:** -360, **Max:** 360, **Default:** 270.0)

Rotates the direction of the velocity force vector (at 0 rotation, the vector is (1,0)).

Vel Add B (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.5)

Scales the magnitude of the velocity force vector.

Remove Net Force (Checkbox **Default:** Off)

When **Off** for every frame the active forces are added to the current state of the force field pushing the image about. Removing the "Net Force" only applies the currently calculated forces to the motion so the fluid will tend to stay in the image and not flow off. If random forces you get static turbulent flow.

Forces2 CONTROL PAGE

Restart (Pushbutton)

Restarts the flow. Please note that this is a physical simulation, the whole point of which is that it evolves over time. It isn't possible to independently calculate frames at random, nor is it at all straightforward to calculate different frames at different randomly chosen resolutions. This can make this effect unsuitable for use in some background rendering systems.

Wind 1 Local (Checkbox **Default:** On)

Make Wind 1 local. It will die away over the distance from Start to End.

Use Wind1 (Checkbox **Default:** Off)

Turn on-off the first wind controls. Values set will not be lost when turned off, only ignored.

Wind 1 Start (Position **Default:** 0.0,0.0)

Position of the start of Wind 1.

Wind 1 End (Position **Default:** 0.5,0.5)

Position of the end of Wind 1.

Wind 1 Strength (Number **Min:** 0.0, **Max:** 1.0, **Default:** 0.1)

The strength of Wind 1.

Wind 1 Tight (Number **Min:** 0.0, **Max:** 200.0, **Default:** 1.0)

The directional tightness of Wind 1.

Use Vortex1 (Checkbox **Default:** Off)

Turn on-off the first vortex controls. Values set will not be lost when turned off, only ignored.

Vortex 1 Center (Position **Default:** 0.333 0.333)

Position of the center of Vortex 1.

Vortex 1 Radius (Number **Min:** 0.001, **Max:** 1.0, **Default:** 0.1)

The radius of Vortex 1.

Vortex 1 Strength (Number **Min:** -1.0, **Max:** 1.0, **Default:** 0.1)

The strength of Vortex 1.