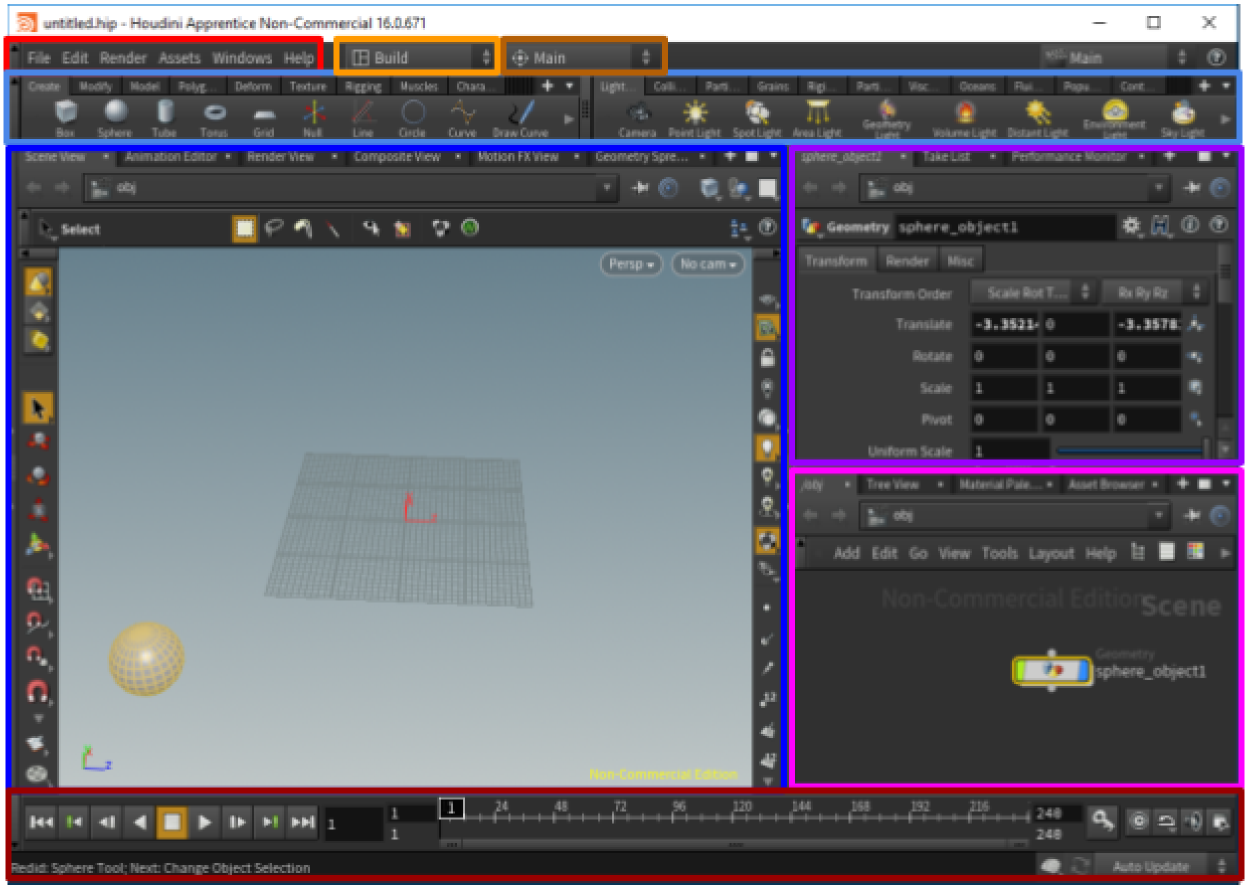
Houdini 16

Basic Interface


untitled.hip - Houdini Apprentice Non-Commercial 16.0.671
File Edit Render Assets Windows Help
Build
Main
Create Modify Model Polyg... Deform Texture Rigging Muscles Chara...
Box Sphere Tube Torus Grid Null Line Circle Curve Draw Curve
Light... Colli... Parti... Grains Rigi... Parti... Visc... Oceans Flui... Popu... Cont...
Camera Point Light Spot Light Area Light Geometry Light Volume Light Distant Light Environment Light Sky Light
Scene View Animation Editor Render View Composite View Motion FX View Geometry Spre...
obj
Select
Persp No cam
Non-Commercial Edition
sphere_object1 Take List Performance Monitor
Geometry sphere_object1
Transform Render Misc
Transform Order Scale Rot T... Rx Ry Rz
Translate -3.3521 0 -3.3578
Rotate 0 0 0
Scale 1 1 1
Pivot 0 0 0
Uniform Scale 1
/obj Tree View Material Pale... Asset Browser
Add Edit Go View Tools Layout Help
Non-Commercial Edition Scene
Geometry
sphere_object1
1 24 48 72 96 120 144 168 192 216 240
240
Redid: Sphere Tool; Next: Change Object Selection
Auto Update

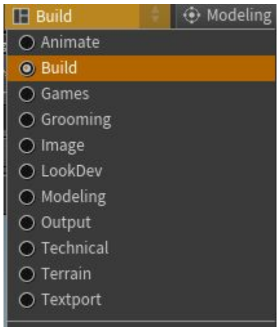
NOTE: The interface shown above is for the Build desktop. It's the default desktop. There are many different types of desktops you can switch between.

Main Menu

Main menu is always there and provides ways to access the basics. Unlike Maya, there doesn't seem to be menusets available. These menus are constant.

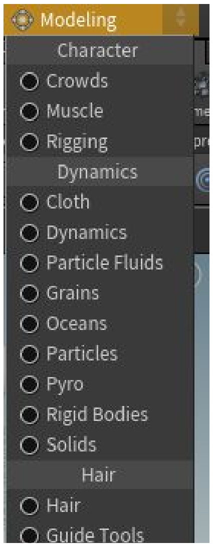
Desktop

Desktops are essentially layouts. They're similar to workspaces in Maya 2017. The Build layout seems to be the default -- this is probably similar to the Maya Classic workspace.



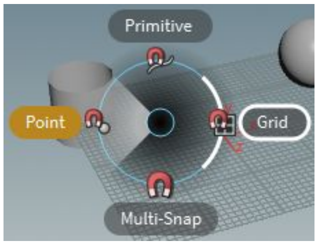
Radial Menu

Radial menus are Houdini's version of Maya's marking menus. You can select what type of work you're doing via this dropdown, and then press C in the viewport (and maybe other places as well?) to show up common features.

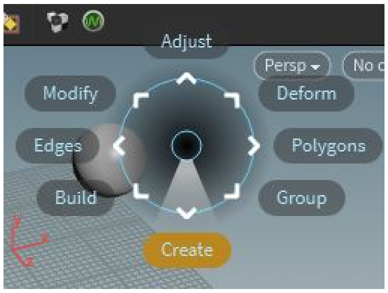


Holding…

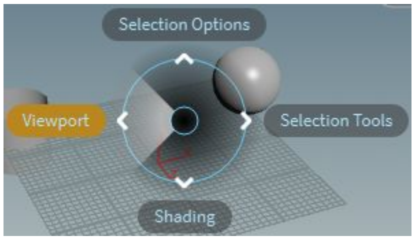
X opens snapping functions



C opens functions for what you selected in this dropdown



V opens viewport functions



HINT: You can also hit these keys instead of holding -- hitting will keep them open until you hit the key again.

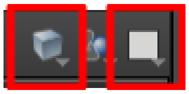
Shelves

Similar to Maya shelves. Contains commonly used things. Note that unlike Maya, there are actually 2 shelf sets here.

Scene View

This is like Maya's main viewport window. You can select objects, move them around, etc…

HINT: The buttons right are the ones that'll let you have multiple view ports and change your viewport views (e.g. enable shaded + wireframe).



Parameter View

This is like Maya's channel view / attribute panel. You can manipulate attributes of whatever the it is you have selected in the viewport / network.

Network View

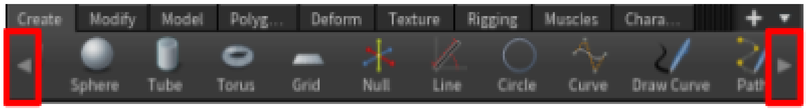
This is like Maya's node editor. When you select something in here, it'll also get selected in the viewport (and vice versa).

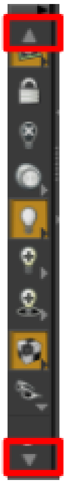
Animation

This is like Maya's animation tools.

Toolbar/Shelf Offscreen Icons

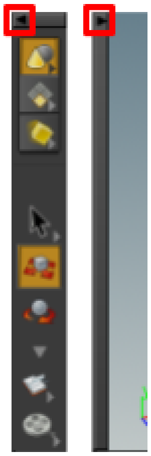
If the Houdini window isn't big enough, icons in the various toolbars/shelves may be missing. You can click the arrows or use the mouse scroll wheel (when hovered over the toolbar/shelf) to expose off screen buttons/icons in the toolbars and shelves.





Toolbar/Shelf/Menu Manipulation

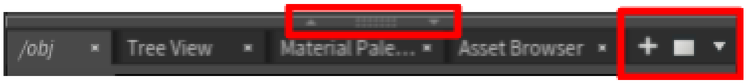
On one side of a toolbar/shelf/menu, there should be a small black arrow that allows you to collapse or un-collapse that panel.



Panel Manipulation

Panels almost always are tabbed views -- meaning that a panel is made up of multiple tabs that you can flip through. Panels can be hidden, maximized, and shifted around via the dividers. The tabs within them can also be manipulated in various ways.

Almost all panels have the following setup...

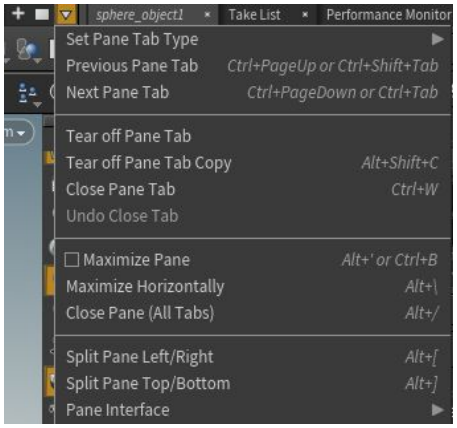


The middle portion of the divider bar (the dotted portion) can be click+dragged to expand/contract the panel. You can also just click it (instead of click+drag) to swap the positions of the panels.

The arrow buttons on the side of the middle portion can be clicked to have a panel take up all the space in that direction (hiding other panels).

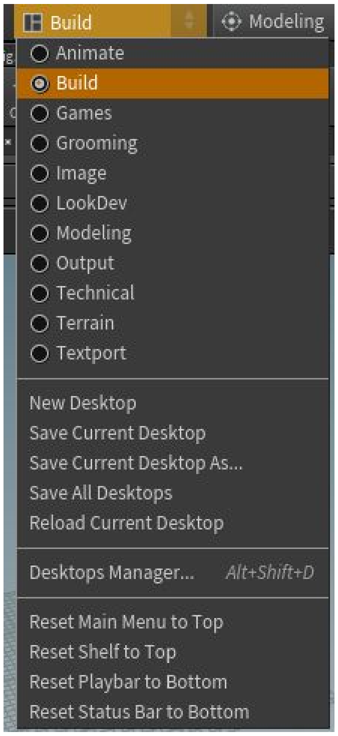
The top-right of a panel contains 3 buttons.

1. The plus button allows you to add new tabs.
2. The window button makes the panel take up the entire houdini window.
3. The drop-down arrow gives you several other panel/tab manipulation options.


sphere_object1
Take List
Performance Monitor
Set Pane Tab Type
Previous Pane Tab Ctrl+PageUp or Ctrl+Shift+Tab
Next Pane Tab Ctrl+PageDown or Ctrl+Tab
Tear off Pane Tab
Tear off Pane Tab Copy Alt+Shift+C
Close Pane Tab Ctrl+W
Undo Close Tab
□ Maximize Pane Alt+' or Ctrl+B
Maximize Horizontally Alt+\
Close Pane (All Tabs) Alt+/
Split Pane Left/Right Alt+[
Split Pane Top/Bottom Alt+]
Pane Interface

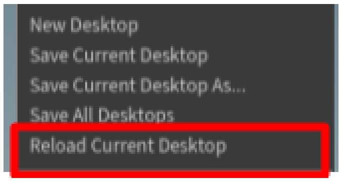
Desktops

Desktops are Houdini’s version of Maya’s workspaces. They’re essentially an arrangement of panels and windows that optimize a certain type of workflow. You can switch between desktops by using the desktop dropdown alongside the main menu…



Reloading Desktops

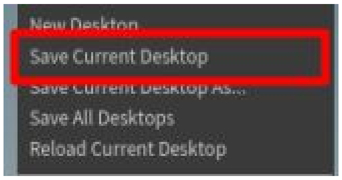
If you ever mess something up with your panels, or accidentally close certain panels and don't know how to get them back, you can go to the desktop dropdown and select Reload Current Desktop. It'll bring back all the panels and tabs and etc… in the way they were originally laid out.



Saving Desktops

If you've organized your layout to something that you're comfortable with and you want to save it so you can use it later, you can use Save Current Desktop As.

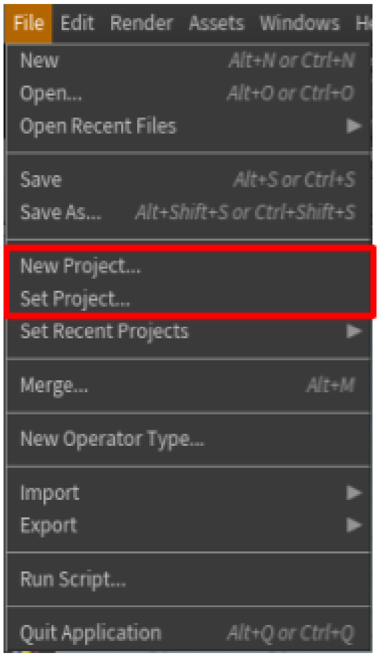
DO NOT USE Save Current Desktop because if you're on one of the prebuilt desktops (e.g. Build) it will override the defaults and you won't be able to go back to those defaults without blowing away all your preferences..

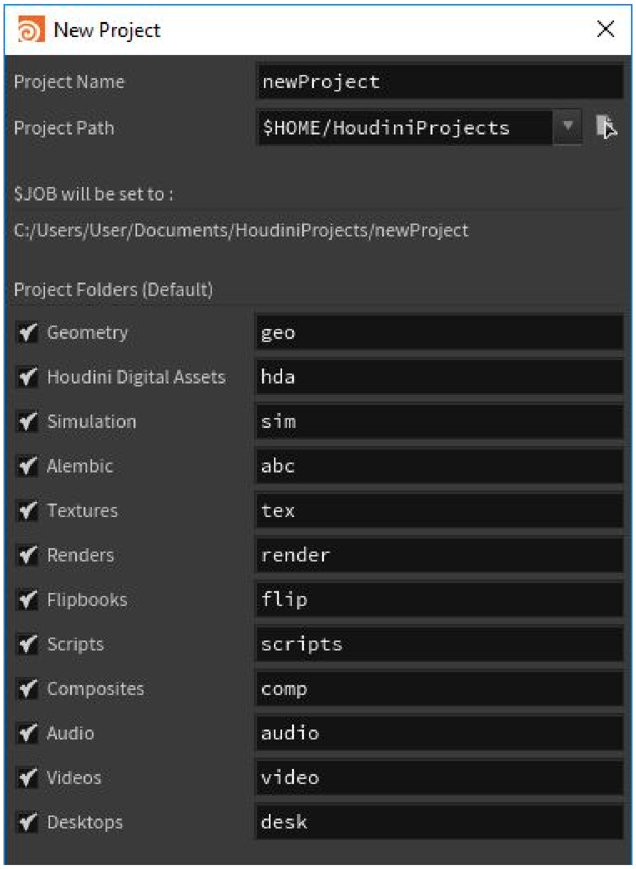


Projects

Project's are Houdini's equivalent of Maya projects. A project is a set of organized folders on your harddrive that Houdini will look at to find different elements in your scene (e.g. textures).

You can create a new project or load an existing project via the File menu…




New Project

Project Name: newProject

Project Path: $HOME/HoudiniProjects

$JOB will be set to :
C:/Users/User/Documents/HoudiniProjects/newProject

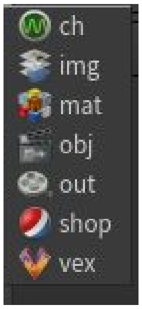
Project Folders (Default)

✓ Geometry	geo
✓ Houdini Digital Assets	hda
✓ Simulation	sim
✓ Alembic	abc
✓ Textures	tex
✓ Renders	render
✓ Flipbooks	flip
✓ Scripts	scripts
✓ Composites	comp
✓ Audio	audio
✓ Videos	video
✓ Desktops	desk

Contexts

Houdini works in what are called "contexts"... Depending on the context you're in, Houdini will provide you different operators/functions that you can use (these are exposed as nodes).

There are many different contexts inside Houdini…

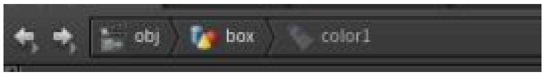


- ch / chops
  Channel Operators
  - Used for manipulation of raw data (e.g. movement of objects) and some audio stuff.
- img / cops
  Compositing Operators
  - Houdini offers an almost Nuke-like compositing package. All of that is container here.
- obj
  ???????????
  - This is where your scene / shapes are generally described.
- out / rops
  Rendering Operators
  - Allows you to set up various types of renders and exports.
- shops
  Shading Operators
  - This is where you deal with most of your shading/material stuff.
- vex / vops
  Vex Operators
  - Provides a node-based interface for doing programming and scripting.

Some extra things to note…

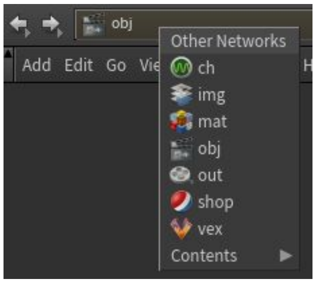
- Some contexts aren't listed here. For example, if you're working with particles, those are called pops (particle operators). Another one is dops: dynamic operators.
- Certain contexts are independent of each other, while other contexts are crossed together (information can flow between them).

Most views in Houdini that support switching between contexts (e.g. scene view) provide a common toolbar.

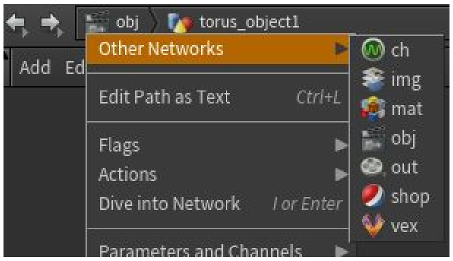


Depending on the view that you're in, you'll be able to switch to different contexts. Some views may only support a subset of contexts (e.g. scene view), while others may support all contexts (e.g. network view).

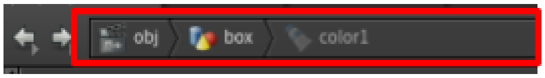
If you’re at the top-level, you can switch to contexts by left-clicking on the top breadcrumb…



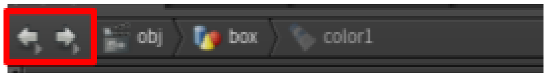
Another way to switch between contexts is to right-click on the top breadcrumb (doesn’t matter if you’re further down in the hierarchy) and go to Other Networks…



The toolbar provides breadcrumbs that you can click on to quickly move back up the context hierarchy…

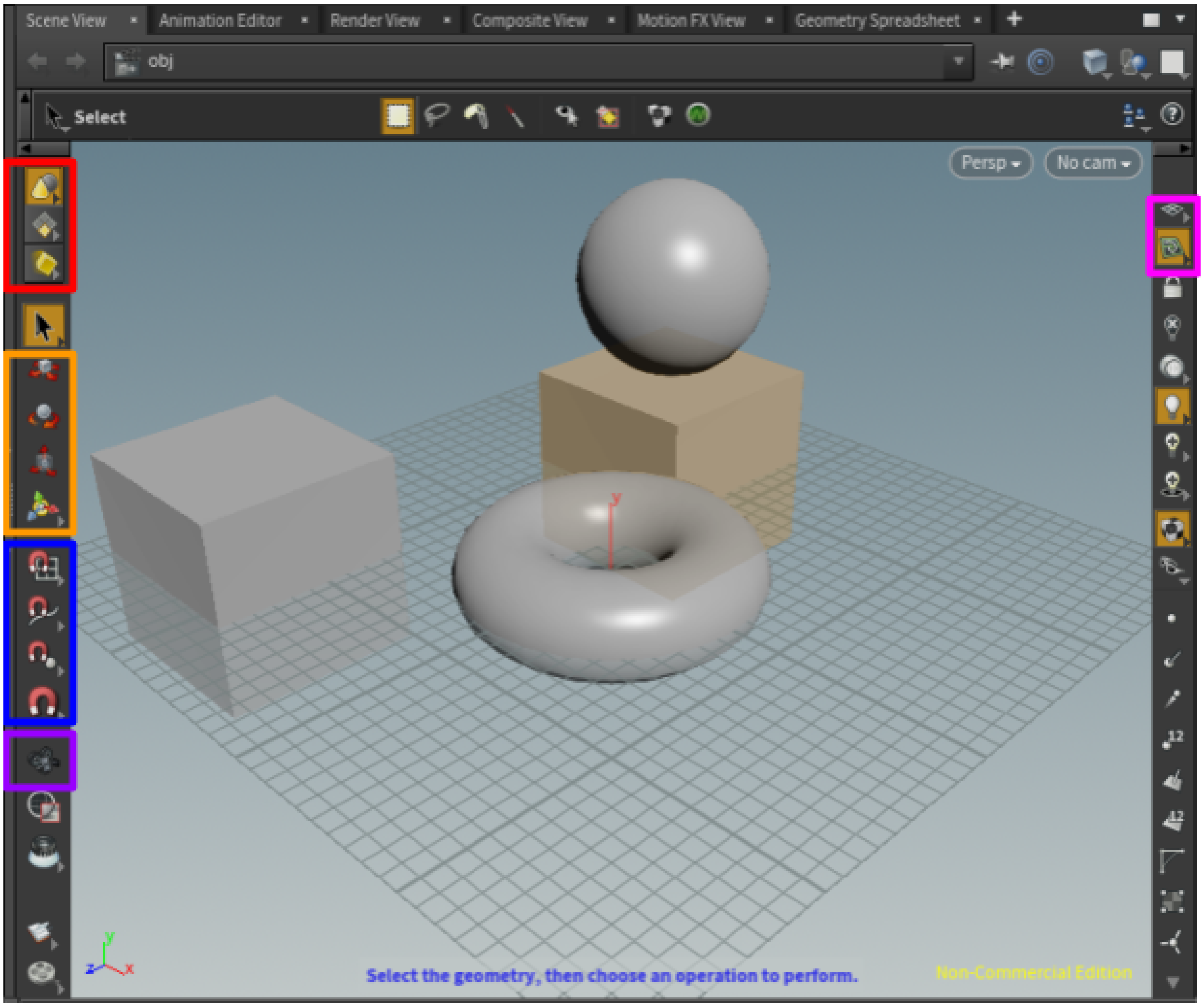


And… just like a web browser, you can also use the back/forward buttons to move between your history of context switches…



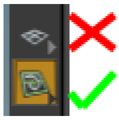
Scene View

The scene view interface gives you a visual representation of your scene -- very similar to Maya’s viewports.

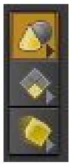


Select the geometry, then choose an operation to perform.

NOTE: It's best to turn off the reference grid and turn on the construction plane. The construction plane will constrain your Y to 0 when placing objects, unless you're holding Shift.



Selection Type Defines how you select.



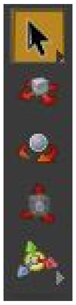
Button 1 is object selection.
Button 2 is component selection (object must be selected first).

Button 3 is ???? selection.

You can right-click on each button to further refine selection parameters. For example, if you want to select by edge, you can right-click on button 2 and specifically select edge as your component type.

Manipulations

Typical selection tools, very similar to the ones in Maya. The last one is called the "Handle" tool and it's a combo of scale/rotate/translate.



NOTE: The hotkeys here are different than the ones in Maya. Instead of QWER, it's STRE...

Select =	S in Houdini	Q in Maya
Translate =	T in Houdini	W in Maya
Rotate =	R in Houdini	T in Maya
Scale =	E in Houdini	R in Maya

Snapping

Typical snapping controls, similar to the ones in Maya.

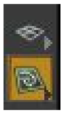
Camera Control

If you want exclusive control to manipulate the camera, select this. But, just like Maya, you can hold Alt+LMB/MMB/RMB to tumble/pan/zoom. You can also hold Space+LMB/MMB/RMB.

Reference Plane

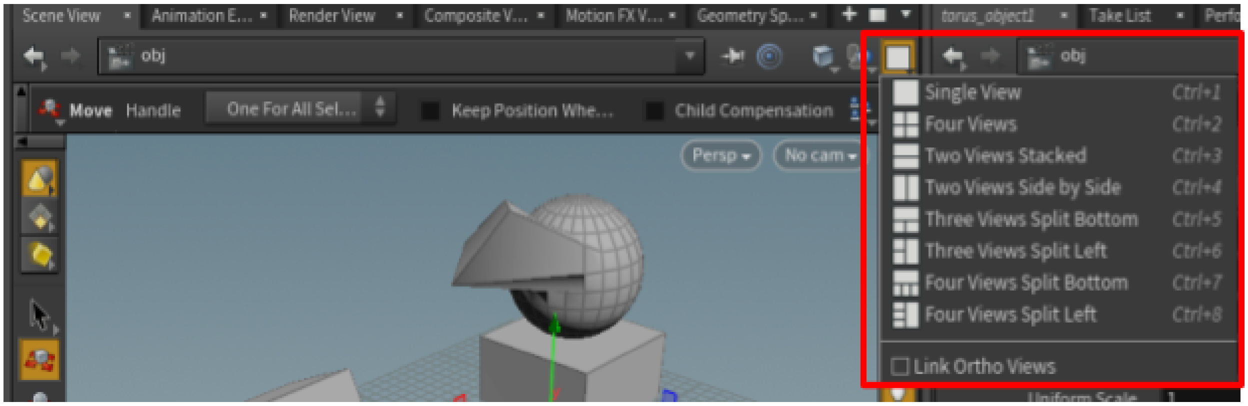
The first button shows the construction plane. It's good to have this on because when you're dropping objects on the viewport they'll be aligned to this plane.

The second button shows the reference grid/plane. It looks like this is just a reference, probably doesn't hurt to have this on.



Multiple Viewports

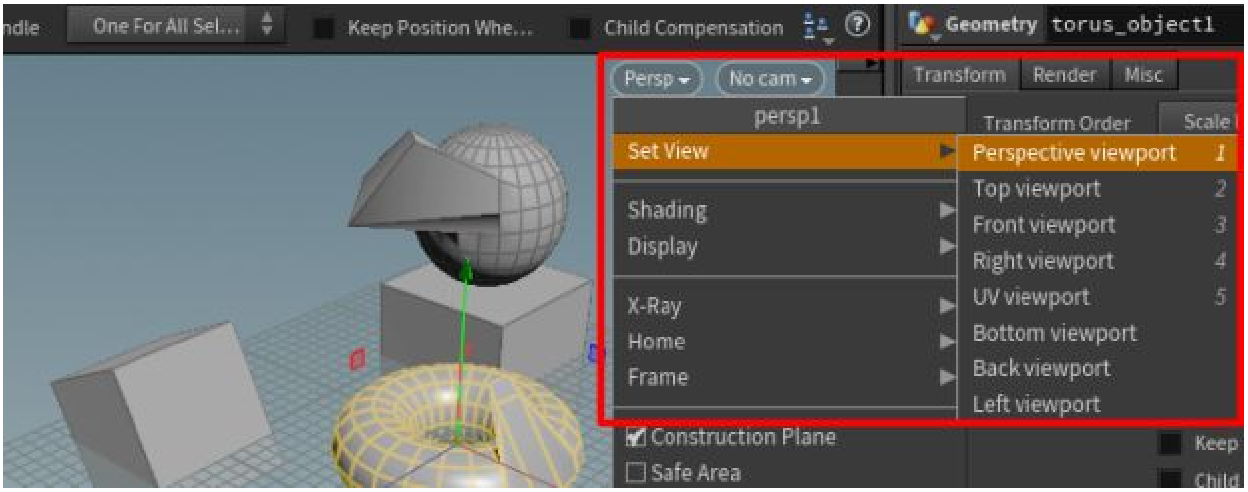
If you want multiple viewports in your scene view (just like you have in Maya), you can use the the viewport dropdown on the upper-right of the panel…



The hotkeys are listed in the image. If you want to switch between single view and 4-panel view (just like in Maya), you can do so via Ctrl+1 and Ctrl+2.

Viewport Orientation

You can change the viewport camera through first dropdown in the upper-right the viewport…



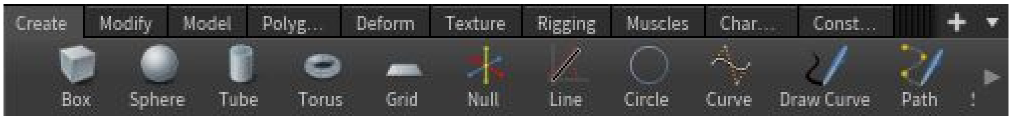
Just like Maya, top/front/right views are orthographic, meaning you won't be able to tumble them.

HINT: The hotkeys don't work unless you're holding Space. For example, Space+1 will take you to perspective.

Creating Objects

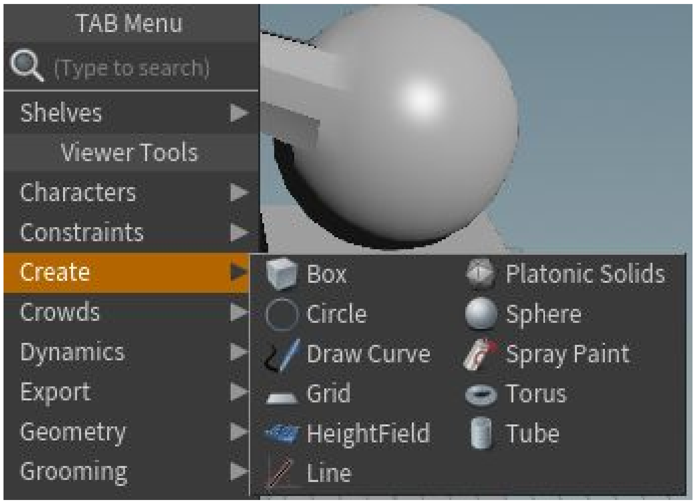
You can create objects either through the…

1. create shelf

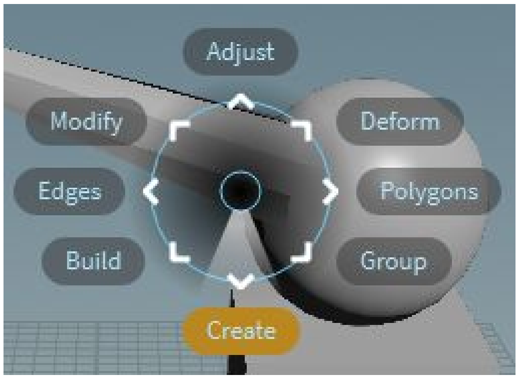


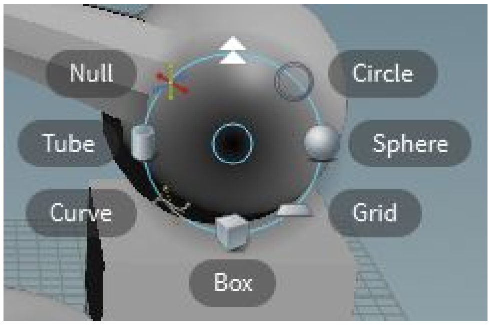
HINT: Hold CTRL when clicking to have it show up at origin vs interactively placing it in the viewport

2. tab menu



3. radial menu (if radial menu is set to Main or Modeling) via Create

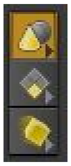




When you’re asked to place in viewport (interactive placement), your placement will be bound to the construction plane. That means your Y will be fixed. You can hold Shift while you’re moving around to change the Y.

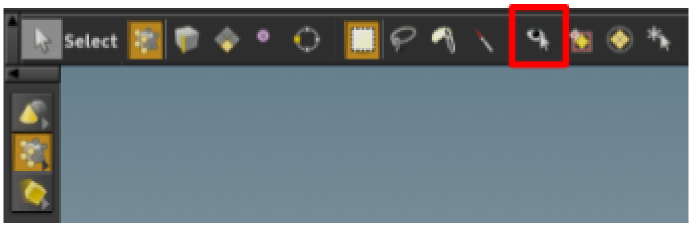
Manipulating Objects and Components

To move objects around in the scene, first change the selection mode to...

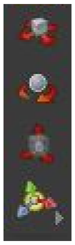


Button 1 is object selection.
Button 2 is component selection (object must be selected first).

> **HINT**: When you're doing marquee selections, be mindful of the <u>Select Visible Geometry Only</u> option. When this is enabled, you'll only select what you can see. So if you're in shaded mode, everything hidden won't be selected. But, if you're in wireframe mode, nothing is hidden so everything will get selected.
>
> Note that THIS IS THE OPPOSITE OF HOW MAYA WORKS. Keep this off if you want consistent behaviour (it seems to be off by default).



Then, select your manipulation tool and manipulate…



The above will let you translate, rotate, and scale.The hotkeys for the manipulation tools are…

- Translate = T
- Rotate = R
- Scale = E
- Handle = Y (press twice to go to secondary mode)

HINT: Hold Shift when dragging manipulation handles to make subtle change instead of normal change.

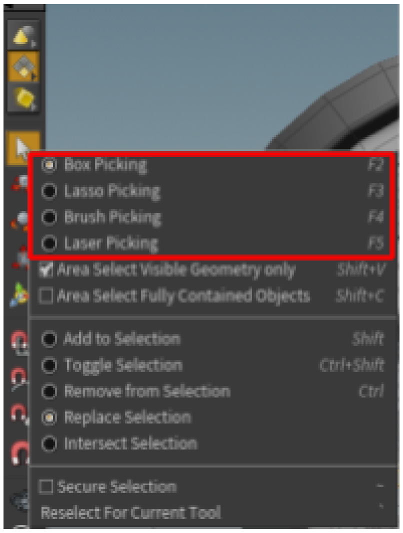
HINT: If you want to interactively move the pivot point (the point around which you rotate), you can select any of the manipulation tools and hit the INS key.

Selecting Components

You can select components just like you do in Maya.

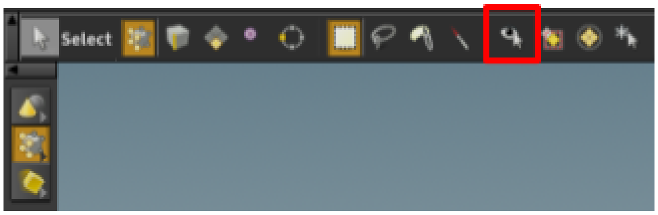
Hold Ctrl and click to remove a selection.
Hold Shift and click to add a selection.

The basic tool for selection is the marquee tool (box picking). But there are several others. You can click-and-hold on the selection tool to get a flyout that'll let you pick which tool you want to select with.


● Box Picking F2
○ Lasso Picking F3
○ Brush Picking F4
○ Laser Picking F5
☑ Area Select Visible Geometry only Shift+V
□ Area Select Fully Contained Objects Shift+C
○ Add to Selection Shift
○ Toggle Selection Ctrl+Shift
○ Remove from Selection Ctrl
● Replace Selection
○ Intersect Selection
□ Secure Selection ~
Reselect For Current Tool `

Selecting Visible Geometry Only

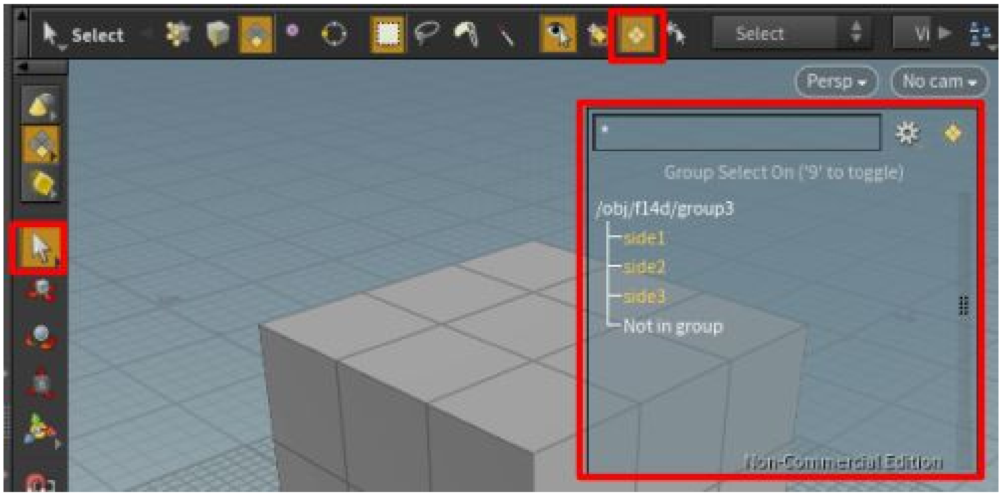
Regardless of which selection tool you use, be mindful of the Select Visible Geometry Only option. When this is enabled, you'll only select what you can see. So if you're in shaded mode, everything hidden won't be selected. But, if you're in wireframe mode, nothing is hidden so everything will get selected.



Selecting by Group

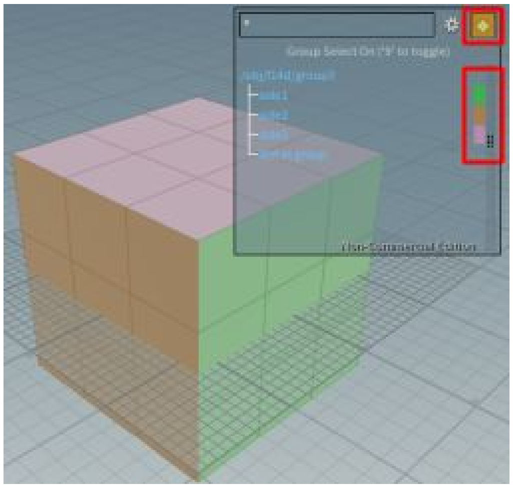
> **NOTE**: Don’t know what groups are used for? Check out the Group nodes sectin for more information.

In addition to both of these, you can view / highlight the various groups in the scene view by choosing the select tool and enabling enabling ‘Select Groups’ in the top toolbar…



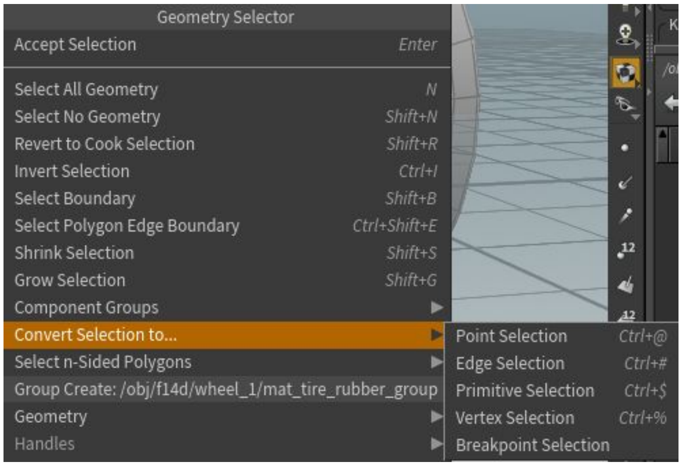
When you do this, a popup will show up on the lefthand side and you’ll only be able to select by groups. You can select groups as a whole by clicking on the actual model causing it to highlight in the popup pane (and vice versa).

Selecting the diamond in the popup causes your models to get colorized by group, which is also super useful…



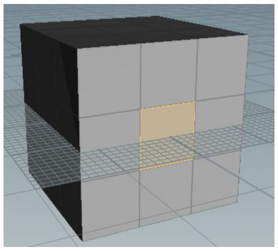
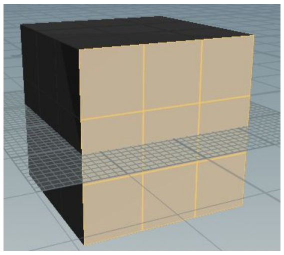
Converting Selection between Component Types

You can change your selection type (e.g. from edges to faces) by right-clicking on your selection and going to Convert Selection to… This is incredibly useful when you need to select geo in something that has lots of pits and grooves and other areas you can’t get into. For example, you can select edge loops that go totally around the object, and then convert that selection to faces.



Growing/Expanding Selection

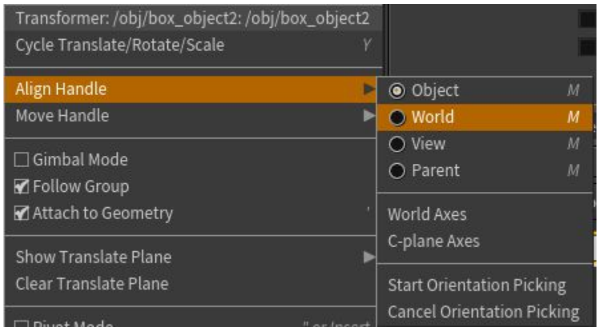
You can grow a selection outwards by hitting Ctrl+G (or right-clicking on the selection and choosing Grow Selection). Note that the shortcut doesn’t always work -- I think if you move your mouse out of scene view and then back in, it won’t work anymore unless you make a new selection.

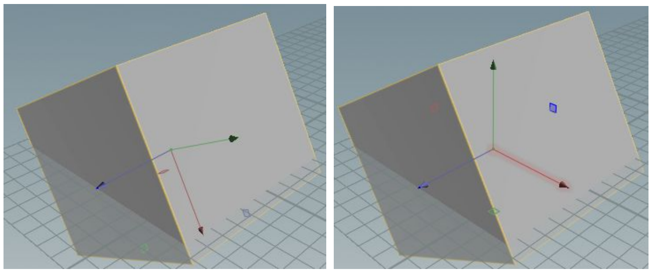



Handle Alignment

One thing to note is that, unlike Maya, the default here for the handles is to be object oriented rather than world oriented. That means that if you rotate your object and then try to translate, the handles will sync with the new orientation of the object.

To fix this, hover over the handles and right-click. Then, go to Align Handle -> World…

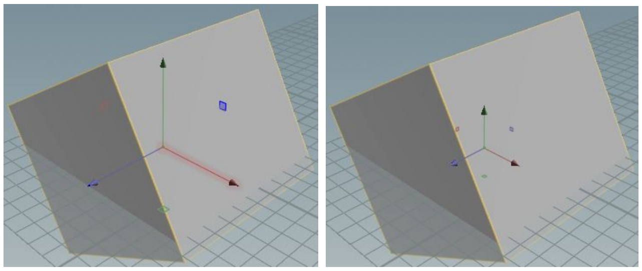




The above example shows the before and after. Note that in the after, the axis are aligned to the construction plane, which is aligned to the world.

Handle Size

If the handle is too small or large, you can use the * key to make it larger and the / key to make it smaller…



Manipulating Camera

You can manipulate the camera the same way you do in Maya…

- Alt+LMB to tumble
- Alt+MMB to pan
- Alt+RMB to zoom

NOTE: You can hold Space instead of Alt. It does the same thing.

In addition to these hotkeys, you can click the camera button to exclusively use the mouse buttons to manipulate the camera. With the camera button selected, you don't have to hold Alt/Space + relevant mouse button to tumble/pan/zoom (you can use the relevant mouse buttons by themselves).

> **HINT**: Using Alt is kind of a blessing and a curse. It helps you avoid having to develop new muscle memory after already learning how to do it in Maya, but certain important hotkeys won't work in with Alt (they'll only work with Space).

Frame up on Selected Object

To frame up on the selected object(s), either have the camera button selected and press F, or hit Space+F.

> **NOTE**: USING F BY ITSELF WON'T WORK. USING ALT+F WON'T WORK EITHER. If you want to use the the hotkey instead of the being in camera manipulation mode, you need to use Space+F.

> **HINT**: Space+G does this as well. Infact Space+G seems to be preferred instead of F because the network view also uses G to do the same thing.

Frame up on All Objects

To frame up on the all object(s), either have the camera button selected and press A, or hit Space+A.

> **NOTE**: USING A BY ITSELF WON'T WORK. USING ALT+A WON'T WORK EITHER. If you want to use the the hotkey instead of the being in camera manipulation mode, you need to use Space+A.

Revert to Home Orientation

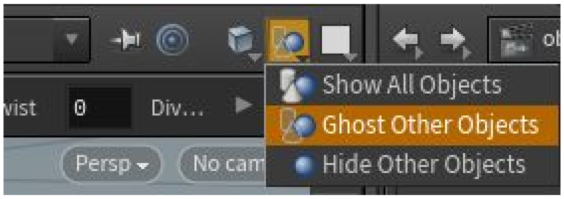
To revert to the original camera orientation framed up on the construction plane, either have the camera button selected and press H, or hit Space+H.

> **NOTE**: USING H BY ITSELF WON'T WORK. USING ALT+H WON'T WORK EITHER. If you want to use the the hotkey instead of the being in camera manipulation mode, you need to use Space+H.

> **NOTE**: Remember that this frames up on the construction plane. If you have stuff outside of the construction plane, it won't come into view.

Changing Viewport Object Ghosting

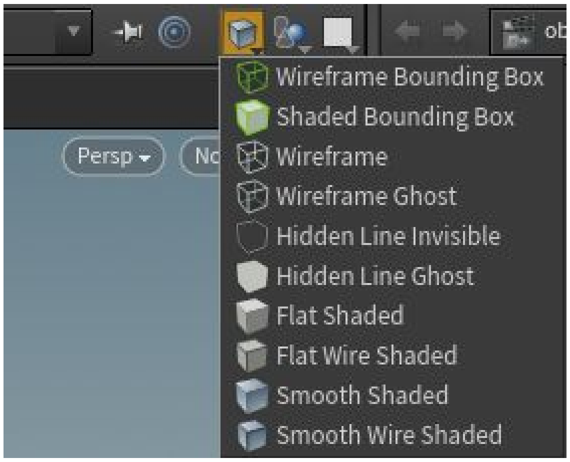
You can choose how objects appear based on your depth in the hierarchy by going to the ghosting type in the top-right of the viewport...



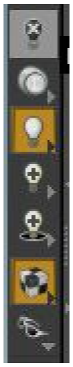
For example, if you have Ghost Other Objects selected, when you dive into an object other objects will become semi-transparent.

Changing Viewport Lighting/Shading

You can choose how things are shaded by going to the shading type in the top-right of the viewport…



You can choose how lights are shown by going to the right toolbar…

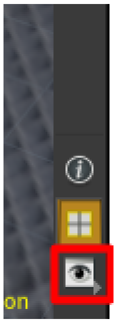


Options are as follows (or you can just look directly at the tooltips to tell what’s what)...

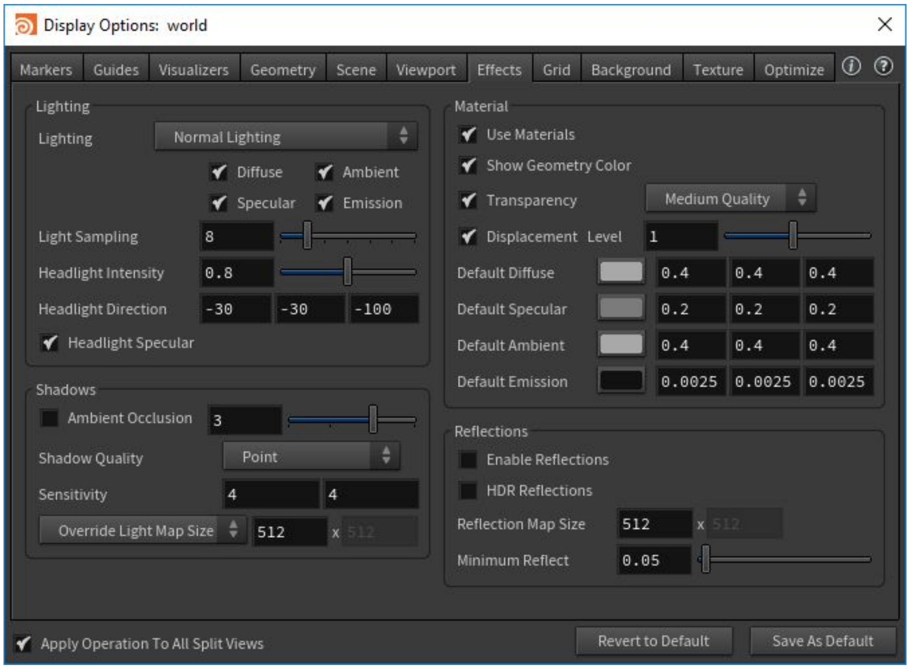
1. Disable lighting (flat shade?)
2. Headlight only
3. Normal lighting
4. High-quality lighting
5. High-quality lighting w/ shadows
6. Display materials on objects

Display Options

You can change the particulars of how your scene is displayed (e.g. sampling quality) by doing to the very last button in the right-hand toolbar (you might need to scroll down to see it)....



A lot of relevant options can be found under Scene and Effects.



Display Options: world

Lighting

Lighting: Normal Lighting

☑ Diffuse ☑ Ambient

☑ Specular ☑ Emission

Light Sampling 8

Headlight Intensity 0.8

Headlight Direction -30 -30 -100

☑ Headlight Specular

Shadows

□ Ambient Occlusion 3

Shadow Quality: Point

Sensitivity 4 4

Override Light Map Size 512 x 512

Material

☑ Use Materials

☑ Show Geometry Color

☑ Transparency: Medium Quality

☑ Displacement Level 1

Default Diffuse 0.4 0.4 0.4

Default Specular 0.2 0.2 0.2

Default Ambient 0.4 0.4 0.4

Default Emission 0.0025 0.0025 0.0025

Reflections

□ Enable Reflections

□ HDR Reflections

Reflection Map Size 512 x 512

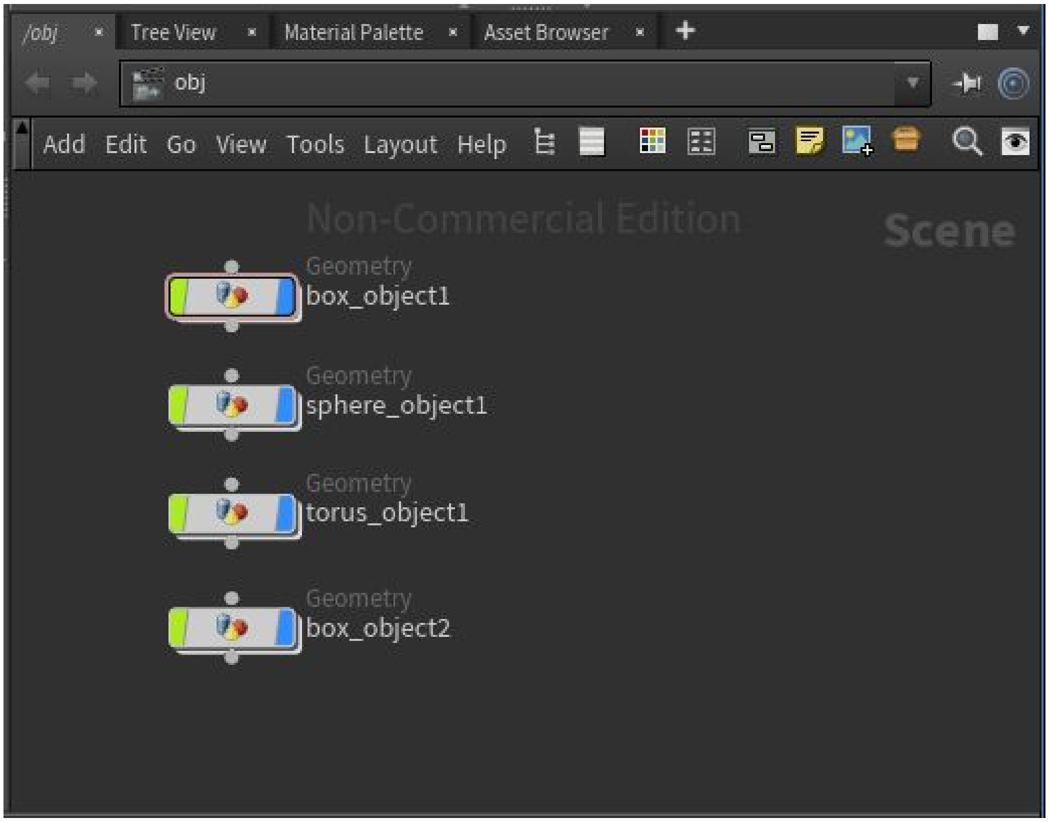
Minimum Reflect 0.05

☑ Apply Operation To All Split Views

Scene will let you turn up AA samples and Effects will let you turn on special rendering options in the scene view (ambient occlusion, reflections, etc..). Depending on your workstation, your rig and the amount of geometry you have, Houdini may not be able to handle the settings you give it.

Network View

The network view interface gives you a node representation of your scene -- very similar to Maya's node editor. Nodes feed into one another in a DAG. Each node represents some item or operations/functions performed on items.



Selecting something in the network view will select it in all other views as well (e.g. scene view/parameter view/etc..).

> **HINT**: If you maximize this view, you can get the parameters panel to show up as a mini panel on the upper-right by pressing the P button (toggles on and off).
>
> **HINT**: You can show a tree view as a panel in the network view by pressing Shift+W. Note that the tree view is Houdini's version of Maya's outliner.

Manipulating View

The controls work similarly to the camera controls in the scene view, except you don't need to hold the Alt/Space button and you can't tumble because this is a 2D view. Hotkeys are as follows…

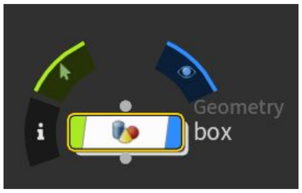
- MMB pans
- RMB zooms
- H brings everything into view
- G brings selection into view

  NOTE: Remember that LMB does nothing here. Neither does F (to frame up use G / to show all use H).

Nodes

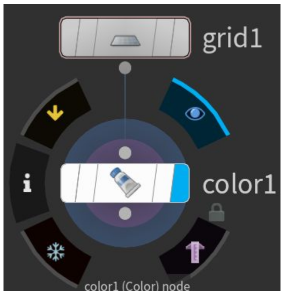
Node Flags

By flipping on/off flags in the network view. For example, if you have an object node...



- Green edge toggles selectability in scene view.
- Blue edge toggles visibility in scene view.

Different node types give you different options. If you go within the object node, you'll see more flags on a node…



The flag edges, from left-to-right, are as follows…

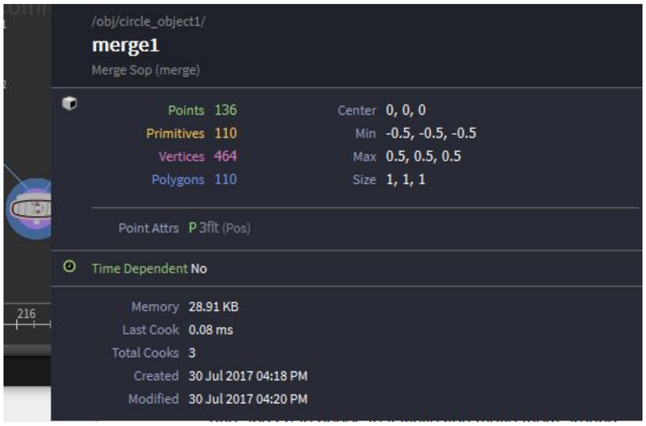
1. Bypass - Bypasses a node so that it doesn't get evaluated in the node tree.
2. Lock - ???
3. Template/Selectable Template - ???

4. Visibility - Whatever is being computed by the graph is visible in the scene view from this node. That is, if you have multiple nodes chained together, the one that has visibility turned on is the point at which it'll show up in the scene view. Only one node in the chain can have this flag set.

The radial menu options when you hover over the node pretty much do the same thing (they're new to Houdini 16). The i button in the radial menu shows information about the node.

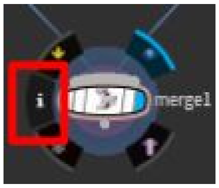
Node Information

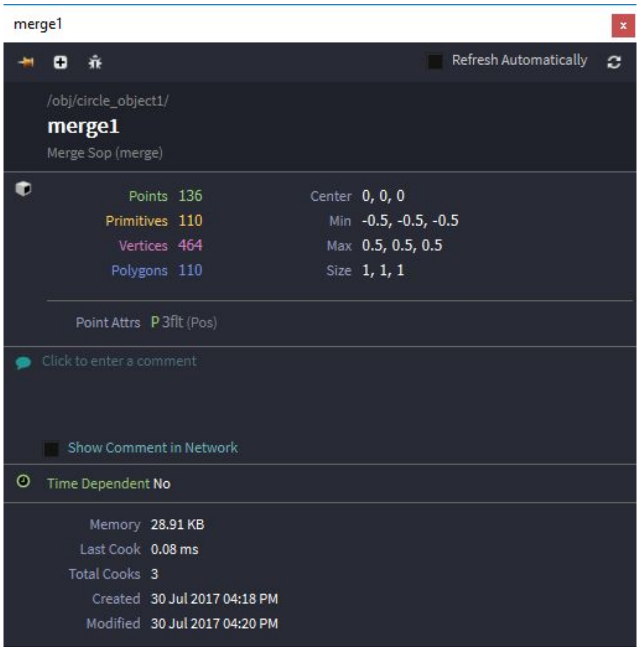
You can get information on a node by click-and-holding MMB in the middle of the node (on the icon) to show a temporary popup…



```
/obj/circle_object1/
merge1
Merge Sop (merge)

     Points  136          Center  0, 0, 0
 Primitives  110             Min  -0.5, -0.5, -0.5
   Vertices  464             Max  0.5, 0.5, 0.5
   Polygons  110            Size  1, 1, 1

Point Attrs  P 3flt (Pos)

Time Dependent No

     Memory  28.91 KB
  Last Cook  0.08 ms
Total Cooks  3
    Created  30 Jul 2017 04:18 PM
   Modified  30 Jul 2017 04:20 PM
```

OR... by clicking the i in the radial menu to open a new window…





merge1

Refresh Automatically

/obj/circle_object1/

merge1

Merge Sop (merge)

Points 136 Center 0, 0, 0
Primitives 110 Min -0.5, -0.5, -0.5
Vertices 464 Max 0.5, 0.5, 0.5
Polygons 110 Size 1, 1, 1

Point Attrs P 3flt (Pos)

Click to enter a comment

Show Comment in Network

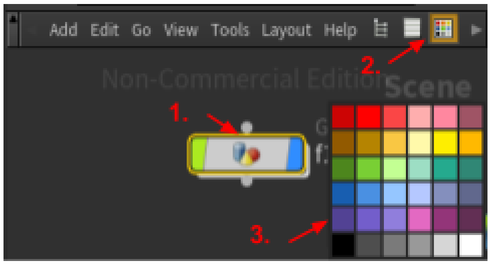
Time Dependent No

Memory 28.91 KB
Last Cook 0.08 ms
Total Cooks 3
Created 30 Jul 2017 04:18 PM
Modified 30 Jul 2017 04:20 PM

Coloring Nodes

You can assign custom colors to nodes.

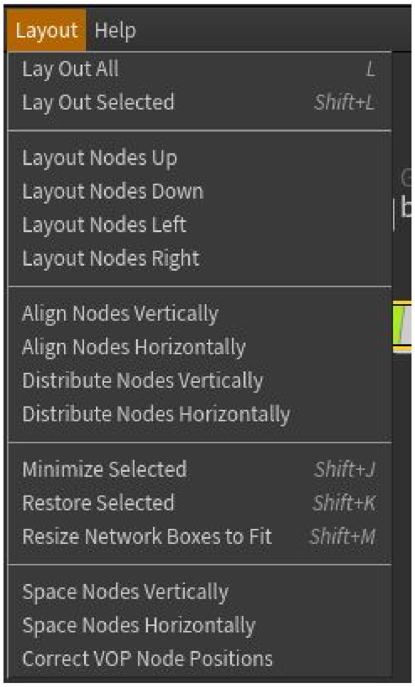
Select the nodes you want to color, then click the color palette toggle in the toolbar and choose the color you want.



Positioning Nodes

You can position nodes by clicking-and-dragging them around.

The layout menu also provides lots of options for laying out your node, including functions you'll find in most vector drawing programs (e.g. inkscape). For example, you can choose to show a grid and snap nodes to it while you move them around.



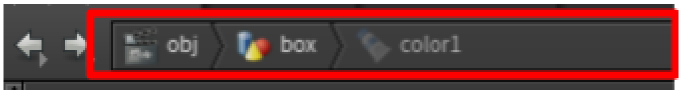
> **NOTE**: L is the auto-layout shortcut. If your node placement is messy, just hit L to organize it.

Stepping Into Nodes

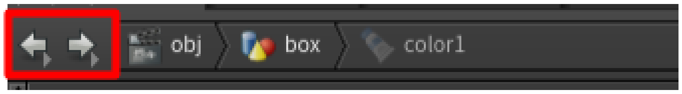
You can step in and out of nodes in multiple ways…

1. Double-click the node to step in.
2. Click the node then hit Enter to go in.
3. Click the node then hit I to go in / U to step out.

You'll also notice that as you step in, most views provide you with breadcrumbs that you can click on to quickly move back up the context hierarchy…



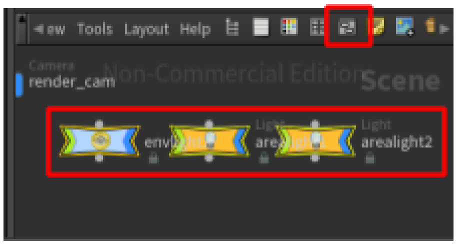
Just like a web browser, you can also use the back/forward buttons to move between your history of context switches…



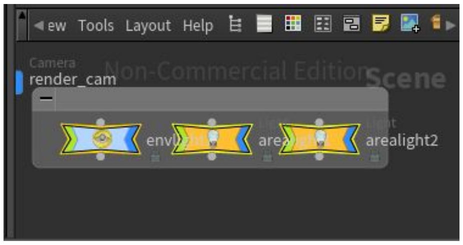
Creating Netboxes

A Houdini netbox is exactly the same thing as a Katana backdrop node.

To use a netbox, select all the nodes you want to put inside of it and hit the netbox button in the toolbar…



→



- Assign the netbox a name by double-clicking the title
- Minimize the netbox using the upper left-hand corner
- Color the netbox just as if you were coloring any other node

When you …

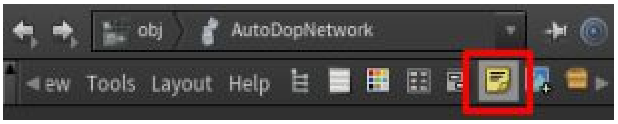
- move a netbox, it shifts everything inside of it with it.
- delete a netbox, it only removes the netbox, not the stuff inside it

  NOTE: There is no J shortcut here like there is in Katana -- where you can jump to a backdrop node / netbox. Infact, netboxes don’t even show up in node searches (Ctrl+F in network view).

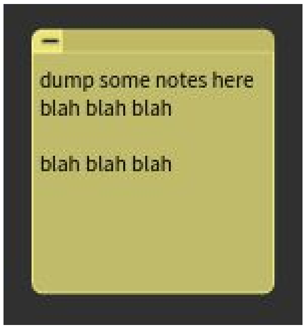
Creating Notes

You can create notes directly in Houdini. They show up as sticky notes, and you can move them around and manipulate them just like normal nodes.

To create a note, click the note icon in the toolbar…



A little yellow box will show up and you can click inside to fill it in. Click the minus sign in the upper right corner to collapse it.

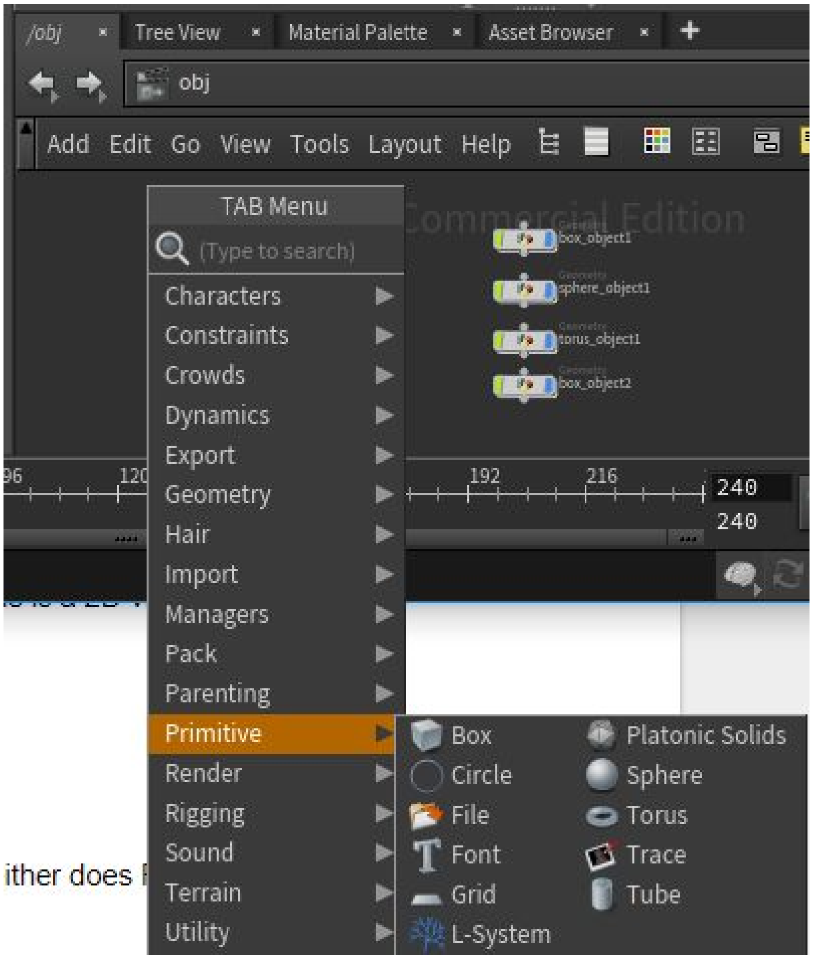


dump some notes here
blah blah blah

blah blah blah

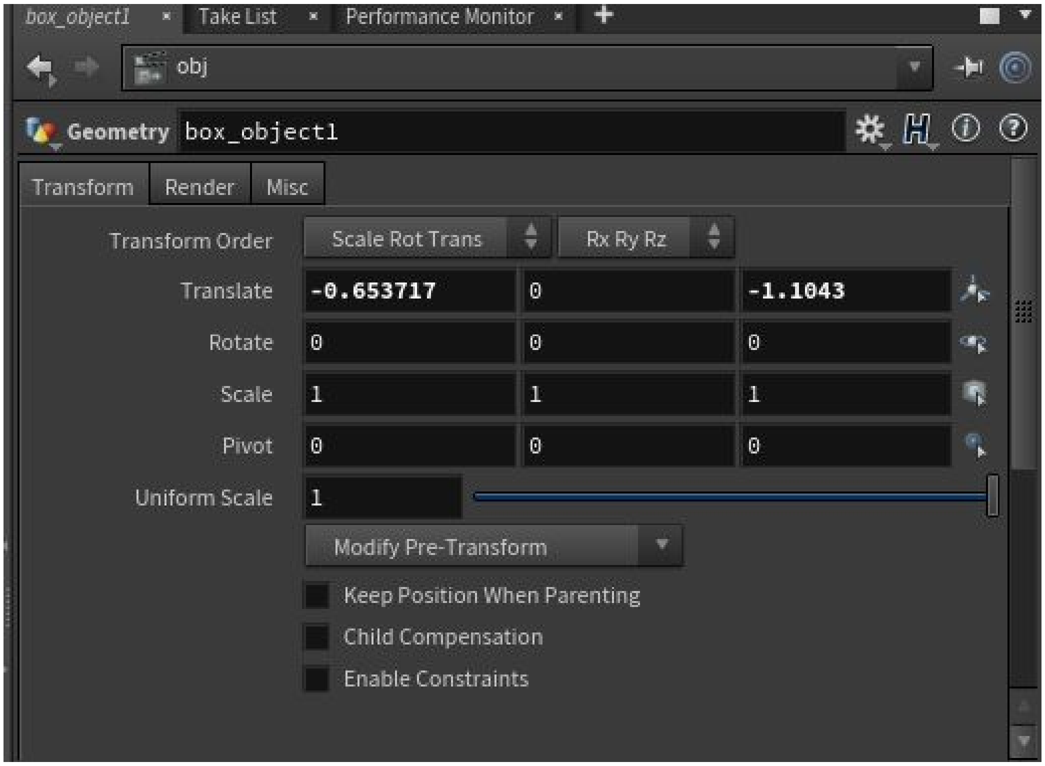
Creating Nodes

You can create objects either through the tab menu. Either right-click to show the tab menu or hit tab, then select whatever it is you want to create...



Parameter View

The parameter view is the equivalent of Maya's attribute editor.



Geometry box_object1

Transform Order Scale Rot Trans Rx Ry Rz

Translate -0.653717 0 -1.1043

Rotate 0 0 0

Scale 1 1 1

Pivot 0 0 0

Uniform Scale 1

Modify Pre-Transform

□ Keep Position When Parenting

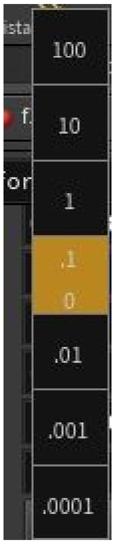
□ Child Compensation

□ Enable Constraints

What's displayed in here is entirely dependent on what you have selected. For example, if you have geometry selected, you'll get something like the selection above: translate/rotate/scale/pivot/etc…

Value Ladder

If you're making changes to numerical parameters, you can use the value ladder to make your adjustments. In the text box that takes in the input, hold down MMB. A scale should pop up.



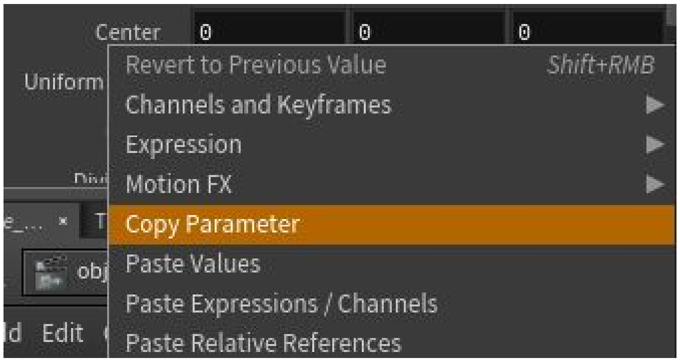
The number in the middle (0 in the example above) is the current value. Move your mouse pointer up/down to the scale you wish to adjust by, and move right to ADD and left to SUBTRACT.

Setting Relative References

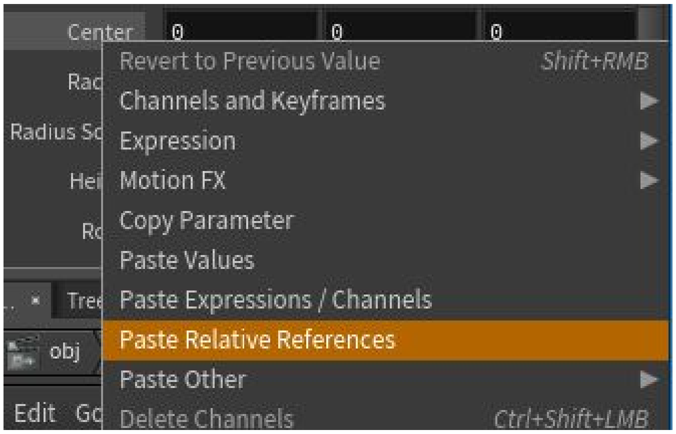
Sometimes, you want to bind certain parameters to the parameters of some other node. For example, you may want your object to be centered around another object.

To create a reference…

1. Go to the parameter(s) to bind from and right-click and choose Copy Parameter…



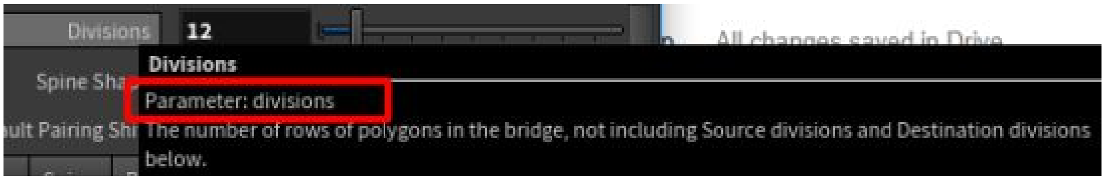
2. Go to the parameter(s) to bind to and right-click and choose Paste Relative References…



When you do this, the parameters that you pasted to will turn green and they'll reference a location.

Notice what's happening here. The parameter value changes to something like ch("path1/path2"). The ch function lets you grab a parameter value from any node in your setup. You just need to specify the path.

HINT: How do you know what the parameter name is? Go to the parameter and hover your mouse over the label. The tooltip that pops up will tell you…

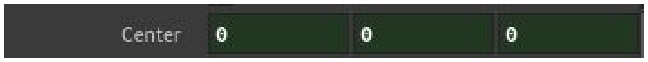


Divisions

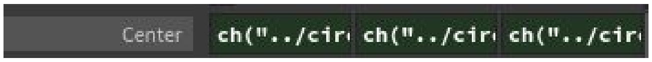
Parameter: divisions

The number of rows of polygons in the bridge, not including Source divisions and Destination divisions below.

HINT: You can switch between seeing the references and seeing the evaluated values by clicking the field set name…



Center	0	0	0

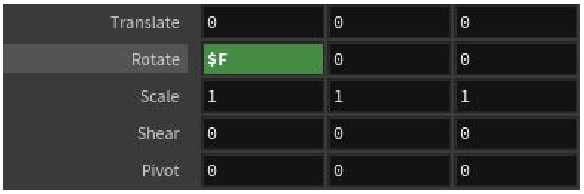


Center	ch("../cir	ch("../cir	ch("../cir

Setting Dynamic References

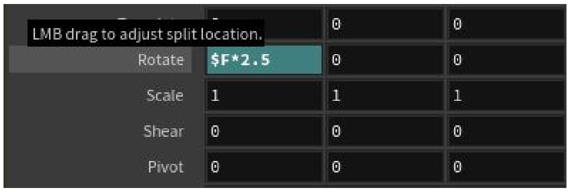
Just like relative references, you can do computations and stuff based on variables. Relative references are just a special case of that.

For example, let’s say you want your X rotation to change based on what animation frame you’re currently at. You can reference the animation frame by using $F.



Translate	0	0	0
Rotate	$F	0	0
Scale	1	1	1
Shear	0	0	0
Pivot	0	0	0

Now let’s say you want to that rotation to speed up. You want it to go 2.5 times as fast. You can change that expression to be $F*2.5…



LMB drag to adjust split location.

		0	0
Rotate	$F*2.5	0	0
Scale	1	1	1
Shear	0	0	0
Pivot	0	0	0

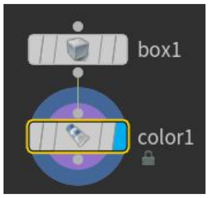
Geometry Spreadsheet View (Attributes)

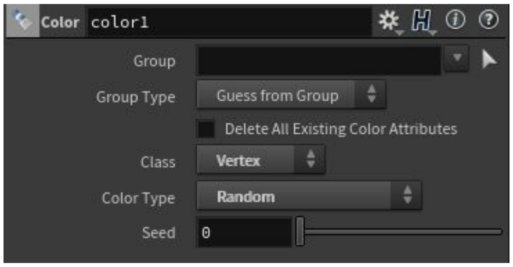
Houdini allows you to set attributes on various levels of the geometry…

- Object (also called detail)
- Primitive (also called face)
- Point

- Vertex

So for example, if you had a piece of geometry and you fed it into a color node, the output of that color node would be the same geometry with some added attributes…





Color color1

Group

Group Type: Guess from Group

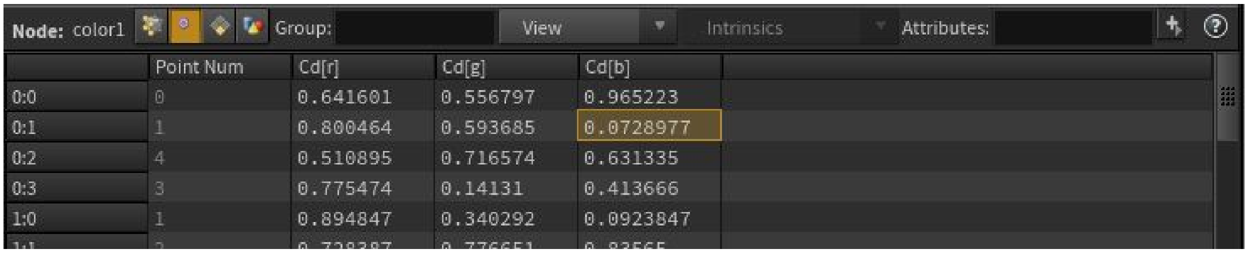
□ Delete All Existing Color Attributes

Class: Vertex

Color Type: Random

Seed: 0

You can see in the properties panel shown above that we're applying random colors to the vertices. If you go into the Geometry Spreadsheet panel (in the same tab set as the scene view), you'll see a bunch of newly added attributes to the vertices that describe the color…



Node: color1 Group: View Intrinsics Attributes:

	Point Num	Cd[r]	Cd[g]	Cd[b]
0:0	0	0.641601	0.556797	0.965223
0:1	1	0.800464	0.593685	0.0728977
0:2	4	0.510895	0.716574	0.631335
0:3	3	0.775474	0.14131	0.413666
1:0	1	0.894847	0.340292	0.0923847
1:1	2	0.728387	0.776651	0.83565

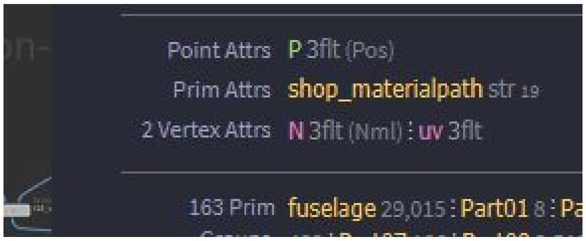
> **NOTE**: The toggle buttons in the toolbar let you switch between point/vertex/primitive(face)/detail(object) attributes.

These attributes will propagate throughout your node network (and may potentially even be changed or removed by other nodes down the line).

> **NOTE**: These are lots and lots of common attribute types: colors, normals, uv, etc.... Check out the Common Attributes section of this page: http://www.sidefx.com/docs/houdini/model/attributes

Manipulating Attributes / Moving Around Attributes

Is your Merge node giving you a warning? Check the attributes of your inputs at various levels via the geometry spreadsheet. Just MMB click-and-hold over the node in Network View and you'll see what attributes it's passing through…



```
       Point Attrs  P 3flt (Pos)
        Prim Attrs  shop_materialpath str 19
  2 Vertex Attrs  N 3flt (Nml) : uv 3flt

        163 Prim  fuselage 29,015 : Part01 8 : Pa
```

Ultimately for a Merge node to not complain, all the attributes need to match up at the correct levels (e.g. if the UVs are on the vertex, then the UVs need to be on the vertex for all of the inputs)....

- If you need to move around an attribute, use the Attribute Promote node
- If you need to delete an attribute, use the Attribute Delete node
- If you need to generate normals, use the Normal node
- If you need to generate UVs, use the UV Unwrap node

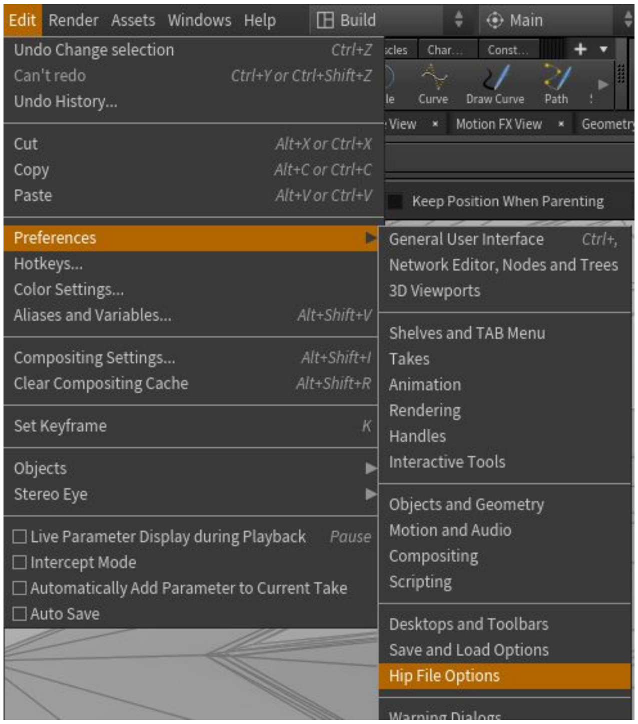
It's super important to remember that the attributes MUST BE APPLIED AT THE SAME LEVEL. Having UVs for the 1st input on points while having them for the 2nd input on vertex IS A MISMATCH and needs to be corrected (use Attribute Promote for this).

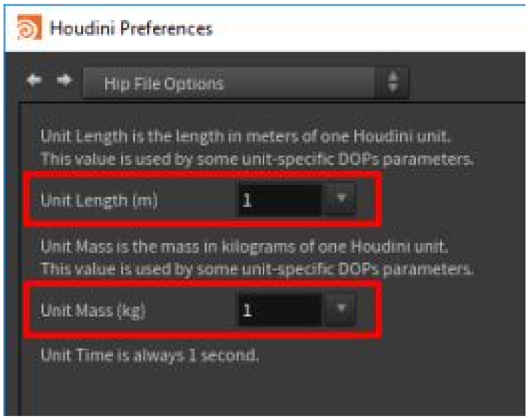
Note that some warnings may be okay. For example, you may have certain groupings (see section on Group nodes and/or materials) in one input but not the other. Groups are put in as attributes by Houdini.

Scene Scale

Scene scale is a super important concept when it comes to Houdini. The size of your objects has a direct effect on the way in which your simulations run. This is exactly like when you were using Box2D on Android.

By default, Houdini is set to treat 1 unit as 1 meter in length / 1 kg in weight. You generally don't want to change these values, but you can do so by going to Edit -> Preferences -> Hip File Options…





Unit Length is the length in meters of one Houdini unit. This value is used by some unit-specific DOPs parameters.

Unit Length (m) 1

Unit Mass is the mass in kilograms of one Houdini unit. This value is used by some unit-specific DOPs parameters.

Unit Mass (kg) 1

Unit Time is always 1 second.

> **NOTE**: If you're having problems visualizing this, think of a building collapsing. How long did it take the world trade center building to fall to the ground? Now imagine if the buildings were shrunken down to the height of a smartphone.
>
> In the new shrunken scale, the building would fall down much faster. The pieces falling to the ground would be much lighter. They wouldn't be in the air as long. They'd have less impact when they hit the ground. They'd have less weight. etc..

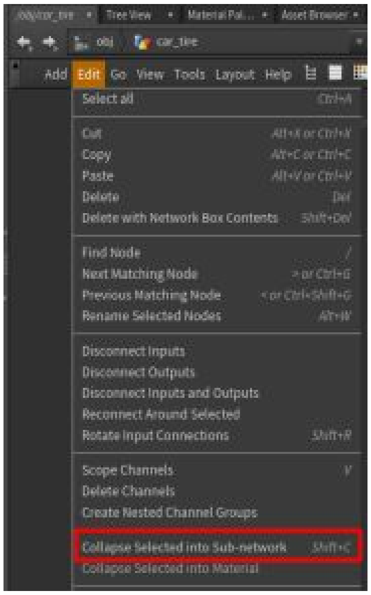
Subnetworks

Subnetworks are essentially groupings of nodes packaged together into a single node.

To create a subnetwork…

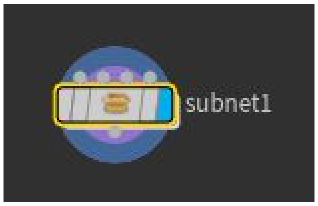
1. Go to the Network View
2. Select the nodes you want to package up

3. Go to Edit -> Collapse Selected Into Subnetwork


/obj/car_tire
Tree View
Material Pal...
Asset Browser
obj
car_tire
Add
Edit
Go
View
Tools
Layout
Help
Select all Ctrl+A
Cut Alt+X or Ctrl+X
Copy Alt+C or Ctrl+C
Paste Alt+V or Ctrl+V
Delete Del
Delete with Network Box Contents Shift+Del
Find Node /
Next Matching Node > or Ctrl+G
Previous Matching Node < or Ctrl+Shift+G
Rename Selected Nodes Alt+W
Disconnect Inputs
Disconnect Outputs
Disconnect Inputs and Outputs
Reconnect Around Selected
Rotate Input Connections Shift+R
Scope Channels V
Delete Channels
Create Nested Channel Groups
Collapse Selected into Sub-network Shift+C
Collapse Selected into Material

HINT: Ctrl+A to select all

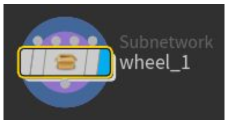
All those nodes will now get put into a single node that you can double-click into…

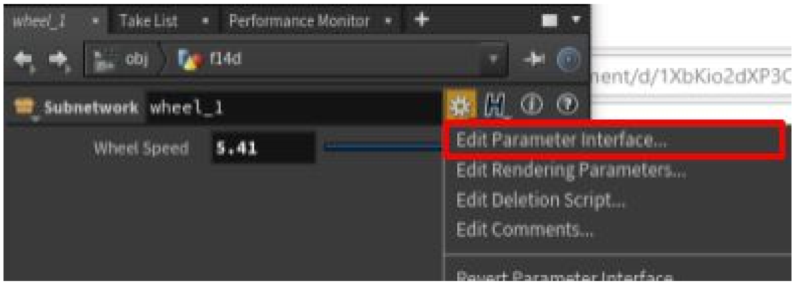


Custom Inputs / Parameters

NOTE: Subnetworks will always have 4 baked in inputs. There's no way to get rid of these or add more. The input connectors will always show up. But, you can add in custom parameters in the parameter view. See the following links for more info…
http://forums.odforce.net/topic/23798-subnetwork-can-i-remove-inputs/
http://forums.odforce.net/topic/10387-more-than-4-inputs-in-a-subnet/

To create custom inputs for your subnetwork, begin by selecting the subnetwork in the network view. Then go into the parameter view, choose the gear, and select Edit Parameter Interface.

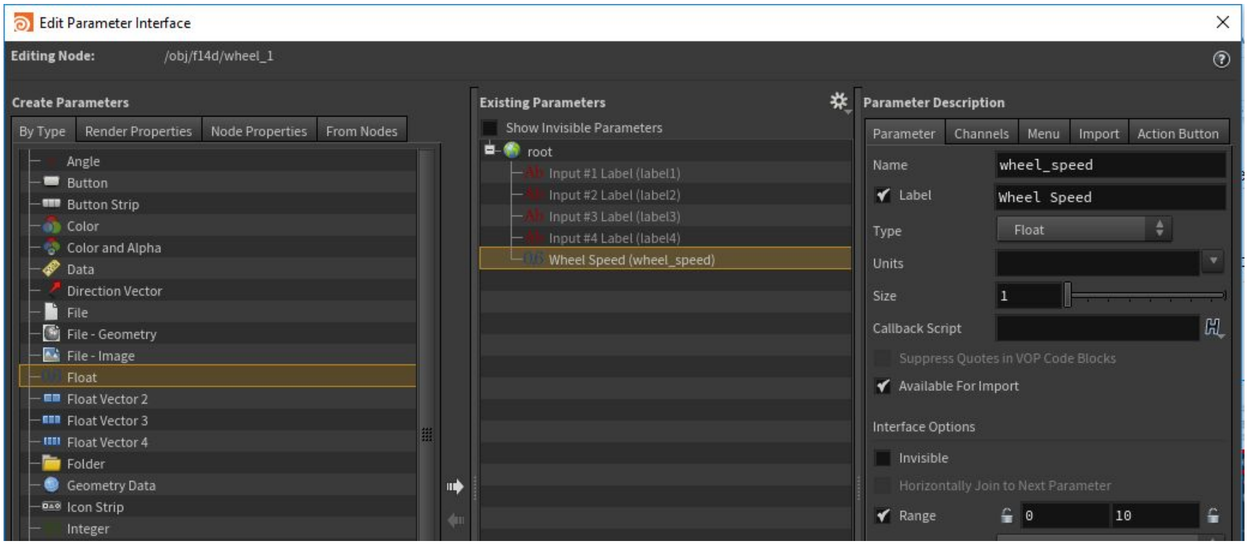




Subnetwork wheel_1

Wheel Speed 5.41

You'll be presented with the Edit Parameter Interface dialog.



Editing Node: /obj/f14d/wheel_1

Name wheel_speed

☑ Label Wheel Speed

Type Float

Size 1

☑ Available For Import

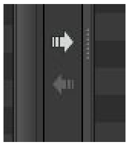
☐ Invisible

☑ Range 0 10

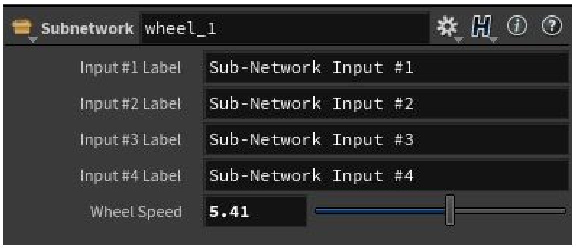
The window is split into 3 sections…

1. Create Parameters
2. Existing Parameters
3. Parameter Description.

To create a new parameter, either drag-and-drop from 1 to 2 or select in one and hit the right arrow in between the two sections….



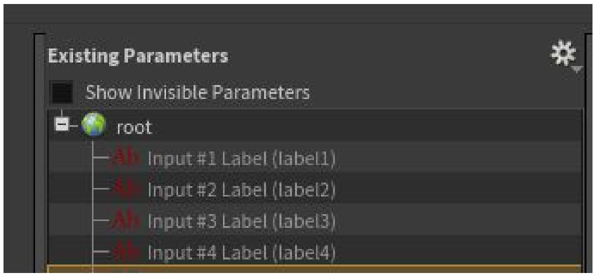
Once created, set a name and label in 3. You'll see your new parameter show up in the Parameter View when you select…



Subnetwork wheel_1

Input #1 Label	Sub-Network Input #1
Input #2 Label	Sub-Network Input #2
Input #3 Label	Sub-Network Input #3
Input #4 Label	Sub-Network Input #4
Wheel Speed	5.41

NOTE: If you want to move over existing parameters from one of the nodes inside the subnetwork, you can click the label of that parameter in the properties pane and drag it to root node in Existing Parameters. The Edit Parameters Interface isn't a modal dialog box (you can still click around in the main Houdini window and do stuff even though the Edit Parameters Interface dialg is open and on top)....

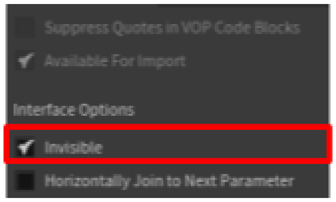


Existing Parameters

□ Show Invisible Parameters

- root
  - Input #1 Label (label1)
  - Input #2 Label (label2)
  - Input #3 Label (label3)
  - Input #4 Label (label4)

NOTE: You can stop the 4 baked inputs (e.g. Input #1 Label) from showing up in the parameter view if you choose them in the Edit Parameter Interface dialog and select to hide them. This only hides them in the parameter view… the input connectors on the subnetwork node will still be there.



□ Suppress Quotes in VOP Code Blocks

✓ Available For Import

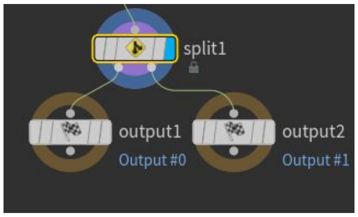
Interface Options

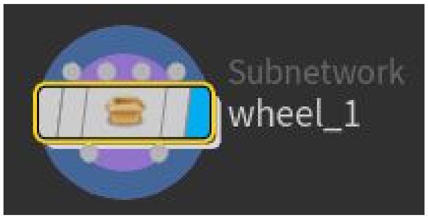
✓ Invisible

□ Horizontally Join to Next Parameter

Custom Outputs

By default, I think Houdini takes the first output of the first leaf node in the tree. If you want multiple outputs, you can drop output nodes in your subnetwork. Each output node will show up as a new output connector on your subnetwork…





Exporting (Digital Assets)

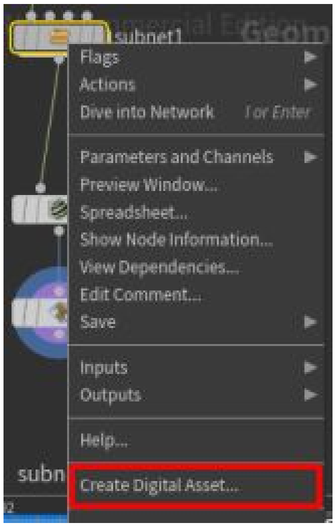
Houdini lets you export your subnetwork as a Houdini Digital Asset. When you do this, you can have it available for reuse in other houdini scenes that you work on / make it available for other artists by going to it in the Tab menu (it'll be just like any of the other hardcoded nodes that Houdini provides).

One of the other good things about Houdini Digital Assets are that you can load them up in other tools (e.g. Maya or Cinema 4D) via Houdini Engine and apply it to whatever it is you're working on in those tools.

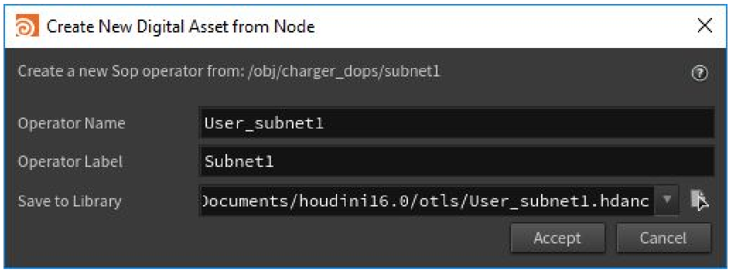
> **NOTE**: Houdini Engine doesn't have an apprentice edition so I couldn't try this out. But, the video showed how it worked with Maya. You just have to load the plugin and then a Houdini menu shows up in the main menu that you can use to load it up and assign stuff to the inputs/properties/etc...

To export your subnetwork as a Houdini Digital Asset…

1. Right-click it in the network view and select Create Digital Asset…



2. You'll be presented with a dialog to provide a name+label+location to save in and hit Accept…


Create New Digital Asset from Node

Create a new Sop operator from: /obj/charger_dops/subnet1

Operator Name: User_subnet1

Operator Label: Subnet1

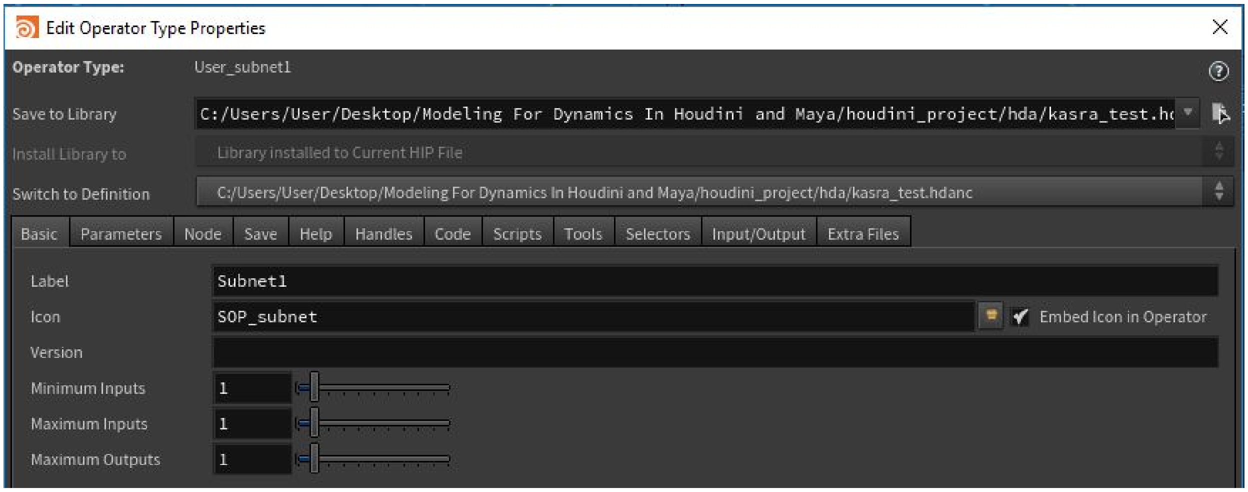
Save to Library:)ocuments/houdini16.0/otls/User_subnet1.hdanc

Operator Name → what shows up in the tab menu
Operator Label → the default name once placed
Save to Library → disk location to save to

NOTE: By default the save location will be the otls subdirectory of your main Houdini preferences folder. If you choose this, it'll be made available to any project you work on. If you only want to make it available to scenes in your current project, you can put it in the hda subdirectory of your current project directory ($HIP/hda/my_node.hdanc).

NOTE: otls stands for Operator Type LibrarieS.

3. You'll be presented with a dialog t o provide the operator properties…

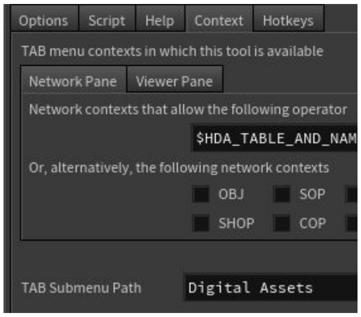


Operator Type: User_subnet1
Save to Library: C:/Users/User/Desktop/Modeling For Dynamics In Houdini and Maya/houdini_project/hda/kasra_test.hc
Install Library to: Library installed to Current HIP File
Switch to Definition: C:/Users/User/Desktop/Modeling For Dynamics In Houdini and Maya/houdini_project/hda/kasra_test.hdanc

Label: Subnet1
Icon: SOP_subnet ☑ Embed Icon in Operator
Version:
Minimum Inputs: 1
Maximum Inputs: 1
Maximum Outputs: 1

There are lots of options here, but the most tabs are…

a. Basic should be pretty straight forward.
b. Help is just a textbox where you document what your node does.
c. Tools is where you can set what Sub-menu of the Tab menu your Digital Asset pops-up under -- TAB Submenu Path under the Context tab.

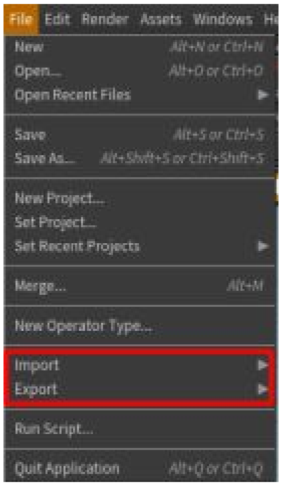


TAB menu contexts in which this tool is available
Network contexts that allow the following operator: $HDA_TABLE_AND_NAM
Or, alternatively, the following network contexts
☐ OBJ ☐ SOP
☐ SHOP ☐ COP
TAB Submenu Path: Digital Assets

d. Input/Output is where you can set the name/label of the input and output connectors on your node.

Importing/Exporting Assets

You can import/export assets through the file menu…

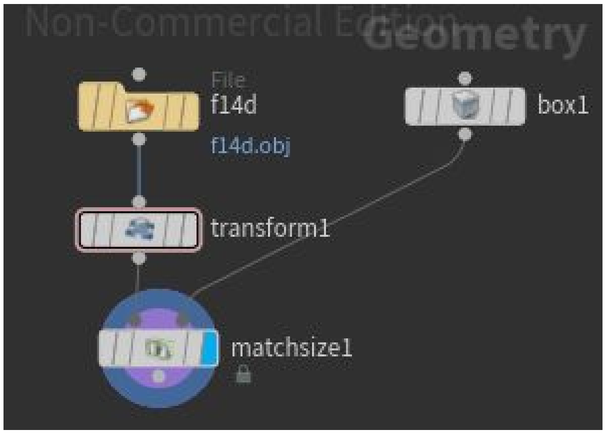


If you're importing assets, make sure that you place your assets inside of your project folder in the CORRECT subfolder. For example, all imported geometry should sit in /geo. It looks like the import isn't a one-time thing -- the file being imported is referenced continually.

Correcting Imported Geometry

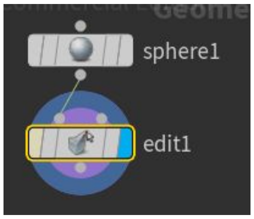
When you're importing geometry, the dimensions of the object may not be what they are in the real world -- meaning that your simulations will come out wrong. If you want to resize your imported geometry to the correct dimensions (assuming you know what those dimensions are)...

1. Go into the imported model's node.
2. Normalize the center/translate/whatever with a transform node (if required).
3. Add a box node -- size it to the correct size.
4. Add a match size node -- feed in the model and the box.



Component Selection and Manipulation

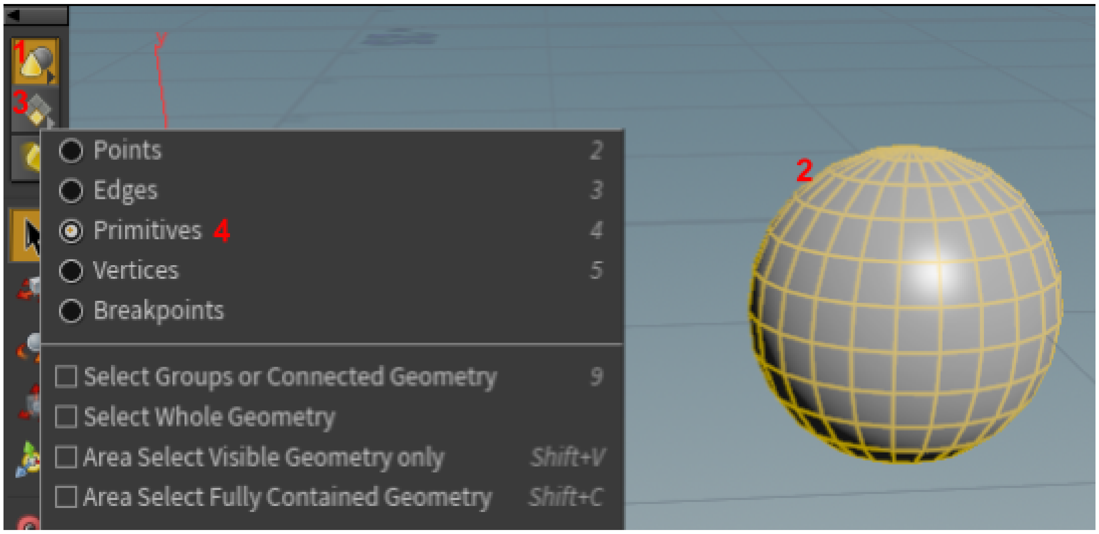
You need to be in the scene view to do this. Note that any modifications on components are added to your network as an Edit node.



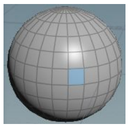
> **HINT**: For all selections (even when you're doing marquee selections). Hold Shift to add to your existing selection / Ctrl to remove from existing selections. The mouse cursor should change to show a mini + or - symbol.
>
> **NOTE**: There's more on selections in the Scene view notes above.

Selecting Primitives (Faces)

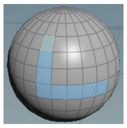
1. Select object selection button
2. Select your object
3. Right-click component selection button (or left-click and hold)
4. Choose Primitives in the pop-up


1
3
○ Points 2
○ Edges 3
● Primitives 4 4
○ Vertices 5
○ Breakpoints
□ Select Groups or Connected Geometry 9
□ Select Whole Geometry
□ Area Select Visible Geometry only Shift+V
□ Area Select Fully Contained Geometry Shift+C
2

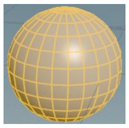
- Click to select a face



- Click to select a face + hold A and select another face to select a path of faces

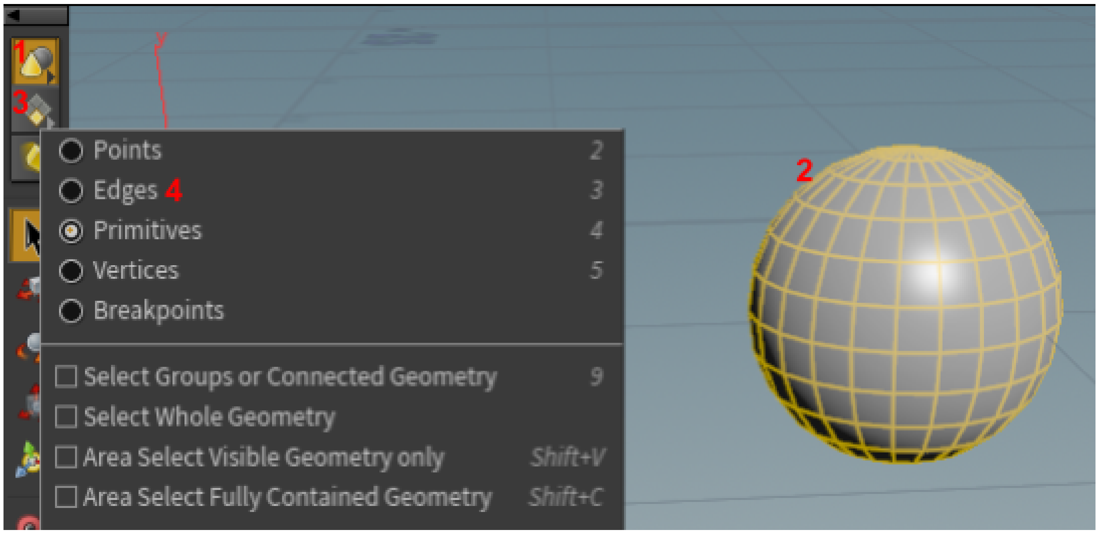


- Double-click to select all faces

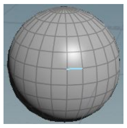


Selecting Edges

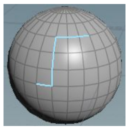
1. Select object selection button
2. Select your object
3. Right-click component selection button (or left-click and hold)
4. Choose Edges in the pop-up


1
3
○ Points 2
○ Edges 4 3
● Primitives 4
○ Vertices 5
○ Breakpoints
□ Select Groups or Connected Geometry 9
□ Select Whole Geometry
□ Area Select Visible Geometry only Shift+V
□ Area Select Fully Contained Geometry Shift+C
2

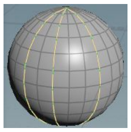
- Click to select an edge



- Click to select an edge + hold A and select another edge to select a path of edges



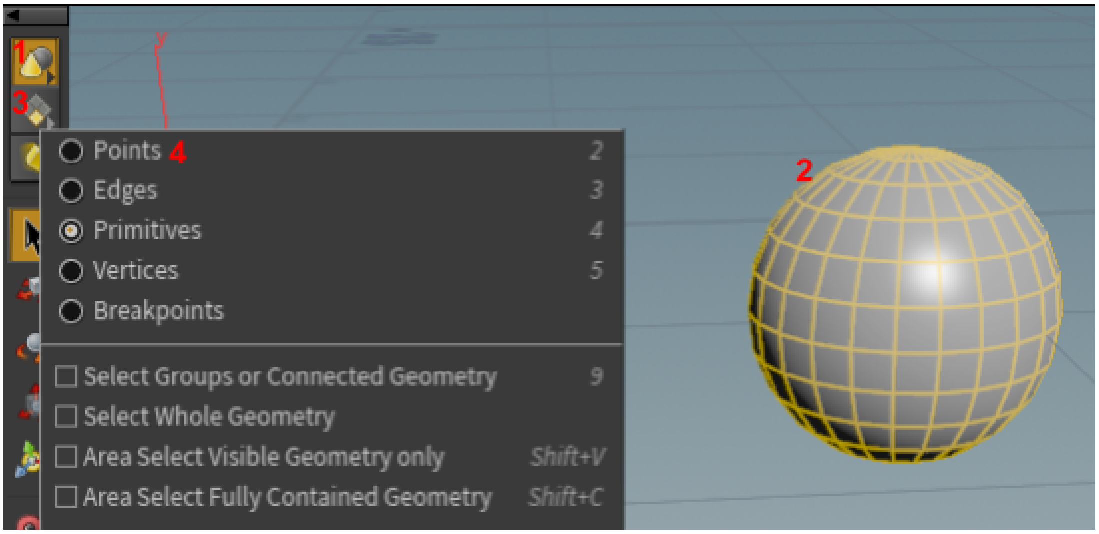
- Double-click to select an edge loop



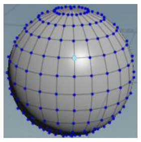
Selecting Points

HINT: A point is simply a point in space as defined by four numbers (X, Y, Z, W). A vertex is a reference to a point. Points are what you want, even though you may think of them as vertices.

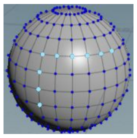
1. Select object selection button
2. Select your object
3. Right-click component selection button (or left-click and hold)
4. Choose Points in the pop-up


1
3
Points 4 2
Edges 3
Primitives 4
Vertices 5
Breakpoints
□ Select Groups or Connected Geometry 9
□ Select Whole Geometry
□ Area Select Visible Geometry only Shift+V
□ Area Select Fully Contained Geometry Shift+C
2
y

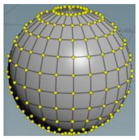
- Click to select a point



- Click to select a point + hold A and select another point to select a path of points

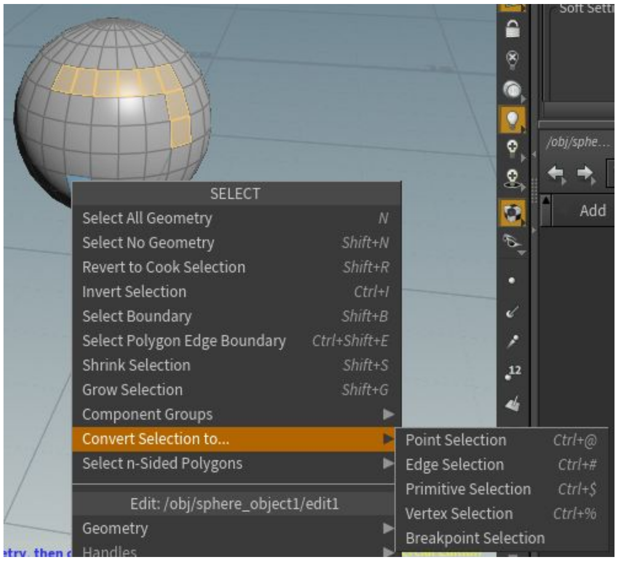


- Double-click to select all points



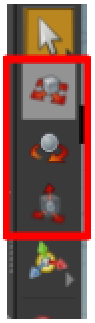
Converting between Primitive/Edge/Point Selections

1. Make your component selection
2. Right-click anywhere in the viewport
3. Choose Convert Selection to -> (component type here)



Scale/Rotate/Translate Tools

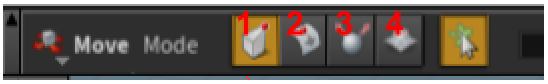
The scale/rotate/translate tools work the way they normally do. Select via the left toolbar…



Or, use the hotkeys…

- E = Scale
- R = Rotate
- T = Translate

When you’re in component mode (instead of object mode), you can select what mode your manipulation tool uses via the top toolbar…



1. Edit mode -- The typical mode (this is what you expect to happen)
2. Slide on surface -- The change is constrained to the surface made up by the surrounding area. Use this to try to retain the shape of your geo.
3. Peak -- Unsure what this does??? The opposite of slide on surface? It just protrudes?

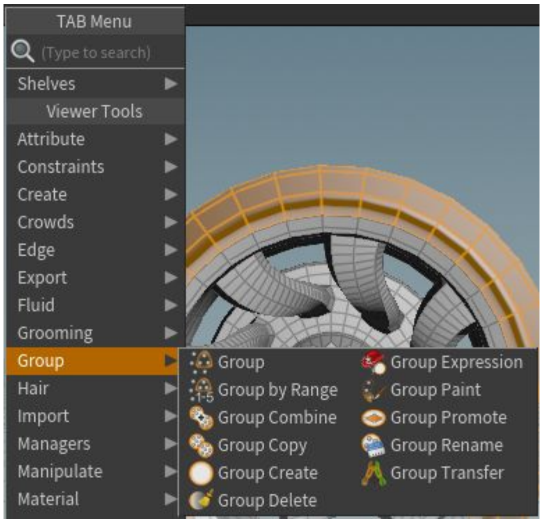
4. Sculpt -- Sculpts. You can choose brush size, etc… I don't know the exact details of this.

Group Nodes

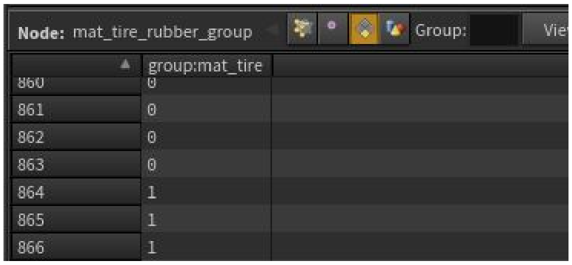
Groups are a way to bundle up geometry components together for whatever reason. Most of the time we want to do this bundling because we want to apply materials to specific parts of the geometry. For example, if you had a mesh of a car, you would want a separate material applied to the glass vs the rest of the car body.

> **NOTE**: You can use groups in pretty much any node that asks you for a group, or you can manually select the components to apply to. Creating a group makes things more organized, and the group created can be used in multiple places using the same reference.
>
> It's like using a constant in your code (private static final int DEFAULT_VALUE = 5) vs using a literal (5).

You can create and modify groups with the various Group nodes. Most of these are self-explanatory, and the important ones are detailed in the subsection below.

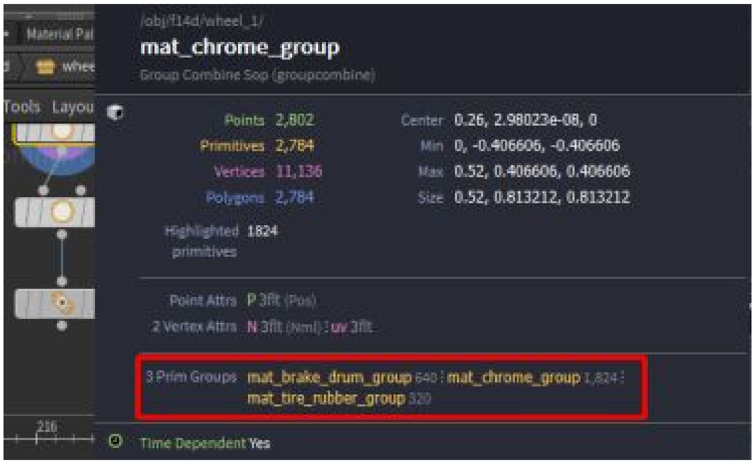


Once your group is available, you'll be able to see it in the geometry spreadsheet as an attribute. If the component is included in the group, you'll see a 1 for the attribute that that group is for. Otherwise you'll see a 0.


Node: mat_tire_rubber_group Group: Vie

	group:mat_tire
860	0
861	0
862	0
863	0
864	1
865	1
866	1

You'll also able to see the group (along with other groups propagated down) by MMB click-and-holding the node in the network view (or by clicking the information radial button -- does the same thing)...



/obj/f14d/wheel_1/

mat_chrome_group

Group Combine Sop (groupcombine)

Points 2,802
Primitives 2,784
Vertices 11,136
Polygons 2,784

Center 0.26, 2.98023e-08, 0
Min 0, -0.406606, -0.406606
Max 0.52, 0.406606, 0.406606
Size 0.52, 0.813212, 0.813212

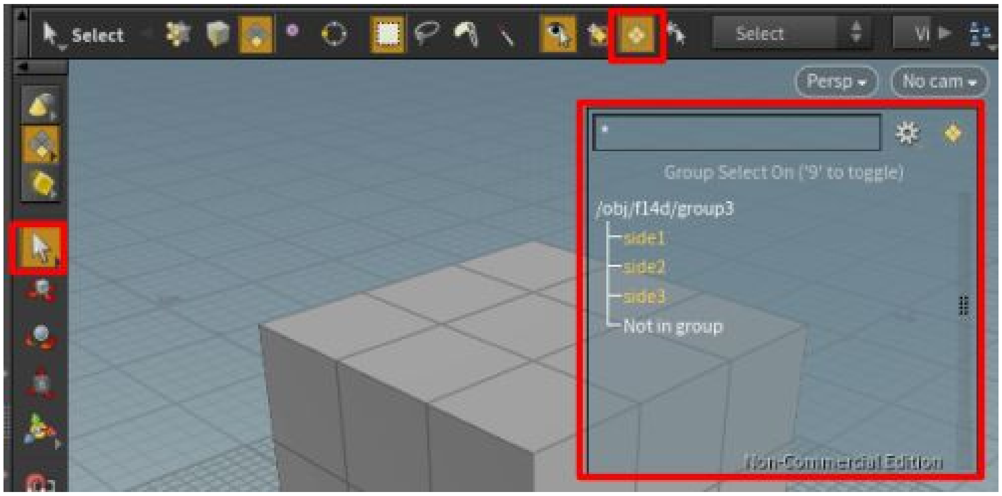
Highlighted primitives 1824

Point Attrs P 3flt (Pos)
2 Vertex Attrs N 3flt (Nml) ⁝ uv 3flt

3 Prim Groups mat_brake_drum_group 640 ⁝ mat_chrome_group 1,824 ⁝ mat_tire_rubber_group 320

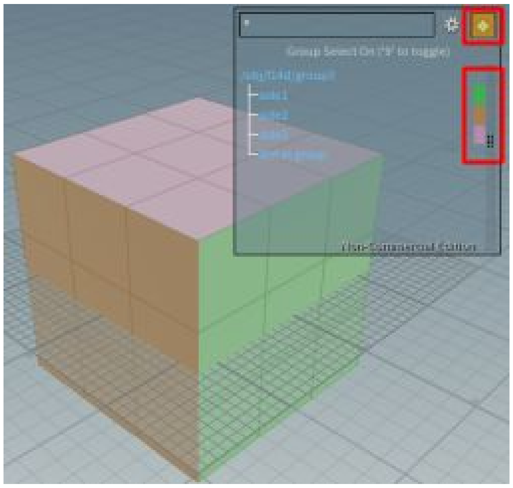
Time Dependent Yes

In addition to both of these, you can view / highlight the various groups in the scene view by choosing the select tool and enabling enabling 'Select Groups' in the top toolbar…



When you do this, a popup will show up on the lefthand side and you'll only be able to select by groups. You can select groups as a whole by clicking on the actual model causing it to highlight in the popup pane (and vice versa).

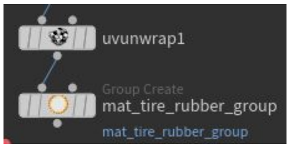
Selecting the diamond in the popup causes your models to get colorized by group, which is also super useful…



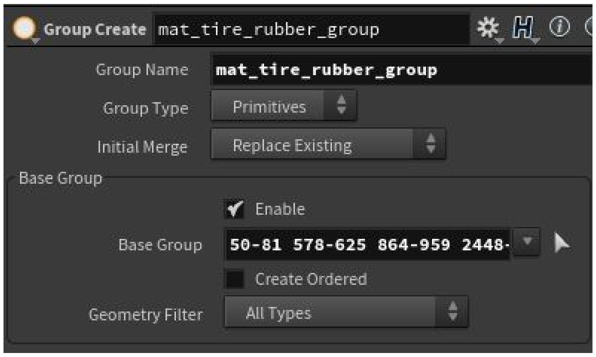
Group (Create component groups)

You can group up primitives(faces)/edges/points together for various operations down the line. For example, you can apply different materials to the various groups in your geometry.

Begin by selecting a the components you want to group together, and then dumping in a Group node and hooking it up to your geo. Doesn't matter where you do it.



Alternatively, you can make your selection after dumping in the node. Go into the properties pane and click the left arrow next to Base Group. Follow the prompts that show up in the Scene view.



Group Create mat_tire_rubber_group

Group Name: mat_tire_rubber_group

Group Type: Primitives

Initial Merge: Replace Existing

Base Group

☑ Enable

Base Group: 50-81 578-625 864-959 2448-

□ Create Ordered

Geometry Filter: All Types

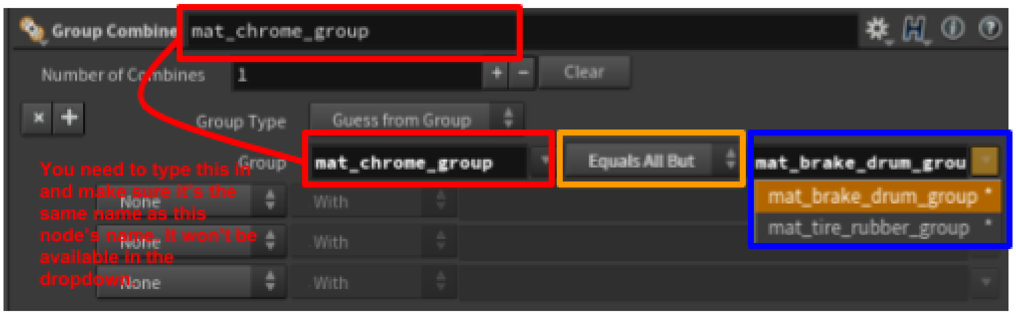
Notice that you can also set the group type here as well (e.g. do you want to group edges or points or primitives(faces)).

> **NOTE**: If you select nothing for the group, the entire geometry will be included in the group.
>
> **NOTE**: There are lots of different ways to group stuff by in the properties pane: keep by normals/edges/etc… I don't know what all of these actually do, but I think manually selecting the faces is the main way this node is used.
>
> **HINT**: It's good to give your groups descriptive names. For example the name mat_tire_rubber_group makes it pretty obvious what this group is for: it's the polys that the tire rubber material should be applied to.

Group Combine

one super important one is Group Combine. Group Combine lets you create groups based on simple math expressions where the operands are other groups that are being propagated down the chain.

One important use case for this is if you want to take all remaining geometry (geo that isn't part of a group) and stick it into some final group. Here's what the properties dialog looks like…



Group Combine mat_chrome_group

Number of Combines 1

Group Type: Guess from Group

Group: mat_chrome_group | Equals All But | mat_brake_drum_grou

mat_brake_drum_group *

mat_tire_rubber_group *

None | With

None | With

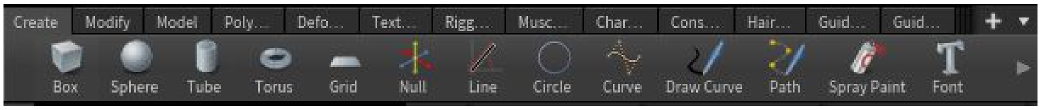
None | With

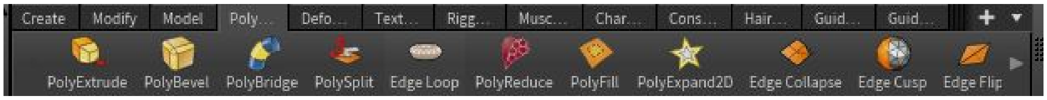
You need to type this in and make sure it's the same name as this node's name. It won't be available in the dropdown.

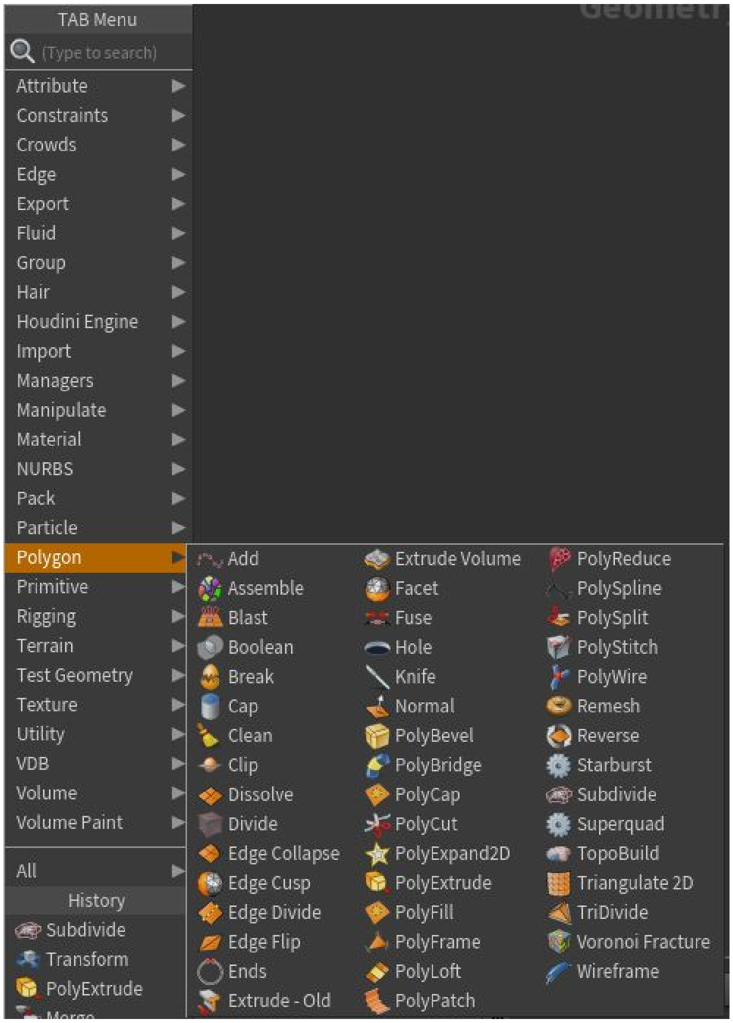
It's basically saying that this group (mat_chrome_group) is equal to everything that isn't in (Equals All But) all previous groups (mat_brake_drum_group and mat_tire_rubber_group -- notice how they both have asterisks at the end, that means that they're both selected).

Creating Geometry

Polygon modeling can be done via the Create and Polygon nodes. You can get to these by either going to the Tab menu (at root /obj level) or via the Create/Polygon shelf.

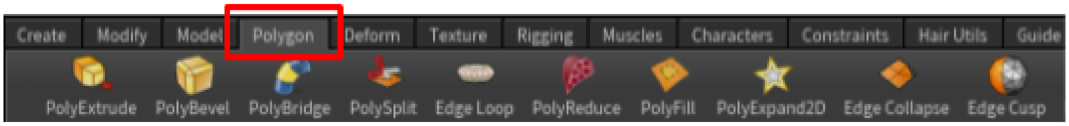






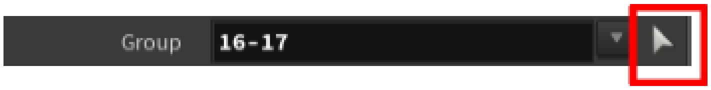
Interactive Modeling (Scene View)

If you prefer to use the Scene view to do your modelling, it is highly recommended that you use the shelf. Adding nodes via the shelf will interactively prompts you to put in the inputs rather than putting inputs via the parameter's view.



Non-Interactive Modeling (Network View)

If you prefer to use the Network graph view to do your modelling, you can still select which components you want the operations to affect in the scene view. All you have to do is go to the parameter view of the node and find where your component inputs should go (e.g. faces) and click the arrow button to the left…

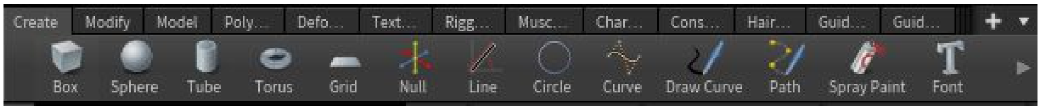


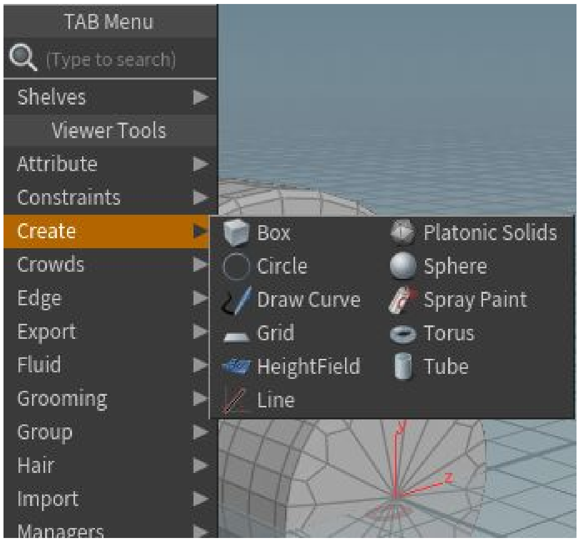
Group 16-17

> **NOTE**: Some tools won't have this. For example, trying to specify edge loops through the parameter window would be a nightmare -- use the shelf to interactively place those instead.
>
> **NOTE**: If you see that the input is a group, you can use pre-determined groups. See the Group Nodes section for more information.

Creating Primitives

To create a primitive, you can use the Create shelf or go to Create in the Tab menu





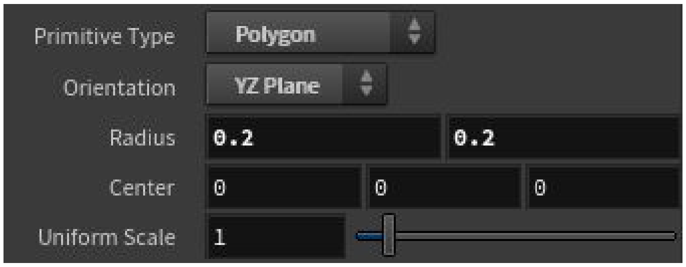
Common things you might want to create (because you're doing box modeling type stuff) is…

- Box
- Sphere
- Tube
- Torus

- Grid ← this is shorthand for plane
- Circle

Another cool thing you can do is put in text as polygons -- you can then do normal polygon operations on this text: extrude, etc.. etc..

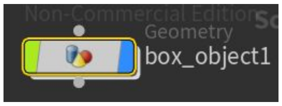
A lot of times, when you create a primitive, you'll need to orient it (move, scale, change the pivot point, rotate, etc..). You can do this through a transform node, but most of the time the primitive itself will give you the ability to orient it. For example, a circle node has the following orientation options built-in…

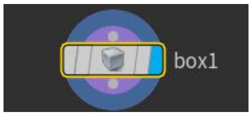


Primitive Type Polygon
Orientation YZ Plane
Radius 0.2 0.2
Center 0 0 0
Uniform Scale 1

Importing Geometry

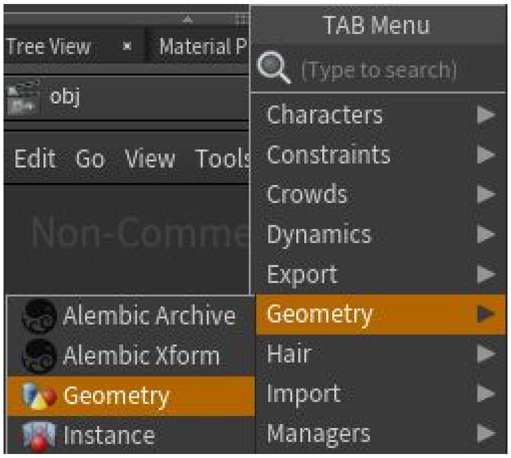
All of the primitives discussed in the previous section are created inside of a Geometry node. So for example, you can ask to create a Box in the scene view, but what that actually does is create a Geometry node called Box and shoves a Box node in there. If you traverse into that Geometry node, you'll actually see the inner Box node that creates the actual box geo.



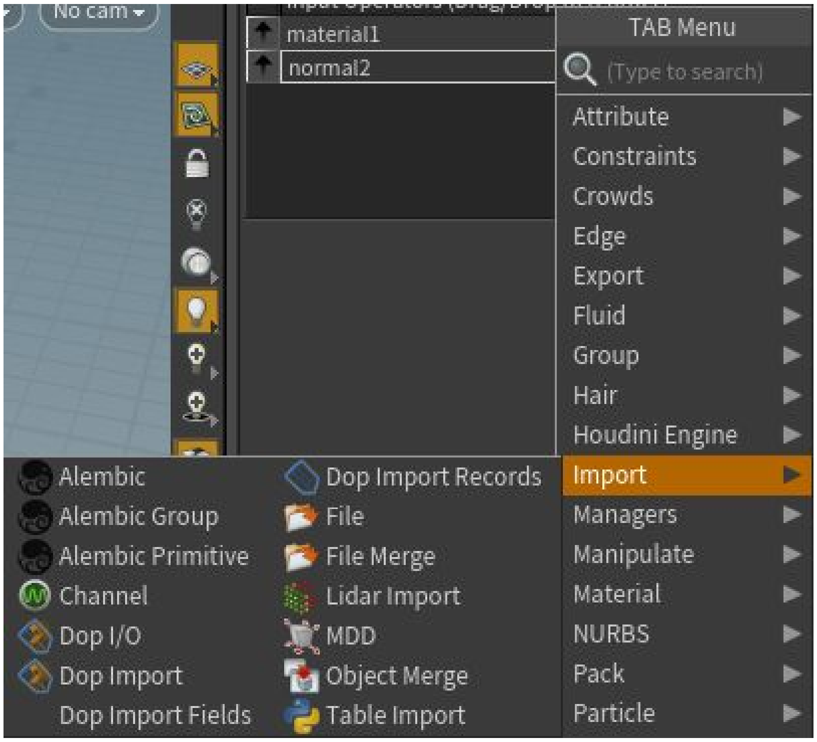
→


In the same vein, if you want to import some geometry, you can dump a basic Geometry node and add import nodes in there instead of nodes to create some primitive. To do so…

1. In the scene view, open the tab menu and search for Geometry (you can find this at the top /obj level in the network view as well -- tab menu).



2. Navigate into the new Geometry node.
3. Open the tab menu again (in the network view) and go under the Import section.



Many of the import options should be self-explanatory…

- Alembic probably imports alembic files
- File brings in files like OBJ
- Object Merge lets you copy over specify geometry from somewhere else in your scene (you have to give it a path)
- Table Import probably lets you shove in some python code or something

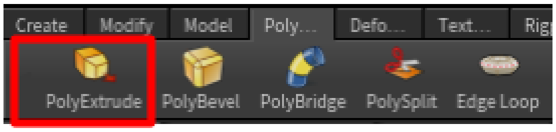
Geometry Nodes

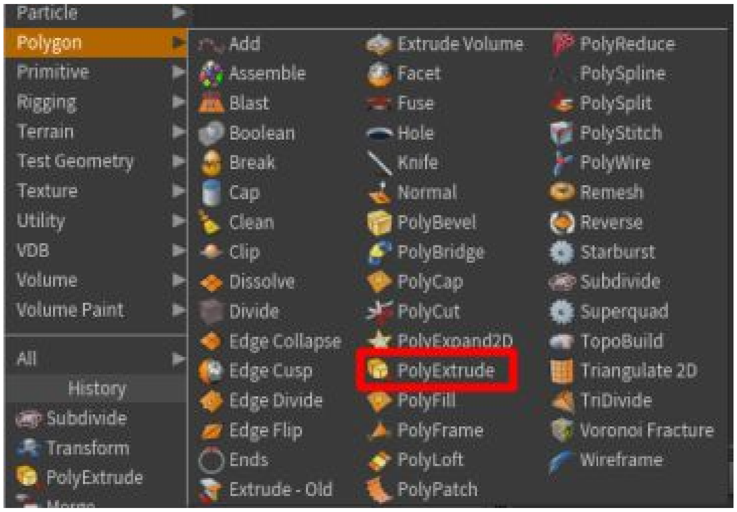
These nodes are for when you go into a geometry node. For example, if you create a Box in your scene view, but what that actually does is create a new geometry node and shoves a box in there.

These nodes will ONLY be available when you traverse into the geometry node!

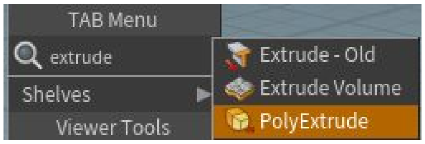
PolyExtrude (Extrude faces/edge)

The PolyExtrude tool extrudes polygon(s). To get to the PolyExtrude tool, you can use the the Tab menu or the Polygon shelf.



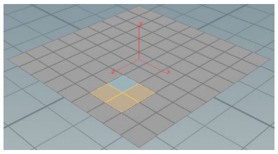


NOTE: Do not choose the other Extrude tools by accident. Those tools are totally different from the PolyExtrude tool.

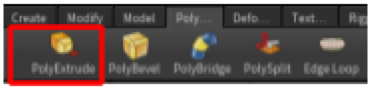


If you're working in interactive mode (via the scene view and Polygon shelf) you can extrude your polygons by…

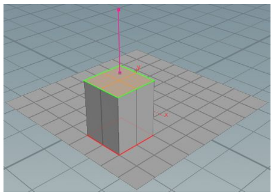
1. Choosing the components (e.g. faces) you want to extrude

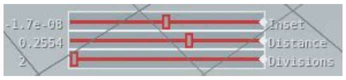


2. Clicking the PolyExtrude tool in the shelf



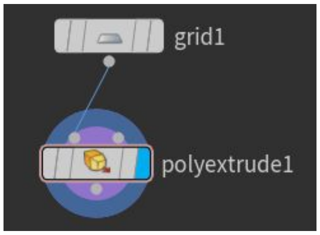
3. Dragging the handle and/or sliders that show up





-1.7e-08 Inset
0.2554 Distance
2 Divisions

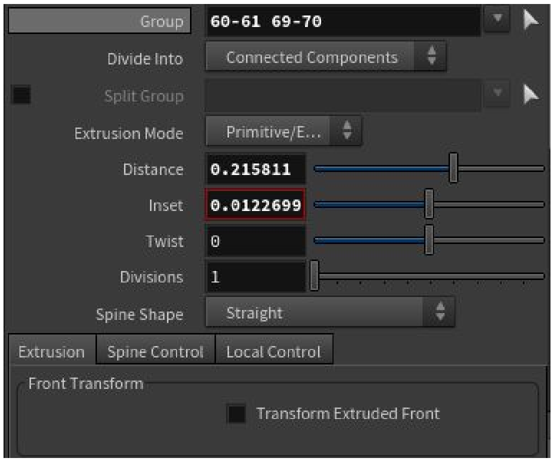
If you’re working in non-interactive mode (via the network view) you can extrude polygons by connecting your geometry to the first input of a PolyExtrude node…



NOTE: The second input is for curving the extrusion. See the Spine Shape section below.

HINT: A lot of times you want to extrude something multiple times quickly. If the last node in your networks graph is a extrude node (e.g. you just finished extruding something) and you want to follow up with another extrusion, hover your mouse in the scene view and press Q. The Q shortcut just adds in another node of the same type again. Just make sure you have the correct components still selected before pressing it.

Regardless of how you did things, the properties panel will give you a bunch of settings that’ll let you change how the extrusion works…



Group: 60-61 69-70

Divide Into: Connected Components

□ Split Group

Extrusion Mode: Primitive/E…

Distance: 0.215811

Inset: 0.0122699

Twist: 0

Divisions: 1

Spine Shape: Straight

Extrusion | Spine Control | Local Control

Front Transform

□ Transform Extruded Front

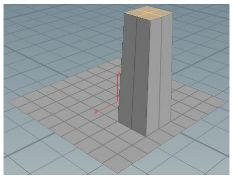
Here's an outline of what each of these means…

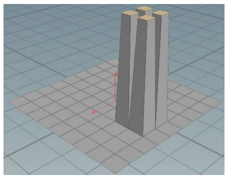
Group Components being extruded.

Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, it extrudes everything.

Divide Into If you want your extrusion to affect the selected components as a whole…

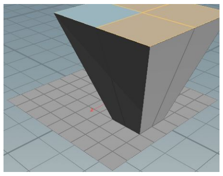




Exclusion Mode ???

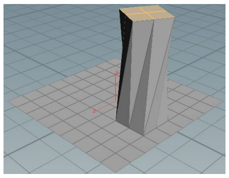
Distance How far to extrude.

Inset How much to expand/contract extruded the components.



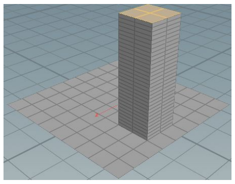
Twist

How much to twist the extrusion by.



Division

How many division the extrusion should have.



Spine Shape

Controls how the extrusion curves. Depending on the option you choose, this can either be…

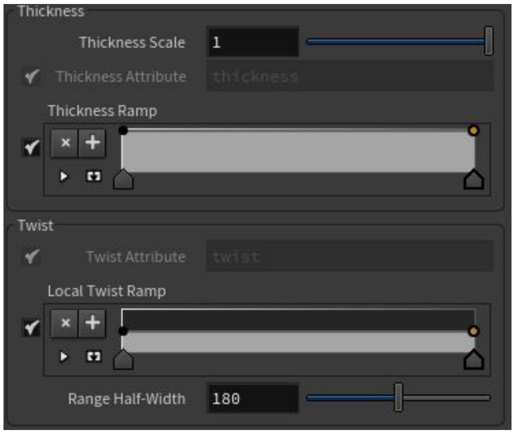
* straight (linear)
* curved (you define properties of the curve in the spline tab)
* curved from 2nd input (you feed in a NURBs curve into the 2nd input of the node)

NOTE: This seems to be very similar to Maya's extrusion.

Transform Extruded

If you choose this, you'll be presented with transform options just below your selection. With this option, you'll be able to transform the faces that you extruded out (e.g. translate/rotate/scale).

In addition to the options shown above, if you go to the Spine Control tab, you have the ability to change the thickness of the extrusion at various points + change the twist at various points. Just make sure that you have enough divisions in your extrusion.



Thickness

Thickness Scale 1

✓ Thickness Attribute thickness

Thickness Ramp

✓

Twist

✓ Twist Attribute twist

Local Twist Ramp

✓

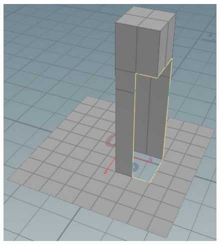
Range Half-Width 180

PolyFill / PolyCap (Fill holes)

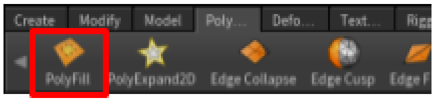
The PolyFill tool (called PolyCap in Houdini 15) lets you create faces between edges in your geometry. So if you have a hole in your geometry, you can use the PolyFill tool to create face(s) to fill in that hole.

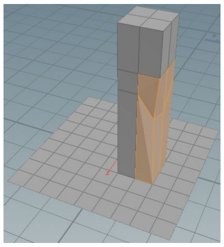
If you're working in interactive mode (via the scene view and Polygon shelf) you can extrude your polygons by…

1. Selecting the edges you want filled in

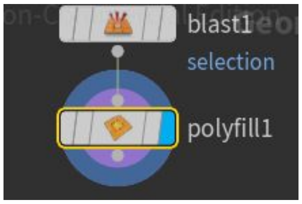


2. Clicking the PolyFill tool in the shelf

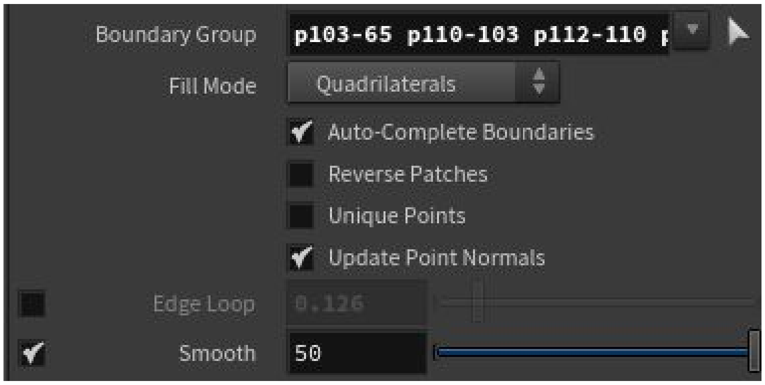




If you're working in non-interactive mode (via the network view) you can fill by connecting your geometry to the input of a PolyFill node…



Regardless of how you did things, the properties panel will give you a bunch of settings that'll let you change how the fill works…



Boundary Group p103-65 p110-103 p112-110 p

Fill Mode Quadrilaterals

☑ Auto-Complete Boundaries

□ Reverse Patches

□ Unique Points

☑ Update Point Normals

□ Edge Loop 0.126

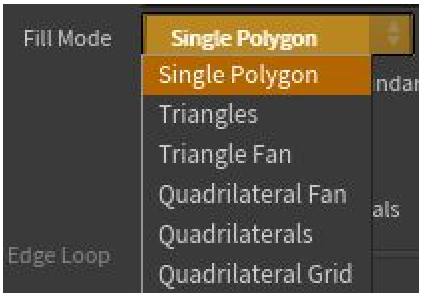
☑ Smooth 50

Boundary Group The edges to create the fill between.

Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, it tries to fill every 'gap' that it finds.

Fill Mode Defines the type of polygon(s) to use for the fill. Options include…



NOTE: Single Polygon means that only 1 polygon will be used. That means that all points on the selected edges have to exist on the same plane. If that isn't the case, it seems like it implicitly chooses the Triangles option, but none of edges for the generated faces will be selectable.

Auto-Complete Boun If your boundary only contains only part of a hole, this option tries to get the edges for the rest of the hole.

Reverse Patches Reverses the orientation of the created polygons. Orientation in this case means what??? It looks like it just creates the polys in reverse order.

Unique Points The generated polygons won't share their points with edges that they were created against. Meaning that technically this won't be a solid piece of geometry? It'll technically be 2 pieces?

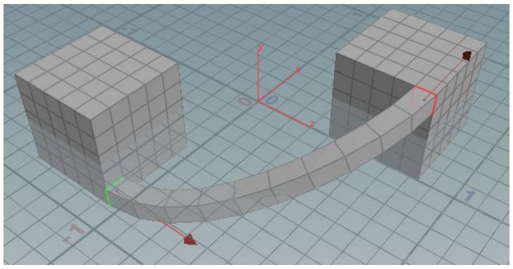
Update Point Norma The normals of the points that are getting filled will be updated to include the new faces?

Edge Loop Seems to add an extra edge around the geo that is created???

Smooth Seems to shift around how the fill geo that is created???

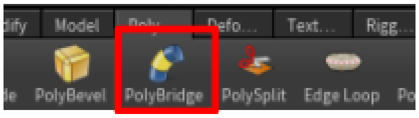
PolyBridge (Bridge between edges)

The PolyBridge tool lets you create a bridge between 2 sets of edges.



To interactively create a bridge (via scene view)...

1. Select the PolyBridge tool from the shelf



2. Select the source edges and press enter (you'll be prompted to do so in the scene view)
3. Select the dest edges and press enter (you'll be prompted to do so in the scene view)

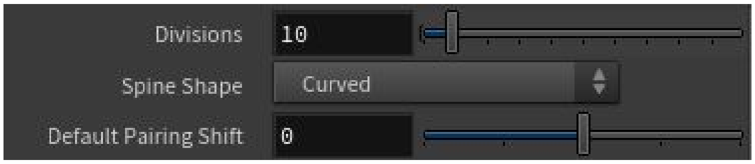
To non-interactively create a bridge (via network view)...

1. Drop a PolyBridge node and connect your geometry to the first input
2. In the properties panel, under the Footing tab, fill in Source -> Group
3. In the properties panel, under the Footing tab, fill in Destination -> Group

NOTE: The second input is for an external curve… if you want your bridge to follow some external curve.

NOTE: If you notice that the bridge is messed up upon creation (e.g. inside out), try clicking the Reverse Winding options (there's 2 of these, one for source and one for destination). If that doesn't work, try fiddling with the Default Pairing Shift slider option.

Regardless of how you did things, the properties panel will give you a bunch of settings that'll let you change how the bridge works…



Divisions 10

Spine Shape Curved

Default Pairing Shift 0

Divisions

The number of divisions you want your bridge to have.

Spine Shape

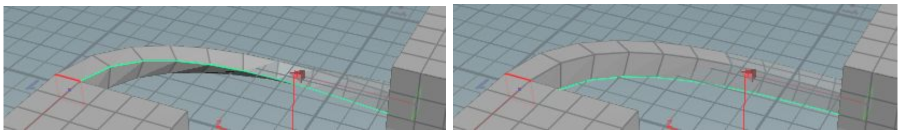
Controls how the bridge curve. Depending on the option you choose, this can either be…

* straight (linear)
* curved (you define properties of the curve in the spine tab)
* curved from 2nd input (you feed in a NURBs curve into the 2nd input of the node)

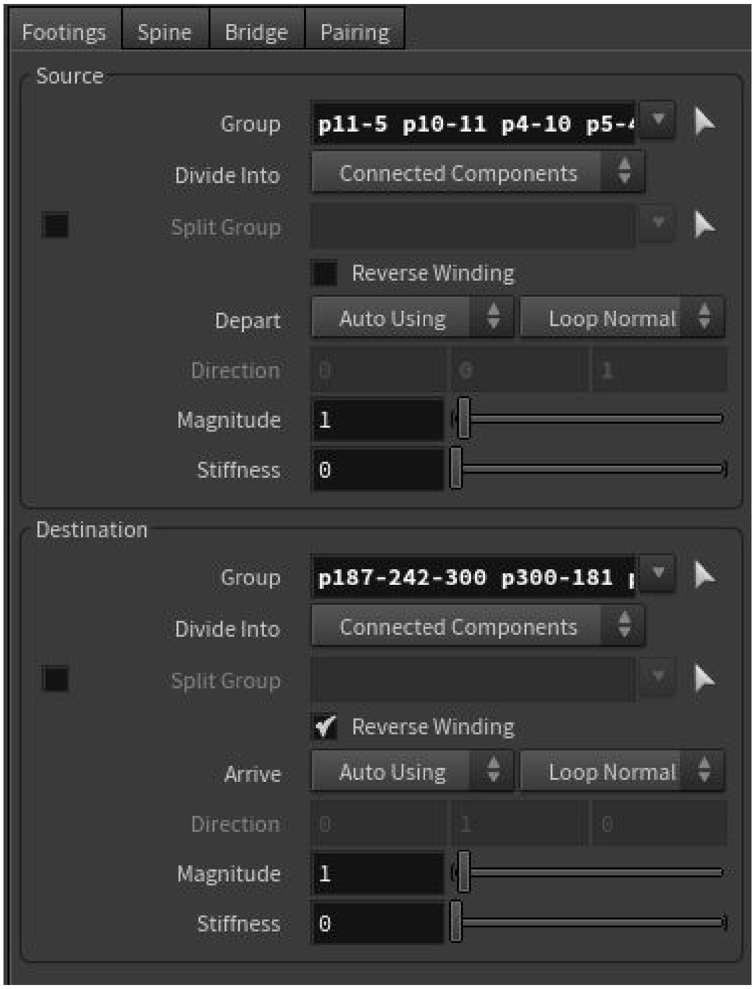
NOTE: This seems to be very similar to Maya's extrusion.

Default Pairing Shift

Shifts the mapping of the source edges to the destination edges. Change this if your bridge is coming out twisted.



The Footing tab in the properties controls the source and destination from which the bridge is created. It's split into 2 sections: Source and Destination. The same options are available for both.



Footings | Spine | Bridge | Pairing

Source

Group: p11-5 p10-11 p4-10 p5-

Divide Into: Connected Components

□ Split Group

□ Reverse Winding

Depart: Auto Using | Loop Normal

Direction: 0 0 1

Magnitude: 1

Stiffness: 0

Destination

Group: p187-242-300 p300-181

Divide Into: Connected Components

□ Split Group

☑ Reverse Winding

Arrive: Auto Using | Loop Normal

Direction: 0 1 0

Magnitude: 1

Stiffness: 0

Group

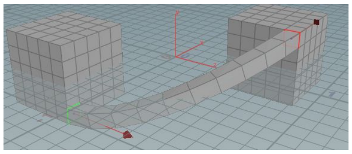
The source/destination edges for the bridge.

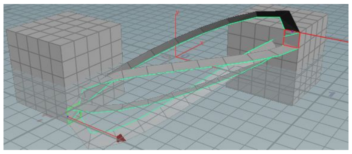
Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, does nothing.

Divide Into

If set to Connected Components, the bridge will be a solid piece.
If set to Individual Components, the bridge will be a exploded by edge.



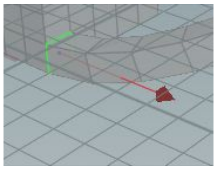
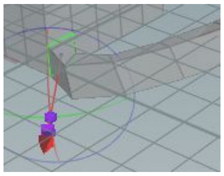


Reverse Winding

Try this if the bridge is coming out twisted or out of place.

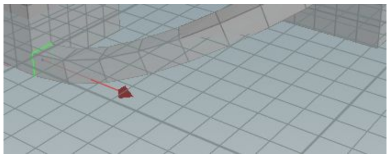
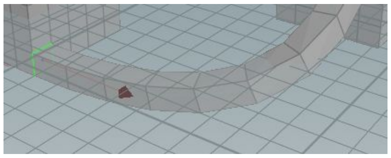
Depart/Arrive

Controls the direction that the bridge this starts/stops from.

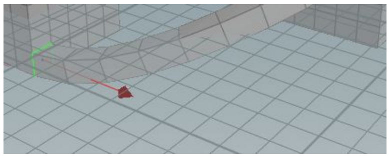
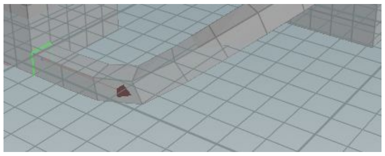
Magnitude

Blasts out (scales) the bridge from the source/destination position.

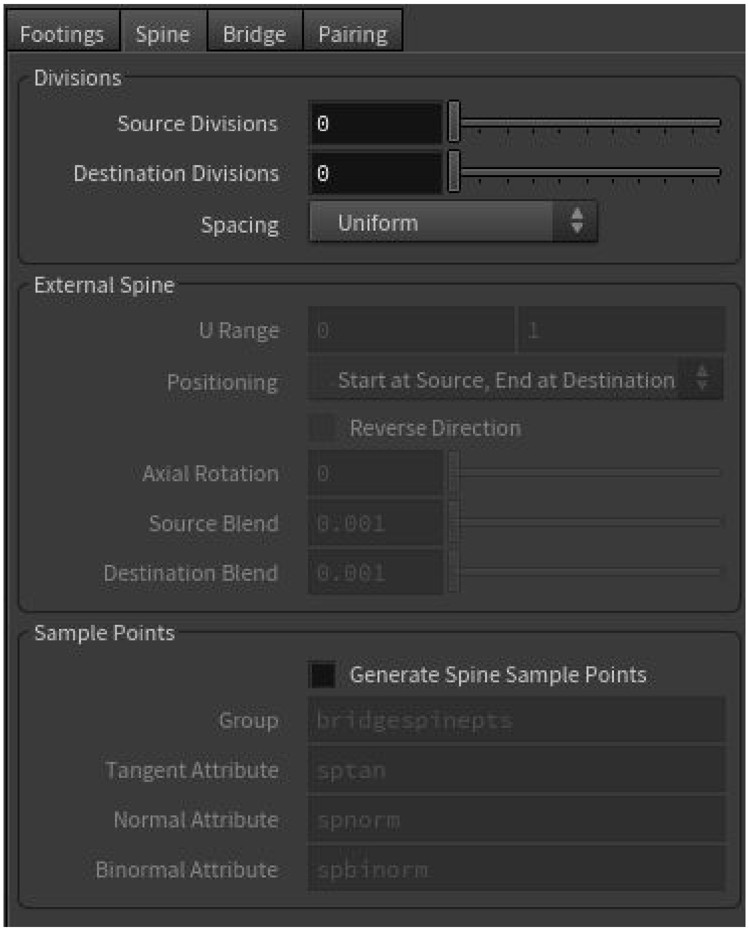



Stiffness

How stiff the bridge is from the source/destination position.

The Spine tab in the properties controls the spine of the bridge that is created.



Divisions

Source Divisions 0

Destination Divisions 0

Spacing Uniform

External Spine

U Range 0 1

Positioning Start at Source, End at Destination

□ Reverse Direction

Axial Rotation 0

Source Blend 0.001

Destination Blend 0.001

Sample Points

□ Generate Spine Sample Points

Group bridgespinepts

Tangent Attribute sptan

Normal Attribute spnorm

Binormal Attribute spbinorm

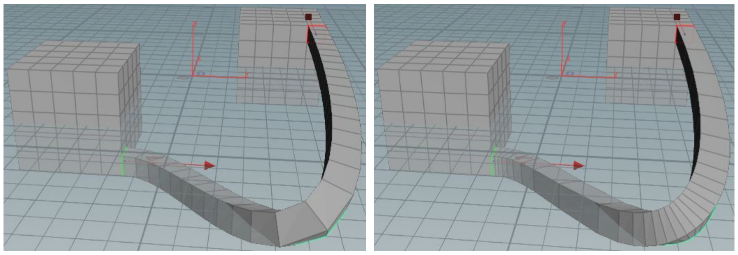
Source Divisions Adds divisions “to the start of the bridge.” I don’t know how this works, the divisions don’t seem to be bound the the start or the end.

Destination Divisions Adds divisions “to the end of the bridge.” I don’t know how this works, the divisions don’t seem to be bound the the start or the end.

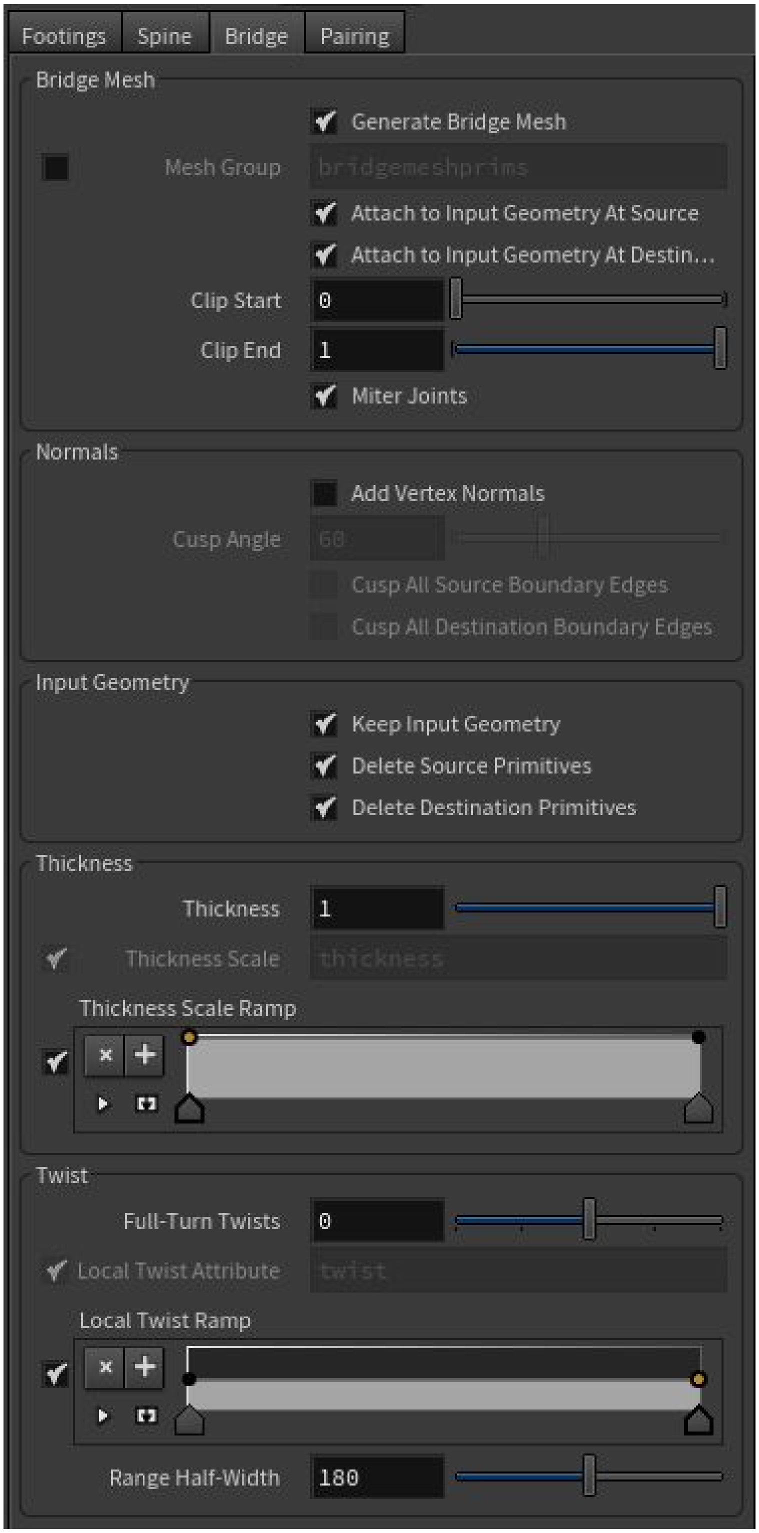
Spacing There are 2 options here...

* uniform
* curve sensitive

Curve sensitive is almost always the one you want. It adds in the divisions where the curve starts to twist and turn (where it needs more divisions). If you choose uniform it'll add it evenly.



The Bridge tab controls how the actual bridge gets created



Bridge Mesh

☑ Generate Bridge Mesh

□ Mesh Group: bridgemeshprims

☑ Attach to Input Geometry At Source

☑ Attach to Input Geometry At Destin...

Clip Start: 0

Clip End: 1

☑ Miter Joints

Normals

□ Add Vertex Normals

Cusp Angle: 60

□ Cusp All Source Boundary Edges

□ Cusp All Destination Boundary Edges

Input Geometry

☑ Keep Input Geometry

☑ Delete Source Primitives

☑ Delete Destination Primitives

Thickness

Thickness: 1

☑ Thickness Scale: thickness

☑ Thickness Scale Ramp

Twist

Full-Turn Twists: 0

☑ Local Twist Attribute: twist

☑ Local Twist Ramp

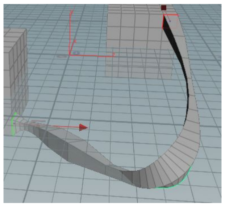
Range Half-Width: 180

Clip Start/End Clips the start/end of the bridge. That means that if this isn't 0, you'll see a gap at the start/end of your bridge.

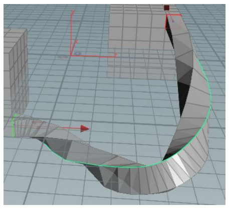
Normals If normals should be added for your bridge???

Delete Primitives Delete the faces/edges which these bridges were generated from???

Thickness How thick you want to be the bridge. You can change thickness in multiple places.

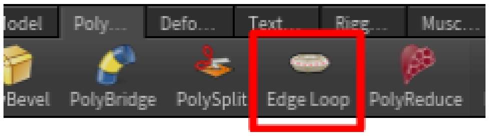


Twist How you want your bridge to twist. You can change the twist in multiple places.

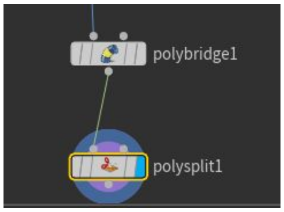


Edge Loop

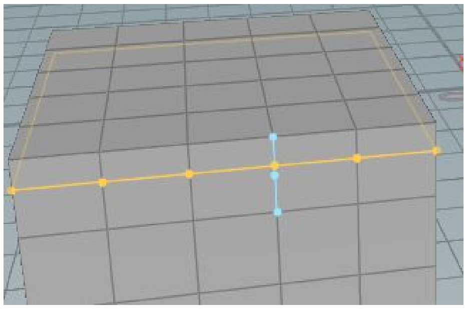
You can create an edge loop by going to the shelf and selecting Edge Loop…



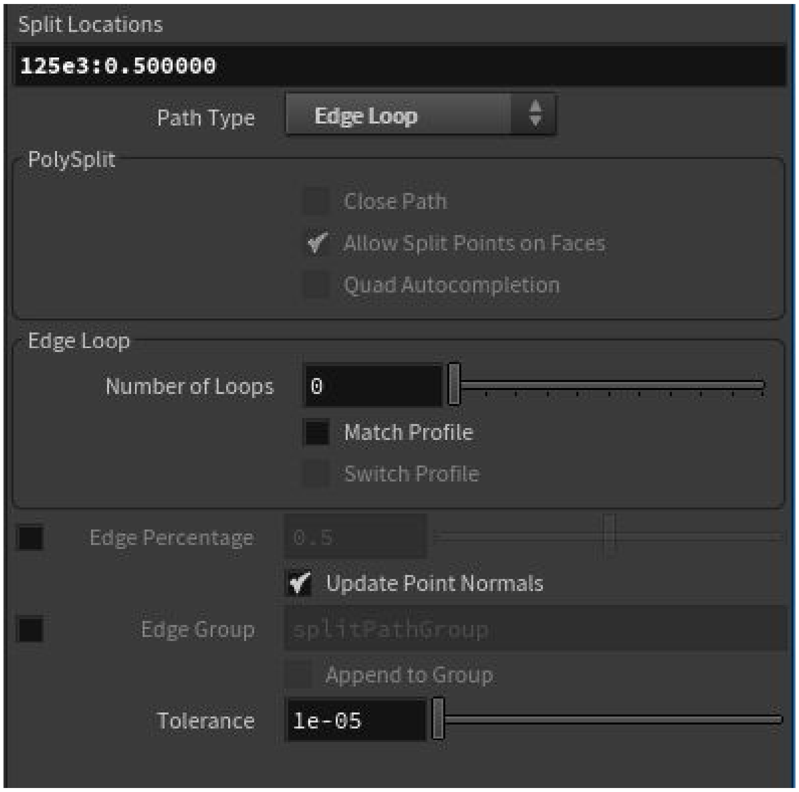
Or by dropping a PolySplit node in the network view and hooking your geometry into it…



Regardless of how you do it, you'll be prompted to select where you want the edge loop created in your scene view. You'll notice that as you move your mouse around your geo, it'll hover a yellow loop around where you're aiming at…



Click once to create the loop. If you need to adjust, go into the properties…



Split Locations

125e3:0.500000

Path Type: Edge Loop

PolySplit

- □ Close Path
- ☑ Allow Split Points on Faces
- □ Quad Autocompletion

Edge Loop

- Number of Loops: 0
- □ Match Profile
- □ Switch Profile

□ Edge Percentage: 0.5

☑ Update Point Normals

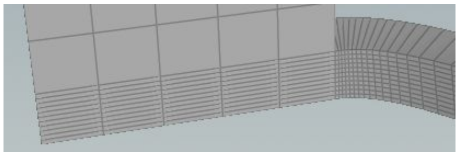
□ Edge Group: splitPathGroup

□ Append to Group

Tolerance: 1e-05

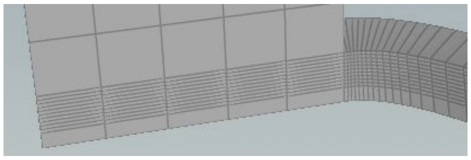
Split Locations The number after the colon is where the edge will get created. Use this if you want to make tiny tweaks.

Number of Loops If this is set to > 0, the number in split location (the one after the colon) is ignored and instead n new edge loops are created equidistant from each other.

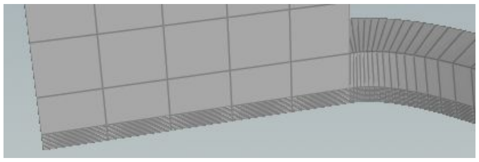


UNLESS Match Profile (just below Number of Loops) is checked. Then it'll create the equidistant loops starting from the split location.

Match Profile Create equidistant loops starting from the split location.



Switch Profile Flips the number of loops so that they END at the split location instead of start at the split location.



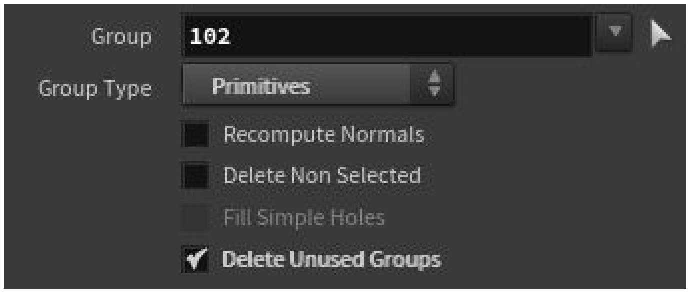
Blast (Delete faces and points)

The blast tool lets you remove faces (and points).

To interactively blast faces (via scene view), select the faces and hit the DEL key.

To non-interactively blast faces (via network view), connect your geo to a blast node.

Regardless of how you did things, the properties panel will give you a bunch of settings that'll let you change how the blast works…



Group 102

Group Type Primitives

□ Recompute Normals

□ Delete Non Selected

□ Fill Simple Holes

☑ Delete Unused Groups

Group The points/faces to delete.

Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, it deletes everything.

Group Type If you’re dealing with faces or points.

Recompute Normals Recomputes the normals for whatever that was a sibling of whatever got deleted.

Delete Non Selected Deletes the inverse of whatever you specified in Group. That means that whatever you specified in Groups is what will be kept.

NOTE: This is super useful for splitting off a single piece of geo for dynamic simulations. For example, imagine you have a building as a whole piece. Select the windows and blast them for just the walls. Then, copy that blast node and select this option so you have just the windows.

You can then import the windows into another Geometry node via an Object Merge node (look it up in the Create Geometry section of this doc) and treat it as a separate dynamics entity from the wall.

Dissolve (Delete edges)

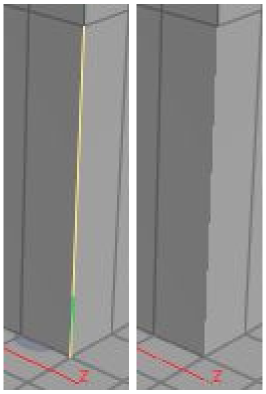
The dissolve tool lets you remove edges.

To interactively dissolve edges (via scene view), select the edge and hit the DEL key.

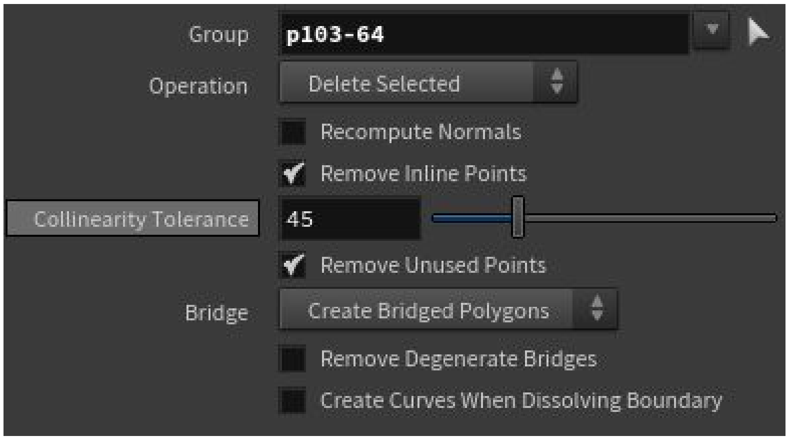
To non-interactively blast faces (via network view), connect your geo to a dissolve node.

> **NOTE**: Remember that an edge can be shared by 2 faces. If you’re going to dissolve an edge , that means that the 2 faces it connects will be merged into 1. The points on that merged face have to sit on the same plane, otherwise it’ll implicitly generate triangles. The following example shows what happens when you delete an edge but the resulting points for the new face aren’t on the same plane (it’s like you never deleted anything at

all)...



Regardless of how you did things, the properties panel will give you a bunch of settings that'll let you change how the dissolve works…



Group p103-64

Operation Delete Selected

☐ Recompute Normals

☑ Remove Inline Points

Collinearity Tolerance 45

☑ Remove Unused Points

Bridge Create Bridged Polygons

☐ Remove Degenerate Bridges

☐ Create Curves When Dissolving Boundary

Group The edges to delete.

Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, it deletes everything.

Operation ???

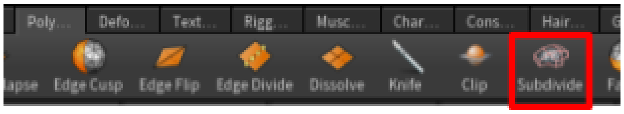
Recompute Normals Recomputes the normals for whatever that was a sibling of whatever got deleted.

Subdivision

Subdivision lets you smooth out your polygons by automatically splitting them.

To interactively subdivide (via scene view)...

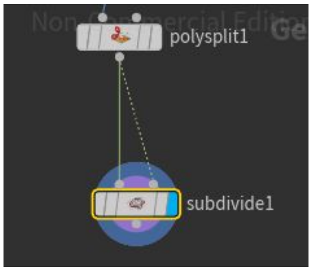
1. Choose Subdivide from the shelf



2. Select your polygons to subdivide (leaving empty subdivides everything)
3. Press Enter

To non-interactively blast faces (via network view)...

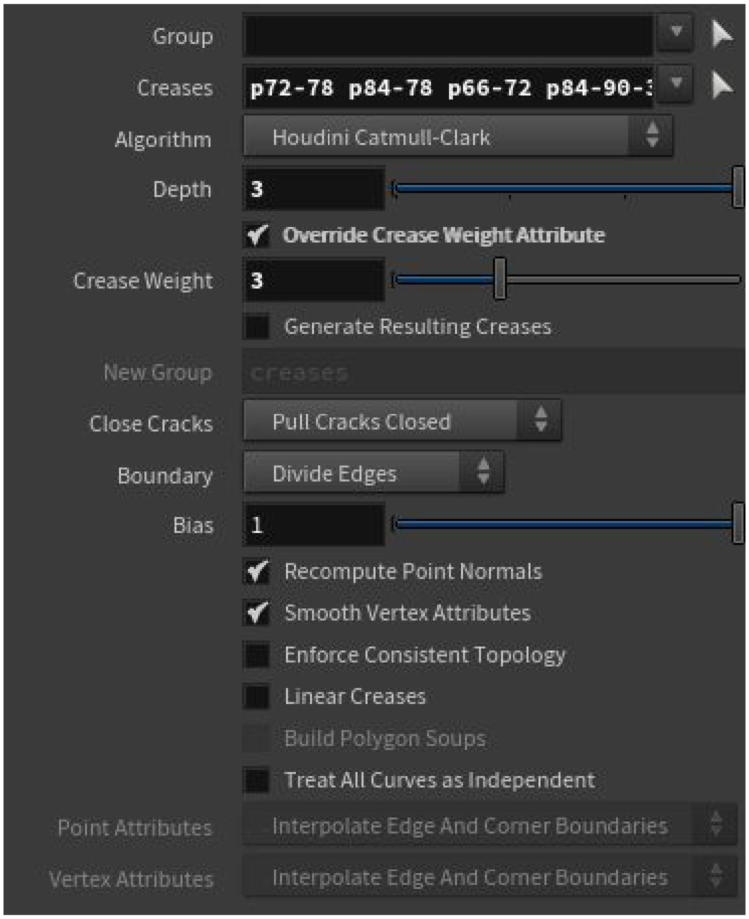
1. Connect your geo to a subdivide node



2. In the properties panel, specify Group (polys to subdivide)
3. In the properties panel, specify Creases (components to avoid subdivision)

   NOTE: The first input is for applying what to subdivide. The second input is for specifying the creases (what to not subdivide). If you don’t have a second input, you can ignore the creases property. You can almost always feed the same geo to both inputs.

Regardless of how you did things, the properties panel will give you a bunch of settings that’ll let you change how the subdivision works…


Group

Creases p72-78 p84-78 p66-72 p84-90-

Algorithm Houdini Catmull-Clark

Depth 3

☑ Override Crease Weight Attribute

Crease Weight 3

□ Generate Resulting Creases

New Group creases

Close Cracks Pull Cracks Closed

Boundary Divide Edges

Bias 1

☑ Recompute Point Normals

☑ Smooth Vertex Attributes

□ Enforce Consistent Topology

□ Linear Creases

□ Build Polygon Soups

□ Treat All Curves as Independent

Point Attributes Interpolate Edge And Corner Boundaries

Vertex Attributes Interpolate Edge And Corner Boundaries

Group

The polygons to subdivide.

Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, subdivides everything.

Creases

The creases are edges where you don’t want subdivide to apply (e.g. hard edges).

Click arrow to select interactively in scene view. Follow the prompts in the scene view.

If empty, it won’t apply creases to anywhere.

NOTE: The second input is for specifying the creases. If you don’t have a second input, you can ignore the creases property. You can almost always feed the same geo to both inputs.

Algorithm Stick to Houdini Catmull-Clark. Otherwise you won't get support for creases.

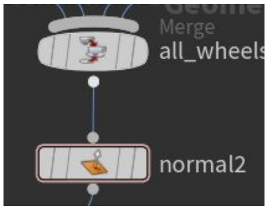
Depth How many iterations of subdivision to apply.

Crease Weight Make sure Override Crease Weight is selected and set Crease Weight to the same value as Depth. This makes sure the creases you selected remain sharp.

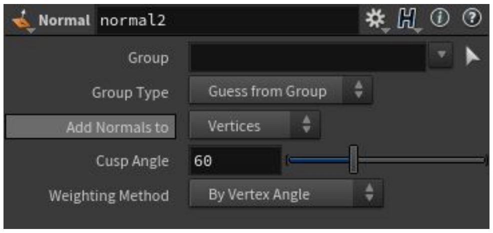
Normal (Generate or correct normals)

Attempts to generate normals on the input.

You can add/fix normals in the network view. Drop a Normal node and connect whatever geometry you want to it...



You can choose what level the normals are applied and how the normals are determined...



Normal normal2

Group

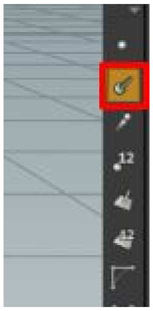
Group Type: Guess from Group

Add Normals to: Vertices

Cusp Angle: 60

Weighting Method: By Vertex Angle

NOTE: To actually see the generated normals, you can use the Scene view and switch on show normals in the toolbar on the right-hand side…



You can also switch to the geometry spreadsheet to see the N attributes added to your points/vertices/faces/whatever.

Utility Nodes

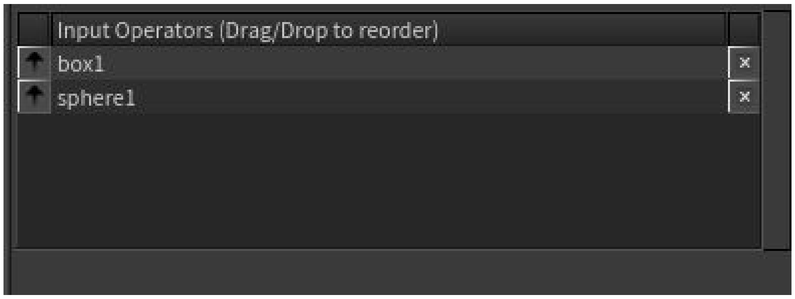
Merge (Combine geometry/outputs)

The merge tool brings two or more separate outputs (e.g. of geometry) together.

> **NOTE**: For geo, how is this different than a Boolean node set to union????? It looks like union won't work unless the geo is intersecting????

You can merge nodes in the network view. Drop a merge node and connect whatever geometry you want to it (no limit). The output of the merge node should contain everything?

There's nothing special in the properties panel for this…



Input Operators (Drag/Drop to reorder)

box1

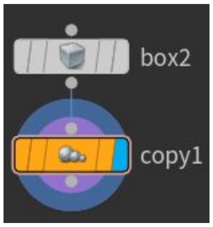
sphere1

> **NOTE**: Is this node giving you a warning? Check the attributes of your inputs at various levels via the geometry spreadsheet. For example, does your first input have normals on primitives (faces) but your other input have normals on vertices? Mismatches and missing attributes may cause warnings to show up, but these may not always be a problem. See the attributes section for more info.

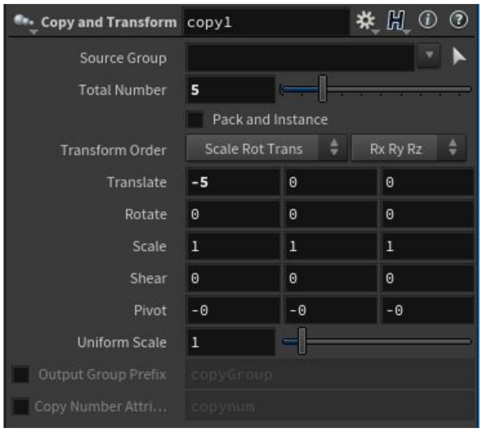
Copy and Transform

Copies input and applies transformations, multiple times (successively). For example, if you input a box, you can choose to copy that box 5 times and offset the X translate by -5 for each copy.

You can copy nodes in the network view. Drop a Copy and Transform node and connect whatever geometry you want to it...



Then, set the number of times you want it to be copied and which transform properties to offset for each copy...



Copy and Transform copy1

Source Group

Total Number 5

□ Pack and Instance

Transform Order Scale Rot Trans Rx Ry Rz

Translate -5 0 0

Rotate 0 0 0

Scale 1 1 1

Shear 0 0 0

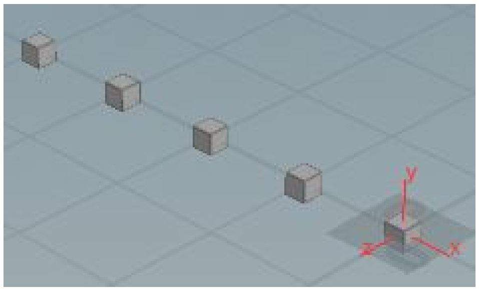
Pivot -0 -0 -0

Uniform Scale 1

□ Output Group Prefix copyGroup

□ Copy Number Attri... copynum

> **NOTE**: A copy count of 1 means no actual copies. You’re just passing through the geometry when you select 1. Select 2 or more to get actual duplication of geometry. This is why the property is called Total Number.

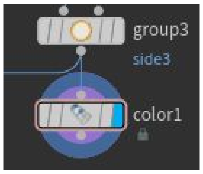


Material Nodes

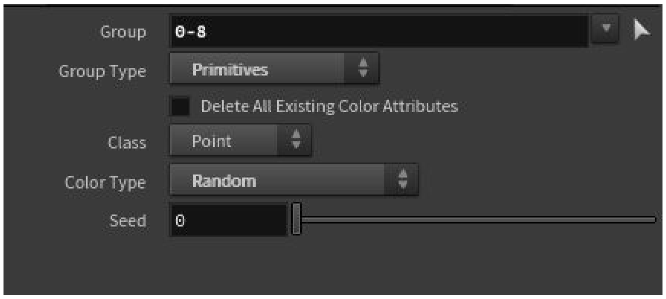
Color

You can apply color(s) to components.

It’s fairly straight-forward what to do with this node. Go into the Network view and drop a Color node, then hook your geometry into it…



In the parameters pane, choose the components which you want the color to apply to. You can do so by clicking the arrow on the right of the Group.



Group 0-8

Group Type Primitives

□ Delete All Existing Color Attributes

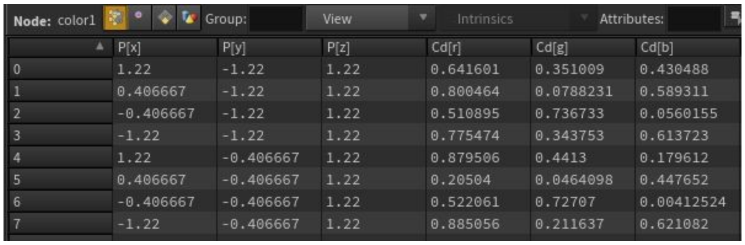
Class Point

Color Type Random

Seed 0

The other parameters here should be straightforward. Class determines what component type to apply the geometry to (e.g. faces or points or edges or whatever). Color type can be set to Constant or Random (or a few others). Everything after that point will be dependent on what color type was set to.

> **NOTE**: If you go into the geometry spreadsheet, you can see the color show up under attributes for whatever component type you used for Class (Cd is the name of the attribute).

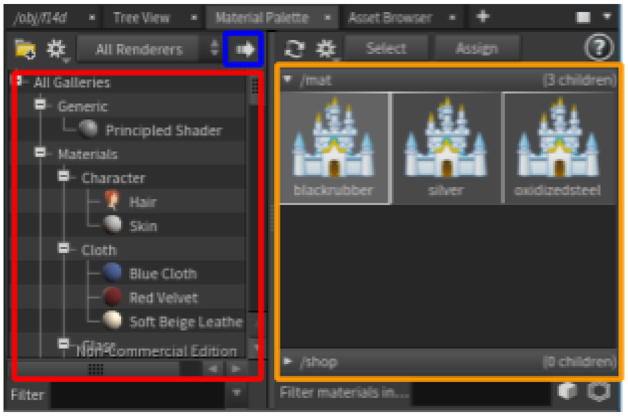


	P[x]	P[y]	P[z]	Cd[r]	Cd[g]	Cd[b]
0	1.22	-1.22	1.22	0.641601	0.351009	0.430488
1	0.406667	-1.22	1.22	0.800464	0.0788231	0.589311
2	-0.406667	-1.22	1.22	0.510895	0.736733	0.0560155
3	-1.22	-1.22	1.22	0.775474	0.343753	0.613723
4	1.22	-0.406667	1.22	0.879506	0.4413	0.179612
5	0.406667	-0.406667	1.22	0.20504	0.0464098	0.447652
6	-0.406667	-0.406667	1.22	0.522061	0.72707	0.00412524
7	-1.22	-0.406667	1.22	0.885056	0.211637	0.621082

Material

Applies materials to components.

> **NOTE**: Before you can apply materials, you need to bring in your materials using the Materials Palette view (in the same section as the network view when under the Build desktop).

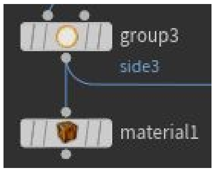


You can select from the left pane, and move over to the right pane by pressing the right arrow or by dragging-and-dropping over. These create the materials for you under the /mat namespace (or the /shop namespace which is collapsed below -- look at bottom lefthand side of right pane), which you can navigate to in the network view and tweak as you see fit.

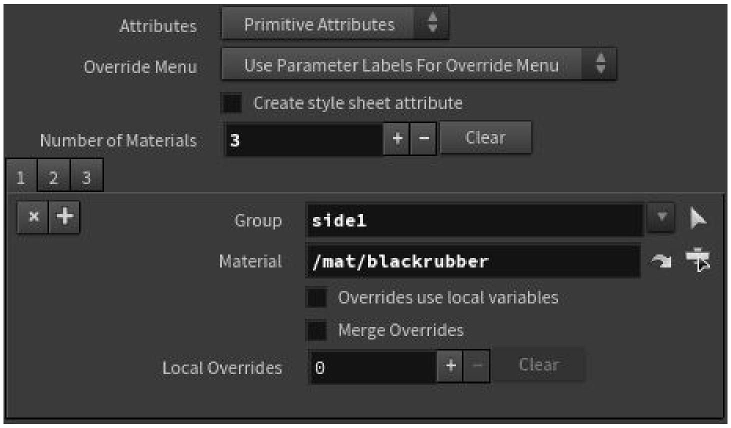
A discussion on materials is outside the scope of this document, but if you switch your network to /mat you can change the properties of these materials. It's very similar to Arnold shaders: sub-surface scattering, specular, reflectivity, diffuse, etc..

NOTE: In Houdini15, these would get created under /shop (shader OPS). Now you can't create these under /shop anymore. Why?????

It's fairly straight-forward what to do with this node. Go into the Network view and drop a Material node, then hook your geometry into it…



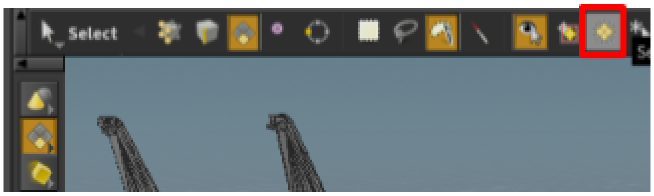
In the parameters pane, choose the components which you want the color to apply to. You can do so by clicking the arrow on the right of the Group. Once you've done that, select the material you want to use by clicking the right-most button next to Material (will open a selection menu of materials under /mat).



Attributes: Primitive Attributes
Override Menu: Use Parameter Labels For Override Menu
□ Create style sheet attribute
Number of Materials: 3 + − Clear
1 2 3
× +
Group: side1
Material: /mat/blackrubber
□ Overrides use local variables
□ Merge Overrides
Local Overrides: 0 + − Clear

Note that you aren't limited to one material. You can add multiple materials for different parts of the geo. In the example above, we have 3 different materials being added to 3 different parts of the geo.

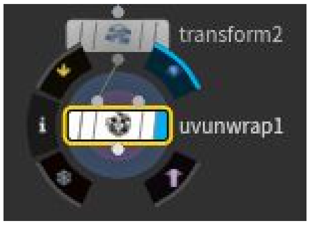
> **NOTE**: You typically would have these groups pre-defined via group nodes (especially for materials). See the Group Nodes section for more information.
>
> **NOTE**: If you click the arrow never to Group, it'll actually ask for a grouping rather than let you put in arbitrary components. You can turn this off by disabling the group selection toggle. You need to do this every time for Material nodes…



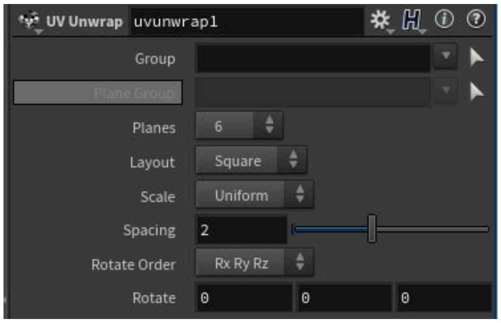
UV Unwrap (Generate or correct UVs)

Attempts to generate UVs for the input.

You can add/fix UVs in the network view. Drop a UV unwrap node and connect whateer geometry you want to it…



You can choose how the UVs get applied…



UV Unwrap uvunwrap1

Group

Plane Group

Planes 6

Layout Square

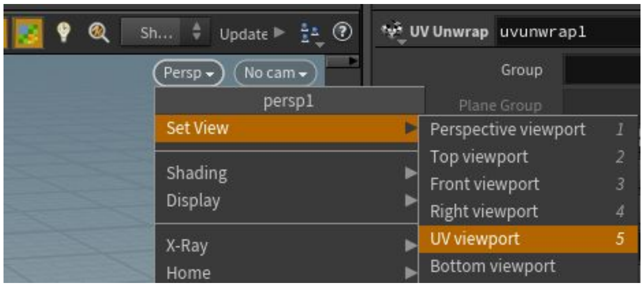
Scale Uniform

Spacing 2

Rotate Order Rx Ry Rz

Rotate 0 0 0

NOTE: To actually see the generated UVs, you can use the Scene view and switch your view from perspective to UV (shortcut is Space+5)…



You can also switch to the geometry spreadsheet to see the UV attributes added to your points/vertices/faces/whatever.

Light and Camera Nodes

All Light Nodes

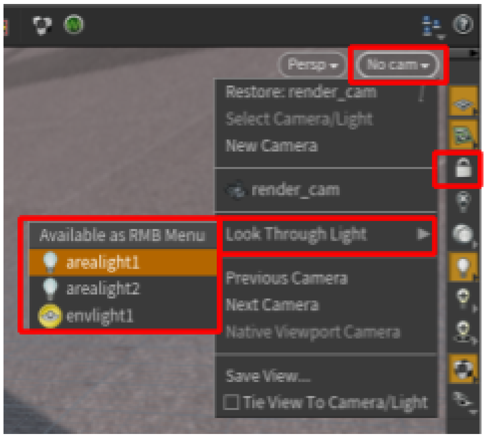
Houdini (mantra) provides a bunch of lights that are very similar to Arnold and Maya lights…

- Point Light → emits light equally in all directions
- Area Light → a rectangle/disk/whatever that emits light
- Geometry Light → similar to having a polygon mesh in Arnold that you set to emit light
- Distant Light → parallel rays, as if the sun were shining
- Environment Light → provide an HDR image and get light emitted based on that -- similar to mentalray's image based lighting or Arnold's skydome light
- Sky Light → how is this different from the environment light above???

Add them to your scene and move them around just like any other scene node. You can find them in the Light shelf (2nd shelf set in the build desktop) or via the tab menu.

An easy to position a light is through the scene view. You can make it so as if you're looking through the light, and orient it as you move around your scene. To do this…

1. Select the light in the camera dropdown on the top-right of the scene view (just below the top toolbar)
2. Toggle the lock camera/light lock button on
3. Orient your view around, just as normally would
4. Toggle the lock camera/light lock button off



Almost all lights come with many properties that are similar to Arnold. They probably work just like Arnold does. The properties panel has sections for increasing the light sampling (if stuff comes out too grainy) and setting intensity/exposure property and a bunch of other stuff.

Intensity and exposure seem to be the main control properties here. Here's the description of it from my Arnold notes…

Intensity

Intensity of your light. This works the same way as a regular Maya light. The higher your intensity is, the more light will be given off.

Exposure

Intensity of your light. This is different from the above setting in that it's measured in f-stops (just like real photography). It's designed to help you interact with real cinematographers. For example, the cinematographer may come and ask the artist to increase something by half a stop -- you can do that directly here in this setting.

This is exponential… each time you go up by 1, it doubles the intensity of your light.

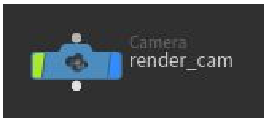
NOTE: This is linked to the intensity in that the exposure uses the current intensity value as the basis for its setup.

Camera

Adds a camera to the scene, which a Mantra node (render node) can use to render from.

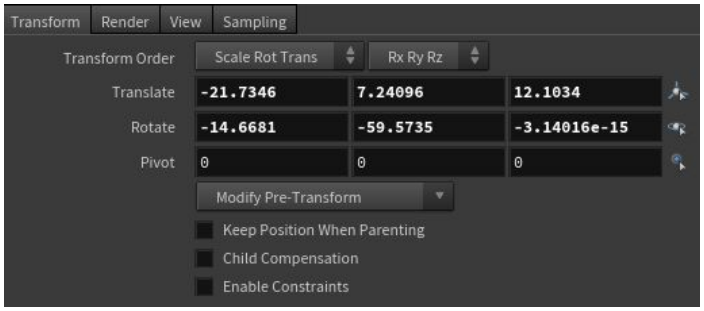
> **NOTE**: A camera is REQUIRED if you want to do renders.

Add a camera node however you want. Do it in the scene view or the network view. It doesn't need to be connected to anything.



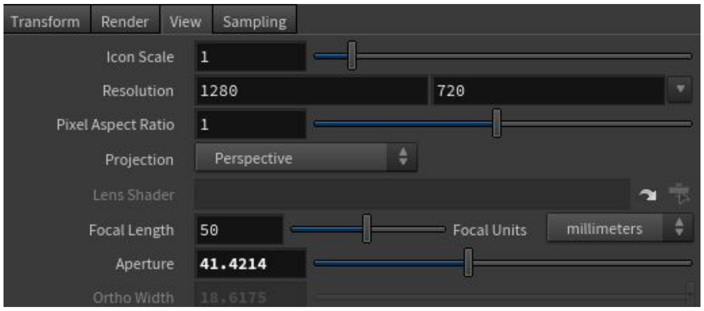
Camera
render_cam

The camera properties contain the basics: positions, focal length, aperture, near clip, far clip, etc… Set these parameters however you see fit. Most of them are self-explanatory…



Transform | Render | View | Sampling

Transform Order: Scale Rot Trans, Rx Ry Rz
Translate: -21.7346, 7.24096, 12.1034
Rotate: -14.6681, -59.5735, -3.14016e-15
Pivot: 0, 0, 0
Modify Pre-Transform
□ Keep Position When Parenting
□ Child Compensation
□ Enable Constraints

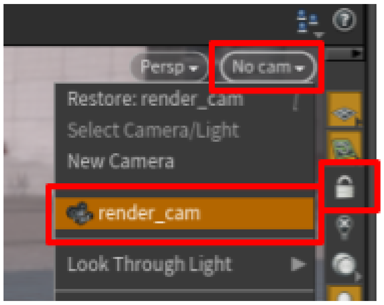


Transform | Render | View | Sampling

Icon Scale: 1
Resolution: 1280, 720
Pixel Aspect Ratio: 1
Projection: Perspective
Lens Shader:
Focal Length: 50 Focal Units: millimeters
Aperture: 41.4214
Ortho Width: 18.6175

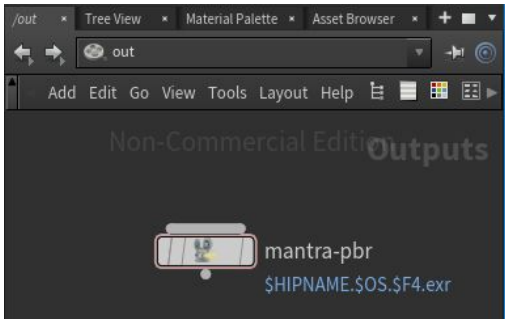
An easy to position the camera is through the scene view. You can look through the camera and orient as if you're looking through it. To do this…

1. Select the camera in the camera dropdown on the top-right of the scene view (just below the top toolbar)
2. Toggle the lock camera/light lock button on
3. Orient your view around, just as normally would
4. Toggle the lock camera/light lock button off



Render Nodes

Render nodes exist in the /out context (render ops / ROPS context).



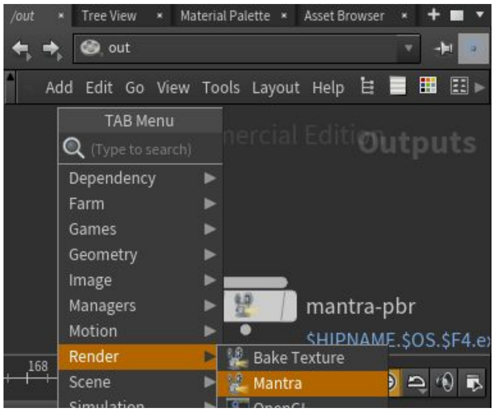
This is where you put in how you want things to be output. Almost always you want to drop nodes in here that specify render settings, but you can dump out geo or pass stuff to a farm or other things as well.

> **NOTE**: Keep in mind that different renderers likely need customized materials and lights. You probably can't use Mantra's principal shader with Arnold and vice-versa.
>
> **NOTE**: Pushing Houdini jobs to cuebot would be done through here via a custom node.

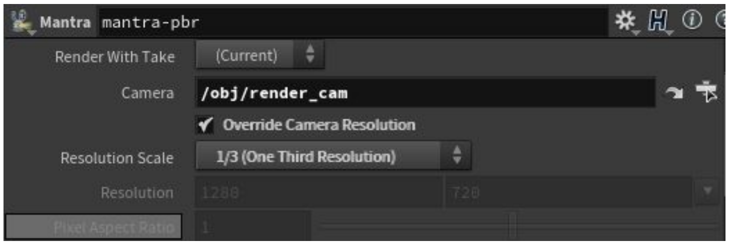
Mantra

Mantra is Houdini's default renderer, and a Mantra node will render the output using that renderer.

You can add a Mantra node by going to the network view, switching context to out, and inserting a Mantra node from the Tab menu…



After you set down the node, YOU MUST SET A CAMERA in the properties pane. That means your scene NEEDS to have at least 1 camera node in it...


Mantra mantra-pbr

Render With Take: (Current)

Camera: /obj/render_cam

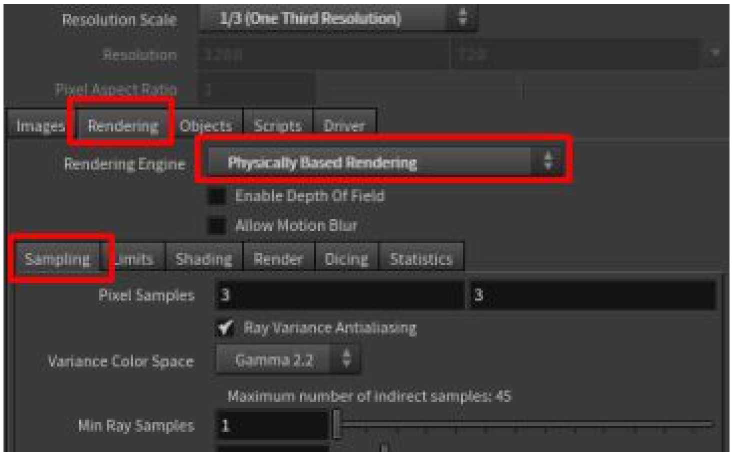
☑ Override Camera Resolution

Resolution Scale: 1/3 (One Third Resolution)

Resolution: 1288 720

Pixel Aspect Ratio: 1

You can set the samples for the render (just like you do with Arnold) under Rendering -> Samples in the properties pane. You should probably set your rendering engine option to Physically Based Rendering…


Resolution Scale: 1/3 (One Third Resolution)

Resolution: 1288 720

Pixel Aspect Ratio: 1

Rendering Engine: Physically Based Rendering

□ Enable Depth Of Field

□ Allow Motion Blur

Pixel Samples: 3 3

☑ Ray Variance Antialiasing

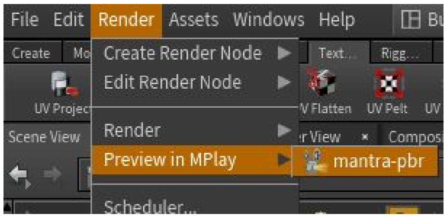
Variance Color Space: Gamma 2.2

Maximum number of indirect samples: 45

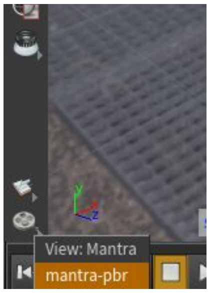
Min Ray Samples: 1

NOTE: There are many ways to trigger a render…

1. In the main menu… Render -> Render -> (render node name)
2. In the main menu… Render -> Preview in MPlay -> (render node name)



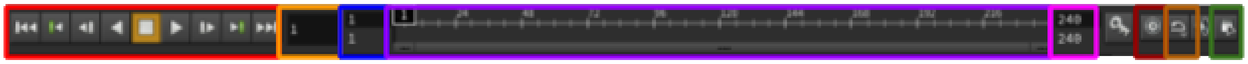
3. In the left toolbar of scene view… click the movie reel (last button in the toolbar) and choose the render node name



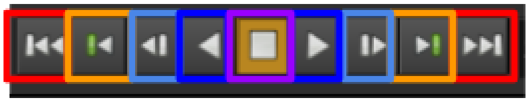
NOTE: MPlay is Houdini's version of fcheck (Maya's image playback thing)????

Keyframe Animation

The lower portion of the Build Desktop holds the animation controls. Almost everything here is exactly the same as Maya's animation controls…



Playback controls Quickly playback, pause, and navigate the animation.



1st outermost buttons = Jump to first/last frame
2nd outermost buttons = Jump to first/last SCOPED frame
3rd outermost buttons = Jump forward/backward by 1 frame
4th outermost buttons = Play backward/forward
Middle button = Stop playing

Current Frame Current frame being shown

Animation Start The 1st number is the first frame of TOTAL animation
The 2nd number is the first frame of SCOPED animation

Scoped means that the scrubber will only show frames from this point onward. Playing will also start from this frame.

Scrubber Typical scrubber -- drag mouse to quickly scrub through animation.

Animation End The 1st number is the last frame of TOTAL animation
The 2nd number is the last frame of SCOPED animation

Scoped means that the scrubber will only show frames up to this point. Playing will also end at this frame.

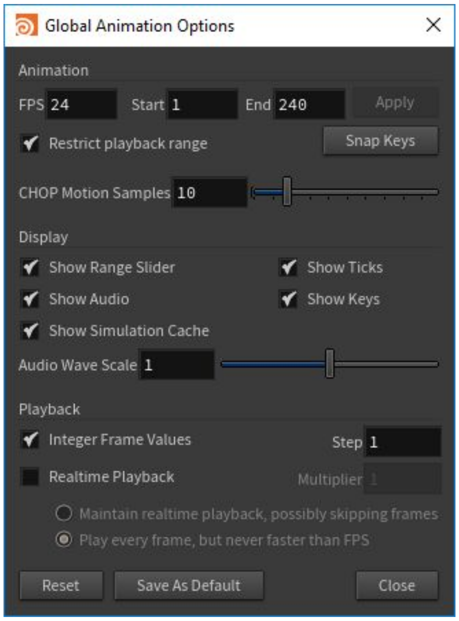
Realtime If enabled, will playback animation at however many FPS the animation is set to.

If disabled, will playback animation as fast as it can compute.

NOTE: Disabled by default. You probably almost always want this on.

Playback type Loop/zig-zag/play once/play backward

Animation settings Go here to set global animation settings (e.g. FPS)



Animation

FPS 24 Start 1 End 240

☑ Restrict playback range

CHOP Motion Samples 10

Display

☑ Show Range Slider ☑ Show Ticks

☑ Show Audio ☑ Show Keys

☑ Show Simulation Cache

Audio Wave Scale 1

Playback

☑ Integer Frame Values Step 1

□ Realtime Playback Multiplier 1

○ Maintain realtime playback, possibly skipping frames

● Play every frame, but never faster than FPS

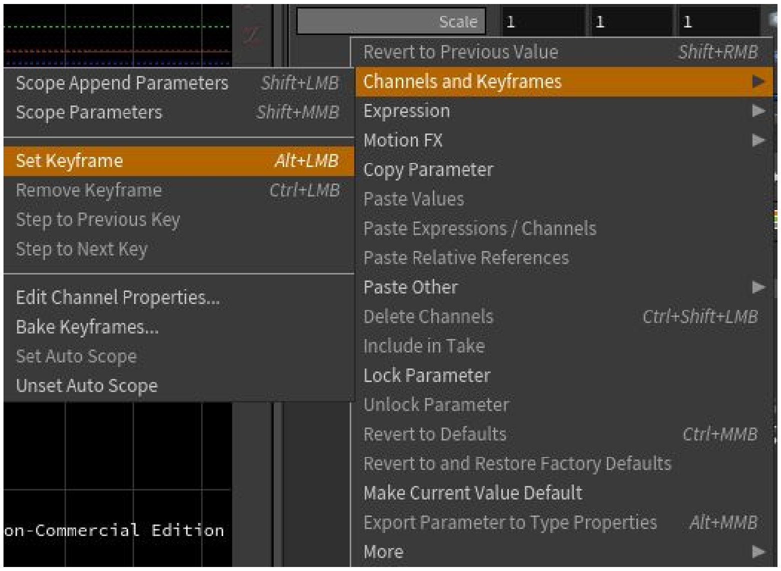
Setting Keyframes

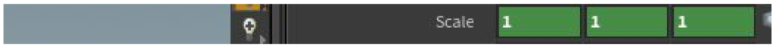
Houdini supports traditional keyframe animation. The interfaces are very similar to what Maya provides.

To animate an object based on some properties…

1. Select the object.
2. In the parameter pane, right-click on the property (or label for a property group) and choose Channels and Keyframes -> Set Keyframe.

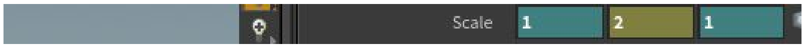
   The property will turn green.

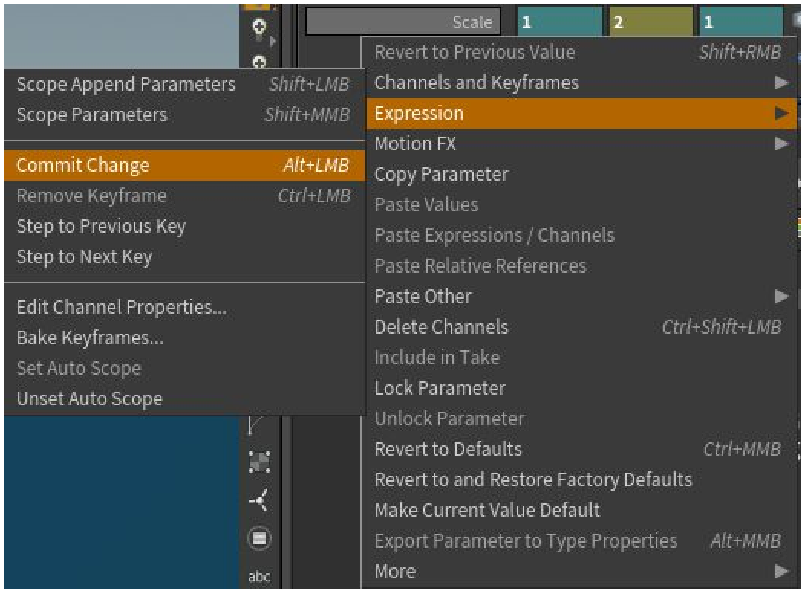

Scale
1
1
1
Scope Append Parameters Shift+LMB
Scope Parameters Shift+MMB
Set Keyframe Alt+LMB
Remove Keyframe Ctrl+LMB
Step to Previous Key
Step to Next Key
Edit Channel Properties...
Bake Keyframes...
Set Auto Scope
Unset Auto Scope
on-Commercial Edition
Revert to Previous Value Shift+RMB
Channels and Keyframes
Expression
Motion FX
Copy Parameter
Paste Values
Paste Expressions / Channels
Paste Relative References
Paste Other
Delete Channels Ctrl+Shift+LMB
Include in Take
Lock Parameter
Unlock Parameter
Revert to Defaults Ctrl+MMB
Revert to and Restore Factory Defaults
Make Current Value Default
Export Parameter to Type Properties Alt+MMB
More



3. Scrub to the point in the animation where you want to change the values for that parameter, change it, and then choose Channels and Keyframes -> Commit Change.

   The property will turn brown when you change it, then back to green when you commit it.




Scale
1
2
1
Scope Append Parameters Shift+LMB
Scope Parameters Shift+MMB
Commit Change Alt+LMB
Remove Keyframe Ctrl+LMB
Step to Previous Key
Step to Next Key
Edit Channel Properties...
Bake Keyframes...
Set Auto Scope
Unset Auto Scope
Revert to Previous Value Shift+RMB
Channels and Keyframes
Expression
Motion FX
Copy Parameter
Paste Values
Paste Expressions / Channels
Paste Relative References
Paste Other
Delete Channels Ctrl+Shift+LMB
Include in Take
Lock Parameter
Unlock Parameter
Revert to Defaults Ctrl+MMB
Revert to and Restore Factory Defaults
Make Current Value Default
Export Parameter to Type Properties Alt+MMB
More
abc

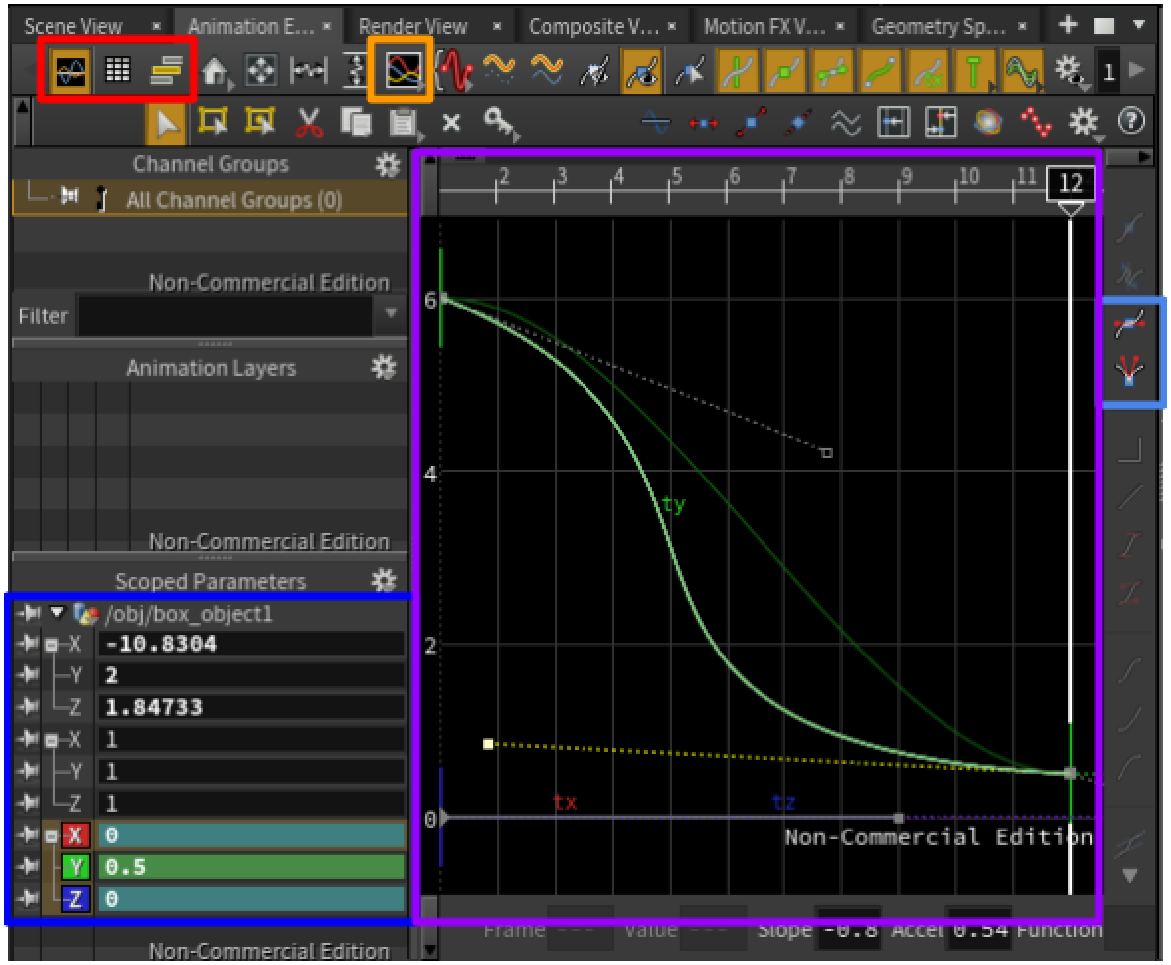
NOTE: Want to remove the keyframe-ability of a property?? Right click and select Delete Channel.

NOTE: Want to remove a keyframe? Right click, Channels and Keyframes, and Remove Keyframes.

Animation Editor

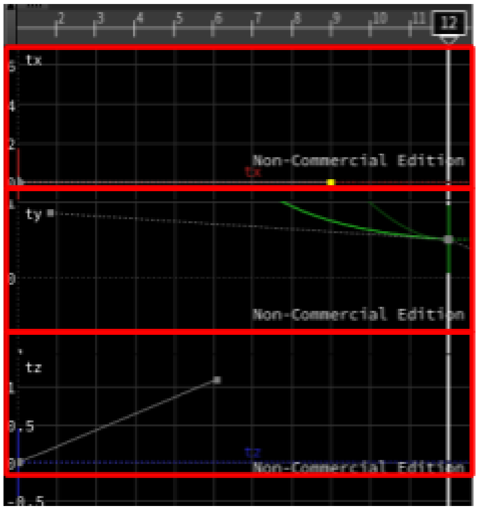
The animation editor contains Houdini's equivalent of Maya's dopesheet + graph editor. It's almost exactly the same.

NOTE: Viewing shortcuts are the same as the network view. G to frame up on channels selected. H to view everything.



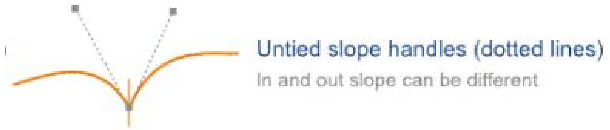
Mode Switch between editing curves, dopesheet, and tabular view.

Multi-graph Display If enabled, each parameter selected will show in its own graph. The graphs will be stacked ontop of eachother.



Tie/Untie Handles

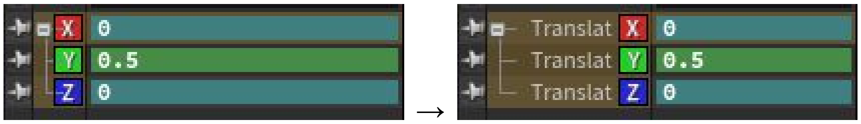
Similar to Maya's feature where you can break up the up the bezier handles. You can use this so to make it easier to have abrupt transitions between keyframes.


Untied slope handles (dotted lines)
In and out slope can be different

Parameters

The parameters showing up in the editor. You can select which parameters show up in the editor via the label.

NOTE: The labels may be super small. If that's the cause, just mouse over between the label and the divider and drag them out....



X	0
Y	0.5
Z	0

→

Translat	X	0
Translat	Y	0.5
Translat	Z	0

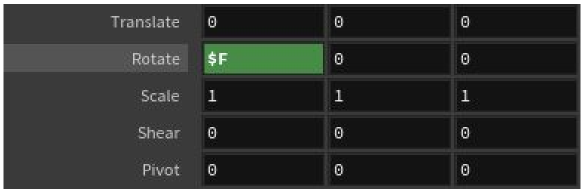
Editor

Just like Maya's editor. Move keyframes and handles around for specific parameters. Basic stuff.

Scripting References

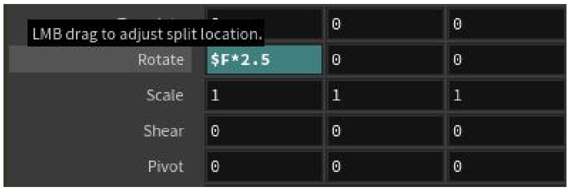
You can set properties based on the current frame number.

For example, let's say you want your X rotation to change based on what animation frame you're currently at. You can reference the animation frame by using $F.



Translate	0	0	0
Rotate	$F	0	0
Scale	1	1	1
Shear	0	0	0
Pivot	0	0	0

Now let's say you want to that rotation to speed up. You want it to go 2.5 times as fast. You can change that expression to be $F*2.5…



LMB drag to adjust split location.

		0	0
Rotate	$F*2.5	0	0
Scale	1	1	1
Shear	0	0	0
Pivot	0	0	0

NOTE: Remember that you can left click a label to show the value of what is computed. So in the example above. If you click the rotate label it'll show you the value for the current frame. This value won't update while you playback the animation, but it will update when you stop the playback.

Physics Animation (Dynamics)

This physics portion of Houdini is called dynamics. Essentially this is just Houdini's interface to the Bullet open source library (and other related physics libraries).

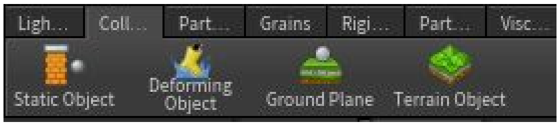
You set physics parameters on objects here, almost exactly the same as those when you were playing with Box2D. There's a generic gravity setting, and then you have impulse (initial velocity?), velocity, angular velocity, density, bounce, friction, etc.. etc..

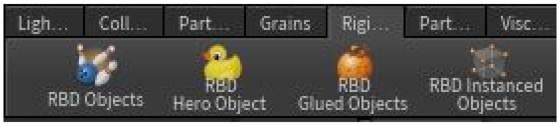
Still unsure how to apply force? I'm sure this will all be detailed in other tutorials.

There are 2 shelves that were detailed here...

1. Collision and Rigid Bodies are for actually setting up the physics properties.

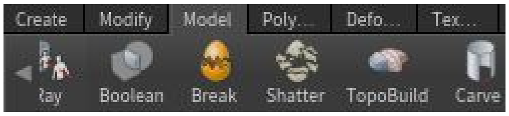
   These are located in the 2nd shelf set if you're using the build desktop.





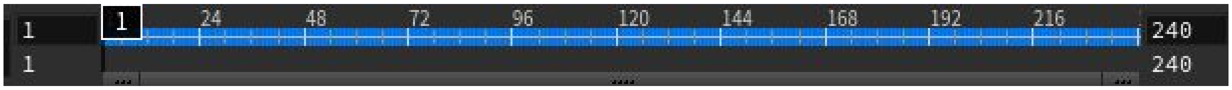
2. Modeling is for doing stuff like breaking or pre-shattering geometry. You can use this for doing things like pre-shattering windows so you can smash stuff through it and watch the pieces fly out everywhere.

   These are located in the 2nd shelf set if you're using the build desktop.

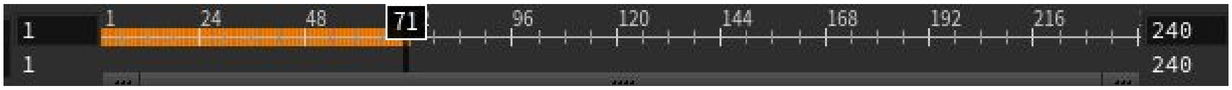


Once you create your dynamic bodies and stuff, the calculations for them be done AS YOU SCRUB YOUR ANIMATION SLIDER / AS YOU PLAY YOUR ANIMATION.

Calculations are done once and then cached. You'll know which frames are calculated because the timeline will show blue for those frames.



If you change something, the cache will get nuked and the computation will happen again the next time you scrub/play from frame 1. YOU HAVE TO GO BACK TO FRAME 1 FOR THE ANIMATIONS TO RECOMPUTE, otherwise you'll play stale frames. You'll know your frames are stale cause they'll change color from blue to orange.



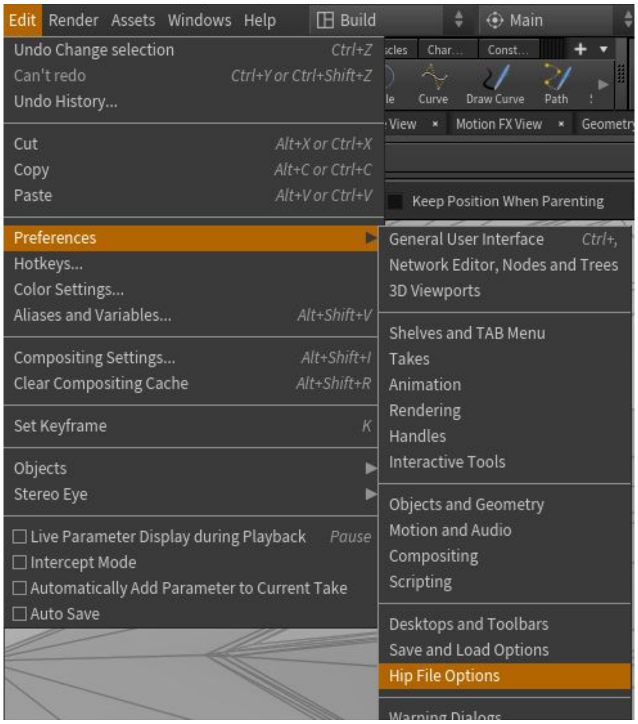
> **NOTE**: Keyframe animations and animations from dynamics don't seem to be mutually exclusive? You can have some geo keyframed and others animated dynamically.

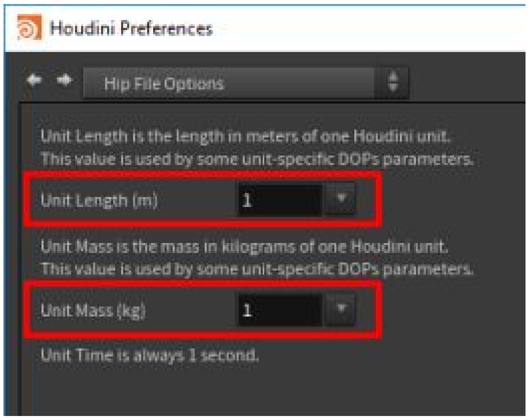
Scene Scale

> **NOTE**: This is a replace of a section way earlier in the document, but it's super important so I copied it here as well. This stuff is straightforward for any physics engine (which Bullet is). There was a similar concept for Box2D as well.

Scene scale is a super important concept when it comes to Houdini. The size of your objects has a direct effect on the way in which your simulations run. This is exactly like when you were using Box2D on Android.

By default, Houdini is set to treat 1 unit as 1 meter in length / 1 kg in weight. You generally don't want to change these values, but you can do so by going to Edit -> Preferences -> Hip File Options…





Unit Length is the length in meters of one Houdini unit. This value is used by some unit-specific DOPs parameters.

Unit Length (m) 1

Unit Mass is the mass in kilograms of one Houdini unit. This value is used by some unit-specific DOPs parameters.

Unit Mass (kg) 1

Unit Time is always 1 second.

> **NOTE**: If you're having problems visualizing this, think of a building collapsing. How long did it take the world trade center building to fall to the ground? Now imagine if the buildings were shrunken down to the height of a smartphone.
>
> In the new shrunken scale, the building would fall down much faster. The pieces falling to the ground would be much lighter. They wouldn't be in the air as long. They'd have less impact when they hit the ground. They'd have less weight. etc..

Creating Dynamics for Objects

At the scene view, you can make any of your geometry nodes objects for use in dynamics.

> **NOTE**: Remember we're messing around with the high-level geometry nodes here, not the stuff in the geometry nodes. The final geometry computation is treated as a whole.

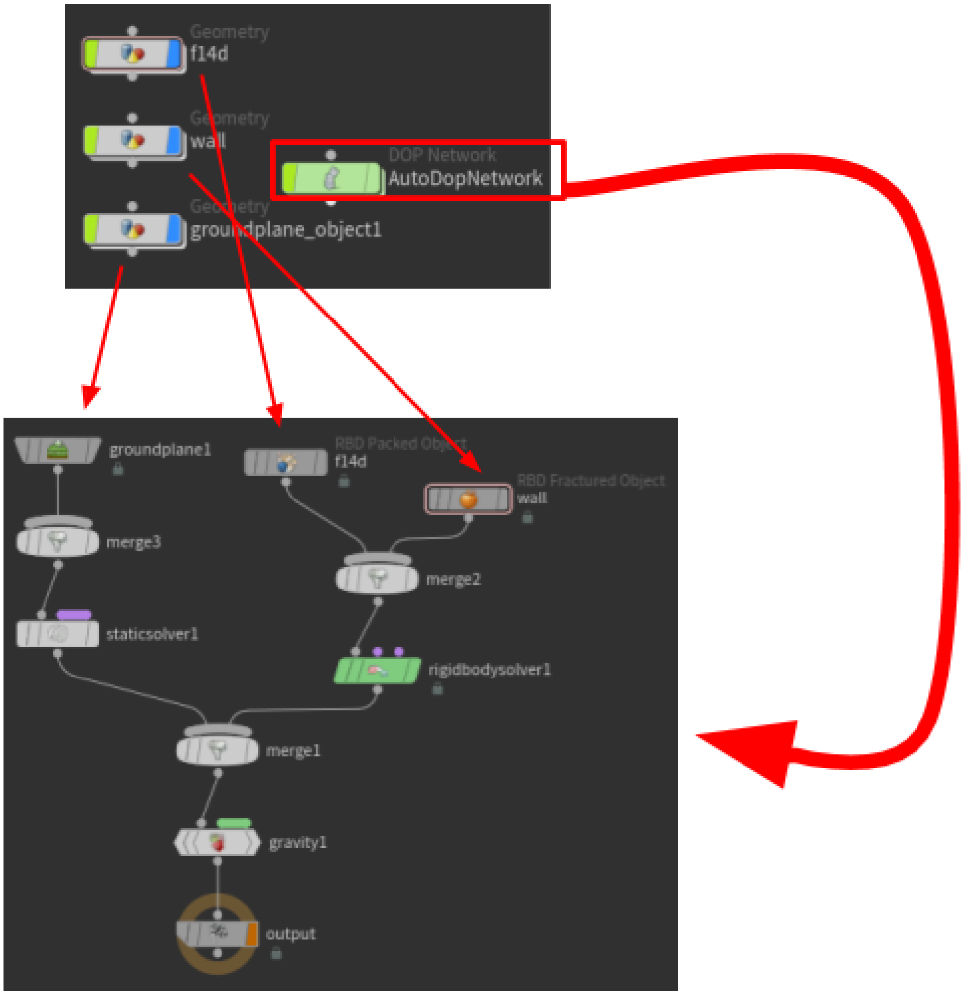
There are multiple types of rigid bodies available:

- RBD Object ← solid objects that have physics applied?
- RBD Fractured Object ← fractured objects that have physics applied? e.g. windows?
- Static Object ← doesn't move but can be interacted with by other dynamic objects
- Ground Plane ← infinite ground plane

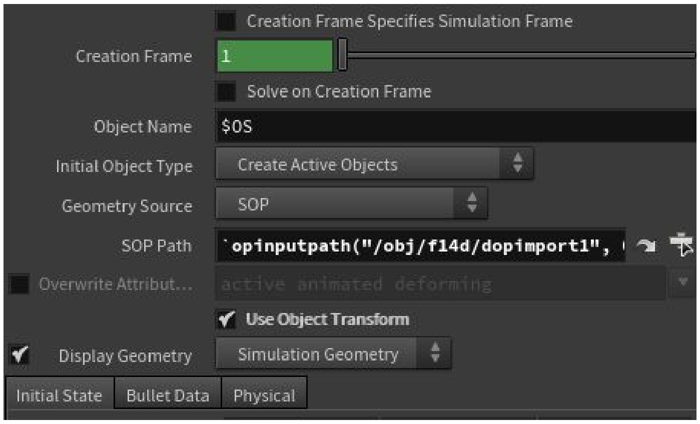
NOTE: Those are 4 we'll be discussing, but there's a ton of shit here and you should probably go over another lesson on it instead of messing around with it.

NOTE: The difference between RBD Object/RBD Fractured Object and Static Object/Ground Plane are that the RBD objects are actually simulated, while the other 2 are things that the simulated objects can interact with (but they don't actually move themselves).

Whenever you create any of these, it'll automatically create a “AutoDopNetwork” node at the root /obj level (if one doesn't already exist). This node will be of type DOP Network (dynamic OPs), and for each rigid body object you have in here, a mirror node will be created.



These mirror nodes inside the AutoDopNetwork will be how you control physics properties. For example, if we checkout the f14d node's parameters…



□ Creation Frame Specifies Simulation Frame

Creation Frame: 1

□ Solve on Creation Frame

Object Name: $OS

Initial Object Type: Create Active Objects

Geometry Source: SOP

SOP Path: `opinputpath("/obj/f14d/dopimport1",

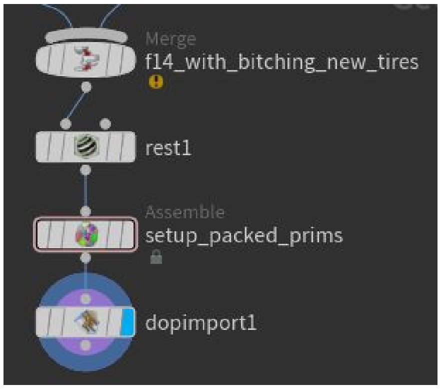
□ Overwrite Attribut... active animated deforming

☑ Use Object Transform

☑ Display Geometry: Simulation Geometry

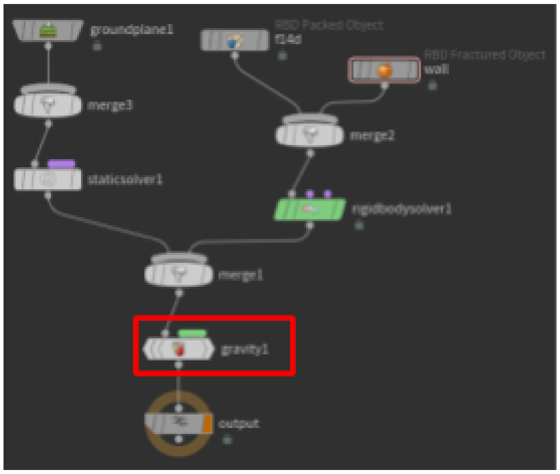
You can see that the node is referencing our original f14d geometry object (/obj/f14d).

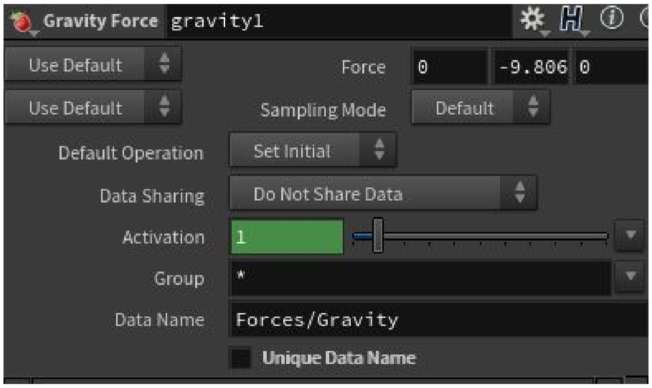
> **NOTE**: Notice how the path specifically checks for /obj/f14d/dopimport1. If you look inside the f14d geometry object, you can see a few new nodes have been added to the end of the chain to help with the dynamics simulations as well…



Setting Gravity

Inside the AutoDopNetwork node, the 2nd last node is gravity1. This is where you'd set your gravity…





Gravity Force gravity1

Use Default — Force: 0 -9.806 0

Use Default — Sampling Mode: Default

Default Operation: Set Initial

Data Sharing: Do Not Share Data

Activation: 1

Group: *

Data Name: Forces/Gravity

□ Unique Data Name

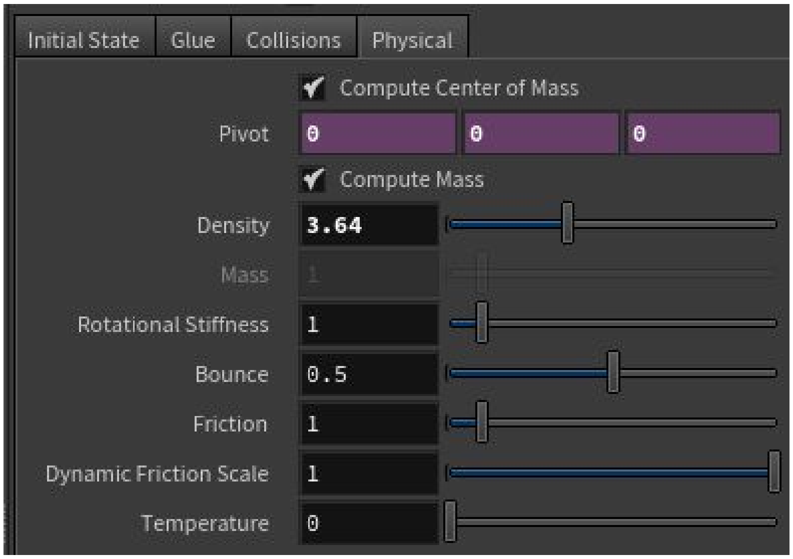
The important property here is Force, and by default it’s set to the correct value (-9.8 meters in the Y-axis). If you want to change it, you can.

Setting Physics Properties for Objects

Remember the for each rigid body object you create, a mirror node will be created in the AutoDopNetwork. These mirror nodes are where you can set your physics properties.

You can find them in tabs within the last section.

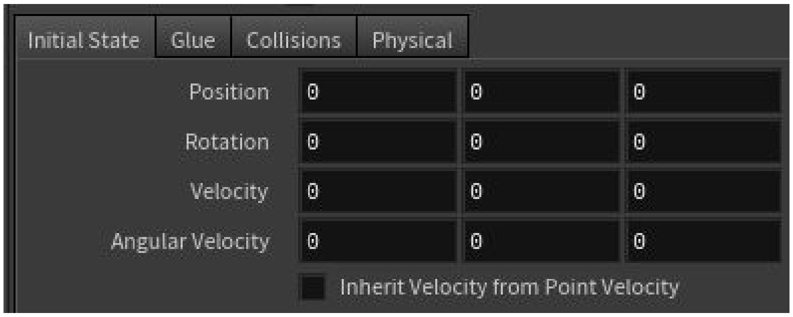
Almost all objects will have a physical tab. The physical tab defines the physics properties of the object.



☑ Compute Center of Mass
Pivot 0 0 0
☑ Compute Mass
Density 3.64
Mass 1
Rotational Stiffness 1
Bounce 0.5
Friction 1
Dynamic Friction Scale 1
Temperature 0

Describing what each of these does is out of scope, but a lot of them should be self-explanatory. They mirror box2d properties.

If your object ISN’T a ground or static object, you’ll also have a initial state tab. This is the state of your dynamic object when the scene first starts…



Position 0 0 0
Rotation 0 0 0
Velocity 0 0 0
Angular Velocity 0 0 0
☐ Inherit Velocity from Point Velocity

Once again, describing what each of these does is out of scope, but it should be self-explanatory. They mirror box2d properties.

NOTE: You almost never ever want to change position. The lesson didn’t even give a single use-case for this.

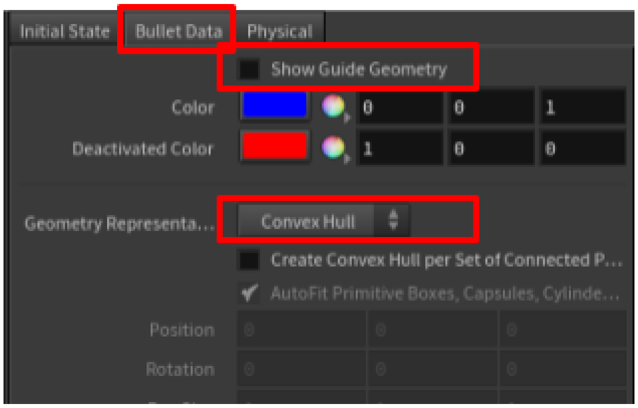
Collision Geometry

The collision geometry put on your object is different from the actual geometry. It’s more simplified.

Remember the for each rigid body object you create, a mirror node will be created in the AutoDopNetwork. These mirror nodes are where you can set your see and tweak your collision geometry.

To see/change the collision geometry….

1. Go to the mirror node in the AutoDopNetwork
2. Go the Bullet Data tab in the parameters pane (this tab may be hidden under the Collisions tab depending on node type)
3. Enable Show Guide Geometry
4. Change Geometry Representation (if desired)



☐ Show Guide Geometry

Color 🟦 0 0 1

Deactivated Color 🟥 1 0 0

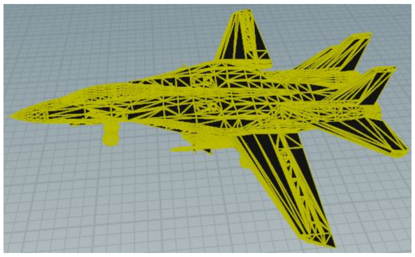
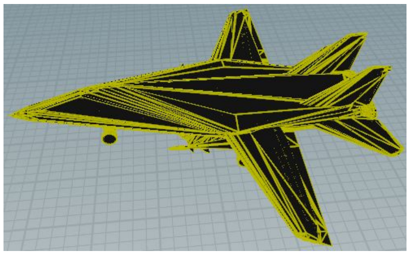
Geometry Representa... Convex Hull

☐ Create Convex Hull per Set of Connected P...

✓ AutoFit Primitive Boxes, Capsules, Cylinde...

Position 0 0 0

Rotation 0 0 0


→


The default collision geometry being produced is ‘convex hull’. It produces geometry around your object such that it doesn’t cave in. There are multiple different options here and you can change them however you see fit. You can change this option

NOTE: Clearly, the example above is caving in on certain areas. I think this is because the geometry object for this plane is made up of multiple distinct meshes -- they are not fused together. Maybe what's happening is a convex hull is being created for each distinct mesh?

A lot of times this type of collision geometry is good enough, but there are cases where this won't give you want you want. For example, imagine you have a soccer goal post. If you use convex hull for the collision geo, the open face of the goal post will be covered (convex means that the geometry doesn't cave in). So if a soccer ball was going into the net, it would hit an invisible wall and bounce off just before entering.

NOTE: Notice the 2 options in the properties pane: "Color" and "Deactivated Color". This shows which objects are "active" during the physics simulations. So if you do some physics and enable "Show Guide Geometry" and scrub through your animation, you'll see which objects are "active" via the colors.

This is EXACTLY like the view in Box2D's debug renderer thing. Objects are at rest for a certain period of time (and aren't around other moving objects?) will no longer be considered for calculations and will be deactivated. If they're collided with, they'll become active again.

Houdini 16 Objects and Collisions

Introduction

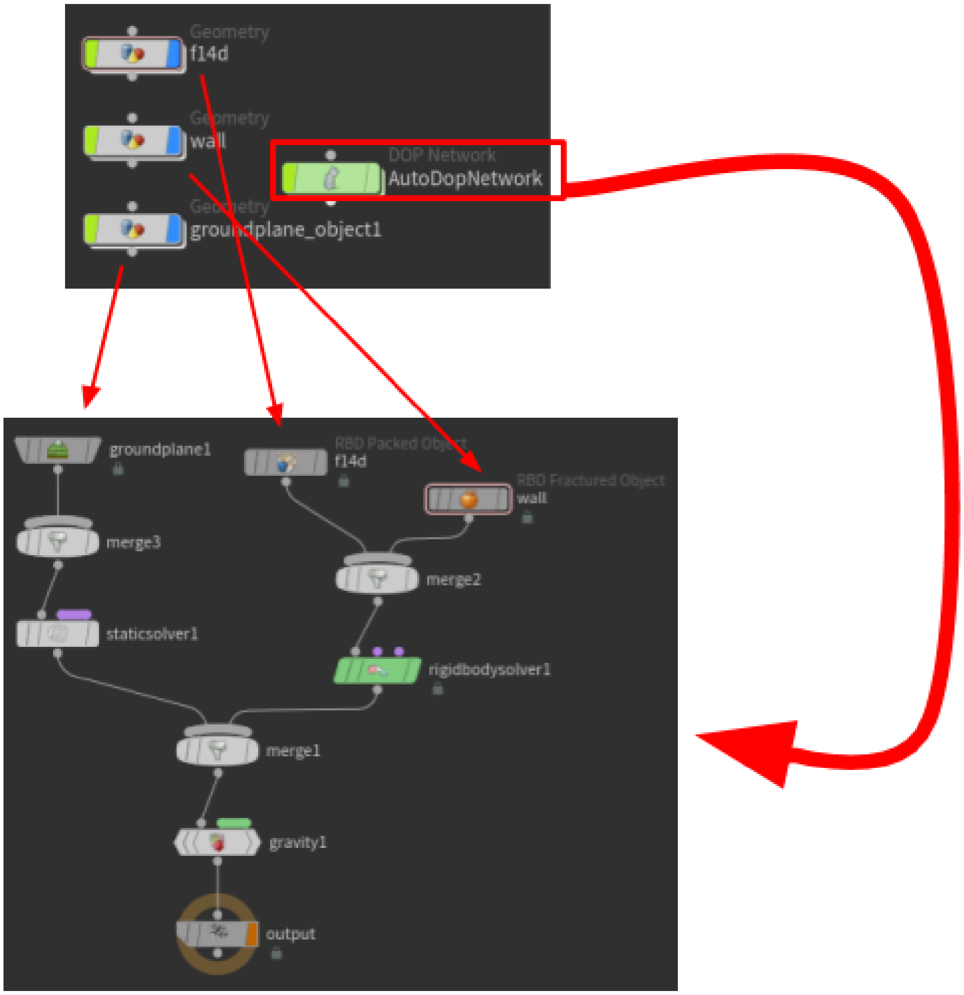
This physics portion of Houdini is called dynamics. There’s a ton of different things you can do with dynamics, but this text is specifically about handling collisions between objects. You can use this instead of normal keyframe animation to get realistic looking animations going. All the basic parameters are there: gravity, velocity, angular velocity, density, bounce, friction, etc.. etc..

The shorthand for dynamics is DOPs (Dynamic OPerators).

Dynamics Network

The following is an example of a DOPS network. This specific network (AutoDopNetwork) is automatically created by Houdini when you use the rigid-body shelf, but networks that you create manually will look very similar.

Essentially what happens is that, you have your various geometry nodes at the /obj level. In your DOP network, you create DOP rigid-body/static/collision nodes that reference those geometry nodes. Those rigid-body/static/collision nodes are then passed to “solvers” and output.



NOTE: If you’re using a AutoDopNetwork and your nodes aren’t organized, hit L to layout the nodes in a nice tree structure.

There are lots of different types of solvers, but the 3 most important ones are…

- Static → only collides with stuff, never moves
- Rigid body → collides with stuff and moves, but is rigid
- Finite element → collides with stuff and moves, but is squishy

  NOTE: What are solvers? Solvers is a fancy name for “rule book”. Essentially all it does is contain the logic to do physics simulations. You’ll notice that there are 2 types of solvers in the example: rigidbody and static.

  Static is for static collision geometry (stuff that can be collided with but never moves). Rigidbody is for objects that have physics actually applied (e.g. rag-doll on a model).

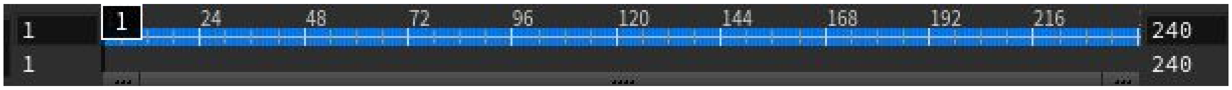
  NOTE: What about gravity? That’s handle by the gravity node just before the output node.

  NOTE: All of this is just an interface to bullet. I imagine bullet’s actual interface is nothing like this and more like what Box2D is like. For example, gravity would likely be a system-wide property that can only be set once. WTF happens if I have 2 gravity nodes?

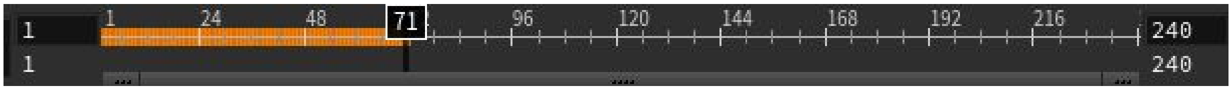
Dynamics Playback

Once you create your dynamic bodies and stuff, the calculations for them be done AS YOU SCRUB YOUR ANIMATION SLIDER / AS YOU PLAY YOUR ANIMATION.

Calculations are done once and then cached. You’ll know which frames are calculated because the timeline will show blue for those frames.

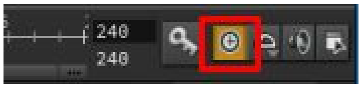


If you change something, the cache will get nuked and the computation will happen again the next time you scrub/play from frame 1. YOU HAVE TO GO BACK TO FRAME 1 FOR THE ANIMATIONS TO RECOMPUTE, otherwise you’ll play stale frames. You’ll know your frames are stale cause they’ll change color from blue to orange.



  NOTE: Keyframe animations and animations from dynamics don’t seem to be mutually exclusive? You can have some geo keyframed and others animated dynamically.

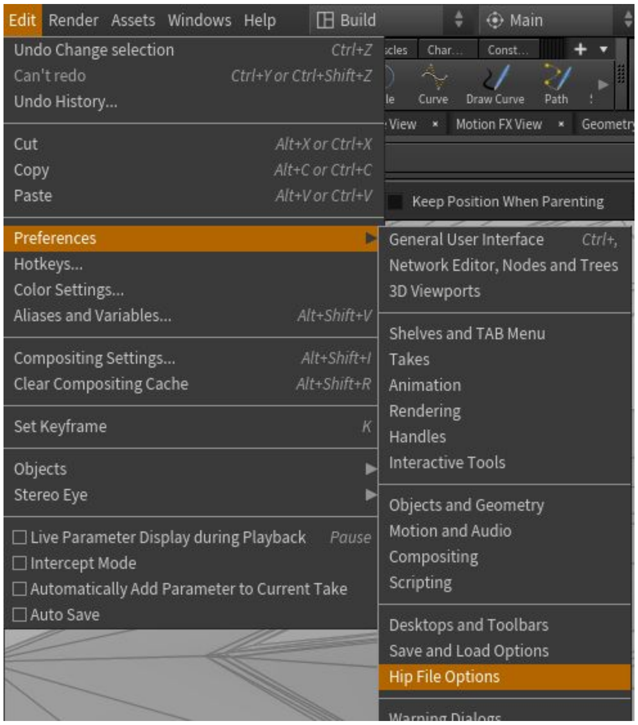
NOTE: Remember to enable the realtime playback button. Otherwise Houdini will playback as fast as possible…

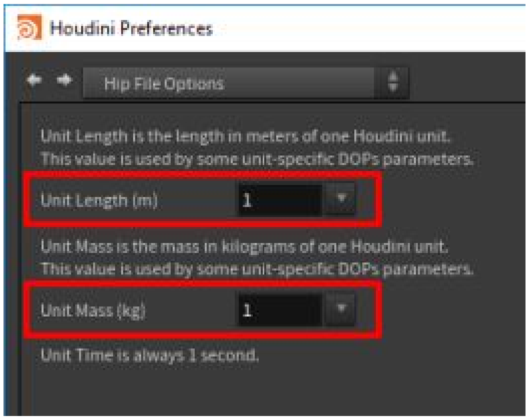


Scene Scale

Scene scale is a super important concept when it comes to Houdini. The size of your objects has a direct effect on the way in which your simulations run. This is exactly like when you were using Box2D on Android.

By default, Houdini is set to treat 1 unit as 1 meter in length / 1 kg in weight. You generally don't want to change these values, but you can do so by going to Edit -> Preferences -> Hip File Options…





Unit Length is the length in meters of one Houdini unit. This value is used by some unit-specific DOPs parameters.

Unit Length (m) 1

Unit Mass is the mass in kilograms of one Houdini unit. This value is used by some unit-specific DOPs parameters.

Unit Mass (kg) 1

Unit Time is always 1 second.

NOTE: If you're having problems visualizing this, think of a building collapsing. How long did it take the world trade center building to fall to the ground? Now imagine if the buildings were shrunken down to the height of a smartphone.

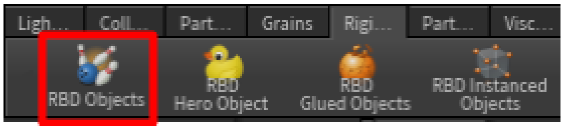
In the new shrunken scale, the building would fall down much faster. The pieces falling to the ground would be much lighter. They wouldn't be in the air as long. They'd have less impact when they hit the ground. They'd have less weight. etc..

Rigid Body Objects

Rigid bodies are physics objects that can move around and collide with stuff. The shape and the volume of the geometry won't change, it'll only be transformed each frame as per the physics calculations.

To create a rigid body object…

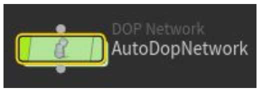
1. Select your object
2. Go to the Rigid Body shelf and select RBD Object

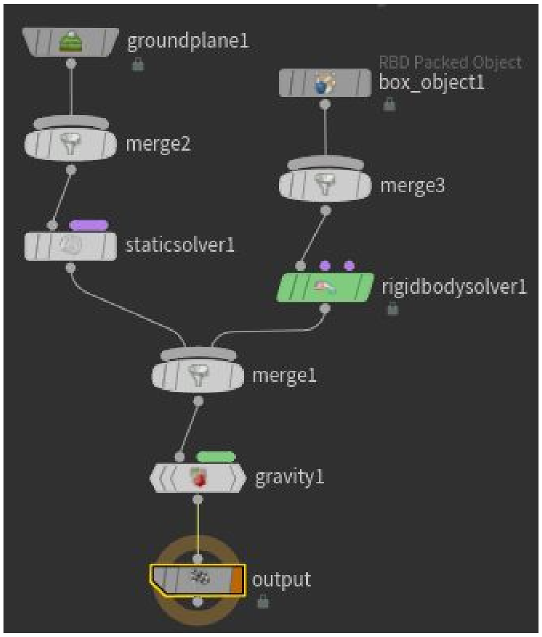


NOTE: Using RBD Objects will create what's called a packed object. Packed objects can't be simulated with finite element objects (objects that go squishy). If you're going to have finite element objects in your scene, use RBD Hero Object. They're less efficient but they work with everything.

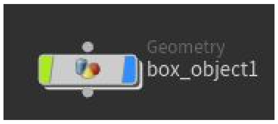
Once you do this, 2 things will happen…

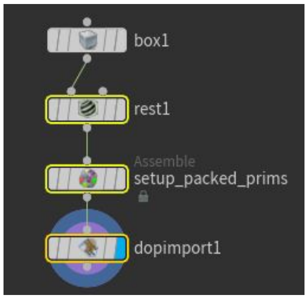
- You'll get a new node called AutoDopNetwork (if it didn't already exist) which contains a node with the same name as the object you've applied RBD to.





- Your object you’ve applied RBD to will get a few nodes added to it





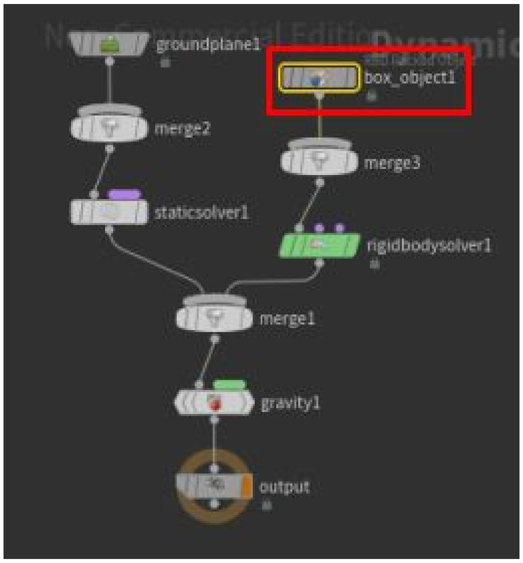
NOTE: The extra nodes that are added into your object, you generally don’t want to touch those. They set things up for use in dynamics and create links between your geometry and dynamics.

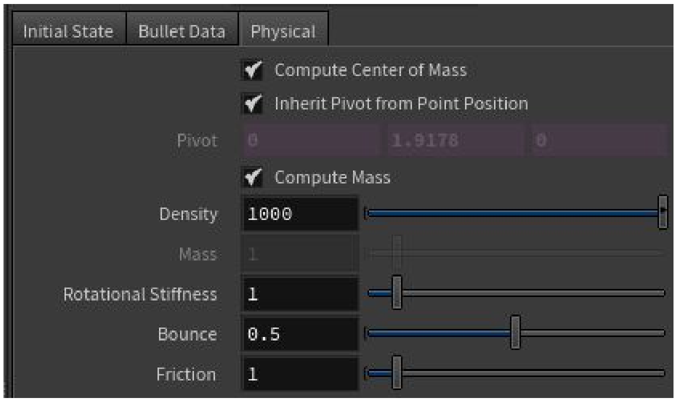
The AutoDopNetwork node is what you traverse into if you want to change the physics properties of your node (e.g. bounciness/friction/etc..)....

Physical Properties

You can change the physical properties for the simulation of your object by…

1. Traversing into the AutoDopNetwork.
2. Selecting the RBD object for that object.
3. In the properties pane, going to the Physical tab.





☑ Compute Center of Mass

☑ Inherit Pivot from Point Position

Pivot 0 1.9178 0

☑ Compute Mass

Density 1000

Mass 1

Rotational Stiffness 1

Bounce 0.5

Friction 1

Bounce Bounce is how bouncy the object is. Note that the value applied on collision is a mix of this value and the bounciness value from the object being collided with.

Friction Friction is how much friction the object has to it (e.g. is it like ice or sandpaper?) . Note that the value applied on collision is a mix of this value and the friction value from the object being collided with.

NOTE: Density = volume * mass. If you disable the Compute Mass checkbox, you can specify mass directly.

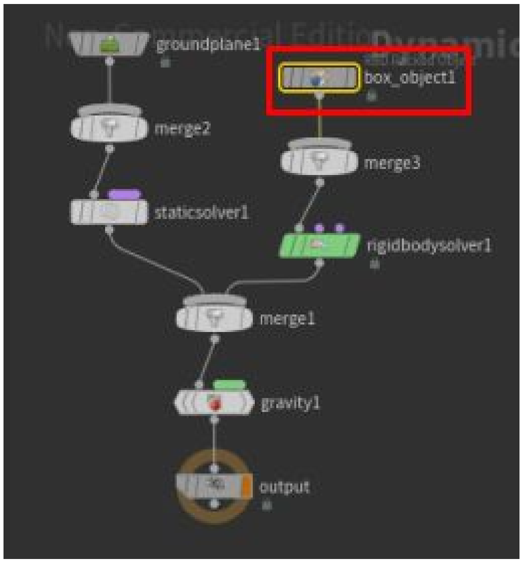
NOTE: No idea what Rotational Stiffness is used for. Best not to change it.

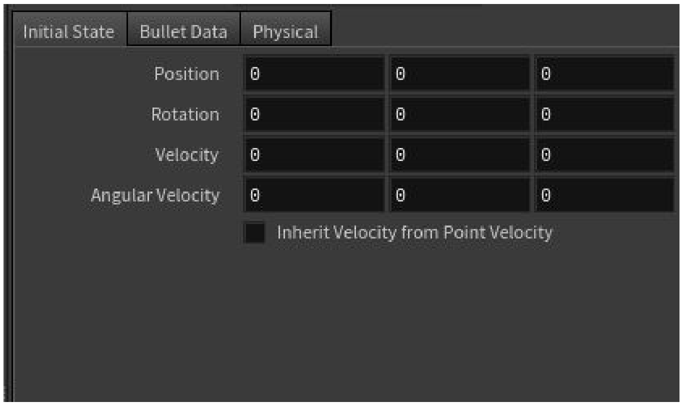
Initial State

Initial state is the physics state of the object at the start of the scene. For example, when the scene starts the object may be throttling down the Z axis.

You can change the physical properties for the simulation of your object by…

1. Traversing into the AutoDopNetwork.
2. Selecting the RBD object for that object.
3. In the properties pane, going to the Physical tab.





Position	0	0	0
Rotation	0	0	0
Velocity	0	0	0
Angular Velocity	0	0	0

□ Inherit Velocity from Point Velocity

Velocity The direction this object is moving + the force it’s moving at.

Angular Velocity How fast this object is spinning + the direction it’s spinning at.

NOTE: You almost always never want to touch Position and Rotation. These are available for artists to make small tweaks.

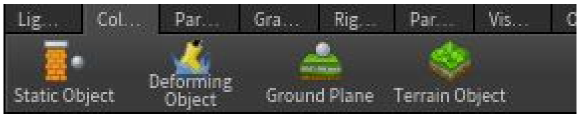
Collisions

Go to the Solver Engines and Collision section and read it THOROUGHLY.

Once you're done that, read the Optimized Collisions section to find out strategies for faster simulations.

Collision Objects

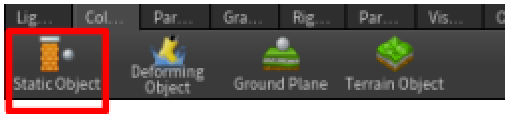
Collision objects are physics objects that can't move around but are used for collisions with other objects. Nothing will change about the geometry, not even the transform.



There are many different types of collision objects you can create…

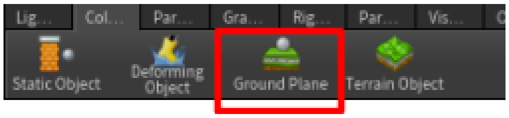
- Static Object → turns your geometry into a collision object

  To create...
  1. Select your geometry object
  2. Got to the Collisions shelf and select Static Object



- Ground Plane → an infinite ground plane not linked to any geometry object.

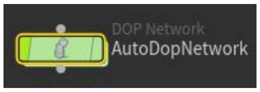
  To create...
  1. Got to the Collisions shelf and select Ground Plane

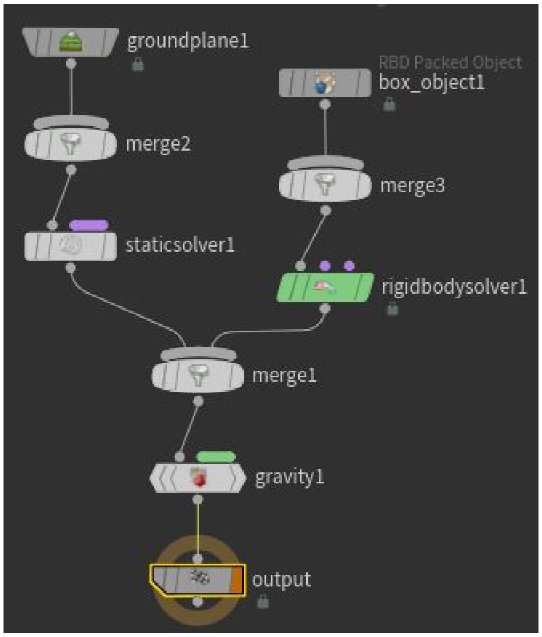


NOTE: What about the other 2?
- No idea what Deforming Object is used for.
- Terrain Object is like a static object but meant specifically for cases where the object is uber thin (e.g. a deformed grid that's meant to represent a terrain) -- apparently this type of static object needs volume applied to it or something so it works better with collisions/solvers/whatever.

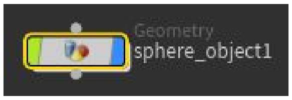
Once you do this, 2 things will happen…

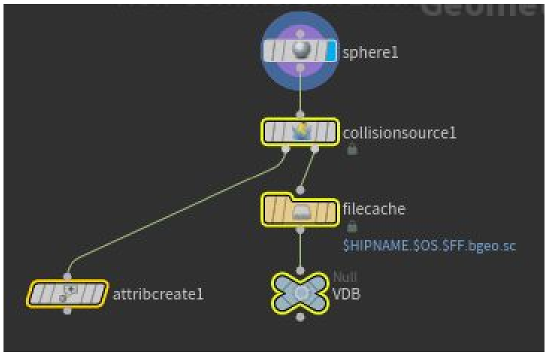
- You'll get a new node called AutoDopNetwork (if it didn't already exist) which contains a node with the same name as the object you've applied collision to.





- Your object you've applied collision to will get a few nodes added to it OR your collision object will generate some new geometry.





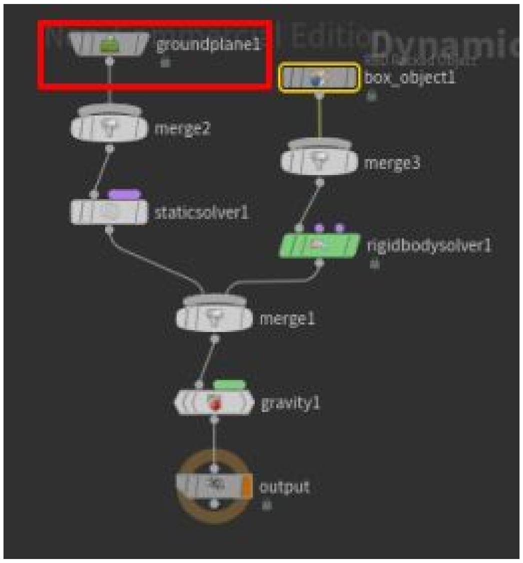
NOTE: The extra nodes that are added into your object, you generally don't want to touch those. They set things up for use in dynamics and create links between your geometry and dynamics.

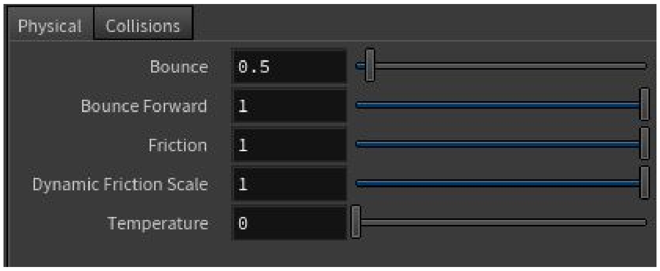
The AutoDopNetwork node is what you traverse into if you want to change the physics properties of your node (e.g. bounciness/friction/etc..)....

Physical Properties

You can change the physical properties for the simulation of your object by…

1. Traversing into the AutoDopNetwork.
2. Selecting the collision object for that object.
3. In the properties pane, going to the Physical tab.





Bounce	0.5
Bounce Forward	1
Friction	1
Dynamic Friction Scale	1
Temperature	0

Bounce Bounce is how bouncy the object is. Note that the value applied on collision is a mix of this value and the bounciness value from the object being collided with.

Friction Friction is how much friction the object has to it (e.g. is it like ice or sandpaper?) . Note that the value applied on collision is a mix of this value and the friction value from the object being collided with.

NOTE: No idea what the rest of the properties are used for. They have descriptions but I don’t really understand what they mean.

Collisions

Go to the Solver Engines and Collision section and read it THOROUGHLY.

Once you’re done that, read the Optimized Collisions section to find out strategies for faster simulations.

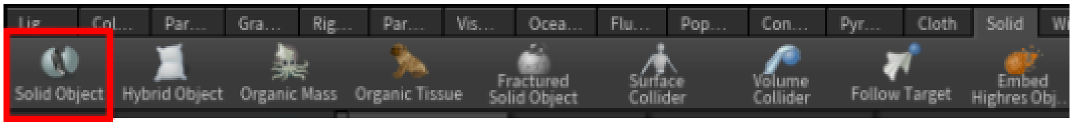
Finite Element Objects

Finite element objects are objects that can move around and collide with stuff (just like rigid body objects). But, the shape and the volume of the geometry can also change. So, in addition to transforming based on physics calculations, the physics calculations will also deform the geometry.

> **NOTE**: Cloth sims use to happen via finite element solver in Houdini 15, but in Houdini 16 it's been updated to use its own solver (called cloth solver).

To create a finite element object…

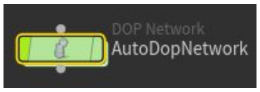
1. Select your object
2. Go to the Solid shelf and select Solid Object

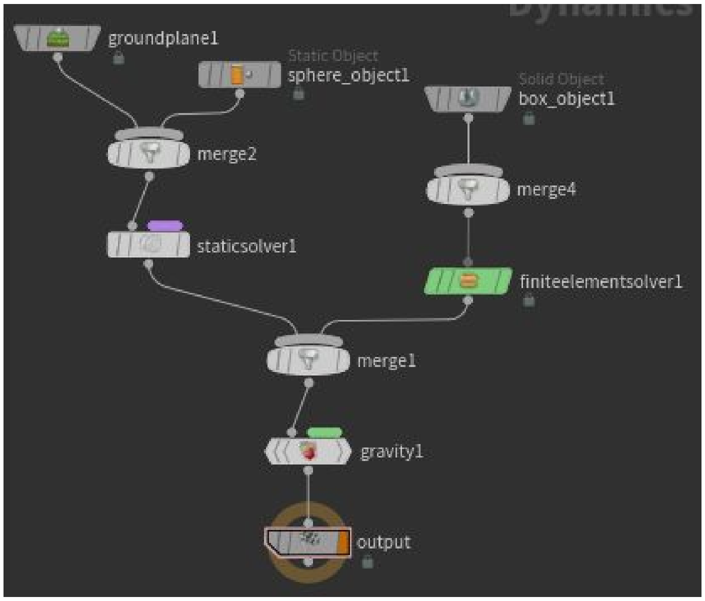


> **NOTE**: It seems that Organic Mass/Tissue will generate the same thing, just with different physics properties (e.g. more stiff / less stiff)?

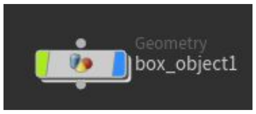
Once you do this, 2 things will happen…

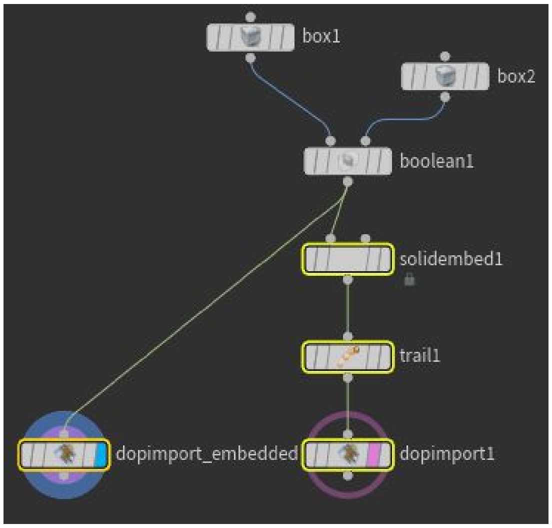
- You'll get a new node called AutoDopNetwork (if it didn't already exist) which contains a node with the same name as the object you've applied RBD to.





- Your geometry object will get a few nodes added to it





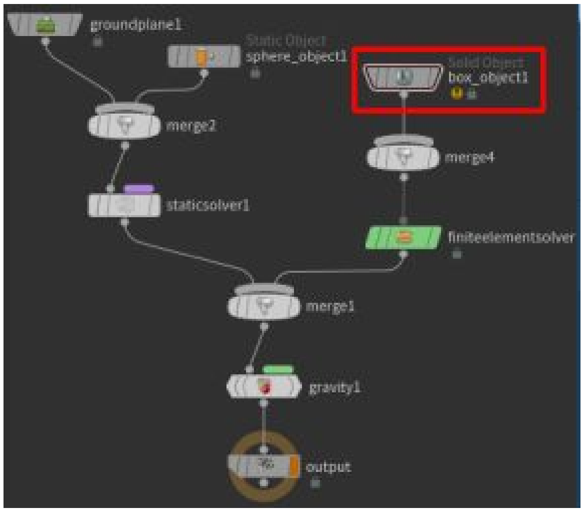
NOTE: The extra nodes that are added into your object, you generally don't want to touch those. They set things up for use in dynamics and create links between your geometry and dynamics.

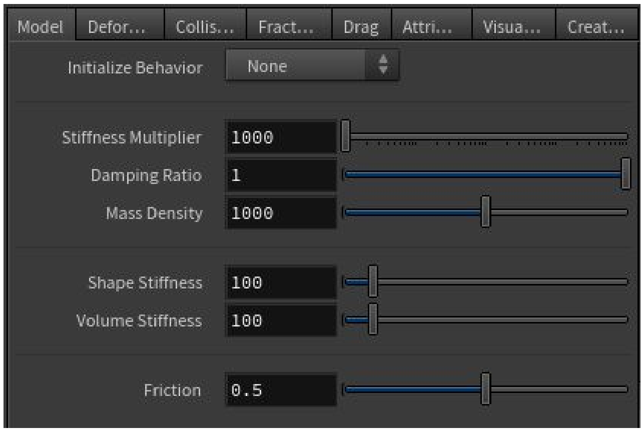
The AutoDopNetwork node is what you traverse into if you want to change the physics properties of your node....

Physical Properties

You can change the physical properties for the simulation of your object by…

1. Traversing into the AutoDopNetwork.
2. Selecting the solid object for that object.
3. In the properties pane, going to the Model tab.





Initialize Behavior: None

Stiffness Multiplier: 1000

Damping Ratio: 1

Mass Density: 1000

Shape Stiffness: 100

Volume Stiffness: 100

Friction: 0.5

NOTE: It looks like these values were really finicky with older versions of Houdini. In certain cases they would cause the deforming geometry to explode. I couldn't get a similar effect to show up in Houdini 16. Houdini 16 seems to have solved that issue?

I'm unsure what all of these settings do. The 'Initialize Behaviour' dropdown up top will change the parameters to represent common materials (e.g. rubber).

Damping Ratio

Officially this is described as "how quickly the object stops deforming". You can also think of this as the springiness of your object.

For example, imagine that you're dealing with a cloth-like piece of geometry that falls/drapes over a sphere. As the cloth drapes, the ends of the cloth may spring back-and-forth heavily. If you jack up the damping ratio, this will happen less?

NOTE: The lower this value is, the more jitter you have to deal with when your object is at rest.

Mass Density

Officially described as "the amount of mass per volume". I assume this means the amount of mass per 1 unit squared of volume.

NOTE: Remember that distance defaults to 1 = 1 meters, so 1 unit of volume is 1 meter squared.

Shape Stiffness

Officially described as how much your object resists changes to its shape. I describe this as how 'melty' your object is. The lower it is, the more your object melts.

Remember that this value is dependent on the mass density value.

Volume Stiffness Officially described as how much your object resists changes to its volume.

Realistically, modifying this value doesn't seem to do much other than maybe make the object less drape-y as it melts (this is with super high values). It's like as the object deforms and goes flat (this happens with a very low SHAPE stiffness or very high mass density), the volume stiffness seems to prevent it from draping over the object. It's almost like throwing a heavy rubber mat over an object.

Friction Friction is how much friction the object has to it (e.g. is it like ice or sandpaper?) . Note that the value applied on collision is a mix of this value and the friction value from the object being collided with.

Accuracy

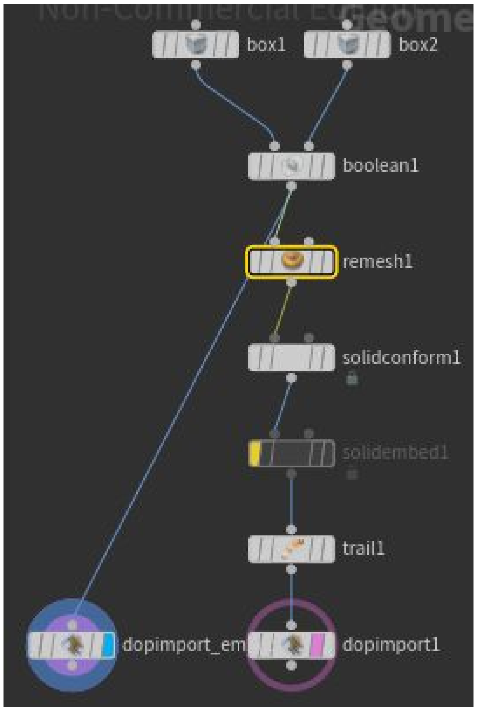
You may notice a small amount of bleedthru with your finite element object when it collides, especially if one of your objects goes really loose and drapes over the other (low surface stiffness).

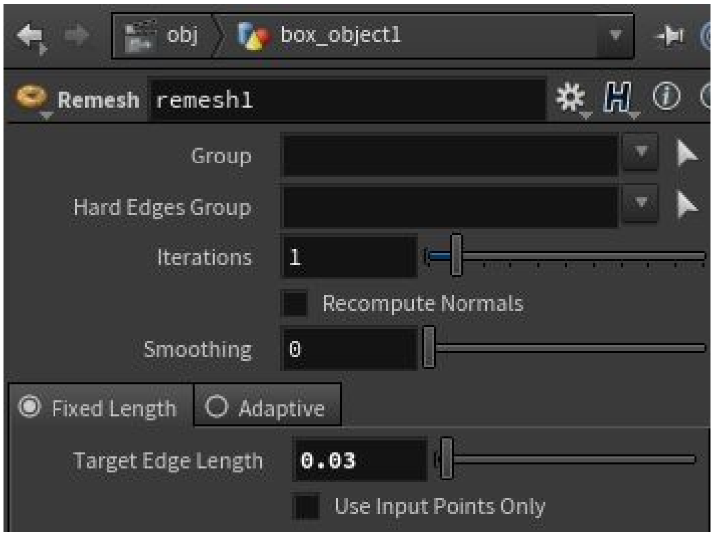
Increasing the poly count of the objects doesn't seem to do anything. What's going on is that the nodes added to the geometry object are automatically creating tetrahedrons based on what the internal algorithms thinks are efficient for the object being passed into the AutoDopNetwork.

Increasing the polys won't increase the number of tets. But, we can MANUALLY increase the number of tets being created for the object by adding a few nodes into the chain. The more tets we have, the more we can stop bleeding (but the sim will also be a lot slower).

To increase the number of tets…

1. Go to your object node
2. Delete the Solid Embed node (or bypass it)
3. Add a Remesh node and a Solid Conform node in its place
4. Go to the Remesh node and change the Target Edge Length property




obj > box_object1

Remesh remesh1

Group

Hard Edges Group

Iterations 1

□ Recompute Normals

Smoothing 0

● Fixed Length ○ Adaptive

Target Edge Length 0.03

□ Use Input Points Only

The lower the Target Edge Length, the more tets will be generated.

> **NOTE**: Feel like doing this to a lot of objects? Shove the nodes inside a subnetwork and make a houdini digital asset out of it. Check out the section on subnetworks in the main document.

Collisions

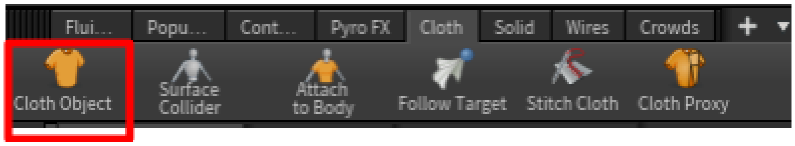
Go to the Solver Engines and Collision section and read it THOROUGHLY. Finite Element objects use RDB, so the objects you're colliding with should have good collision geometry setup for RDB.

Also, the number of test in your model can be boosted (see the Accuracy subsection above) to help if certain parts (tight areas and hard edges?) of your finite element object is being clipped through.

Cloth Objects

As of Houdini 16, cloth no longer uses the finite element solver for simulations. Cloth objects have their own cloth solver that they use. I'm unsure how collisions are done here (Bullet solver? RBD solver? other?).

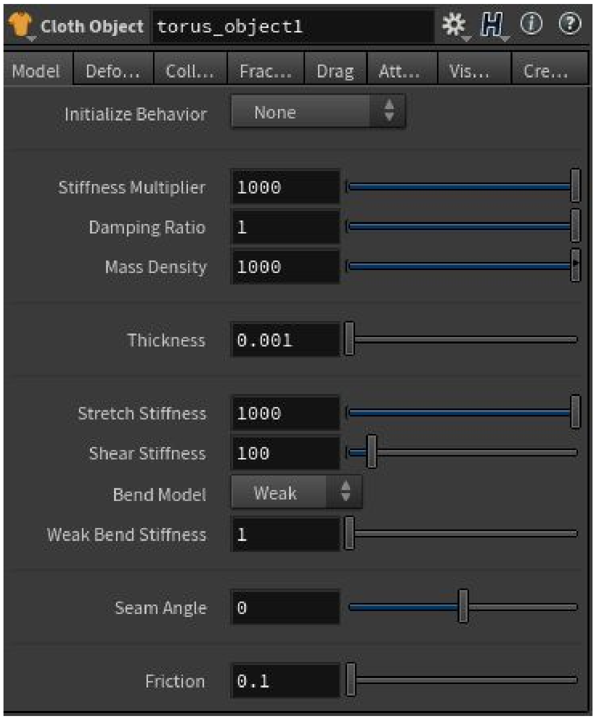
You can turn an object into "Cloth" by going to the Cloth shelf and selecting Cloth Object…



NOTE: Unlike finite element objects, this uses polygons directly. IT DOES NOT CONVERT TO TETRAHEDRONS. If you want more accurate simulations, just up your polycount.

Physical Properties

If you go to the AutoDopNetwork and look at the properties of whatever it was you cloth'd up, you'll see properties similar to those for finite element objects…



Cloth Object torus_object1

Initialize Behavior: None

Stiffness Multiplier: 1000

Damping Ratio: 1

Mass Density: 1000

Thickness: 0.001

Stretch Stiffness: 1000

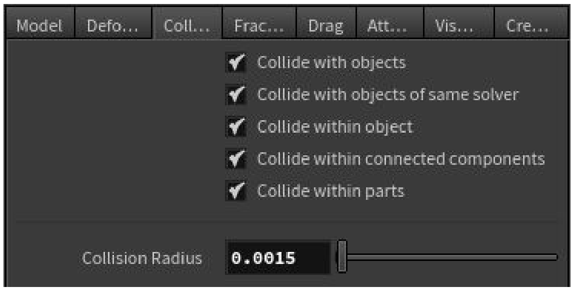
Shear Stiffness: 100

Bend Model: Weak

Weak Bend Stiffness: 1

Seam Angle: 0

Friction: 0.1



☑ Collide with objects

☑ Collide with objects of same solver

☑ Collide within object

☑ Collide within connected components

☑ Collide within parts

Collision Radius: 0.0015

The Initialize Behavior dropdown lets you set these parameters to specific types of cloth (e.g. silk). Also, if you notice any bleed-through happening, you can try fiddling with the Thickness option or go under the Collision tab and fiddle with the Collision Radius option.

Collisions

Go to the Solver Engines and Collision section and read it THOROUGHLY. Finite Element objects use RDB (I think), so the objects you're colliding with should have good collision geometry setup for RDB.

Solver Engines and Collision

There are 3 different solver engines that dynamics can use to calculate how things move and collide:

- Bullet ← open-source physics engine
- RBD ← Houdini's internally built physics engine
- ODE ← open-source physics engine that only deals with shape primitives (e.g. sphere)

Which one gets used is dependent on the solver settings / what's being interacted with. Ultimately, if you're going to be doing anything serious with Houdini, you'll need appropriate collision geometry being generated for both RBD and Bullet.

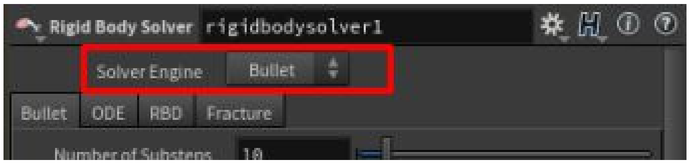
Just remember these two points…

1. Bullet is used for rigid body interactions with other rigid bodies and collision objects
2. RBD is used for everything else (fluids, particles, volumes, cloth, finite element, etc..)

Collision Geometry for Bullet

By default, the rigidbody solver uses the Bullet open-source physics engine to do physics calculations. Bullet seems to generate collision geometry that's more efficient for its calculations vs your real geometry.

> **NOTE**: You can change which engine the solver uses by going to the rigid body solver node and choosing something other than Bullet for the solver engine. If change it to RDB, none of this collision geometry stuff in here will apply to objects being fed into it -- instead see the RDB subsection below.

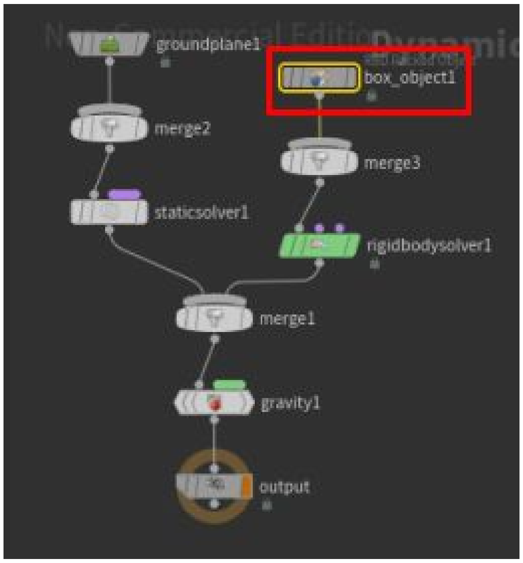


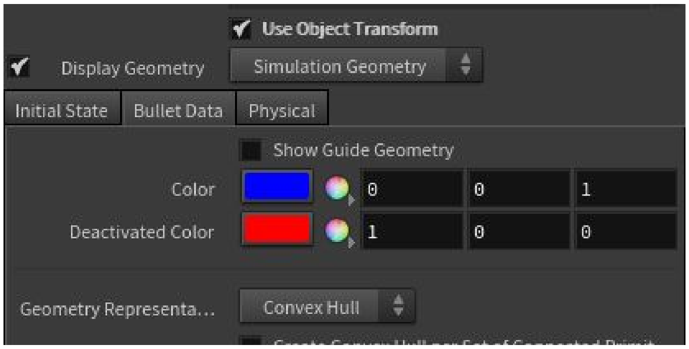
Solver Engine: Bullet

Number of Substeps: 10

You can get to the collision geometry used for the simulation of your object by…

1. Traversing into the AutoDopNetwork.
2. Selecting the RBD object (or Static object) for that object.
3. In the properties pane, going to the Bullet Data tab.





☑ Use Object Transform

☑ Display Geometry: Simulation Geometry

Initial State | Bullet Data | Physical

□ Show Guide Geometry

Color: 0 0 1

Deactivated Color: 1 0 0

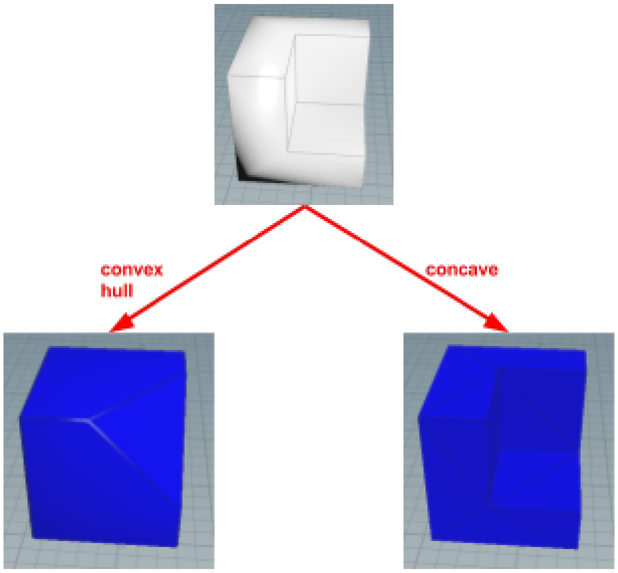
Geometry Representa…: Convex Hull

To show the collision geometry being generated for your object…

1. Uncheck Display Geometry (so only your collision geo will show)
2. Check Show Guide Geometry

   NOTE: If you play your dynamics animation while guide geo is showing, the Color and Deactivated Color will be used to show if Bullet is using your object for computations. Objects that are at rest will be deactivated until they're interacted with again.

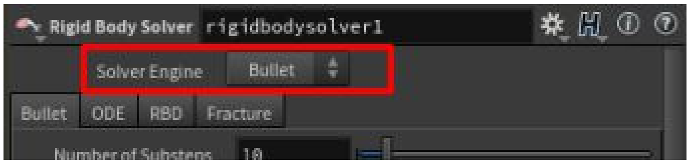
To change your collision geometry…
Use the Geometry Representation dropdown. By default Convex Hull is used, which is a highly reduced version of your geo that doesn't cave in at all. If this isn't good enough, you can choose Concave.



Collision Geometry for RBD

For anything that isn’t a rigidbody-to-rigidbody or rigidbody-to-static interaction, the RBD engine is used. So for example, if your rigid body collides with a finite element object/particles/flip fluids/volumes/etc.., RBD is used.

> **NOTE**: If your rigidbody solver is set to use RBD via its Solver Engine property, then your rigidbody-to-rigidbody and rigidbody-to-static interactions will also use RBD (it defaults to Bullet).

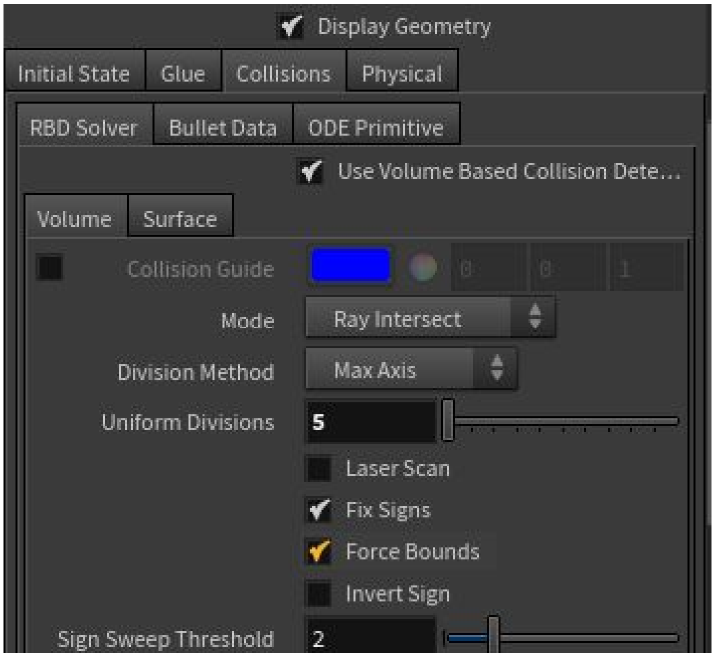


Rigid Body Solver: rigidbodysolver1

Solver Engine: Bullet

Number of Substeps: 10

The RBD Solver tab of your static/rigidbody object will have 2 tabs: Volume and Surface...



☑ Display Geometry

☑ Use Volume Based Collision Dete...

□ Collision Guide: 0 0 1

Mode: Ray Intersect

Division Method: Max Axis

Uniform Divisions: 5

□ Laser Scan

☑ Fix Signs

☑ Force Bounds

□ Invert Sign

Sign Sweep Threshold: 2

> **NOTE**: Remember to uncheck Display Geometry if you’re going to show the Collision Guide

By default, finite element objects use “Volume Based Collision”. I think this tries to approximate a simple volume based on the geometry.

According to the lesson, this is fast but it may not be accurate. You can turn up the uniform division count to make it more accurate, but if your object has a lot of large holes it still won’t be all that accurate. Furthermore it seems that in certain cases colliding geometry will just slowly bleed through the object instead of coming to rest on collision (e.g. fluids or cloth).

If you want more accurate collision geometry…

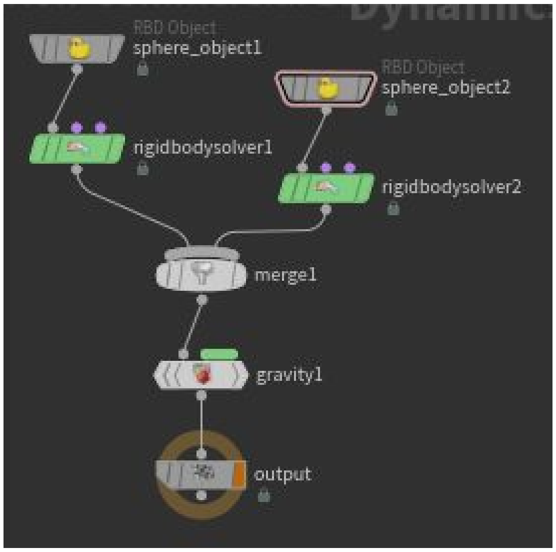
1. Uncheck Use “Volume Based Collision Detection”
2. Switch to the Surface tab

   NOTE: Is there anything worth adjusting in the Surface tabs? You can switch from Points to Edges? The video says depending on the number of points you have vs the number of edges, one may be more efficient than the other? But wouldn’t this be an easy thing to automatically calculate? Why are they making you manually pick it? I think in some cases I saw that you could set a collision radius for surface collisions that’d give the collision surface some extra padding.

Or, you can try generating simpler collision geometry and use that for the volume based collision instead. Simpler geometry means better volumes are generated. See the section on Optimized Collisions for more information.

Solver Gotchas

As far as I know, rigidbodysolver is the only solver node that lets you pick the physics engine to use for your simulations. So what happens when we have 2 rigidbodysolver nodes being merged, one set to use Bullet and the other set to use RBD?



The behaviour slightly varies between the the various options…

- Bullet vs RBD
- RBD vs Bullet
- Bullet vs Bullet
- RBD vs RBD

Ultimately I’m not 100% sure what’s going on here. From experimenting it looks like it uses the engine from the LAST input to the merge (remember that you can reorganize inputs into the a merge by going to the properties).

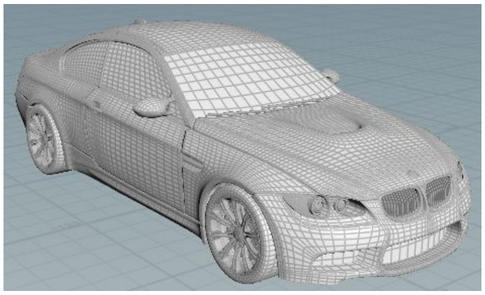
> **NOTE**: I was able to come to this conclusion by having 2 same sized spheres but jacking up the Bullet collision padding property on the second sphere. If it used the padded collision geometry for the collision (instead of sinking into it until it hit the actual sphere), it was using Bullet. No gravity was involved in the test.
>
> **NOTE**: Each one of the Bullet/RBD combos above seemed to act a little differently? No idea what’s going on here.

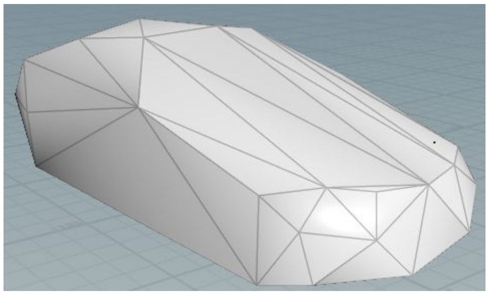
Optimized Collision Geometry

If you’re using highpoly models in your scene, you may not necessarily want to use those same models for your simulations. Your simulations may be able to get away with using much lower resolution models.

Why would you want to do this? It’ll boost the speed of your simulations by a significant amount if you use lower resolution geometry.

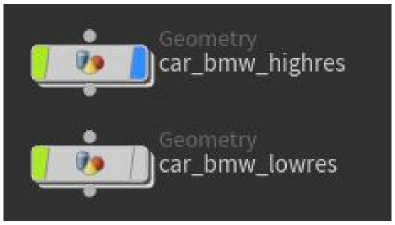
For example, imagine we had 2 models given to us by our modeller: a high-res model for rendering and a low-res model for our rigidbody simulations…





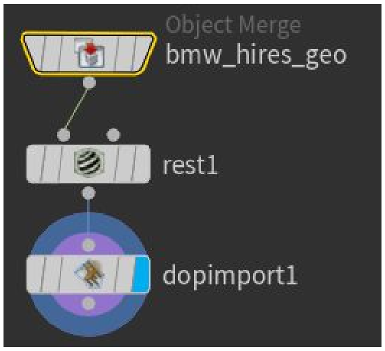
How do we do this?

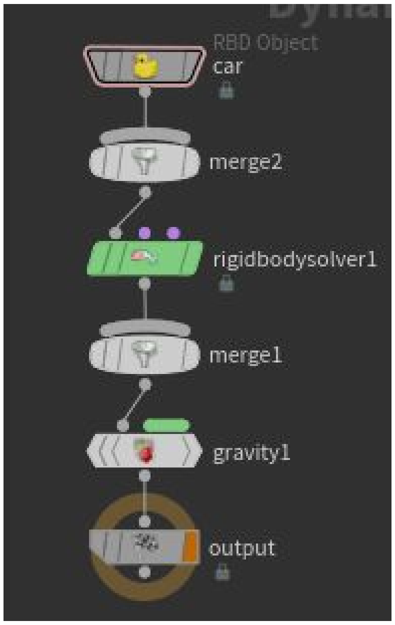
1. Import both models into your scene…



2. Convert the high-res version intro a rigid-body/static object...

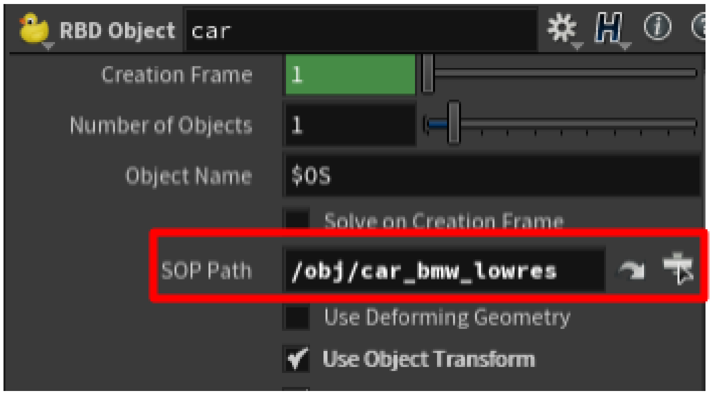
   You'll end up getting the typical nodes in your Geometry node and your AutoDopNetwork





3. Point the RBD Object created in the AutoDopNetwork to the lowres version..

   You can do this by changing the SOP Path property.



RBD Object car

Creation Frame 1

Number of Objects 1

Object Name $OS

□ Solve on Creation Frame

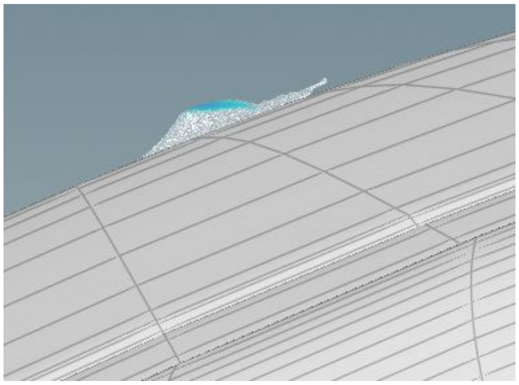
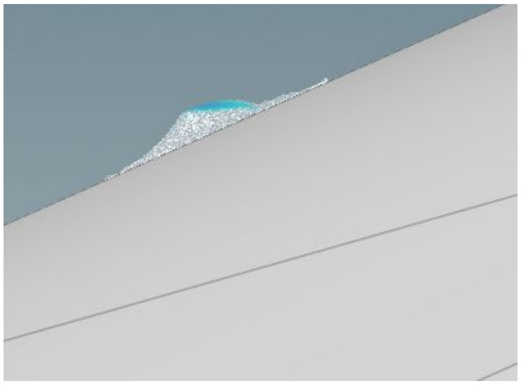
SOP Path /obj/car_bmw_lowres

□ Use Deforming Geometry

☑ Use Object Transform

   NOTE: You can use the little button on the rightmost next to the text field to get a outliner window to quickly pick the geo...

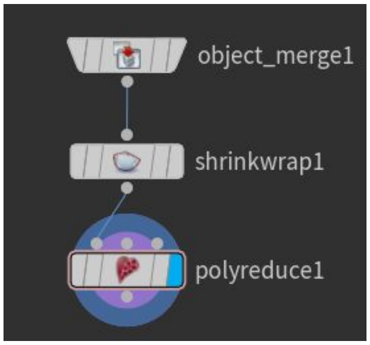
Once you do this, you'll see the low-res version if you go into your AutoDopNetwork, but the high-res version when you come out…


..vs..


Notice how the water isn't really colliding exactly with the highres geometry, but it is with the lowres geometry. A lot of times this is perfectly fine for a render (depends on the angle and various other things), and if you wanted a more accurate collision you can ask for some more polygons in your lowres version.

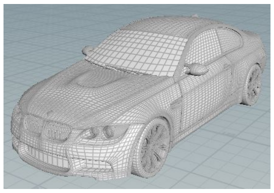
Shrinkwrap Method

If we don't have a simpler geometry given to us by our modeler, we can use Houdini directly to generate some…

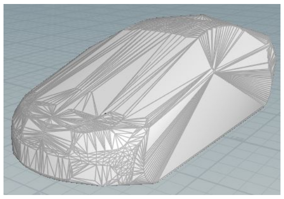


So what's going on here? Here's what each node does…

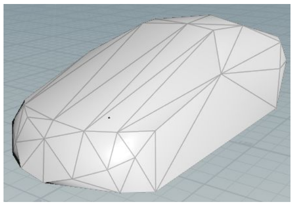
1. Object Merge → pulls in the high-res geometry



2. Shrinkwrap → creates a convex hull polygon mesh out of the geometry



3. Polyreduce → reduces the poly count on the convex hull



You'll need to go into the properties for this node and manually turn down the Keep % number to something you think is appropriate

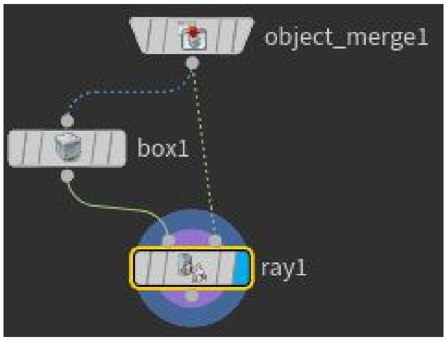
Just as you would have done if lowres geometry was provided to you, point your RBD Object in your DOP network to this new low-res geometry node (or, if you did this in the same geometry node, point directly to the polyreduce node).

This produces geometry very similar to Bullet's default Convex Hull collision geometry. Why would you want to use this instead? If you don't care about the holes in your object for your collision (for example, my car's rims have holes but I'm not doing any specific collision with those in my scene), this will generate a much better RBD collision volume and finish the simulation much faster.

> **NOTE**: Feel like doing this to a lot of objects? Shove the nodes inside a subnetwork and make a houdini digital asset out of it. Check out the section on subnetworks in the main document.

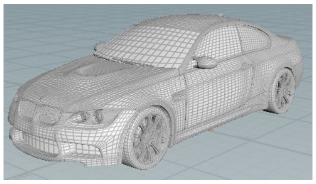
Ray Method

If we don't have a simpler geometry given to us by our modeler, we can use Houdini to directly generate some…

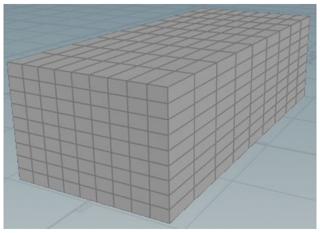


So what’s going on here? Here’s what each node does…

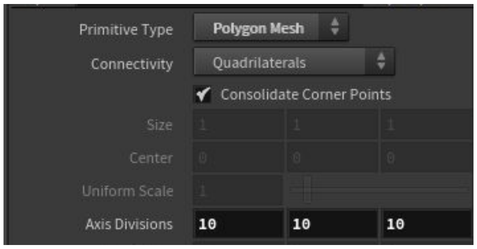
1. Object Merge → pulls in the high-res geometry



2. Box → generates a new polygon box

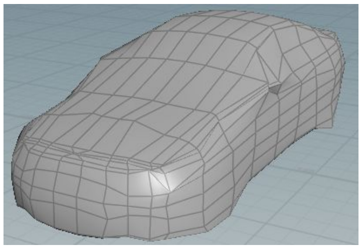


By feeding in our original geometry, the box’s size will match the bounding box of the geometry. You’ll need to go into the properties for this node and make sure the Primitive Type is set to Polygon Mesh and the Axis Divisions will give you the number of faces you want for your low-res geo…

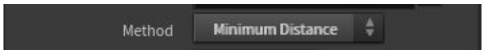


Primitive Type: Polygon Mesh
Connectivity: Quadrilaterals
☑ Consolidate Corner Points
Size: 1 1 1
Center: 0 0 0
Uniform Scale: 1
Axis Divisions: 10 10 10

3. Ray → projects rays from first input to the second input (in the direction of its normal) and moves the geometry to where the rays hit



You'll need to go into the properties for this node and manually set the Method to Minimum Distance...



Method Minimum Distance

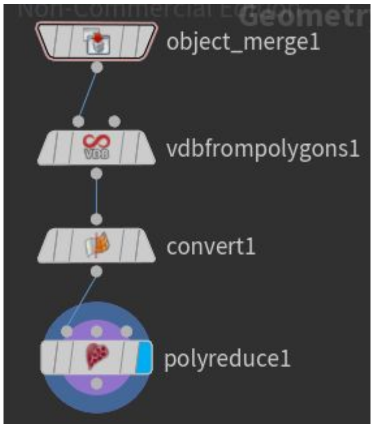
Just as you would have done if lowres geometry was provided to you, point your RBD Object in your DOP network to this new low-res geometry node (or, if you did this in the same geometry node, point directly to the polyreduce node).

This produces geometry that isn't a Convex Hull, so you can use it for those cases where you need a bit more detail for your collisions (e.g. it's important for the colliding object to hit the bottom of the car exactly). Much like the shrinkwrap method, this will generate a much better RBD collision volume and finish the simulation much faster.

> **NOTE**: You may want to add in a polyreduce node at the end of this chain, just because it seems edges seem to accumulate more of the faces than are needed.

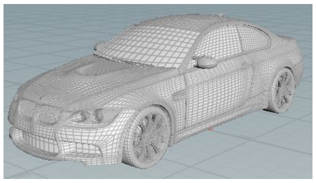
VDB Convert Method

If we don't have a simpler geometry given to us by our modeler, we can use Houdini directly to generate some…

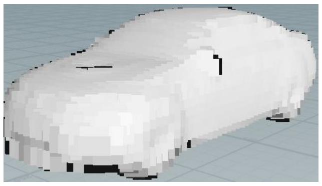


So what's going on here? Here's what each node does…

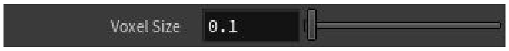
1. Object Merge → pulls in the high-res geometry



2. VDB from Poly → converts your polygon model into voxels

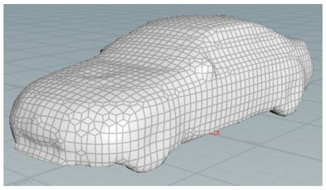


If you have a very large model, this may take a long time to cook during your first run. You'll want to jack up your voxel size in the node's properties so that you can reduce detail…

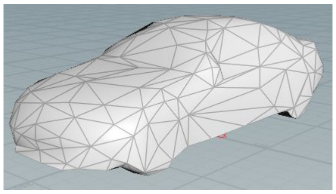


Voxel Size 0.1

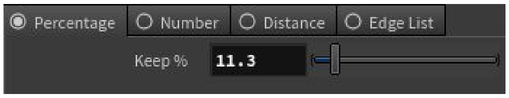
3. Convert → converts your voxels back to polygons



4. Polyreduce → reduces the polycount



You'll need to go into the properties for this node and manually turn down the Keep % number to something you think is appropriate…

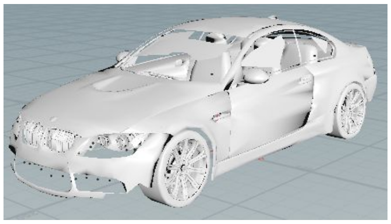


● Percentage ○ Number ○ Distance ○ Edge List

Keep % 11.3

Just as you would have done if lowres geometry was provided to you, point your RBD Object in your DOP network to this new low-res geometry node (or, if you did this in the same geometry node, point directly to the polyreduce node).

This produces geometry that isn't a Convex Hull, so you can use it for those cases where you need a bit more detail for your collisions (e.g. it's important for the colliding object to hit the bottom of the car exactly). Much like the shrinkwrap method, this will generate a much better RBD collision volume and finish the simulation much faster.

> **NOTE**: What the fuck is the point of this? Why don't I just use a polyreduce node directly? Apparently it causes gaps??? Here's what happened when I fed the high-res geometry directly into a polyreduce node and jacked the polycount down…
>
> 
>
> It looks like it just randomly kills faces?

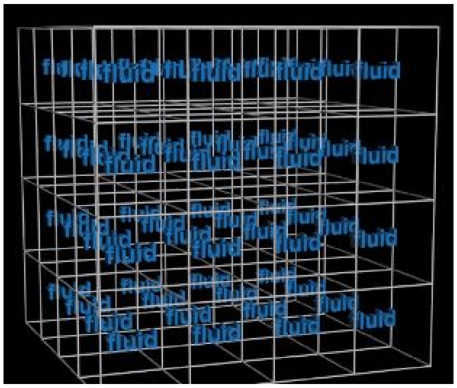
> **NOTE**: Feel like doing this to a lot of objects? Shove the nodes inside a subnetwork and make a houdini digital asset out of it. Check out the section on subnetworks in the main document.

Houdini 16 Fluids

Introduction

Houdini's fluid model is called FLIP (FLuid Implicit Particle). It's a hybrid of 2 different toolsets. In the old days of Houdini, artists were given 2 ways of creating fluids: voxels are particles.

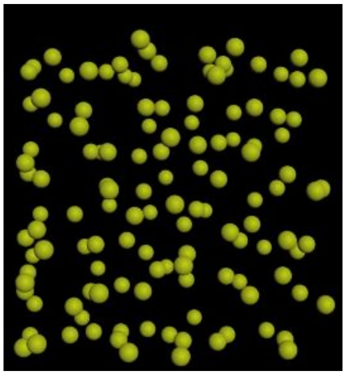
Voxels (also known as volume grid / volume box) were essentially a 3D stack of boxes where each box would contain information about the fluid: direction, speed, pressure, surface shape, etc…



Although this produced good/predictable results for a very long time, it had some problems…

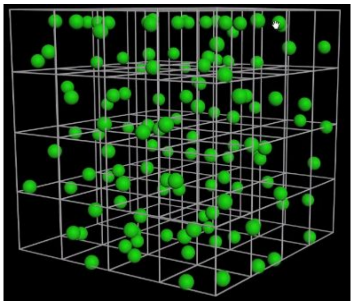
- if your fluid had to go somewhere outside of the box, it couldn't -- it would just disappear.
- if your fluid's voxel resolution was too low, certain parts could disappear at random.
- it was inefficient in that you would have to know the largest possible volume your fluid would consume before hand -- even if you were using a small piece of that volume in the majority of your frames, calculations for the entire volume would have to be done every frame.

Particles were essentially a cloud of points that would make up a fluid.



Unlike voxels, they weren't confined to a certain volume (they could go anywhere) and they were fast to calculate. But, they were also very unpredictable in that they could explode or otherwise act erratically for no apparent reason + they required the extra step of having to generate a surface.

FLIP is a hybrid of both the particle and voxel systems. FLIP essentially calculates each particle's movement inside a voxel, and the overall voxel shape is defined by where the particles are defined in space.



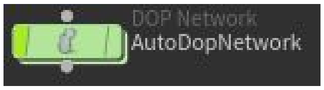
So essentially FLIP is just the volume system and the particle system talking back-and-forth to each other. One is dependent on the other.

> **NOTE**: The only particle thing to note here is that you can apply both volume forces and particle forces to your fluids. This is described in the forces section.

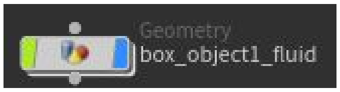
Create Fluids

The various ways to create fluids are documented below. Keep in mind that when you do create a fluid, a few things end up happening…

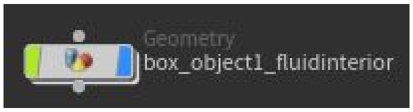
- AutoDopNetwork will get created (if it doesn't exist already).

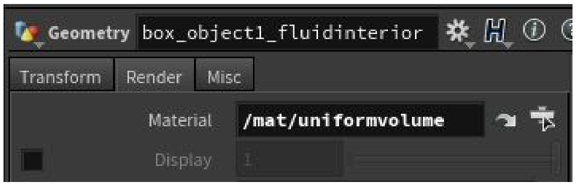
  

- *_fluid geometry will get created -- show you your fluid particles.

- *_fluidinterior geometry will get created -- used for rendering the final fluid material.

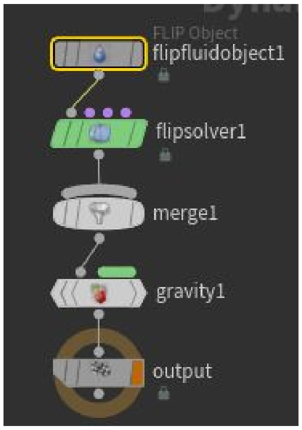
  

  Material /mat/uniformvolume

  Display 1

- Any original geometry you used as part of the creation <u>may</u> have stuff added to it to make it work with the fluid stuff in the AutoDopNetwork.

The inside of your AutoDopNetwork will have a portion added to it for FLIP fluids…

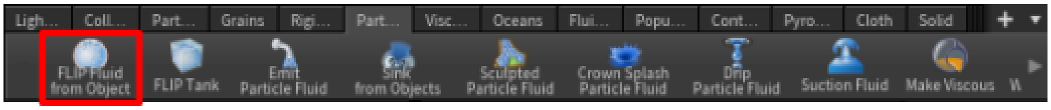


The important thing with this is that FLIP fluids will be confined to a certain bounding box. If they go past the extent of the bounding box they will either disappear or collide (depends on your settings).

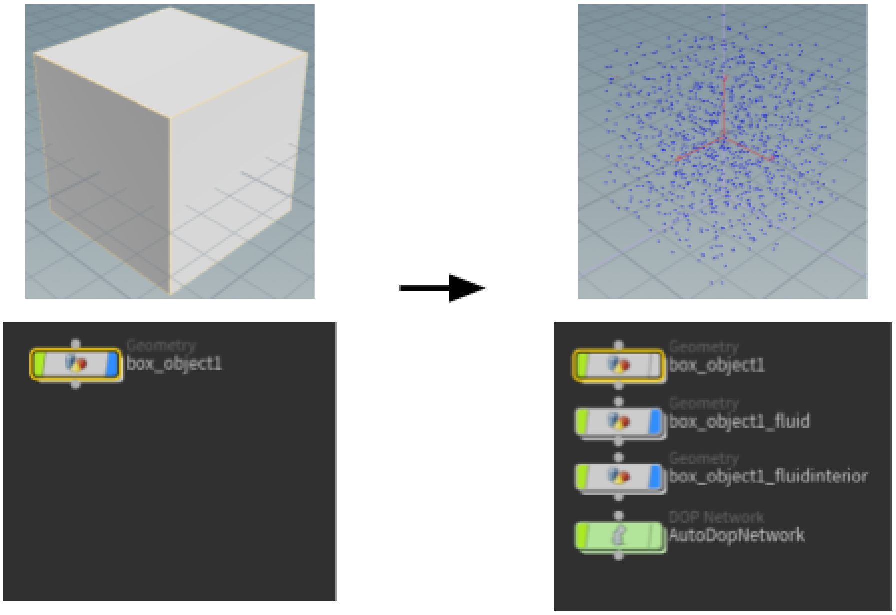
See the Fluid Bounding Box section for more information.

Create

You can convert pretty much any shape into a finite amount of fluid via FLIP Fluid from Object in the Particle Fluids shelf…



If you don't already have an object selected when you do this, it'll ask you to select one and press Enter (this is done inside the scene view). Once it does the conversion, your object will get hidden and in its place will be the particles for your fluid.

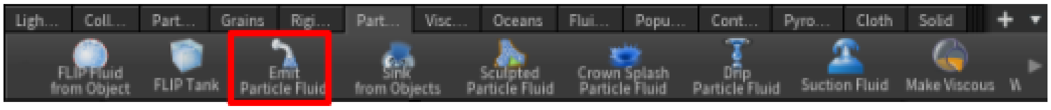


Notice that the original geometry (box_object1) gets hidden once the conversion is complete. If you wanted to expand or reshape or otherwise manipulate the shape, you need to do it on this hidden original geometry.

> **NOTE**: Unsure what fluid and fluidinterior are for? Check the parent section.

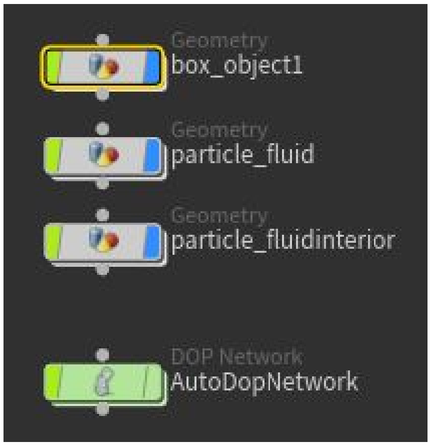
Emit

Just like how you could emit particles from some shape, you can also emit fluid from some shape. You can do this using the Emit Particle Fluid item in the Particle Fluids shelf...

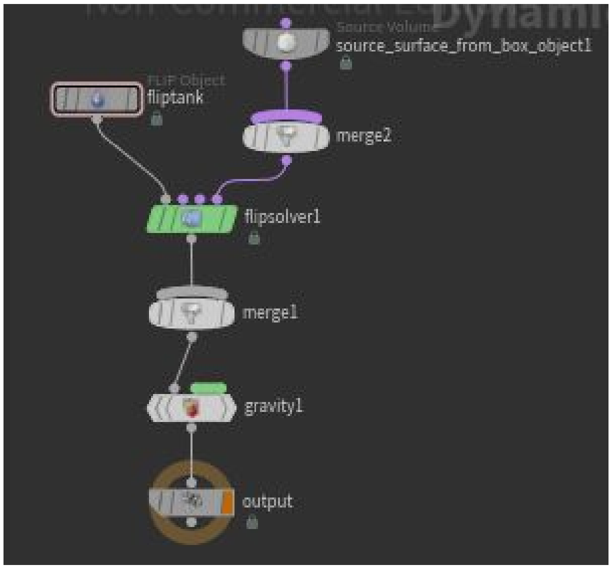


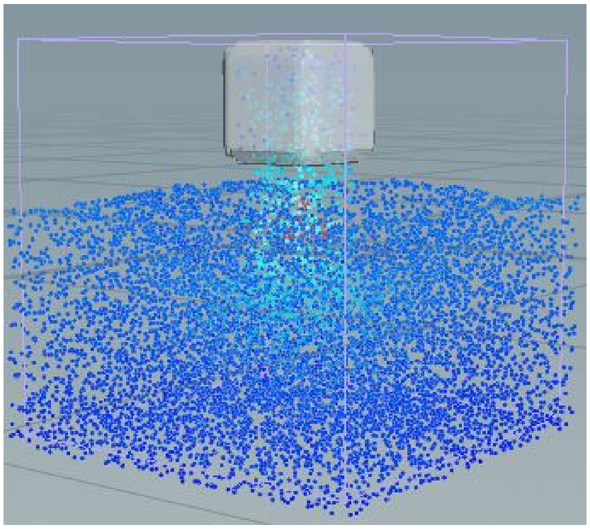
If you don’t already have an object selected when you do this, it’ll ask you to select one and press Enter (this is done inside the scene view).

It’ll then ask you to select the fluid object you want to emit into (if any). For this, go into your AutoDopNetwork and select a FLIP object that feeds into the solver that you want your generated particles to go into, then go back into the scene view and hit Enter.

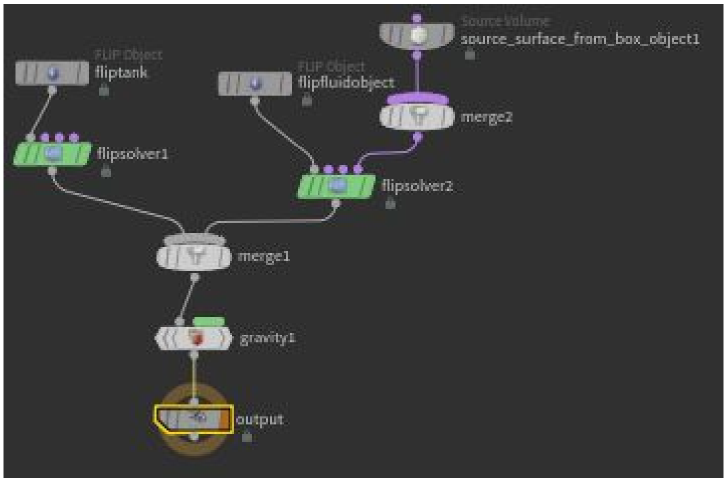


NOTE: Unsure what fluid and fluidinterior are for? Check the parent section.





NOTE: If you don't want it to have a destination, don't select anything and hit Enter. The AutoDopNetwork will create a brand new solver and merge it with any existing flipsolvers. When this happens, the water particles from the different flipsolvers won't end up interacting with each other -- they'll just go through as if it was thin air.

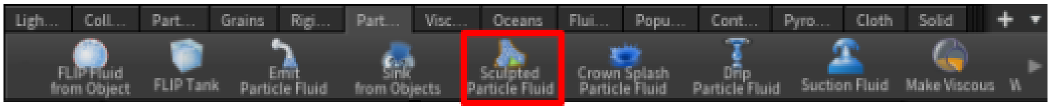


If you want to produce more water particles at once, you'll have to scale up the object you're emitting from.

NOTE: This particular way of creating water lets you set a TTL on the particles. See the Fluid Destruction section for more information.

Fill/Sculpt

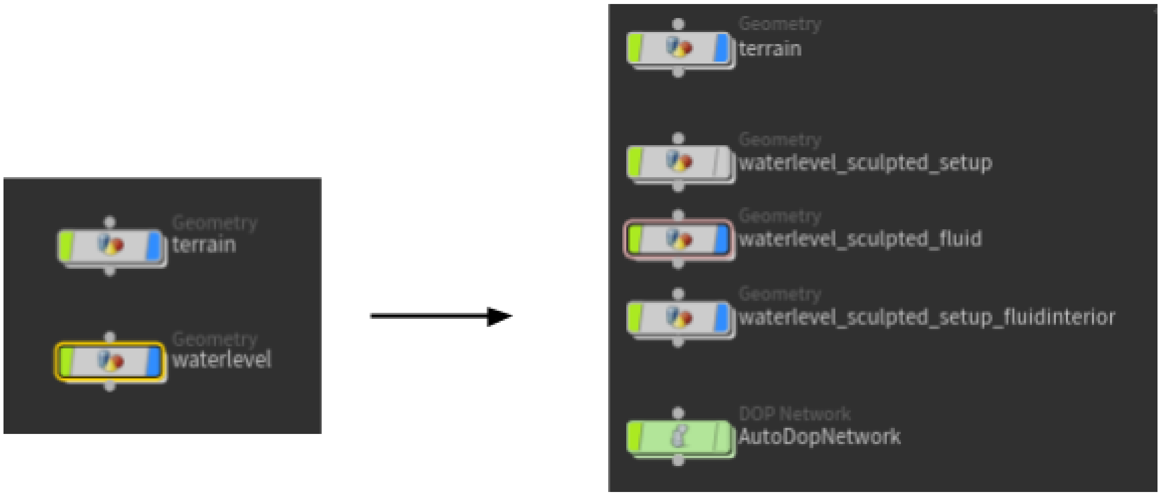
You can make it so that you have a specific volume that gets filled with water in the very beginning (without having to spend frames filling it up with an emitter or anything like that). This is done via Sculpted Particle Fluid in the Particle Fluids shelf...

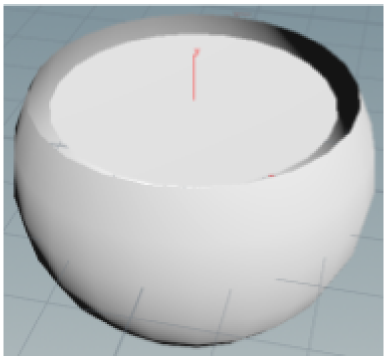


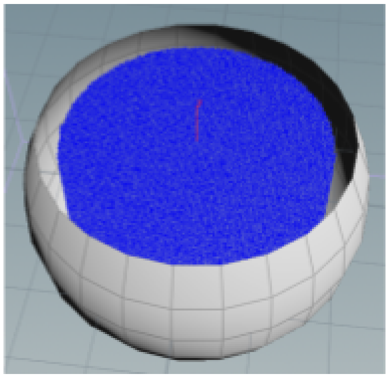
Make sure you have nothing selected when you click this shelf item. When you do click it, you'll prompted in the scene view to make 3 selections (pressing Enter after each selection)

- object that represents how high the water should be
- object that represents the terrain
- object(s) that represents obstructions the water can hit

NOTE: Be careful with what you select as a terrain object. It determines what direction it should try filling in by the normals on that object. So if you notice water filling on the outside of your terrain instead of the inside, you need to reverse the normals. You can do this by shoving a Reverse node into your geometry...



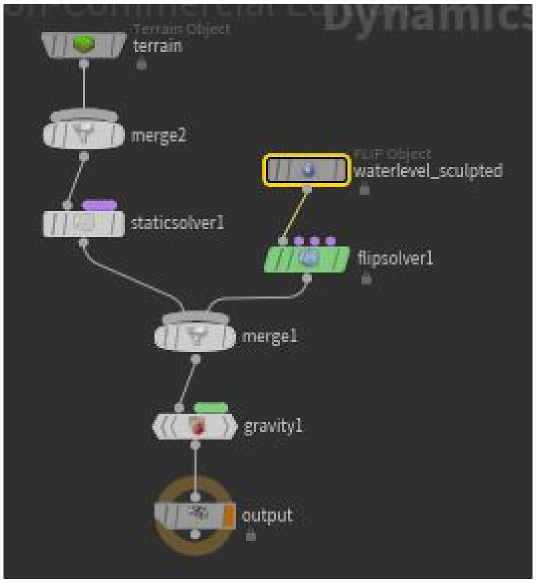




NOTE: Unsure what fluid and fluidinterior are for? Check the parent section.

A few things happen once you do this…

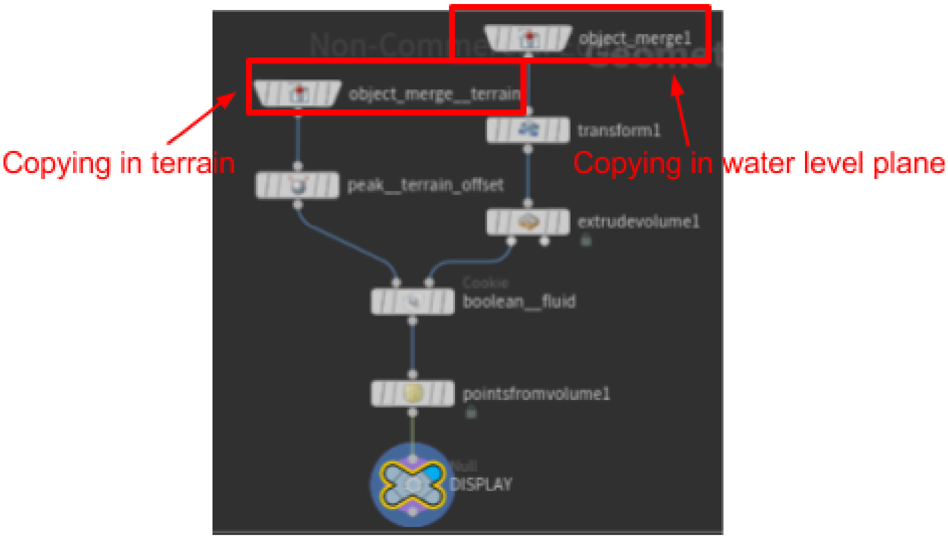
1. your terrain will get a Terrain collision object associated with it (as if you went under the Collisions shelf and chose Terrain).



2. your obstructions get static objects associated with them (as if you went under Collisions shelf and chose Static).

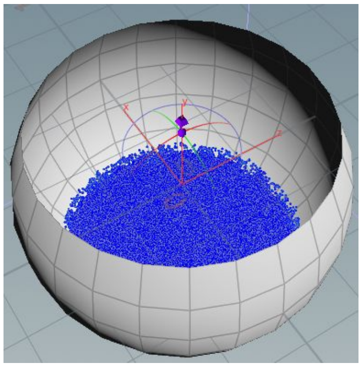
   (no picture here because I didn't choose any, but this would be getting fed into the staticsolver along with the terrain)

3. your water level geometry node gets renamed with a "_sculpted_setup" suffix and stuff gets added to it such that it generates particles in the terrain shape up the water level you specified.



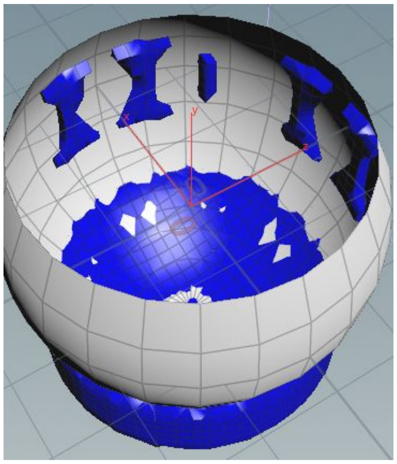
There are lots of problems here…

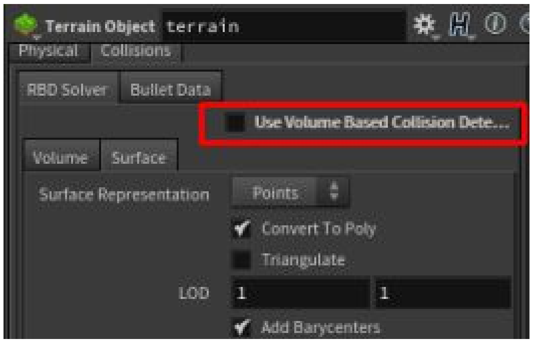
The end result of the example above was that the water would not stay in the basin. It seems that the particles would seep and gradually get stamped down (or disappear?) as the simulation progressed.



NOTE It almost seems as if the water is going down to some invisible basin underneath the sphere. See the picture of the volume based collision geometry below.

The collision geometry generated on the terrain in the example above is extremely messed up. It uses volume based collision by default and does a terrible job of creating the collision geometry. Switching to surface based collision seems to help only sometimes. The problem is that you don't have the option to set Collision Radius on a Terrain object like you do with a Static Object, which means in certain cases seeping of water between the seams still happens.





Terrain Object terrain

□ Use Volume Based Collision Dete...

Surface Representation: Points

☑ Convert To Poly

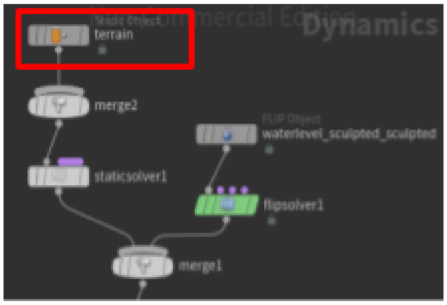
□ Triangulate

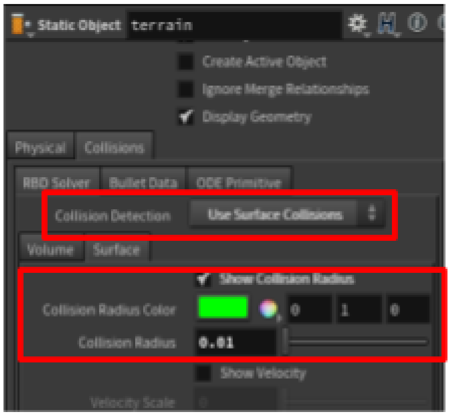
LOD 1 1

☑ Add Barycenters

NOTE: Adding thickness to the terrain geometry (polyextrude node) did nothing.

NOTE: One potential workaround I found to the collision issue was to manually remove the terrain object from the AutoDopNetwork (including whatever was added to the Geometry node to get it in there) and place it back in as a Static object. That way you'd get access to the extra options for Surface collisions. BUT, the problem here is that the fluid particles just disappear as gravity pulls them down and they collide -- it's as if they've hit the bottom of the fluid bounding box. I don't know how to fix this but there's probably a way.

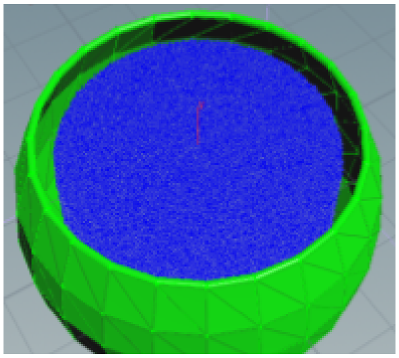



Static Object terrain

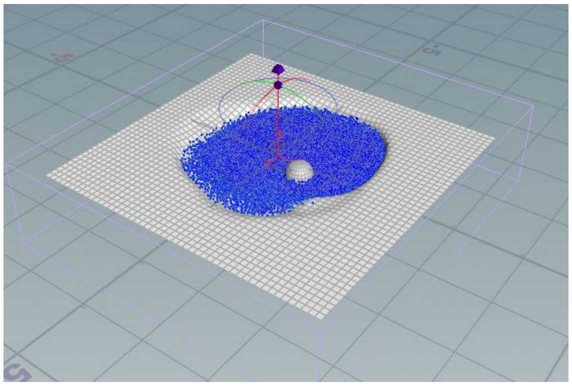
□ Create Active Object
□ Ignore Merge Relationships
☑ Display Geometry

Collision Detection: Use Surface Collisions

☑ Show Collision Radius
Collision Radius Color 🟩 0 1 0
Collision Radius 0.01
□ Show Velocity
Velocity Scale 0



NOTE: Okay, so after continued experimentation, it looks like that it may be this is expecting terrain-like geometry. That is, if I can a normal grid and sculpt some grooves into it, then use another grid for the water-level, everything works perfectly fine. Surface collision mode on the terrain object also seems to work well. I don't know why my original cut-off circle geometry didn't work.



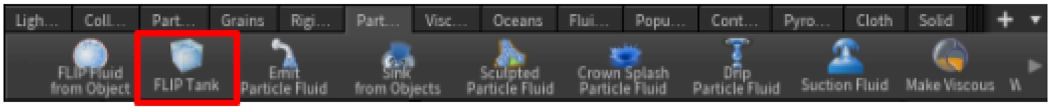
For best results you should give your terrain/static objects some depth (make sure they aren't thin like paper)

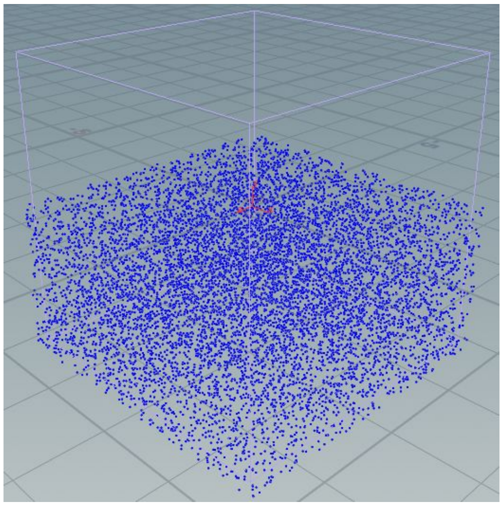
Sandbox (FLIP Tank)

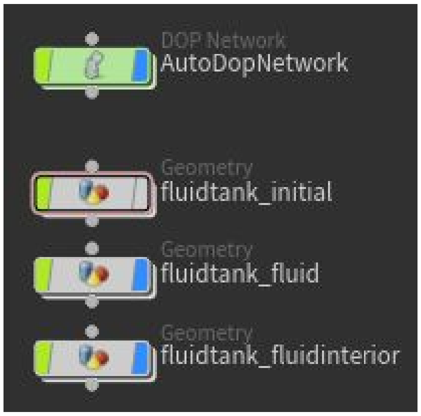
A FLIP tank is almost the same thing as creating a box and converting it to a fluid, except that…

- the bounding box for the solver will be linked to the bounds of the box object (this is called fluidtank_initial)
- the fluid will fill roughly 50% of the bounding box
- the fluid will collide with the bounding box instead of go through it

FLIP tanks are often used as a sandbox to play with water. You can do create a FLIP tank via the FLIP Tank item in the Particle Fluids shelf...







Destroy Fluids

There are a few ways to destroy fluid particles.

Bounding Box Collision

You can control what happens to your fluid particles when they hit the walls of your fluid solver's bounding box. To see how to do this, check out the Fluid Bounding Box section.

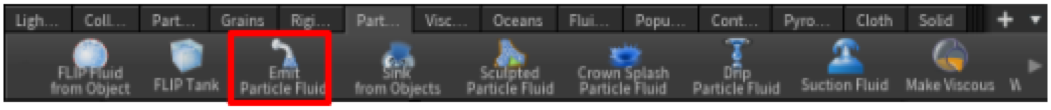
The default for whether your fluid disappears or not depends on the exact way your fluid was created.

For example, if you created your fluid from the FLIP Fluid Object item in the Particle Fluid shelf, the fluid particles disappear when they hit the bounding box walls. But, if you created your fluid using the FLIP Tank item under the same shelf, they'll collide instead of disappear.

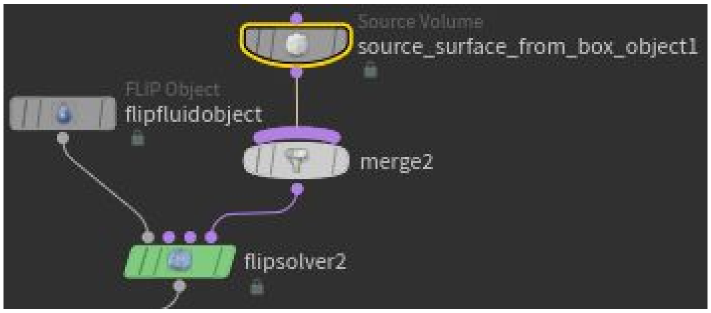
NOTE: It should be obvious why this is. FLIP Tank is meant to be a sandbox playground. What good is a water testing playground if there's no water.

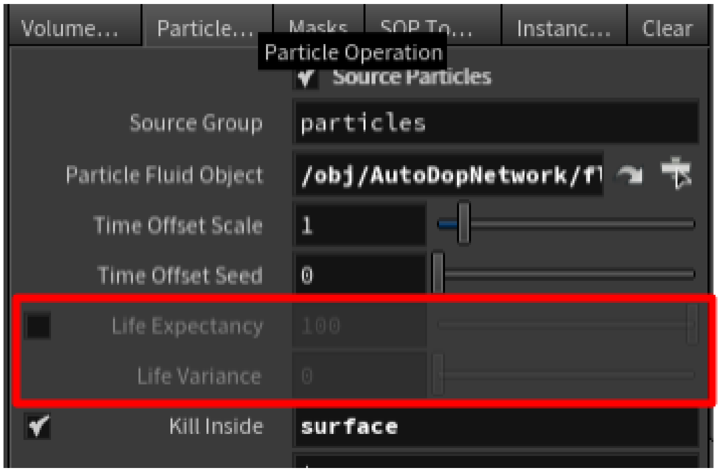
Particle TTL (time-to-live)

You can set TTLs on fluid particles generated from an emitter (if you chose the Emit Particle Fluid item in the Particle Fluids shelf)...



If you want to kill water particles after a certain amount of time (control lifespan), you can do so by going to the Source Volume node in your AutoDopNetwork and selecting the Particle Operation tab. Enable Life Expectancy and set the value to whatever you need to (value is in seconds).





☑ Source Particles

Source Group: particles

Particle Fluid Object: /obj/AutoDopNetwork/fl

Time Offset Scale: 1

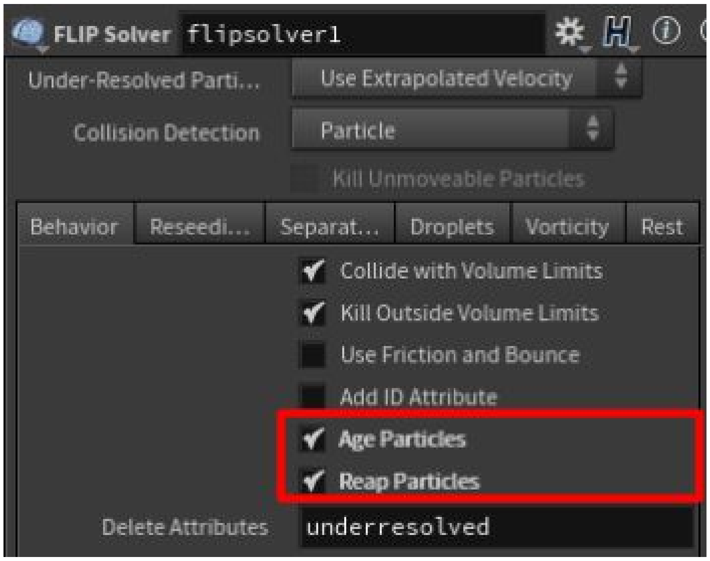
Time Offset Seed: 0

□ Life Expectancy: 100

Life Variance: 0

☑ Kill Inside: surface

Once you set this, you also need to go to the solver and specifically tell it to age particles and reap particles.



FLIP Solver flipsolver1

Under-Resolved Parti... Use Extrapolated Velocity

Collision Detection Particle

□ Kill Unmoveable Particles

☑ Collide with Volume Limits

☑ Kill Outside Volume Limits

□ Use Friction and Bounce

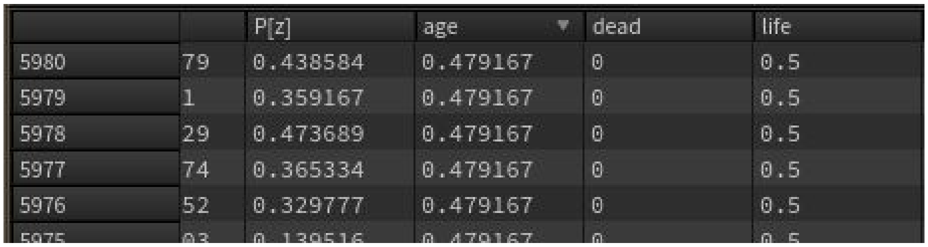
□ Add ID Attribute

☑ Age Particles

☑ Reap Particles

Delete Attributes underresolved

NOTE: If you set the solver to enable age+reap particles, technically you can add an age parameter to particles generated from anything? If you look at the geometry spreadsheet, you can see what extra attributes were added to each particle to get them to age…

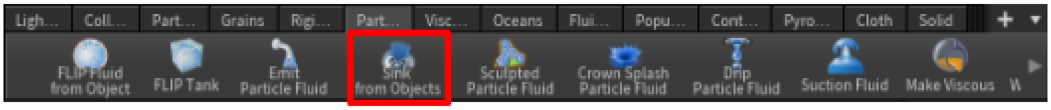


		P[z]	age	dead	life
5980	79	0.438584	0.479167	0	0.5
5979	1	0.359167	0.479167	0	0.5
5978	29	0.473689	0.479167	0	0.5
5977	74	0.365334	0.479167	0	0.5
5976	52	0.329777	0.479167	0	0.5
5975	93	0.139516	0.479167	0	0.5

The life attribute seems to be the big one here. It tells the solver how long to keep the particle around. There doesn't seem to be a node available in the dynamics context to add an attribute to these particles, so it must be done via VEX or python? I don't know enough about Houdini to do that yet, but I think it's definitely doable.

Sink

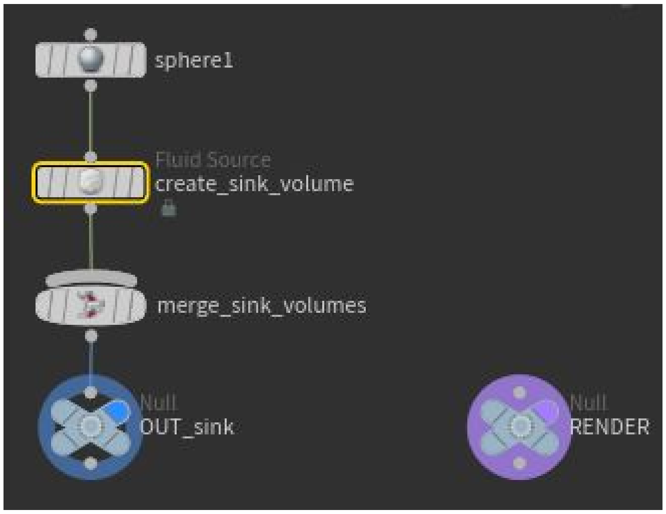
Just like how you could emit fluid particles from some shape, you can turn some shape into a sink for fluid particles. You can do this using the Sink from Objects item in the Particle Fluids shelf...



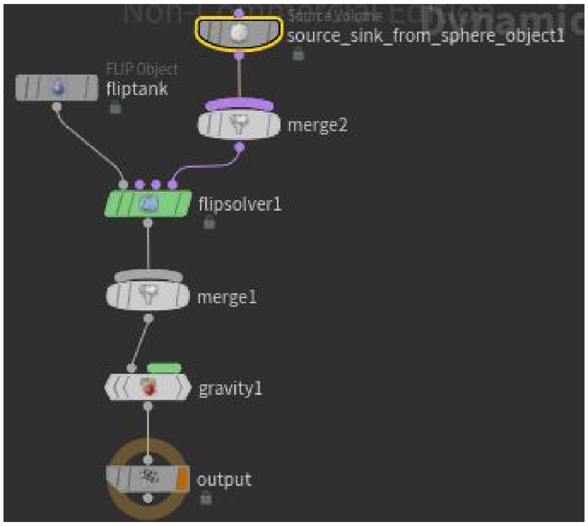
If you don't already have an object selected when you do this, it'll ask you to select one and press Enter (this is done inside the scene view).

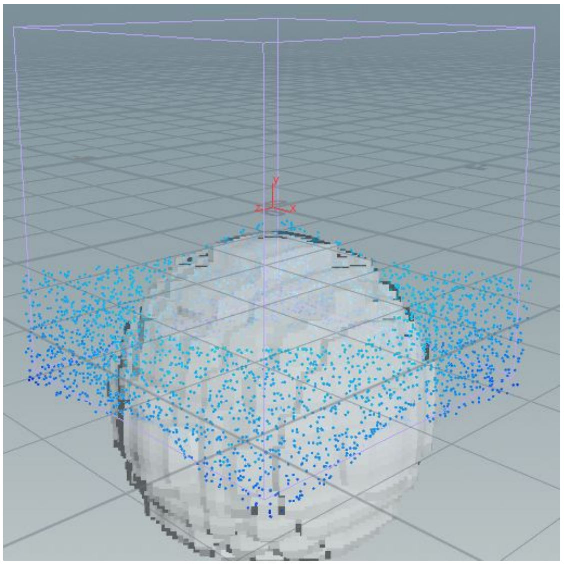
It'll then ask you what "smoke object" you want your sink added to. This is probably a bug. What it's asking for is the fluid solver you want your sink added to. For this, go into your AutoDopNetwork and select a FLIP object that feeds into the solver that you want your fluid particles disappear from, then go back into the scene view and hit Enter.

You shouldn't get any new objects under your /obj context. But, the object you selected to be your sink will have some stuff added to it…



In addition to that, in your AutoDopNetwork, you'll find a new "Source Volume" node added to the flipsolver you chose. If you run your simulation, you'll notice that the water particles will start draining into the object.





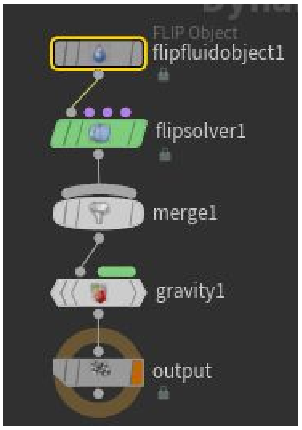
NOTE: If you don't see the particles draining, it may be that your sink object isn't big enough. Jack up the size.

> **NOTE**: Notice that, just like for an emitter, a sink adds a Source Volume node. The difference (I guess) is how the node is set up. It's probably set to kill particles that enter it vs generate particles from it.

If you want to consume more water particles at once, you'll have to scale up your sink object.

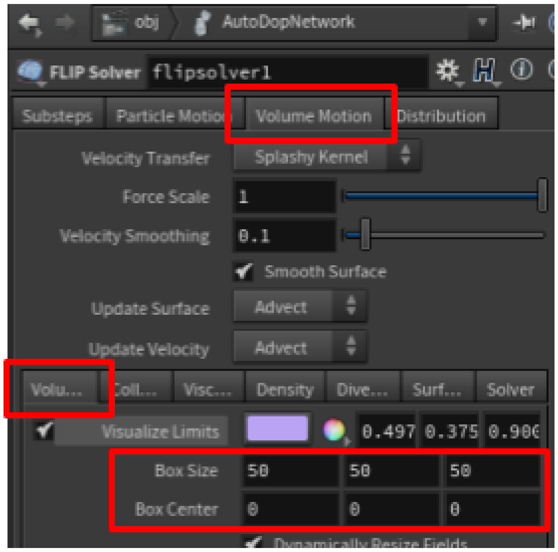
Fluid Bounding Box

The important thing with this is that FLIP fluids will be confined to a certain bounding box. If they go past the extent of the bounding box they will either disappear or collide (depends on your settings).



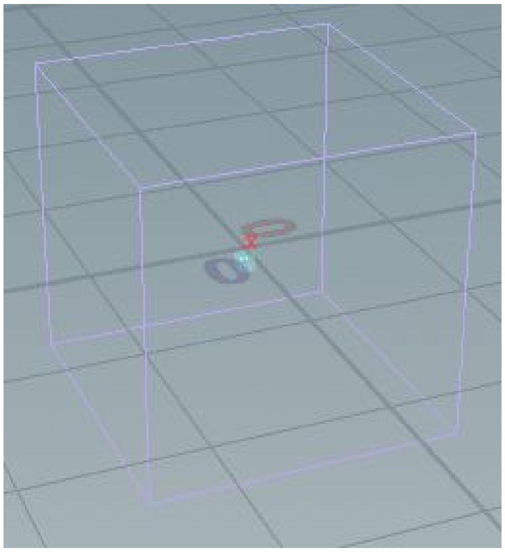
Bounding Box Size

To control the size of the bounding volume box, go to the flipsolver node. Under Volume Motion -> Volume Limits you'll find the Box Size and Box Center properties…

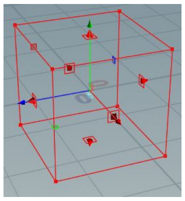


flipsolver1

Velocity Transfer: Splashy Kernel
Force Scale: 1
Velocity Smoothing: 0.1
✓ Smooth Surface
Update Surface: Advect
Update Velocity: Advect
✓ Visualize Limits 0.497 0.375 0.90€
Box Size: 50 50 50
Box Center: 0 0 0
✓ Dynamically Resize Fields

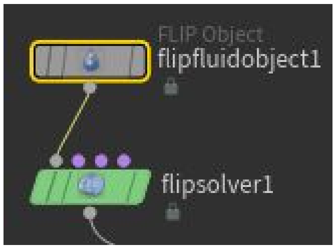


NOTE: You can also use the handle tool and change this directly in the scene view…

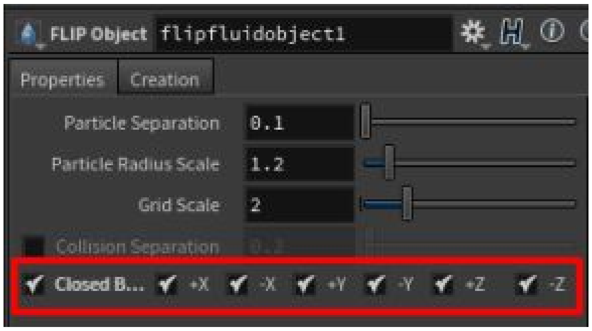


Bounding Box Collision Behaviour

To control whether particles collide or disappear when they hit the limits of the bounding box, you need to go to whatever FLIP object it is that's being fed into the solver. Under the Properties tab, you'll see an option called Closed Boundaries that'll let you choose which parts of the box you want to treat as closed (collides with) or open (disappears when hit).



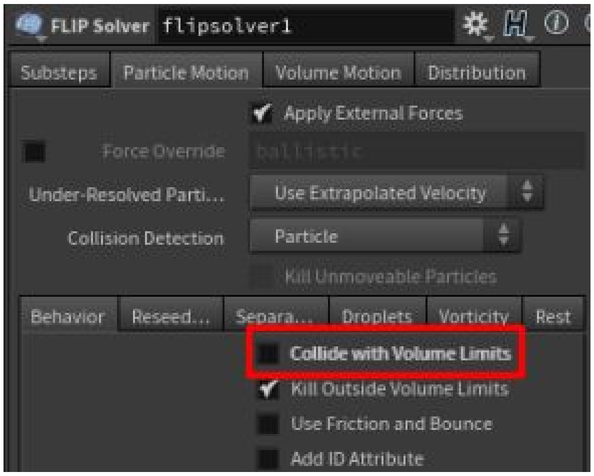
FLIP Object
flipfluidobject1
flipsolver1



FLIP Object flipfluidobject1

Particle Separation 0.1
Particle Radius Scale 1.2
Grid Scale 2
☐ Collision Separation 0.1
☑ Closed B... ☑ +X ☑ -X ☑ +Y ☑ -Y ☑ +Z ☑ -Z

If you choose to have them collide with the boundary, you can set it so that they force of the water doesn't get reflected back when it hits that boundary by unchecking the Collide with Volume Limits checkbox in the flipsolver (under Particle Motion -> Behaviour)...



FLIP Solver flipsolver1

☑ Apply External Forces
☐ Force Override ballistic
Under-Resolved Parti... Use Extrapolated Velocity
Collision Detection Particle
☐ Kill Unmoveable Particles

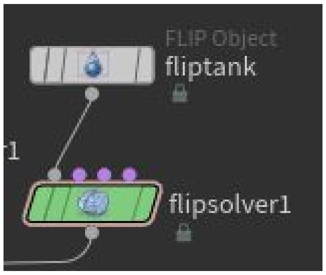
☐ Collide with Volume Limits
☑ Kill Outside Volume Limits
☐ Use Friction and Bounce
☐ Add ID Attribute

Fluid Collisions with RBDs

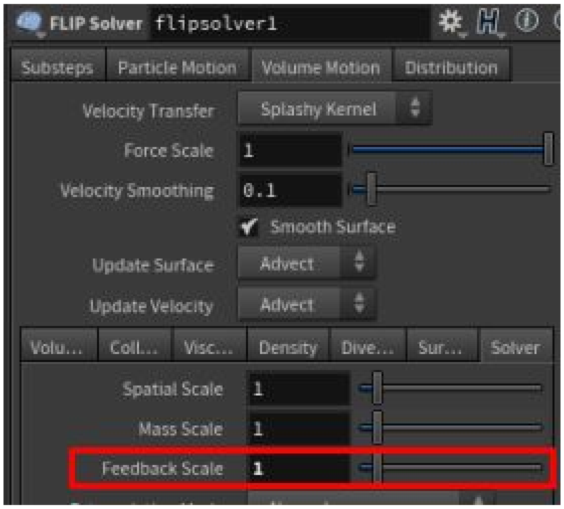
There doesn't seem to be a foolproof way of having FLIP fluids properly interact with collision objects. Ultimately that means that if you're trying to simulate an object floating on or inside the water, you're going to have a tough time. The following sections are an overview of the possible techniques to deal with this.

Feedback Scale, Density, and Collision/Particle Separation

You can adjust how much 'feedback' the water gives to collision objects based on the Feedback Scale value in the flipsolver…



In your solver, you can find the Feedback Scale slider under Volume Motion -> Solver. The Feedback Scale goes from 0 to 1, where 0 is 0% and 1 is 100%. I noticed that somewhere up high (around 0.9 or 1.0) is a good value.

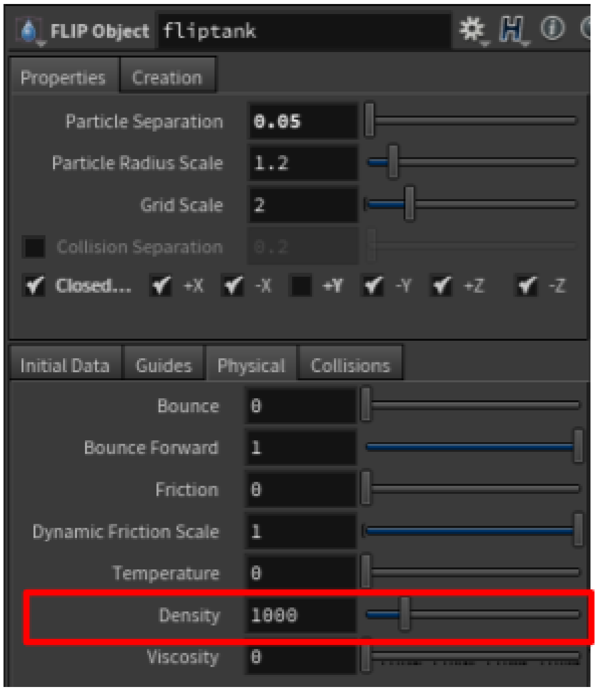


FLIP Solver flipsolver1

Velocity Transfer: Splashy Kernel
Force Scale: 1
Velocity Smoothing: 0.1
☑ Smooth Surface
Update Surface: Advect
Update Velocity: Advect

Spatial Scale: 1
Mass Scale: 1
Feedback Scale: 1

Once you do this, you'll need to potentially tweak the density of your water and the RBD objects you throw in them. You can tweak the density of your water by going to Physical tab of your flipobject. This density value will interact with the density of your RBD object (and the feedback scale to determine) to determine if your object should float or sink or whatever.



FLIP Object fliptank

Particle Separation 0.05
Particle Radius Scale 1.2
Grid Scale 2
□ Collision Separation 0.2
✓ Closed... ✓ +X ✓ -X □ +Y ✓ -Y ✓ +Z ✓ -Z

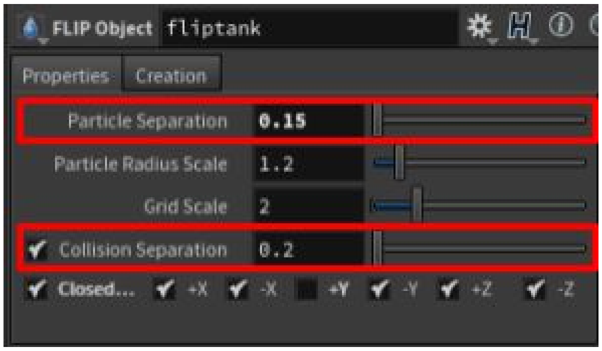
Bounce 0
Bounce Forward 1
Friction 0
Dynamic Friction Scale 1
Temperature 0
Density 1000
Viscosity 0

NOTE: You can find the density of your RBD object in the Properties tab of your RBD object. Remember that changing the density means that it'll your mass is being modified, which means that you'll potentially mess with other physics-related stuff.

Once you do this, you may notice highly erratic behaviour in your simulation with the way the RBD objects react to fluids. There are a couple of things you can tweak under the Properties tab of the flipobject to help with this.

The first is the Particle Separation value. A low Particle Separation value of your flipobject means more particles, which means a more accurate simulation (but also one that's more computationally expensive).

The second is the Collision Separation value. I don't know exactly what this does, but enabling it seems to get rid of all the erratic behaviour without having to lower the Particle Separation value.

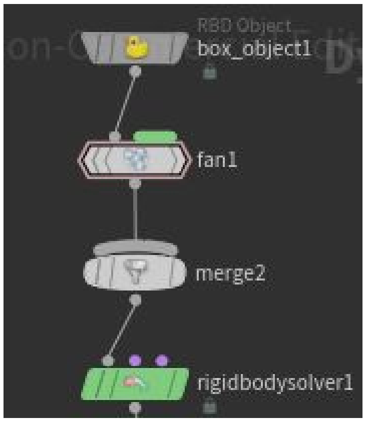


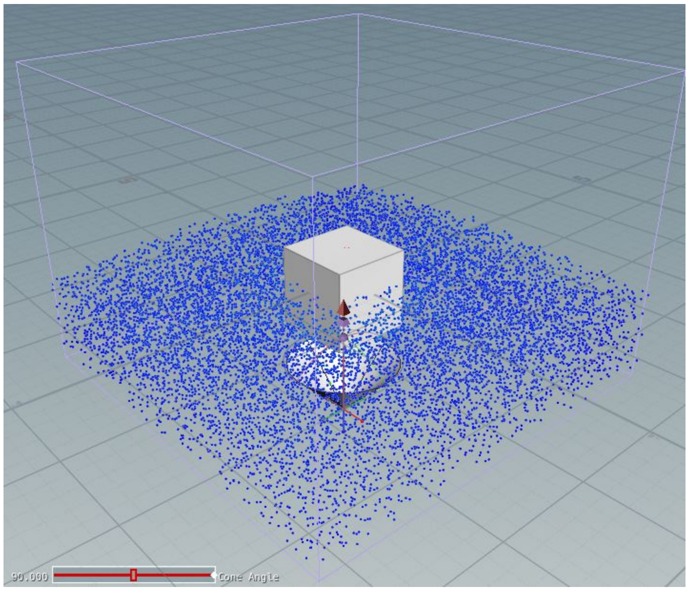
FLIP Object fliptank

Particle Separation 0.15
Particle Radius Scale 1.2
Grid Scale 2
✓ Collision Separation 0.2
✓ Closed... ✓ +X ✓ -X □ +Y ✓ -Y ✓ +Z ✓ -Z

Apply Artificial Force to RBD Object

One option is to having your object interact with water would be to apply a force to the object to make it seem like the water is resisting it. For example, you can stick a Fan Force node after your RBD object to simulate buoyancy in the water. The water will still move around properly as the box goes up-and-down inside it, making it look realistic.

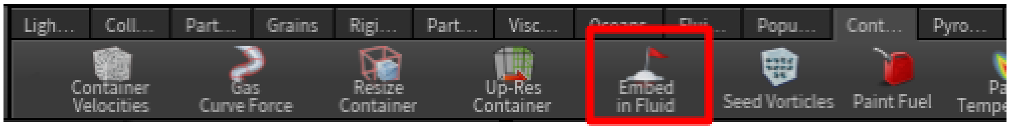




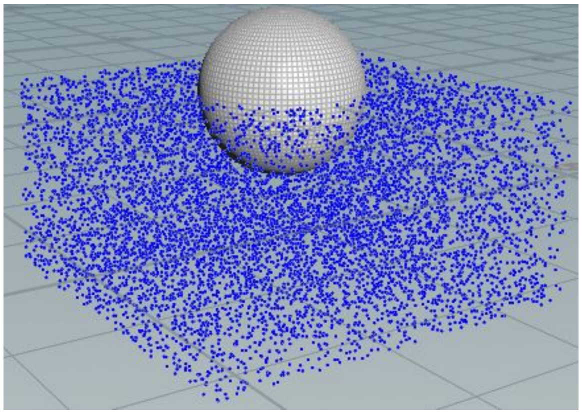
NOTE: The fan is beneath the box. It's being partially obstructed by the water particles. Remember the force that's being applied to the RBD does not have an effect on the water.

Embed In Fluid Tool

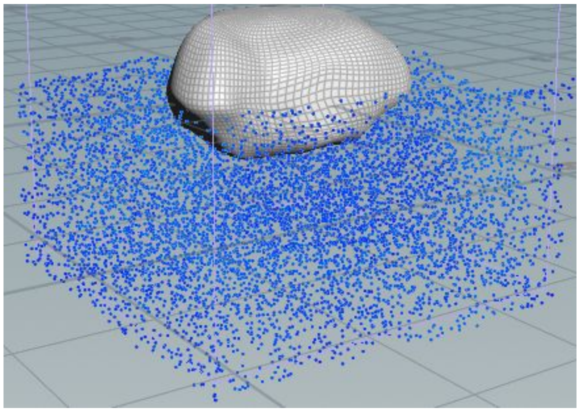
One way to get your objects to interact with water would be to use the Embed in Fluid item under the Container Tools shelf.



This is almost always NOT what you want. What it does is that it takes the POINTS of the object and embeds it alongside the FLIP fluid particles, causing your object to deform over time rather than interact with the water…



→



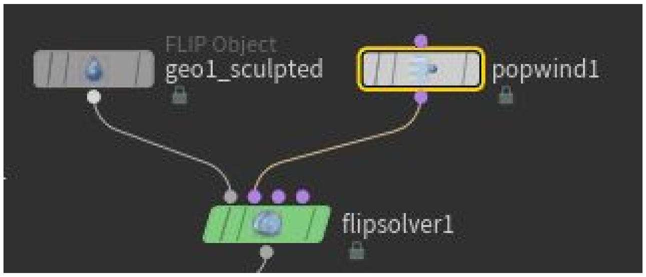
Nevertheless, if you want to use this…

1. make sure you have nothing selected
2. click the shelf item
3. select the object to embed and press Enter
   (you'll be prompted in the scene view)
4. select the flipobject in your AutoDopNetwork and press Enter
   (you'll be prompted in the scene view to do this)

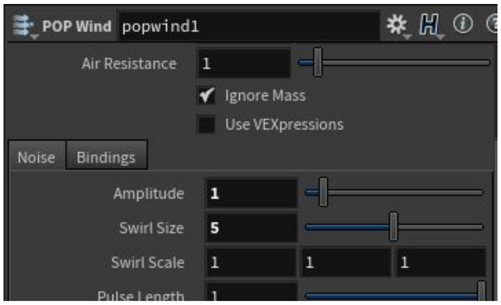
Fluid Forces

Particle Forces

Remember that FLIP fluids are a combo of Houdini's old volume-based fluids and particle-based fluids. As such, you can apply any of the POP forces to FLIP fluids. These forces get plugged into the second input of a fluid solver (the Particle Velocity input)…



> **HINT**: A common technique to create turbulent water is to connect a popwind and change up the noise amplitude and swirl…



POP Wind popwind1

Air Resistance 1

☑ Ignore Mass

□ Use VEXpressions

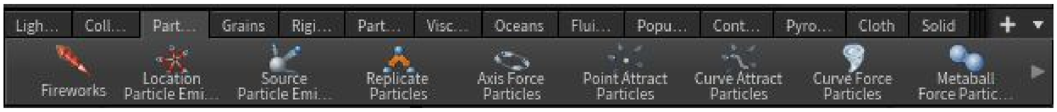
Amplitude 1

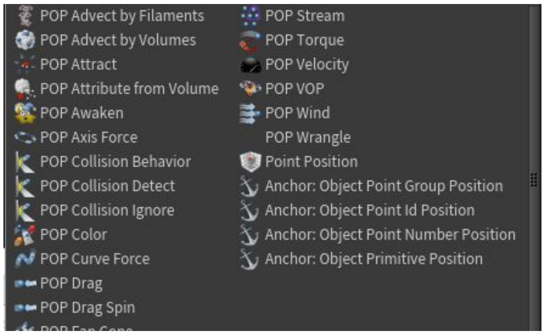
Swirl Size 5

Swirl Scale 1 1 1

Pulse Length 1

The following is the a copy of the forces section of the particles document. It gives an overview of the particle forces available. You can find these all under the Particles shelf or via the Tab menu search (search for “POP”)...





There a ton of different forces you can apply to particles. It seems like these forces ONLY effect particles and nothing else.

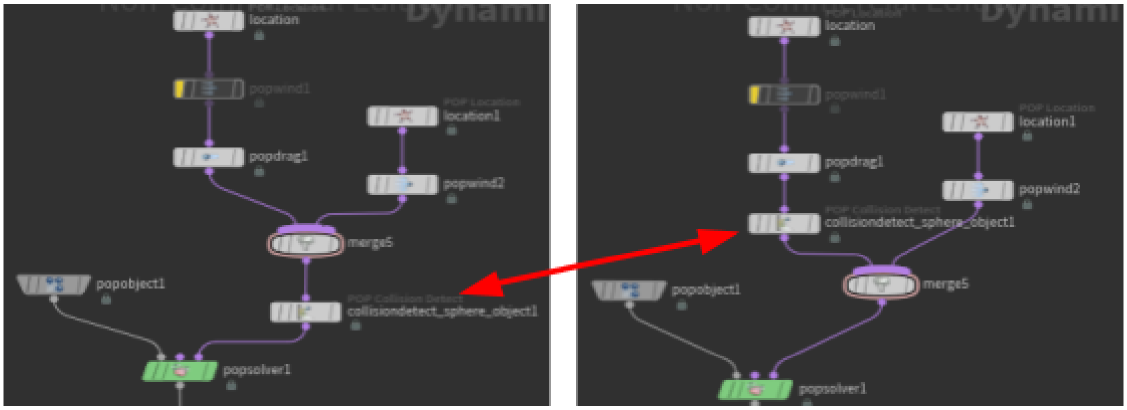
> **NOTE**: The gravity node is applied to particles as well. If you don’t want it applied, you need to bypass it. Keep that in mind when applying these forces.

For example, rigid body objects are not affected by these objects. The particles may change direction as a result of the force and hit the rigid body differently (causing the rigid body to move differently), but the force itself won’t move the rigid body.

Here are the types of forces you can apply…

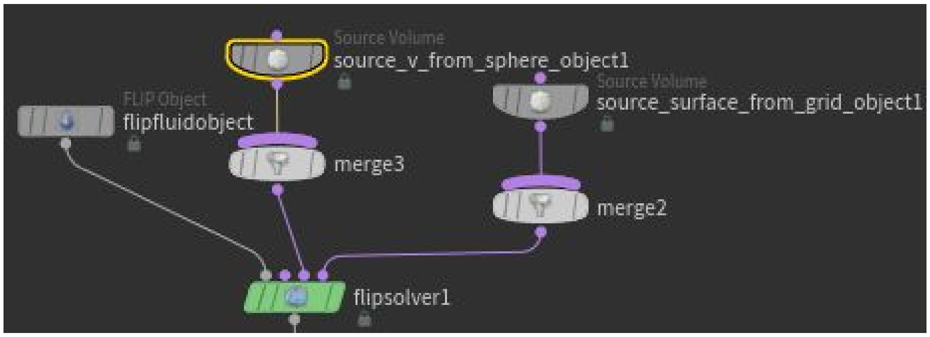
- Axis Force → for spinny things like hurricanes/turbines/dust devils/etc..
- Wind/Drag → applies a constant force to particles regardless of location
- Fan → Like Wind/Drag but applied to a certain area
- Force → something akin to gravity? How’s this different than Wind/Drag?
- Flock → flocking behaviour for particles
- Curve Force → creates forces generated from a curve????
- Point/Curve Attract → attracts particles to point or curve

NOTE: See the section on Collision w/ Non-Dynamics… Just like how the placement of the POP Collision Detect node is important, the placement of your force node is also important. If it's after the final merge before the pop solver, it means all particles being fed into the popsolver will follow the force rules added by this node. If you only want these rules only applied to particles emitted from certain emitters, move it so that it's in the path of those emitters (before the final merge).



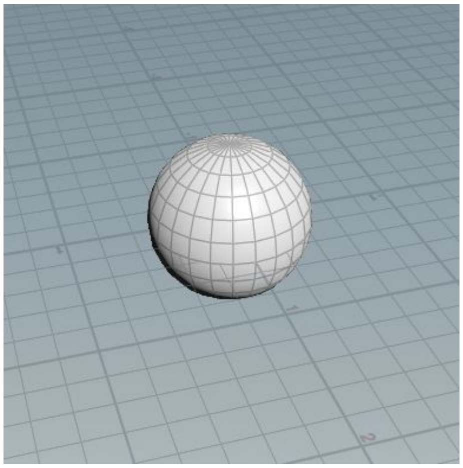
Volume Forces

Remember that FLIP fluids are a combo of Houdini's old volume-based fluids and particle-based fluids. As such, you can apply any of the volume forces to FLIP fluids. These forces get plugged into the third input of a fluid solver (the Volume Velocity input)…

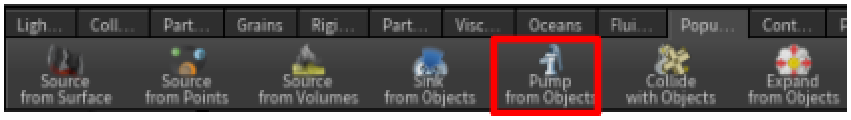


The easiest way to apply a volume force to an object is to use a Pump. You can do this by…

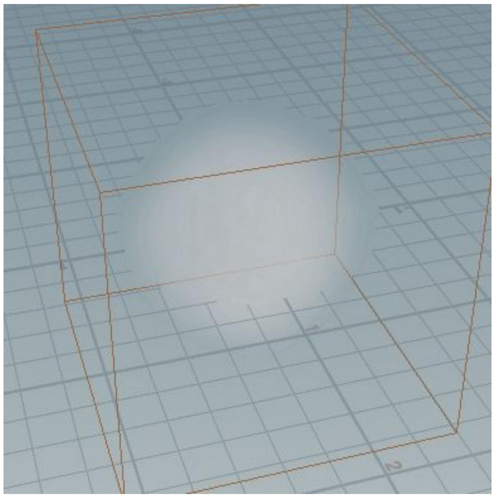
1. create an object in your scene (e.g. sphere)

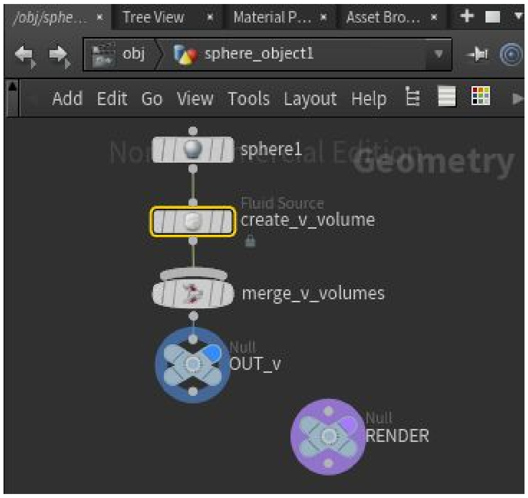


2. make sure nothing is selected
3. select the Pump from Object item in the Populate Containers tab

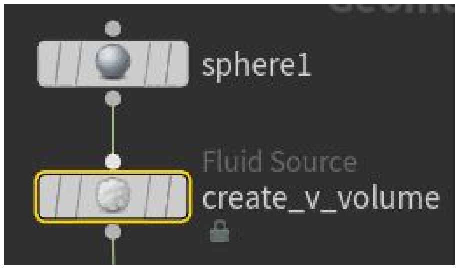


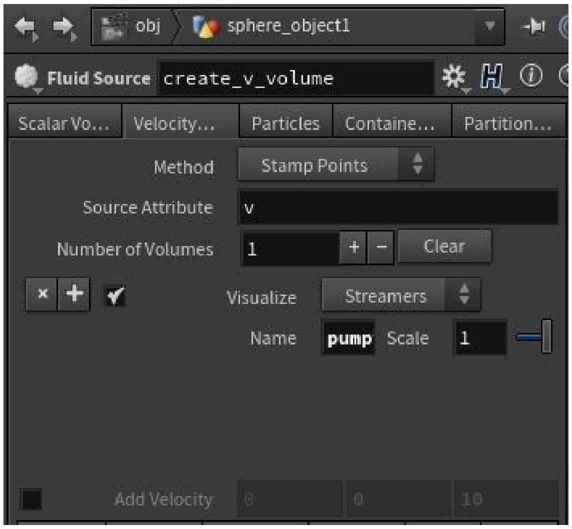
4. select your object and press Enter
   (you'll get a prompt to do so in the scene view)
5. select the FLIP fluid object being fed into your flipsolver in the AutoDopNetwork and press Enter
   (you'll get a prompt to do so in the scene view -- the prompt says gas instead of fluid, this is probably just a bug)
6. your object should show up as mist now -- dive into it
   (this won't show up in the final render + you can hide it in your scene view if you want -- it will still function)





7. select the Fluid Source node and go to the Velocity Volume tab





Fluid Source create_v_volume

Method: Stamp Points

Source Attribute: v

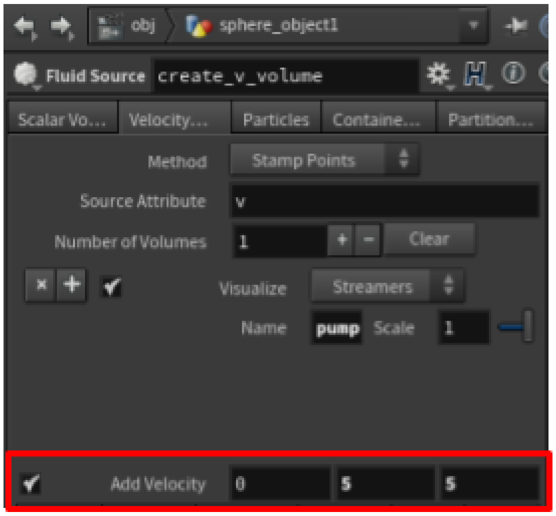
Number of Volumes: 1

Visualize: Streamers

Name: pump Scale: 1

□ Add Velocity 0 0 10

8. enable Add Velocity and punch in a force vector
(red guides will show up in the scene view once you do this)



Fluid Source create_v_volume

Method: Stamp Points

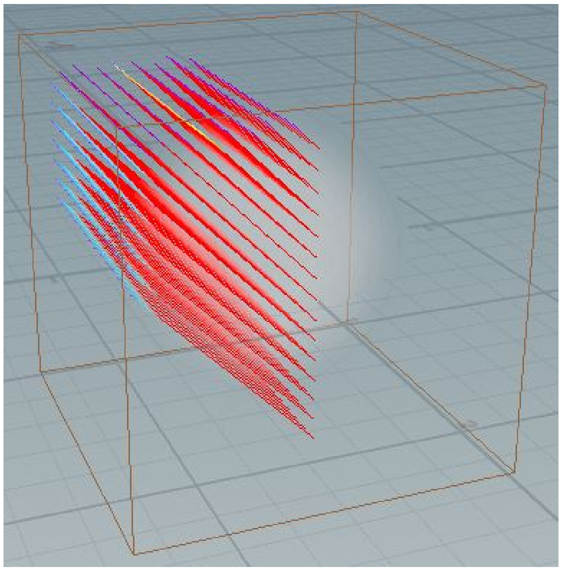
Source Attribute: v

Number of Volumes: 1

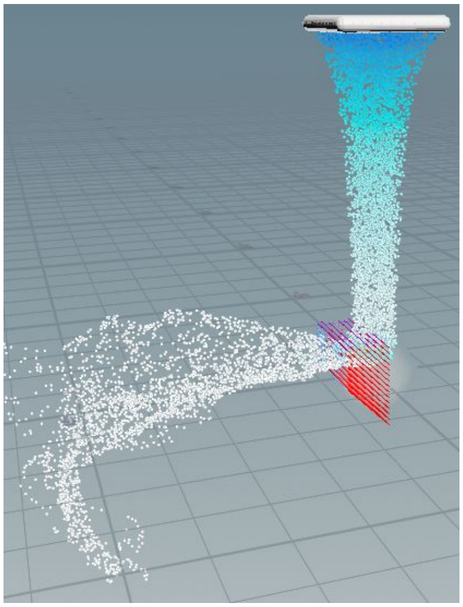
Visualize: Streamers

Name: pump Scale: 1

☑ Add Velocity 0 5 5



9. fluids coming into this area will now have this force applied

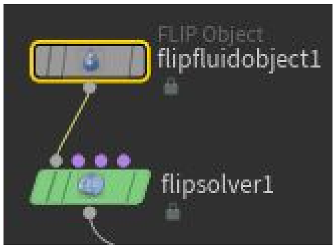


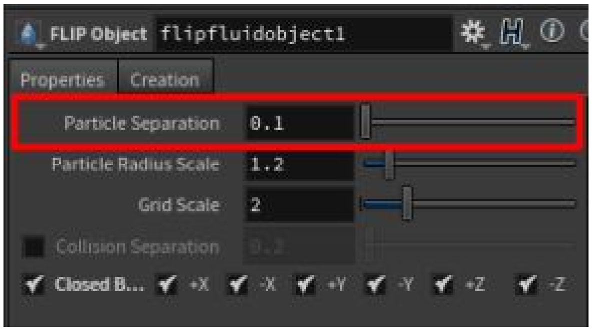
Fluid Accuracy

Particles

To control the accuracy of your FLIP fluid simulation, you can change the size of the particles that get generated. The more particles you have in your fluid simulation, the better it will look. It sounds like the more movement/collisions you have, the more particles you want.

To set the particle size, you need to go to whatever FLIP object it is that's being fed into the solver. Under the Properties tab, you'll see an option called Particle Separation. The smaller this is, the more particles you'll have.





flipfluidobject1

Particle Separation 0.1

Particle Radius Scale 1.2

Grid Scale 2

Collision Separation 0.2

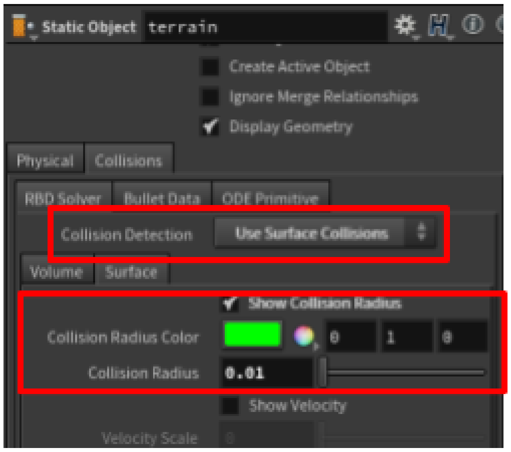
✓ Closed B... ✓ +X ✓ -X ✓ +Y ✓ -Y ✓ +Z ✓ -Z

Obviously… the downside with doing this is that the more particles you have the longer it'll take to compute.

Collision Geometry

You'll notice a lot of times that your fluids will seep through objects instead of properly colliding with them. The reason for this is very likely because the objects your colliding with have bad collision geometry.

If you're dealing with rigid bodies or collision objects (e.g. ground object / terrain object / static object), you can use surface collisions instead of volume collisions to get better results. You can read the document on collisions for a detailed overview on this…


Static Object terrain

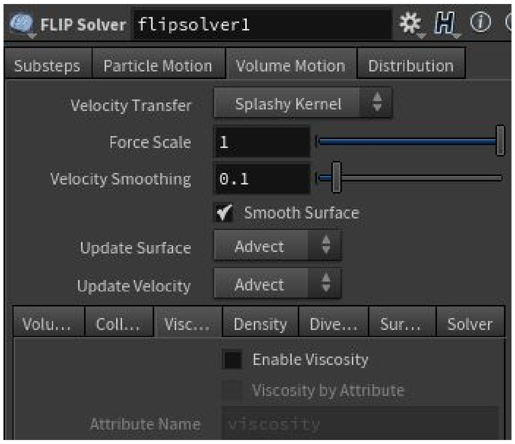
□ Create Active Object
□ Ignore Merge Relationships
☑ Display Geometry

Collision Detection: Use Surface Collisions
☑ Show Collision Radius
Collision Radius Color: 🟩 0 1 0
Collision Radius: 0.01
□ Show Velocity
Velocity Scale: 0

> **NOTE**: One other option to use (if you want) is to try to make your geometry thicker, that way the volume-based collision may have an easier time generating correct collision geo.

Fluid Viscosity

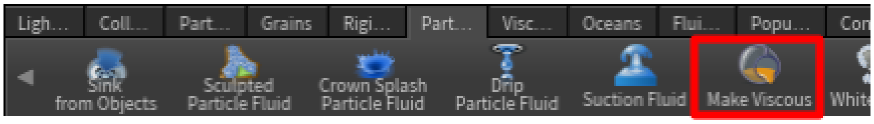
Enabling fluid viscosity manually isn't a simple process. The flipsolver has a Enable Viscosity checkbox under Volume Motion -> Viscosity, but enabling just that does nothing.


FLIP Solver flipsolver1

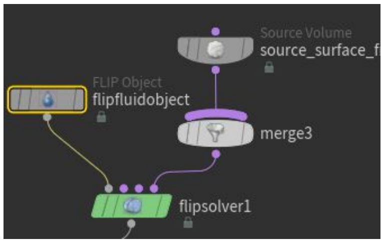
Velocity Transfer: Splashy Kernel
Force Scale: 1
Velocity Smoothing: 0.1
☑ Smooth Surface
Update Surface: Advect
Update Velocity: Advect

□ Enable Viscosity
□ Viscosity by Attribute
Attribute Name: viscosity

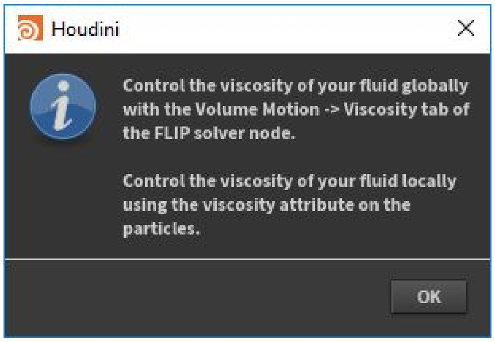
It turns out that there are a bunch of different places you have to enable viscosity to get it to actually function. There's a handy item under Particle Fluids shelf that does this for you called Make Viscous…



Make sure you have nothing selected and click it. In the scene view you'll get prompted to select the FLIP Object(s) to make viscous and press Enter. Go into your AutoDopNetwork, select them, and hit Enter.



You'll get the following message box show up...


Houdini

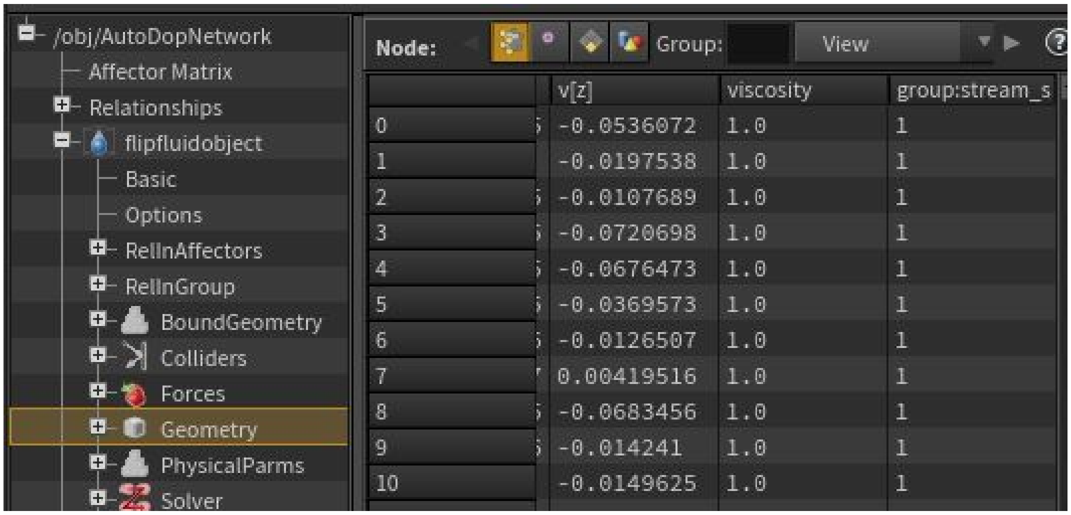
Control the viscosity of your fluid globally with the Volume Motion -> Viscosity tab of the FLIP solver node.

Control the viscosity of your fluid locally using the viscosity attribute on the particles.

OK

Essentially all this is saying is that there's 2 ways to set viscosity…

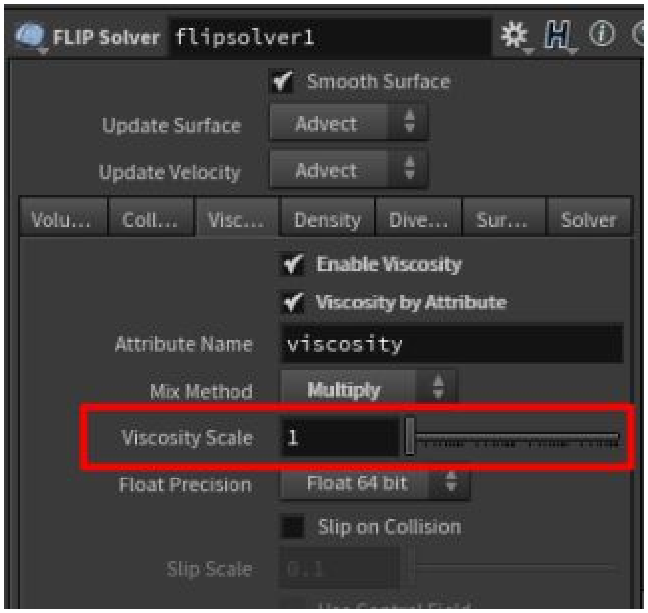
If you want to have different particles be different viscosities, you need to add to the viscosity attribute to your particles. If you look at the geometry spreadsheet, you'll see that the particles generated by your FLIP Object now each have a viscosity attribute of 1.0…



	v[z]	viscosity	group:stream_s
0	-0.0536072	1.0	1
1	-0.0197538	1.0	1
2	-0.0107689	1.0	1
3	-0.0720698	1.0	1
4	-0.0676473	1.0	1
5	-0.0369573	1.0	1
6	-0.0126507	1.0	1
7	0.00419516	1.0	1
8	-0.0683456	1.0	1
9	-0.014241	1.0	1
10	-0.0149625	1.0	1

NOTE: There is no node in the dynamics context to add to an attribute. You'll have to do this via VEX, which I don't know how to do yet.

If you want all your fluids to have the same viscosity, you can use the Volume Motion -> Viscosity Tab of your flip solver…



FLIP Solver flipsolver1

☑ Smooth Surface

Update Surface: Advect

Update Velocity: Advect

☑ Enable Viscosity

☑ Viscosity by Attribute

Attribute Name: viscosity

Mix Method: Multiply

Viscosity Scale: 1

Float Precision: Float 64 bit

☐ Slip on Collision

Slip Scale: 0.1

The more you jack up the Viscosity Scale value, you more viscose your fluid will be. For example, something like 10000 would represent honey. You can get it high enough to represent something like bread dough.

NOTE: Remember from above that your fluid particles will have a viscosity of 1.0. Note the Mix Method -- what's happening is that your viscosity attribute is being multiplied by the Viscosity Scale value you set. If they're always defaulting to 1.0, you don't have to worry. If they're something else, you may want to use a different Mix Method.

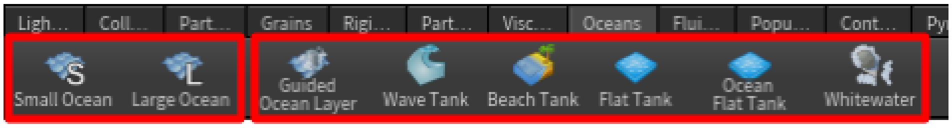
Houdini 16 Oceans

Introduction

Houdini’s Ocean tools (formerly called Ocean FX) are a set of tools to help artists with oceans. These tools are broken down into 2 main categories:

1. Simulating the surface of an ocean
2. Fluid tanks (very similar to a FLIP Tank)

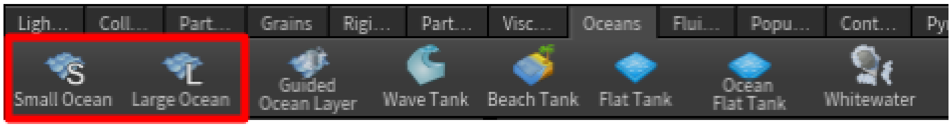
You can find them under the Oceans shelf.



NOTE: Not 100% sure if these tools are actually used by most artists. This part of Houdini could be similar to Maya's muscle system (in that no one uses it for anything).

Ocean Surface

You can simulate the surface of an ocean using the first 2 options in the Oceans shelf.

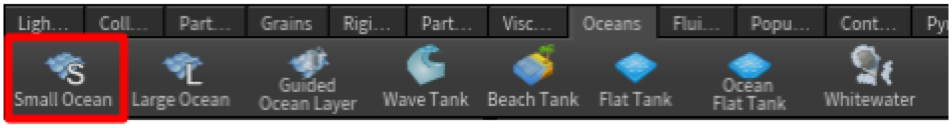


This part of the document will focus on the "Small Ocean" item, but the "Large Ocean" item is probably very similar.

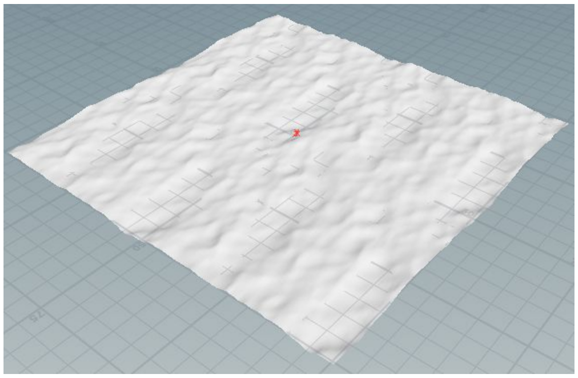
The surfaces generated here ARE NOT CONTROLLED BY A DOPs NETWORK! They are entirely simulated on their own, and they can't really interact with any other fluids or rigidbodies or anything else that requires a DOP setup.

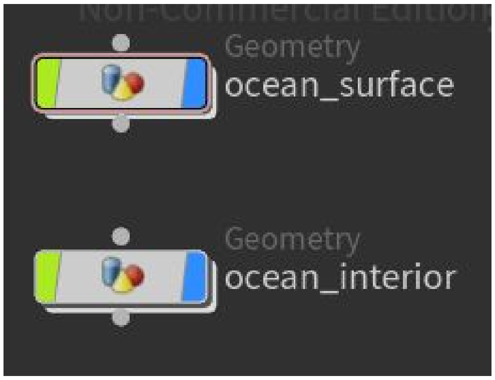
Create

You can create a surface by using the Small Ocean item in the Oceans shelf.



Once you do, you'll end up getting what looks like a patch of terrain and 2 new nodes in your /obj context…





- ocean_surface is the surface of your ocean

- ocean_interior is the ‘inside’ of the ocean (essentially a volume extrude of the surface that’s hidden from the scene view but visible in renders)

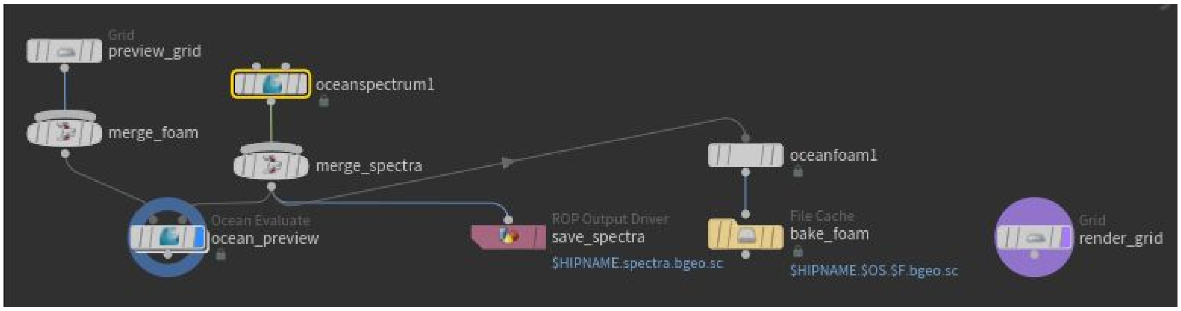
  NOTE: Notice how there’s no AutoDopNetwork here. This is independent of DOPs and can’t interact with other physics as if it were a FLIP fluid.

If you play your animation timeline now, you’ll see the ocean surface animating as if it were a real ocean’s surface.

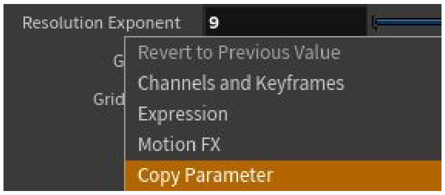
Setup Resolution

Once you’ve created your ocean, you need to do one extra thing: you need to make sure that the grid that makes up the surface has enough quads for our ocean surface's “resolution”...

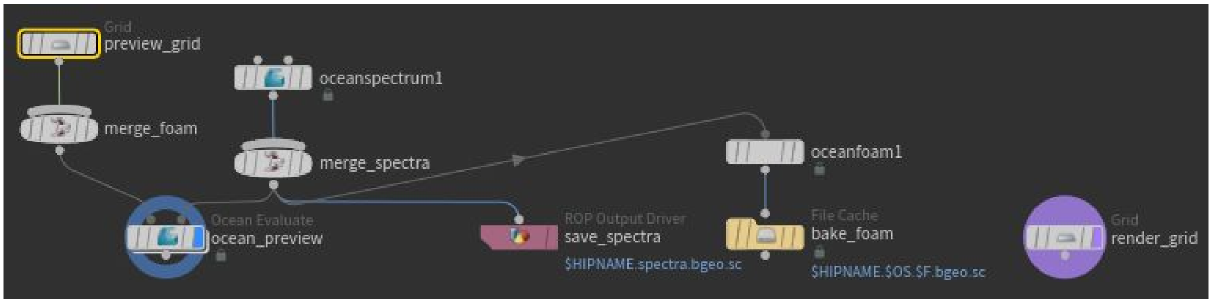
1. Go into the ocean_surface node...
2. Go to the oceanspectrum node (highlighted in image below)...



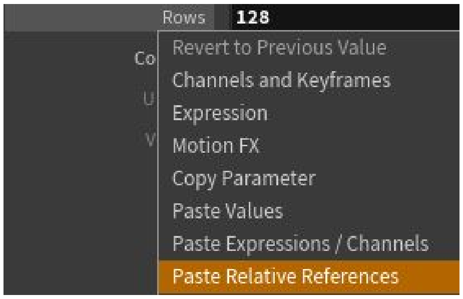
3. The first property you’ll see is Resolution Exponent. Right-click on the label and choose Copy Parameter...


Resolution Exponent 9

4. Go to the preview_grid node (highlighted in image below)...

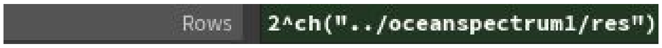


5. Right-click on the label for Rows property and choose Paste Relative References...



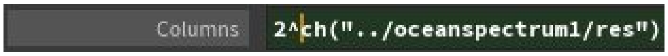
Rows 128

6. The Rows property should now read ch("../oceanspectrum1/res"), change it to 2^ch("../oceanspectrum1/res")...



Rows `2^ch("../oceanspectrum1/res")`

7. Repeat steps 5 and 6 for the Columns property



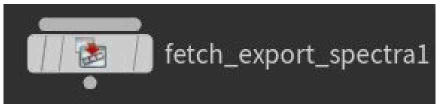
Columns `2^ch("../oceanspectrum1/res")`

NOTE: It's unsure where the formula 2^exponentres came from, but that's what the lesson recommended. With this, we can now make sure that if we jack up the surface simulation's resolution, we'll jack up the number of faces in our grid that the surface gets mapped to (so we end up seeing the extra detail).

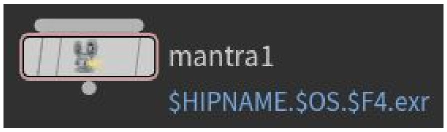
Setup Render

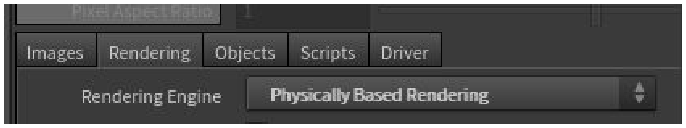
Once you've created your ocean, you need to do one extra thing to get it to render properly: you need to feed the displacement map generated for the ocean's surface into your render node so that you get a proper render.

1. Go to the /out context -- a node should have been added by the Ocean tool called fetch_export_spectra (it controls the displacement map for the ocean surface)…



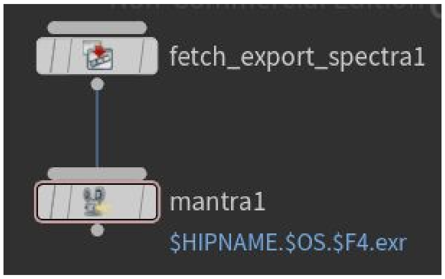
2. Create your mantra node if you don't have one already (make sure the renderer is set to Physically Based Rendering)...



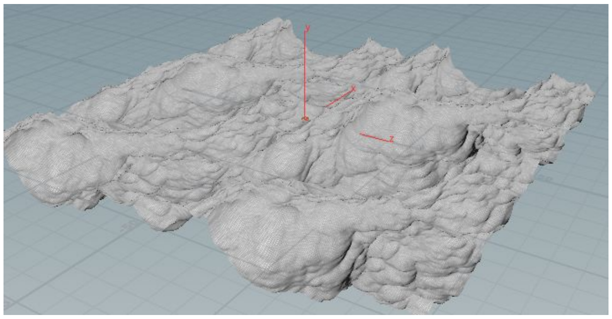


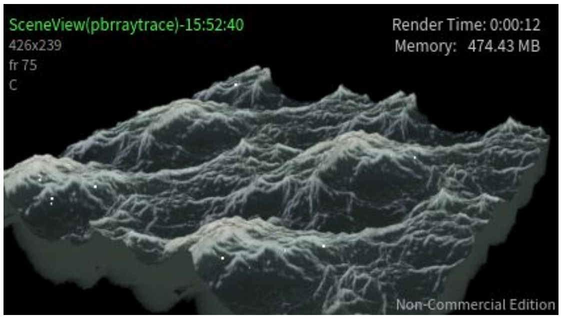
Rendering Engine: Physically Based Rendering

3. Hook the created node into your mantra node…



Once you do this, you'll get proper renders...

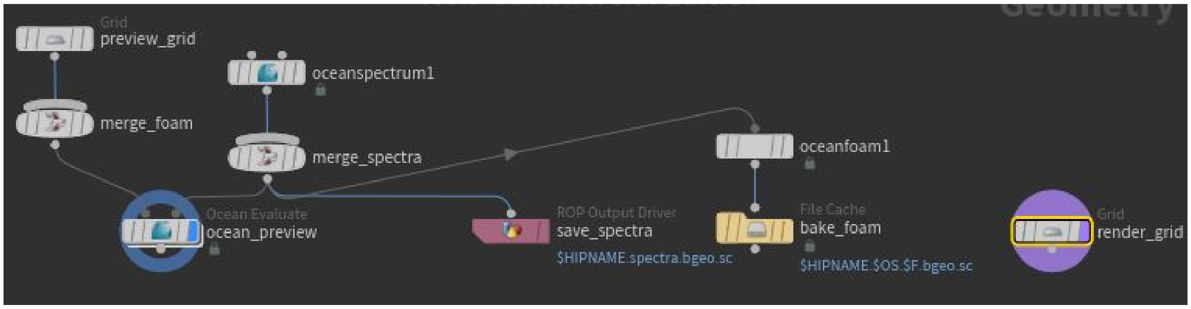




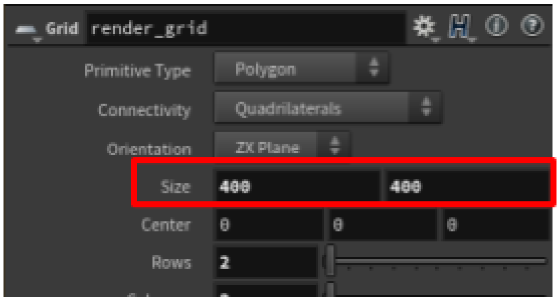
NOTE: DO NOT USE THE RAY TRACE RENDERER. It seems to have problems and ends up dipping into the volume element that's underneath the surface.

Setup Size

You can make the grid that makes up your ocean as large as you want. You can do so by going into your surface node (in the /obj context) and navigating to the render_grid node…



Change the size of this grid to however large you want your surface to be…



Grid render_grid

Primitive Type: Polygon

Connectivity: Quadrilaterals

Orientation: ZX Plane

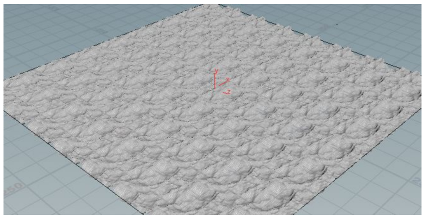
Size: 400 400

Center: 0 0 0

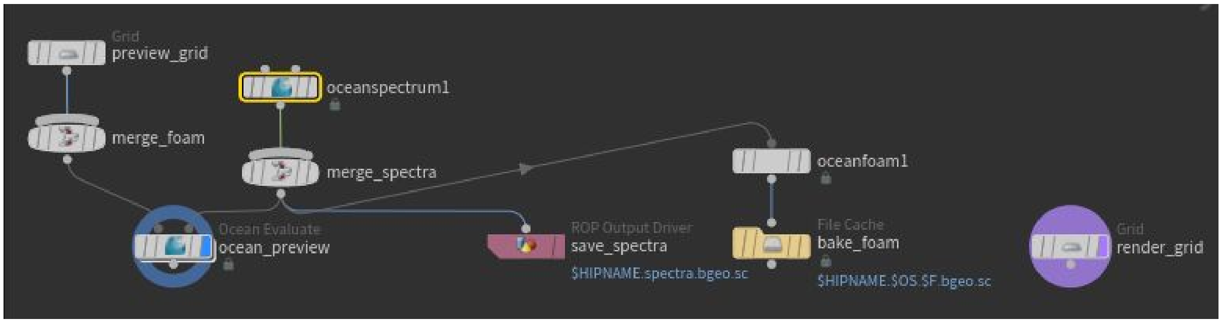
Rows: 2

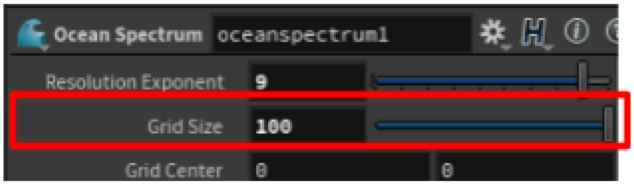
NOTE: Don't worry about this node that being connected to anything. Other nodes in this network have a channel reference to the size here, so everything will just magically work once you set the new size.

One thing you may notice after you do this… the waves are being tiled together (there's an obvious pattern)...



You can increase the size of the tile being generated (to make it less obvious) by going to the oceanspectrum node and increasing the Grid property…





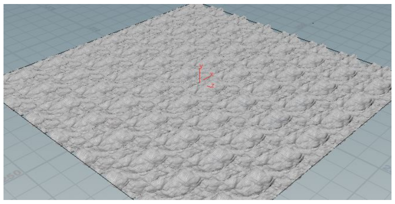
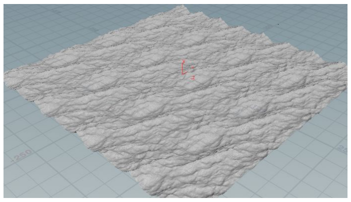
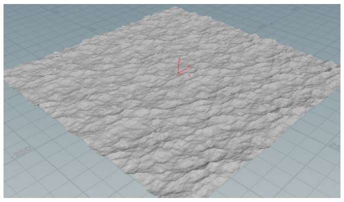
Ocean Spectrum oceanspectrum1

Resolution Exponent: 9

Grid Size: 100

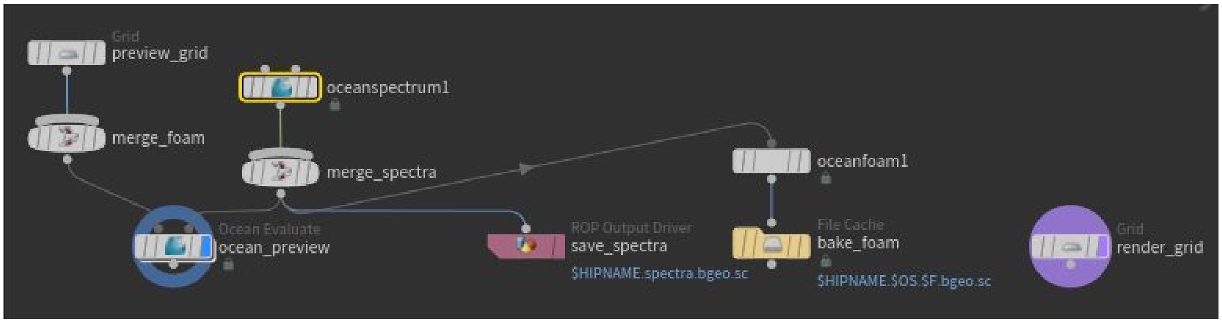
Grid Center: 0 0

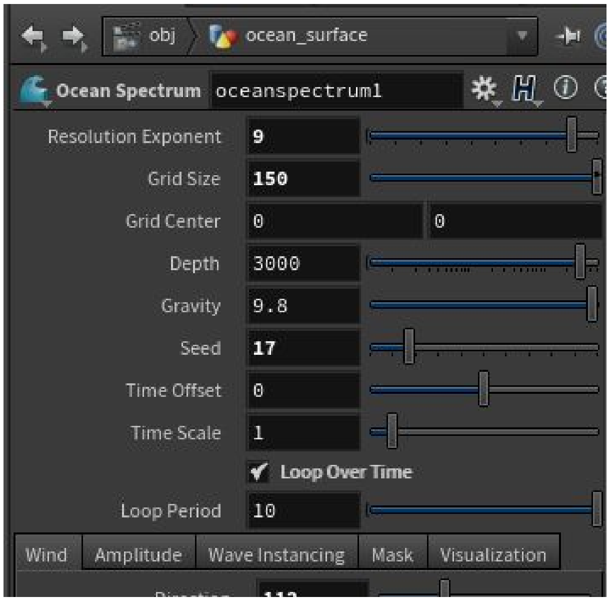
Here's an example with a Grid Size property of 50/100/150 (this is for a 400x400 grid plane).

 →  → 

Control

You can gain control over the surface of your water by going inside the ocean_surface node and to the oceanspectrum1 node. In here, you can find lots of options that reference real-world properties…



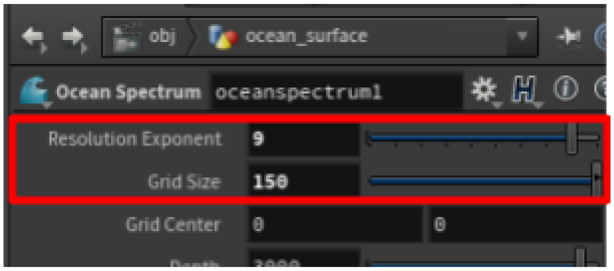


Ocean Spectrum oceanspectrum1

Resolution Exponent	9
Grid Size	150
Grid Center	0 0
Depth	3000
Gravity	9.8
Seed	17
Time Offset	0
Time Scale	1
	☑ Loop Over Time
Loop Period	10
Direction	112

Resolution

As discussed in the Create section, you can jack up the resolution of your surface via the Resolution Exponent and Grid Size properties…



Ocean Spectrum oceanspectrum1

Resolution Exponent 9

Grid Size 150

Grid Center 0 0

Depth 3000

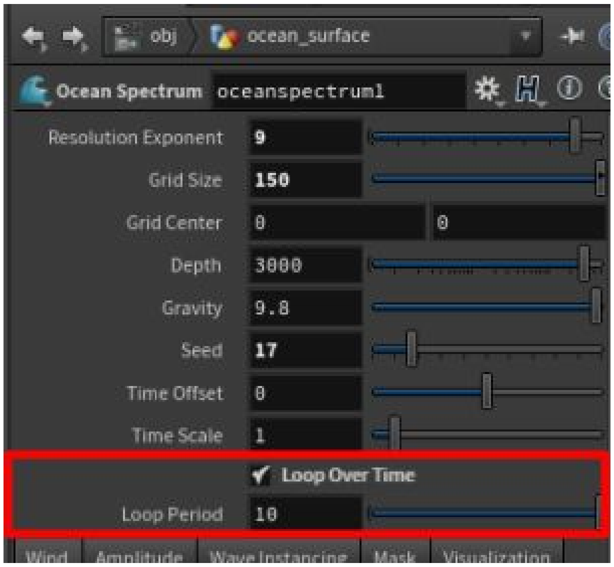
NOTE: All of this is predicated on PROPER SETUP. See the Create section and make sure you did all the setup properly.

Resolution Exponent defines how detailed your surface is. The higher this number is, you can have smaller and smaller waves propagate over your surface, resulting in much finer/realistic detail.

Grid Size defines the tile size for the surface of your ocean. So if you set up your ocean to be super large, Grid Size will control how much repetitiveness there is on the surface of your ocean. See the Setup Size section for more information.

Perfect Loops

You can make it so that the animation on the surface of your ocean loops perfectly via the Loop Over Time and Loop Period properties...



Ocean Spectrum oceanspectrum1

Resolution Exponent 9

Grid Size 150

Grid Center 0 0

Depth 3000

Gravity 9.8

Seed 17

Time Offset 0

Time Scale 1

✓ Loop Over Time

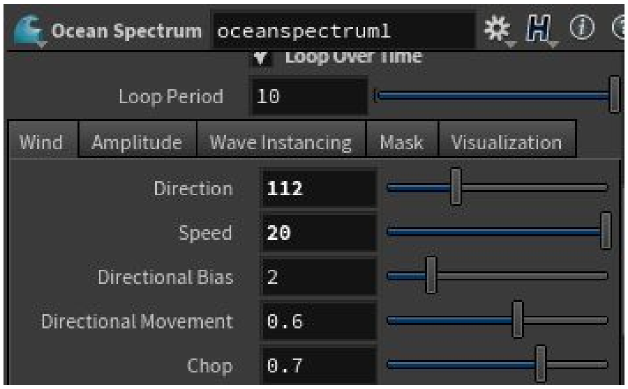
Loop Period 10

If you check Loop Over Time, you can fill in the Loop Period value. Loop Period is the number of seconds before the animation should loop.

So for example, if you have 240 frames in your animation and you set your FPS to 24 (these are the defaults), you want to set this to 10.

Wind

All wind related properties are under the Wind tab...



oceanspectrum1

Loop Over Time

Loop Period 10

Direction 112

Speed 20

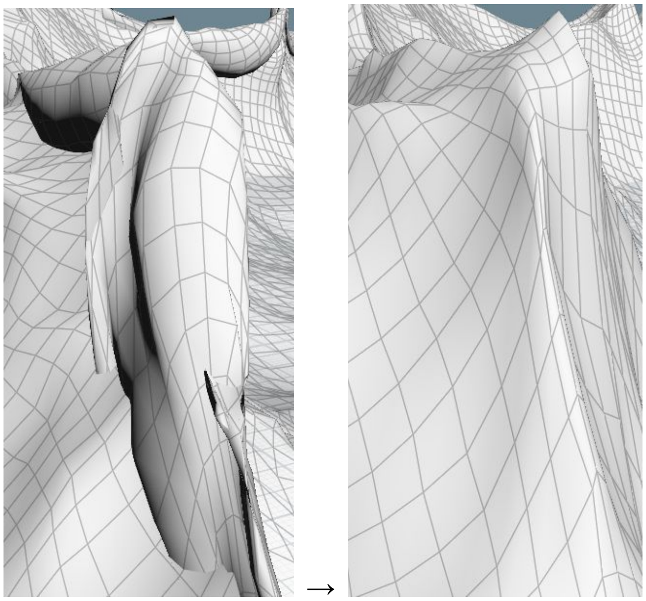
Directional Bias 2

Directional Movement 0.6

Chop 0.7

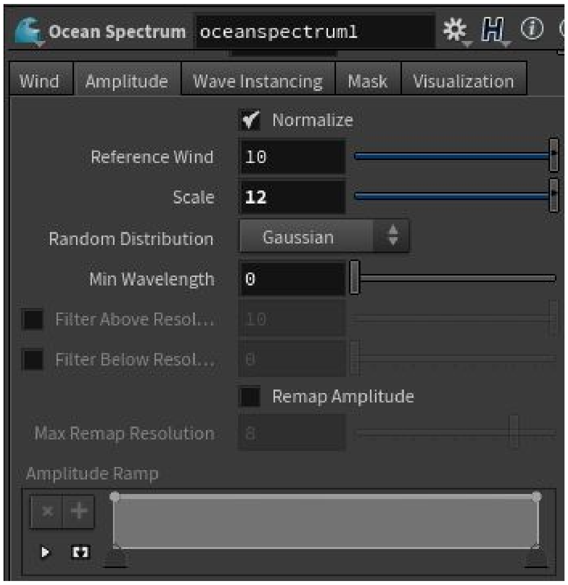
Most of the properties here should be self-explanatory…

- Direction and Speed refer to the wind being applied.
- Directional Bias controls the amount of waves that “align” with the wind direction... how many of the waves being generated for the surface follow the direction property -- if this value is zero, your ocean won’t have a direction? each wave goes in its own direction?
- Directional Movement dampens waves that are going against the wind -- 1.0 means all waves are going in the same direction, 0.0 means waves are going equally both in the direction of the wind and against the direction of the wind.
- Cusp controls how rounded the peaks of your wave are -- in certain cases you’ll need to round your peaks because your peaks will be so sharp and high that they’ll protrude through themselves… here’s an example of the same wave using a high cusp value vs a low cusp value



Waves

All wave related properties are under the Amplitude tab...



oceanspectrum1

☑ Normalize

Reference Wind 10

Scale 12

Random Distribution Gaussian

Min Wavelength 0

☐ Filter Above Resol... 10

☐ Filter Below Resol... 8

☐ Remap Amplitude

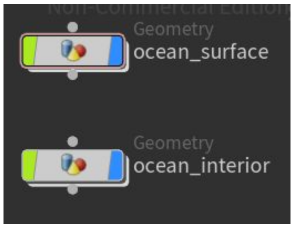
Max Remap Resolution 8

The big property here is Scale. The higher the scale is, the larger your waves will be. Don't bother touching anything else.

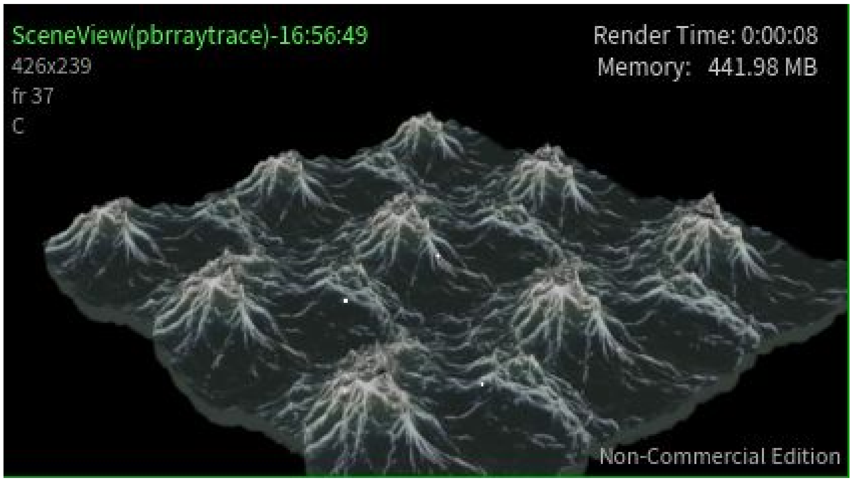
> **NOTE**: The Min Wavelength option takes out waves that are smaller than a certain size. It'll cause you to smooth out your water the higher it gets, almost as if you're dropping the Resolution Exponent value. Unsure why this would ever be used? Maybe to get a cartoonish look to water?

Shaders

Remember that 2 geometry nodes got created for your ocean surface…



ocean_surface is the top of your ocean, while ocean_interior is a murky volume that protrudes slightly underneath it. You can see both if you do a render…

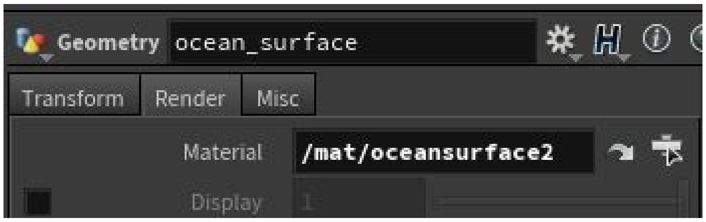


SceneView(pbrraytrace)-16:56:49
426x239
fr 37
C

Render Time: 0:00:08
Memory: 441.98 MB

Each geometry node is controlled by a shader. You can use these shaders to customize how your water looks.

To see which shaders are being used, go to the geometry node's properties and check the Material property under the Render tab…

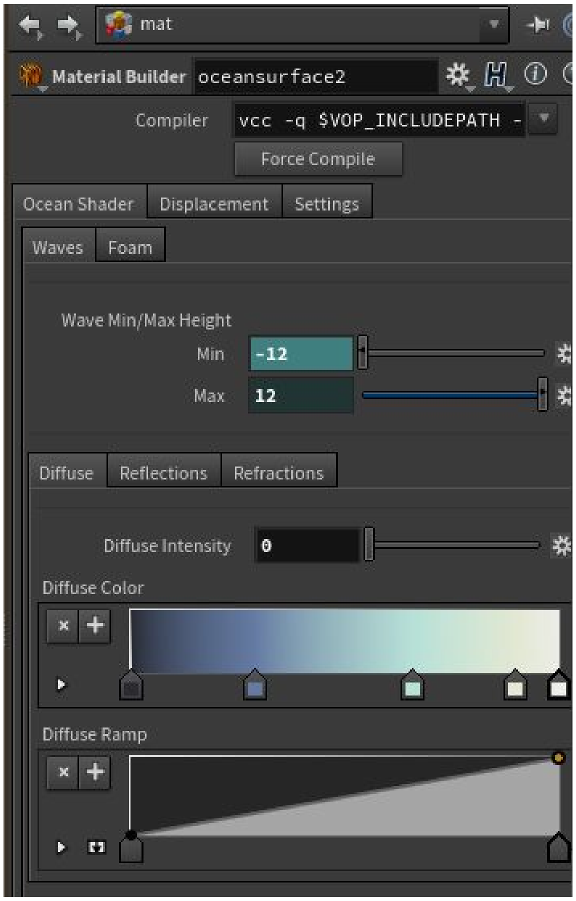


Geometry ocean_surface

Material /mat/oceansurface2

Display 1

HINT: Clicking the little arrow on the right will take you to the material node.


mat

Material Builder oceansurface2

Compiler: vcc -q $VOP_INCLUDEPATH -

Wave Min/Max Height

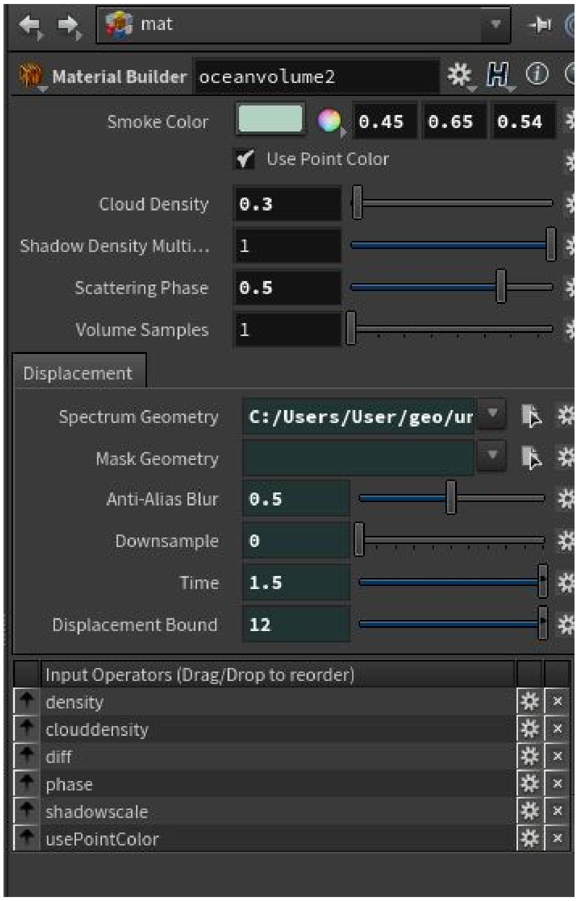
Min: -12

Max: 12

Diffuse Intensity: 0

Diffuse Color

Diffuse Ramp


mat

Material Builder oceanvolume2

Smoke Color: 0.45 0.65 0.54

☑ Use Point Color

Cloud Density: 0.3

Shadow Density Multi...: 1

Scattering Phase: 0.5

Volume Samples: 1

Displacement

Spectrum Geometry: C:/Users/User/geo/ur

Mask Geometry:

Anti-Alias Blur: 0.5

Downsample: 0

Time: 1.5

Displacement Bound: 12

Input Operators (Drag/Drop to reorder)

- density
- clouddensity
- diff
- phase
- shadowscale
- usePointColor

The following is the material properties for the surface and interior.

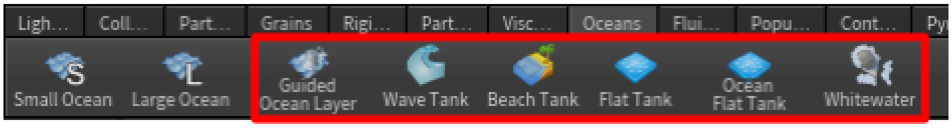
For oceansurface (the material for the surface geometry node), only stuff under Ocean Shader -> Waves seems to do something. Foam doesn’t seem to do anything anymore -- it seems to be totally ignored.

For oceanvolume (the material for the interior geometry node), the Cloud Density property controls how murky the interior volume is.

Fluid Tanks

Fluid tanks are essentially FLIP Tanks (see the document on fluids) with some special stuff added to them such that they follow along with some animated object and create/sink particles as they move with that object.

Every shelf item highlighted below (except for Whitewater) is a variation of the Flat Tank option, so that’s the one we’ll be focusing on.

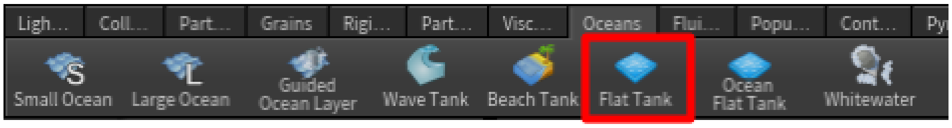


NOTE: Whitewater is used for generating realistic particles for foam/bubbles/etc.. as your object moves in the Flat tank.

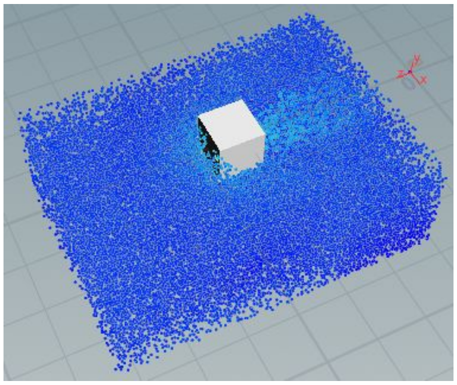
Create

First, make sure you have an animated object in your scene.

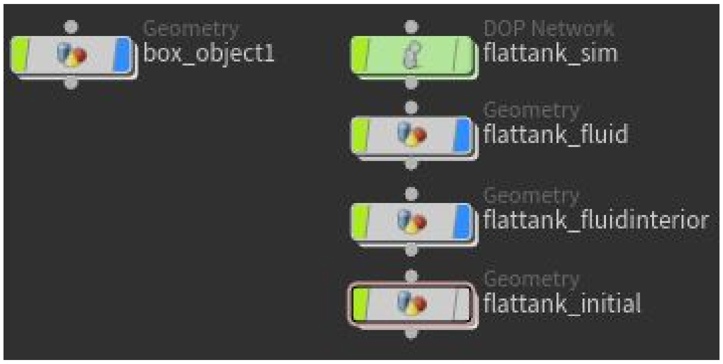
Select the object and choose the Flat Tank shelf item in the Oceans shelf…



A tank of water particles should surround your object now. If you play your animation, the tank will follow along with your object and the water particles will react as if it is actually moving…



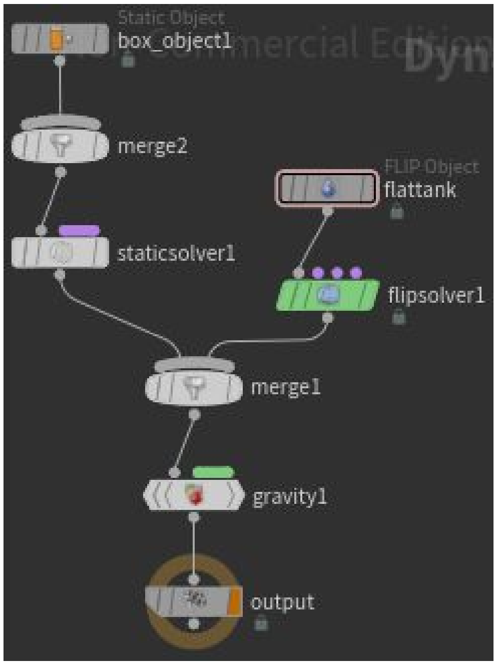
4 new nodes should be added to your /obj context…



- box_object1 is the original object the flat tank was added to.

- flattank_sim is the DOP network that handles the fluid physics.

  This exactly like an AutoDOPNetwork, except that it's focused entirely on the dynamics for the flattank and this object. Infact, if you go into this DOP network you'll see that all it does is add the object as a static object and feeds the flattank into a flipsolver as if you created a FLIP Tank. If you really wanted to you can just move these items into your other DOP network (if you had one)...

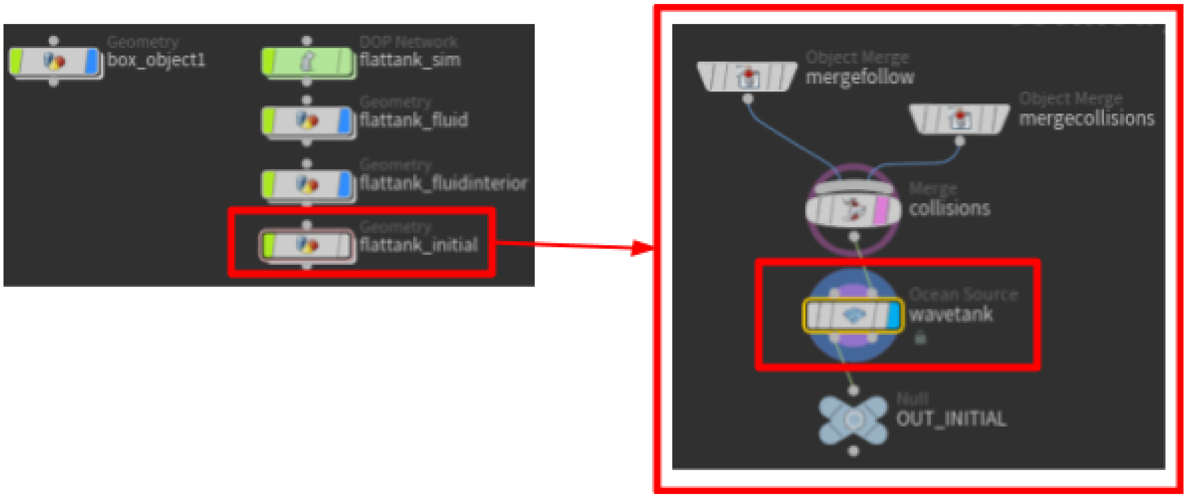


- flattank_fluid shows the particles in your scene view (exactly the same thing that happens when you create FLIP fluids from the Fluid Particles shelf)

- flattank_fluidinterior is for rendering the water in your final render (exactly the same thing that happens when you create FLIP fluids from the Fluid Particles shelf)

- flattank_initial contains the details of your tank

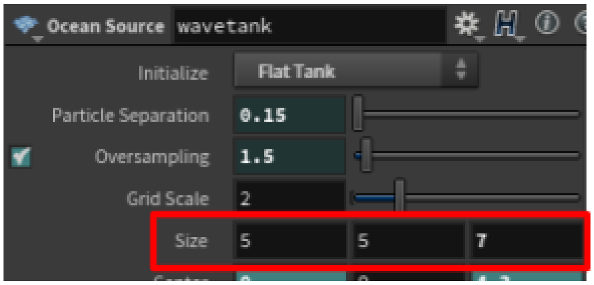
To set the details on your tank, step into the flattank_initial node and then to the wavetank node.



Setup Tank Size

First, step into the flattank_initial node and then to the wavetank node (see end of parent section for more details).

You can control your tank size by changing the Size property…



Ocean Source wavetank

Initialize: Flat Tank

Particle Separation: 0.15

☑ Oversampling: 1.5

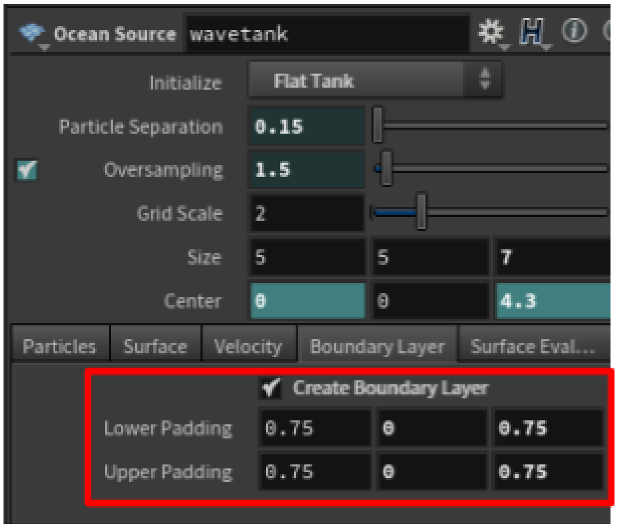
Grid Scale: 2

Size: 5 5 7

Setup Source/Sink Water Rate

First, step into the flattank_initial node and then to the wavetank node (see end of parent section for more details).

You can control how many water particles your tank creates/eats as it moves by setting the boundary around your tank. This is under the Boundary Layer tab...



Ocean Source wavetank

Initialize: Flat Tank

Particle Separation: 0.15

☑ Oversampling: 1.5

Grid Scale: 2

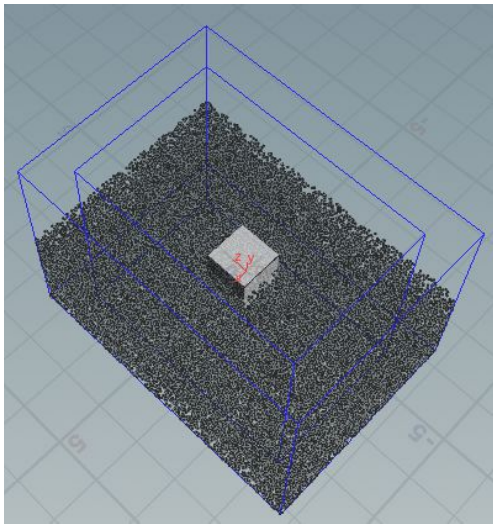
Size: 5 5 7

Center: 0 0 4.3

✓ Create Boundary Layer

Lower Padding: 0.75 0 0.75

Upper Padding: 0.75 0 0.75

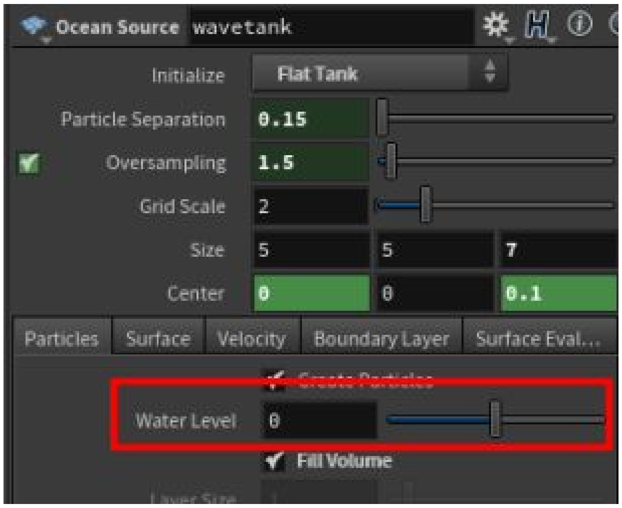


NOTE: The inner box is the water used for the simulation, the outer box is the water that's generated. You may need to increase the size of the outer box depending on how fast you're moving because you'll need to generate more water particles.

Setup Water Level

First, step into the flattank_initial node and then to the wavetank node (see end of parent section for more details).

You can control how many water level in your tank by setting the Water Level property under the Particles tab..



Ocean Source wavetank

Initialize: Flat Tank

Particle Separation: 0.15

☑ Oversampling: 1.5

Grid Scale: 2

Size: 5 5 7

Center: 0 0 0.1

✓ Create Particles

Water Level: 0

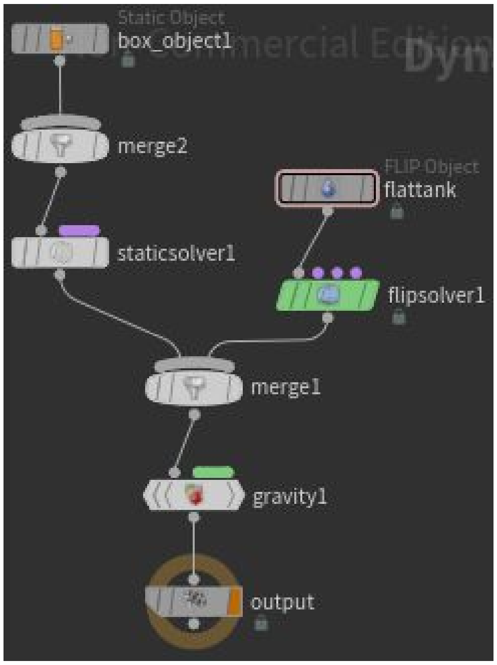
✓ Fill Volume

Layer Size

NOTE: 0 means exactly the middle

Setup FLIP Fluid Details

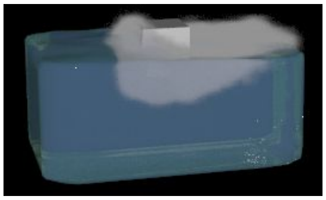
You can set the details on the FLIP fluid by going into the flattank_sim DOP network and fiddling with the FLIP object and flipsolver. This was covered extensively in the Fluids document.



Whitewater

Whitewater controls the dynamics/physics of a flat tank's foaming, spray, and bubbles as your object moves through it.

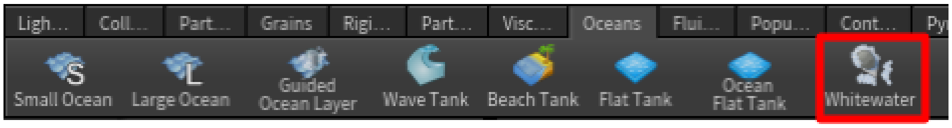
> **NOTE**: This only controls the movement of the particles. The actual shaders on those particles are myst sprites -- they look nothing like what they're suppose to.



Create

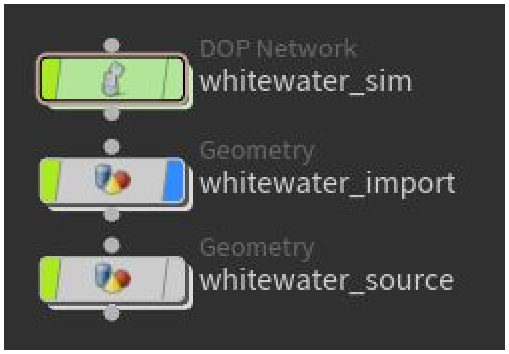
First, make sure you have an nothing select in your scene.

Select the object and choose the Whitewater shelf item in the Oceans shelf…



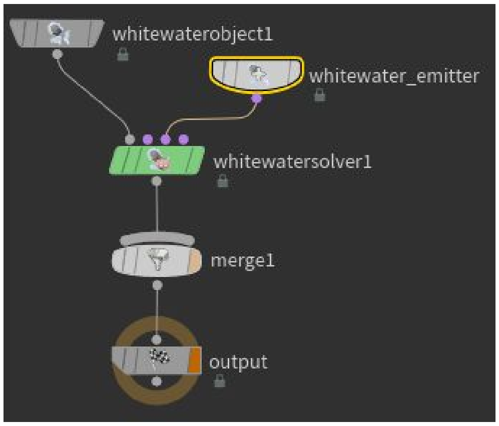
Select your flat tank in your scene and press Enter (or go into the flattank_sim node in /obj and select the FLIP Object for the flat tank that's being fed into the solver, then go back to your scene and press Enter).

3 new nodes should be added to your /obj context…



- whitewater_sim is the DOP network that handles the fluid physics.

  This exactly like an AutoDOPNetwork, except that it's focused entirely on the dynamics for the whitewater. If you really wanted to you can just move these items into your flat tank's DOP network (or maybe a larger DOP network if you had one). The only thing to be aware of is that THERE IS NO GRAVITY NODE HERE.

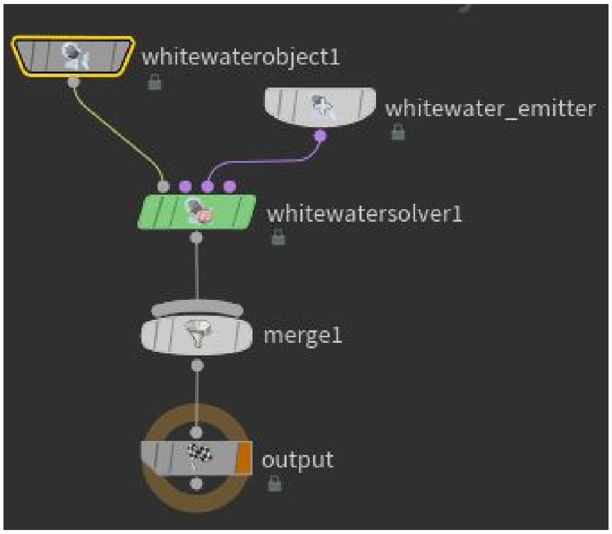
  

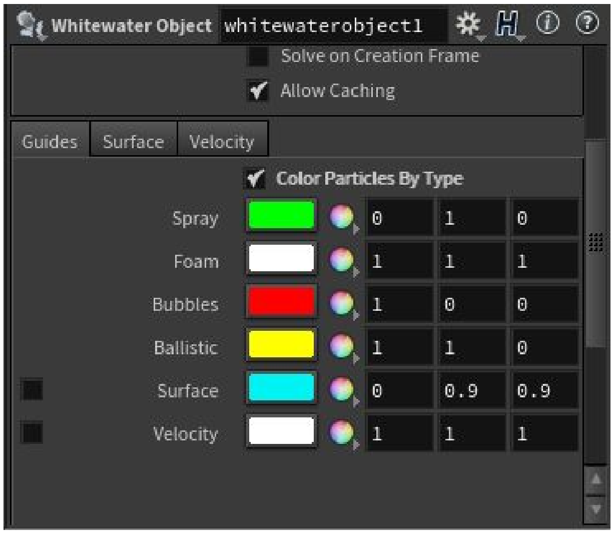
- whitewater_import shows the particles in your scene view (copied over from the whitewater_sim DOP network).

- whitewater_source is (I think) a copy of your flattank's details that eventually get fed into the whitewater_sim network.

Particle Guides

If you want to colorize each particle by type (e.g. spray should be green, bubbles should be red, etc..), you can go to the whitewaterobject node inside the whitewater_sim DOP network. There'll be a Guides tab with an option called Color Particles by Type…




Whitewater Object whitewaterobject1

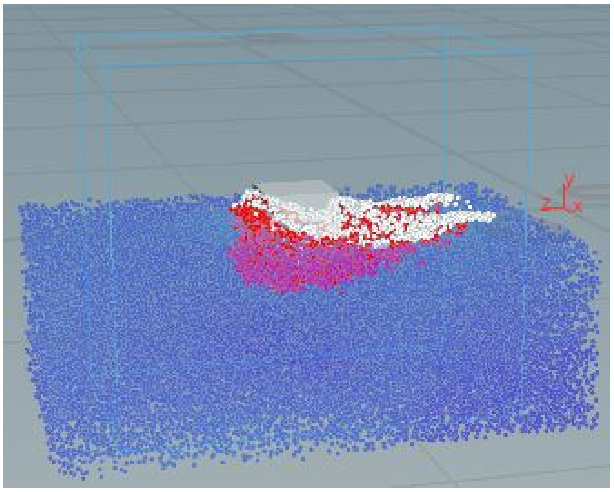
□ Solve on Creation Frame

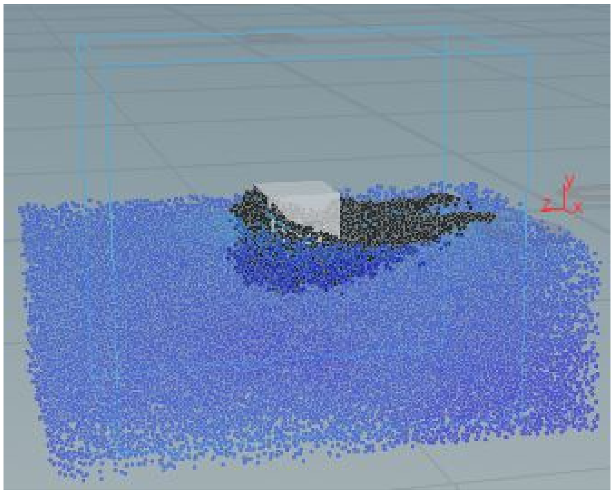
☑ Allow Caching

☑ Color Particles By Type

	Spray	🟩	0	1	0
	Foam	⬜	1	1	1
	Bubbles	🟥	1	0	0
	Ballistic	🟨	1	1	0
□	Surface	🟦	0	0.9	0.9
□	Velocity	⬜	1	1	1

Here are colorized guides vs normal guides…

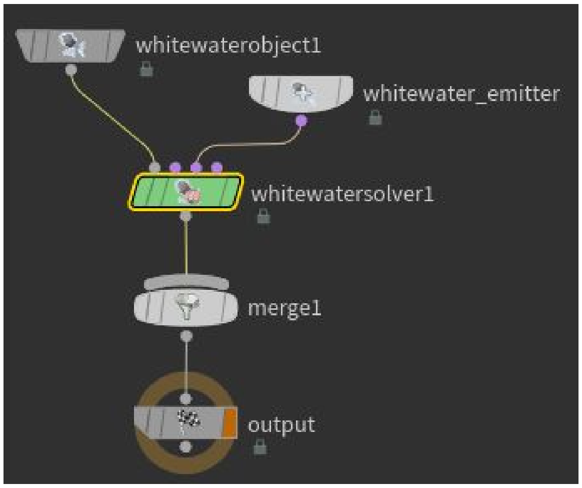


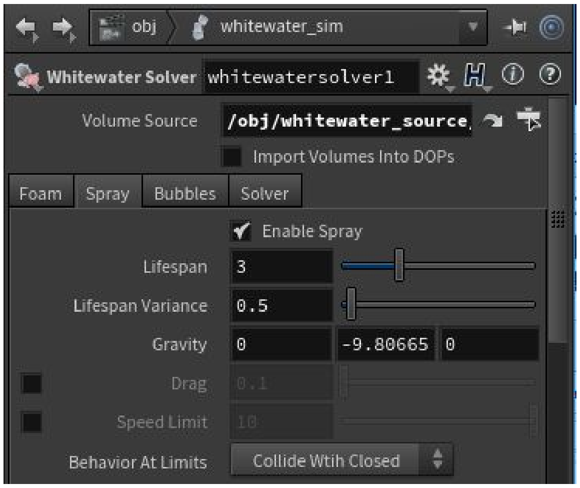


Essentially these colorized guides just give you a quick idea of how things will look in your final render.

Particle Properties

You can go to the whitewatersolver node inside the whitewater_sim DOP network and find 3 tabs that control the foam, spray, and bubble properties.





Whitewater Solver whitewatersolver1

Volume Source /obj/whitewater_source

□ Import Volumes Into DOPs

☑ Enable Spray

Lifespan 3

Lifespan Variance 0.5

Gravity 0 -9.80665 0

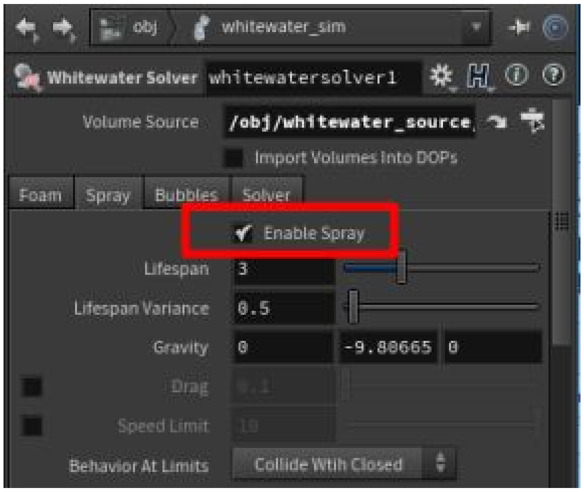
□ Drag 0.1

□ Speed Limit 10

Behavior At Limits Collide Wtih Closed

Enable/Disable

You can choose to enable or disable foam/spray/bubbles by going to the related tab and choosing the Enable (type) checkbox.



Whitewater Solver whitewatersolver1

Volume Source /obj/whitewater_source

□ Import Volumes Into DOPs

☑ Enable Spray

Lifespan 3

Lifespan Variance 0.5

Gravity 0 -9.80665 0

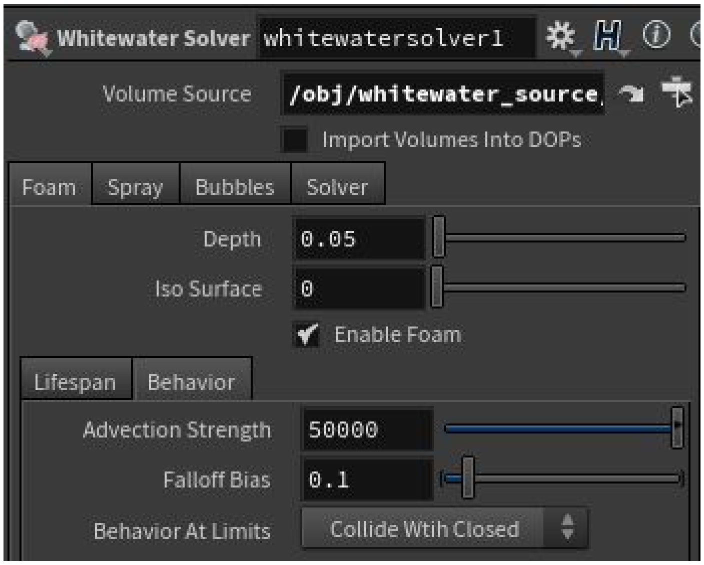
□ Drag 0.1

□ Speed Limit 10

Behavior At Limits Collide Wtih Closed

Foam

Foam particles sit on the surface of the fluid. Foam particles are primarily affected by the movement of the water and your object. You can control foam by going to the Foam tab…



Whitewater Solver whitewatersolver1

Volume Source /obj/whitewater_source

□ Import Volumes Into DOPs

Depth 0.05

Iso Surface 0

☑ Enable Foam

Advection Strength 50000

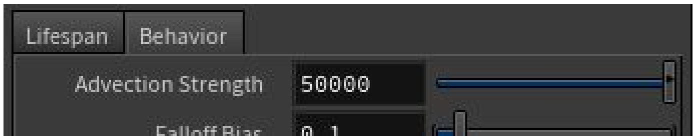
Falloff Bias 0.1

Behavior At Limits Collide Wtih Closed

Obviously, stuff under the Lifespan tab controls the lifespan of the particles and stuff under the behaviour tab controls the behaviour of the particles.

There are 2 important attributes here…

1. Advection Strength (under Behaviour)
   This controls the motion of foam particles. The higher the advection strength, the more that the foam bubbles will move with the fluid.

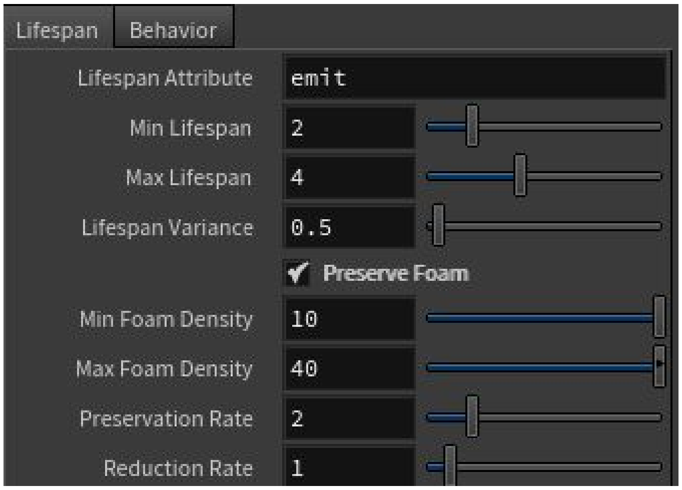


Advection Strength 50000

Falloff Bias 0.1

2. Preserve Foam (under Lifespan)
   This overrides the minimum lifespan you set in Min Lifespan if the foam is part of a large clump of foam. The size of the clump and how much time is added to the lifespan is

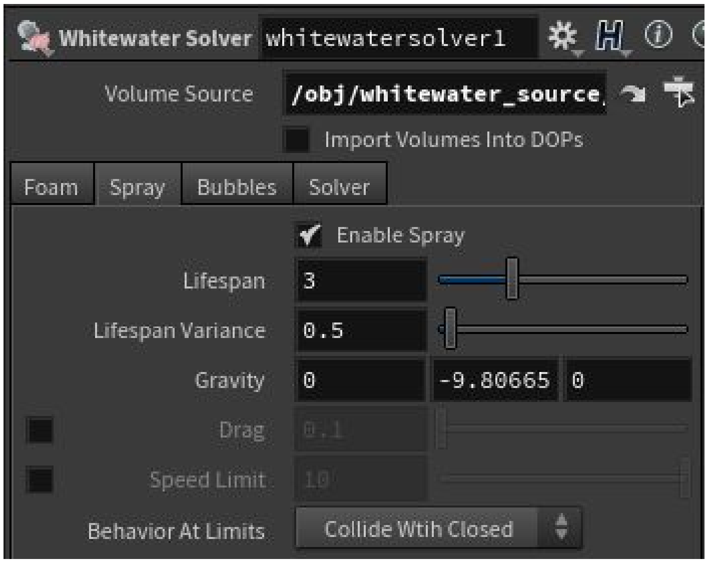
determined by the parameters below the Preserve Foam checkbox.



Lifespan Attribute: emit
Min Lifespan: 2
Max Lifespan: 4
Lifespan Variance: 0.5
☑ Preserve Foam
Min Foam Density: 10
Max Foam Density: 40
Preservation Rate: 2
Reduction Rate: 1

Spray

Spray particles live above the fluid. Spray particles are primarily affected by the force of gravity. You can control spray by going to the Spray tab…

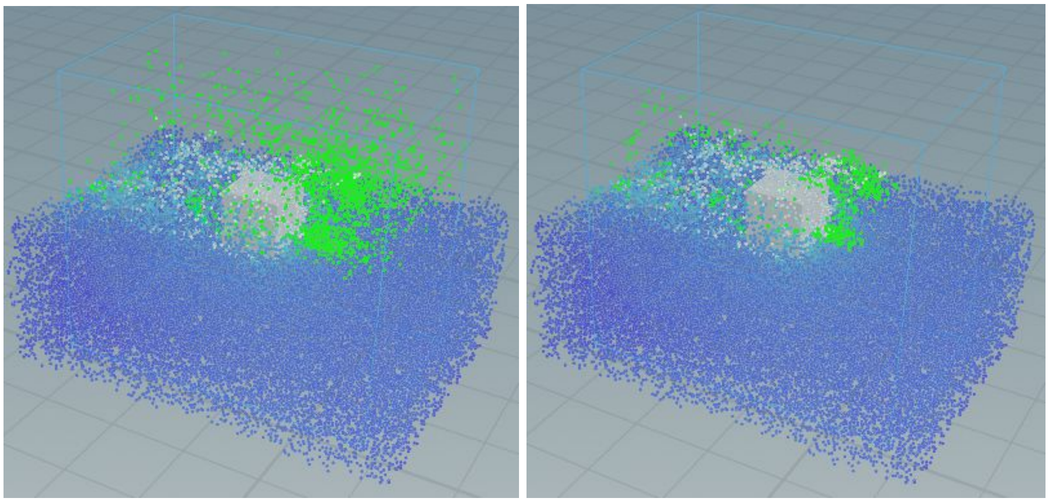


Whitewater Solver: whitewatersolver1
Volume Source: /obj/whitewater_source
□ Import Volumes Into DOPs
☑ Enable Spray
Lifespan: 3
Lifespan Variance: 0.5
Gravity: 0 -9.80665 0
□ Drag: 0.1
□ Speed Limit: 10
Behavior At Limits: Collide Wtih Closed

Obviously, stuff lifespan attributes controls the lifespan of the gravity is the force of gravity.

> **NOTE**: This solver is not feeding into a gravity node. It’s applying its own gravity locally. If you had this inside of a DOP network with gravity, you would likely need to make links between the gravity node and this.

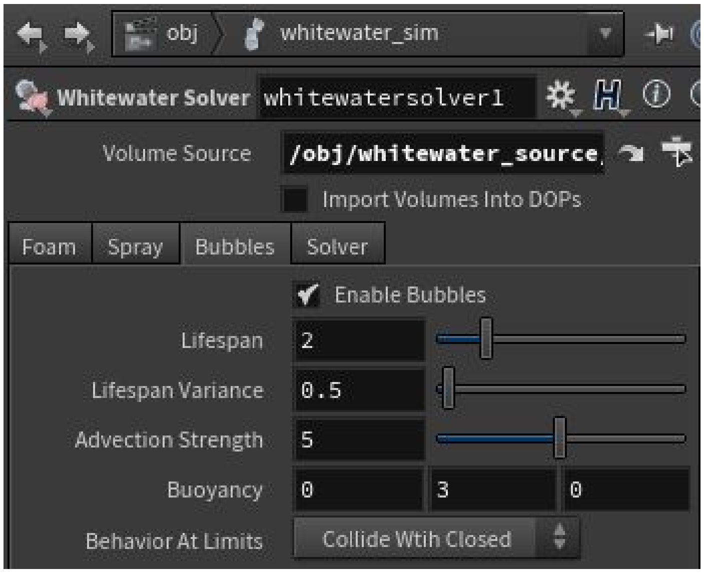
The important property here is Drag. The higher drag is set, the more narrowed those spray particles will be. Here’s an example of 0 drag vs a drag of 2 for a relatively fast moving object…



It’s almost as if the particles are being dragged back, which is I guess what you want. The spray particles are probably not taking into account how fast your object is moving since your object’s movement is relative to the flat tank. The drag simulates the particles staying in place as your object is moving forward (instead what’s happening is that the particles are being dragged back and your object is staying in place -- ultimately the end result is the same).

Bubbles

Bubble particles live below the fluid. Bubble particles are primarily affected by bouyance (this is a property you set on the solver). You can control bubbles by going to the Bubbles tab…



Whitewater Solver whitewatersolver1

Volume Source /obj/whitewater_source

□ Import Volumes Into DOPs

☑ Enable Bubbles

Lifespan 2

Lifespan Variance 0.5

Advection Strength 5

Buoyancy 0 3 0

Behavior At Limits Collide Wtih Closed

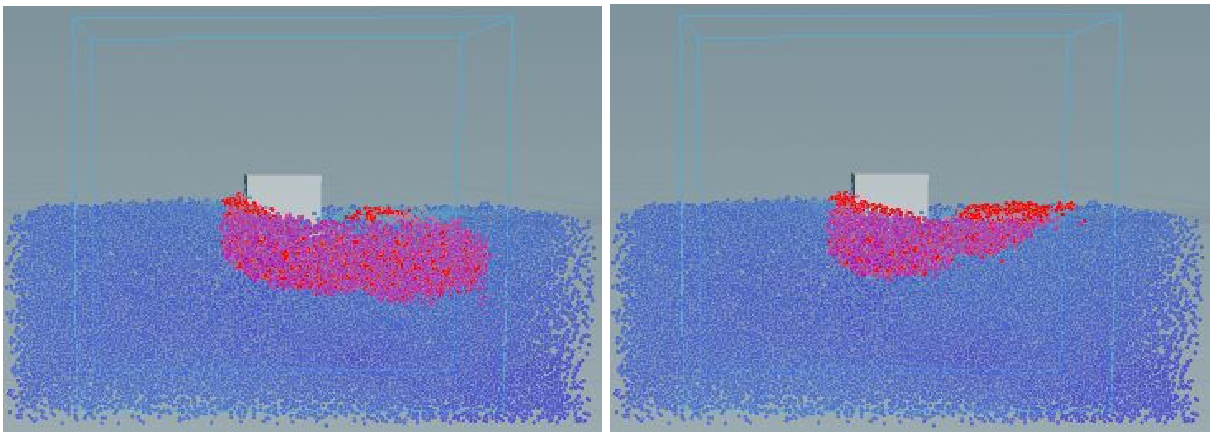
> **HINT**: Bubbles are only visible underneath the surface of the water. If you’re going to have a murky style water shader (e.g. swamp?), you can save yourself some simulation time by disabling bubbles completely.

Obviously, lifespan attributes controls the lifespan.

The important properties here are Advection Strength and Buoyancy.

1. Advection strength controls how the bubbles move with the water (just like foam) -- the higher the advection strength, the more that the bubbles will move with the fluid instead of scattering.
2. Buoyancy is how fast the bubbles come up -- the higher the buoyancy, the faster they come up?

Here’s an example of a buoyancy of 0,0,0 vs a buoyancy of 0,3,0…



NOTE: The actual amount of influence that buoyancy has diminishes significantly as the object moves faster and faster. The example above was a very slow moving object. For a fast moving object, I couldn’t tell the difference between 0,3,0 and 3000,3000,3000.

Houdini 16 Particles

Introduction

This physics portion of Houdini is called dynamics. There's a ton of different things you can do with dynamics, but this text is specifically about particles. You can think of particles as points/objects with no volume. When bound to sprites, they can be used to to simulate things like snow, rain, smoke, dust, etc…

Like all dynamics, particles have friction, bounce, and velocity attributes. Maybe mass/density attributes as well?

The shorthand for dynamics is DOPs (Dynamic OPerators). Particles seem to work under the dynamics portion of Houdini, but I've also seen them get called their own thing: POPs (Particle OPerators)?

Scene Scale and Dynamics

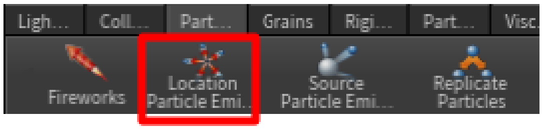
See the following section of the Houdini 16 Collisions document: Scene Scale, Dynamics Networks, and Dynamics Playback.

Not going to repeat those sections here -- all of them still apply.

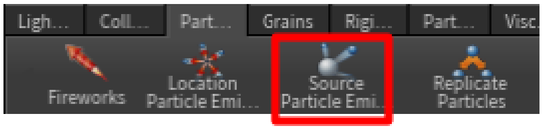
Particle Emitters

You can emit particles either from a point or from faces of a geometry.

To emit from a point, use Location Particle Emitter from the Particles shelf…



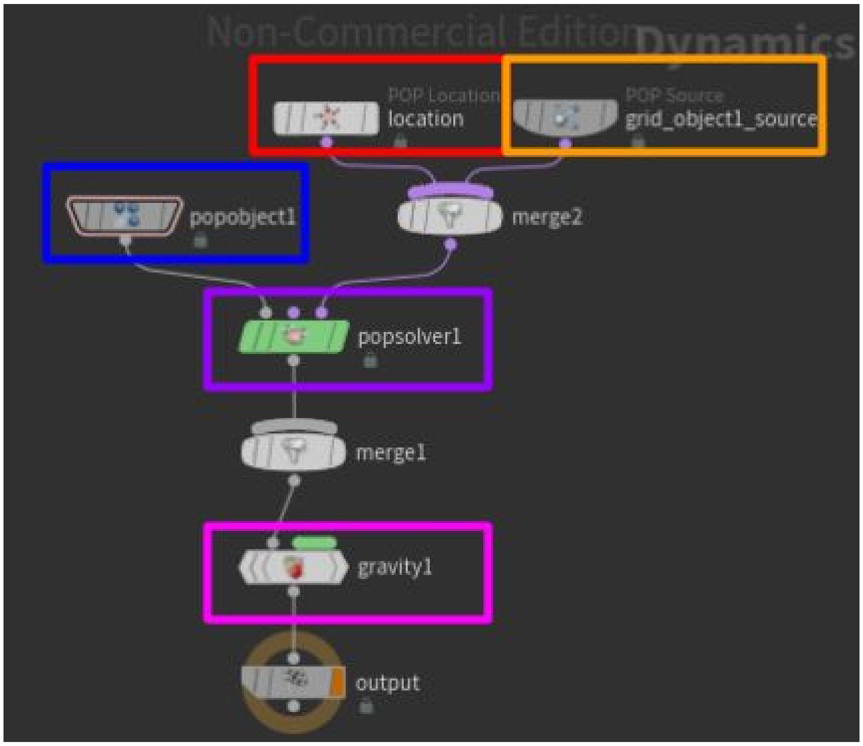
To emit from a set of polygons, select your geometry and use the Source Particle Emitter from the Particles shelf…



Regardless of which one you use, you'll get an AutoDopNetwork at /obj level.

> **NOTE**: If you don't remember, an AutoDopNetwork is a dynamics network that Houdini creates for you when you first use dynamics / adds to as you use more dynamics.

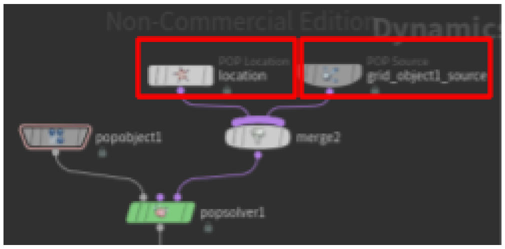
Inside the AutoDopNetwork, you'll see your particle emitters along with a popobject node being fed into a POP solver (Particle OPerator solver).



Here's what the DOP network setup means…

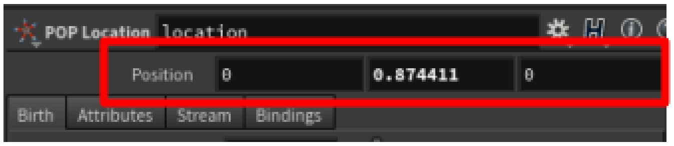
- POP Location controls particles coming out of a Location Particle Emitter
- POP Source controls particles coming out of a Source Particle Emitter
- POP Object makes the particles being generated interactable with other stuff in the dynamics -- e.g. collision with rigid bodies
- POP Solver performs the physics computations for the generated particles
- Gravity is what applies gravity (part of any AutoDopNetwork, not specific to particles)

The Location Particle Emitter and the Source Particle Emitter both have the same set of properties for how particles are generated. To access them, traverse into the AutoDopNetwork and select the node for whichever emitter it was you created (either POP Location or POP Source)...



Location

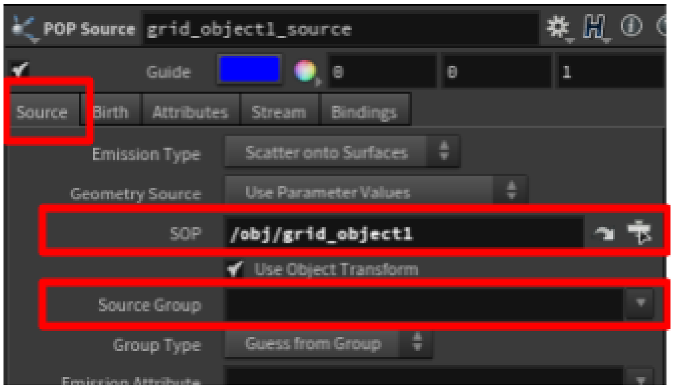
If you used a Location Particle Emitter (POP Location), you can control which point the particles are generated from via the Position property…



POP Location location

Position 0 0.874411 0

If you used a Source Particle Emitter (POP Source), you can control which faces the particles are generated from via the Source tab...



POP Source grid_object1_source

Guide 0 0 1

Emission Type: Scatter onto Surfaces

Geometry Source: Use Parameter Values

SOP: /obj/grid_object1

☑ Use Object Transform

Source Group:

Group Type: Guess from Group

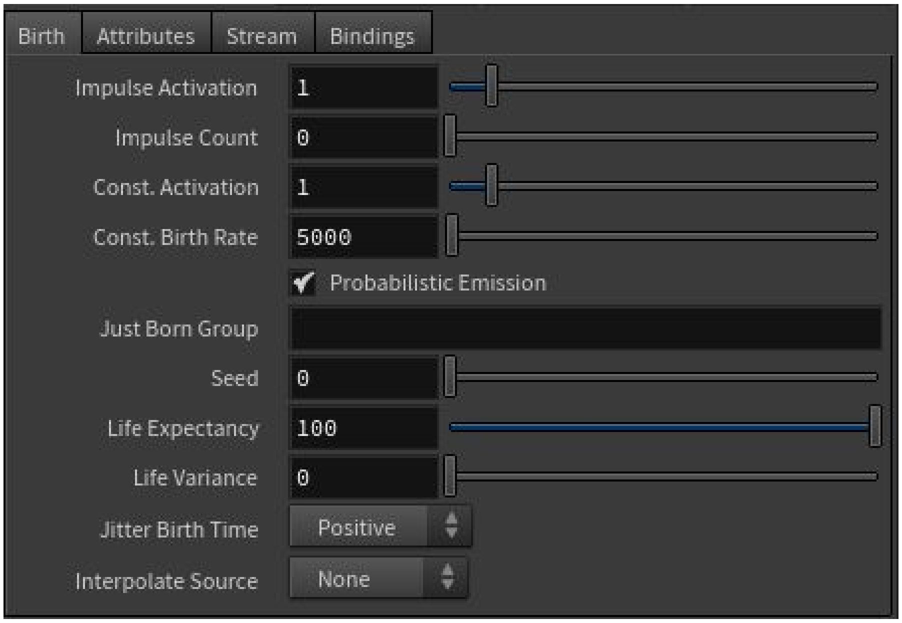
Emission Attribute:

SOP is a reference to the geometry object, and Source Group is a reference to a group in that geometry (if empty selects the entire geometry).

> **NOTE**: By default it looks like particles are generated at random across the geo. You can try fiddling with the Emission Type dropdown to change this.

Creation/Destruction

You can control when/where particles are created and destroyed via the Birth tab…



Impulse Activation	1
Impulse Count	0
Const. Activation	1
Const. Birth Rate	5000
	☑ Probabilistic Emission
Just Born Group	
Seed	0
Life Expectancy	100
Life Variance	0
Jitter Birth Time	Positive
Interpolate Source	None

Impulse Act/Count * Activation is a on/off value (0 off, anything else = 1) that determines if this feature is enabled.
* Count is the number of particles to generate per frame.

NOTE: It says each time the node cooks… nodes cook per frame, right?

NOTE: This CAN be used along side Const. Activation / Birth Rate

Const Act/Birth Rate * Activation is a on/off value (0 off, anything else = 1) that determines if this feature is enabled.
* Count is the number of particles to generate per second.

NOTE: It doesn't generate all the nodes at the start of each second. Instead it generates the relevant amount at each frame. So if our playback was set to 24 FPS and our count was set to 5000, each frame we'd get 5000/24 particles.

NOTE: This CAN be used along side Impulse Activation / Count

Seed Seed for random number generator used for determining where particles show up.

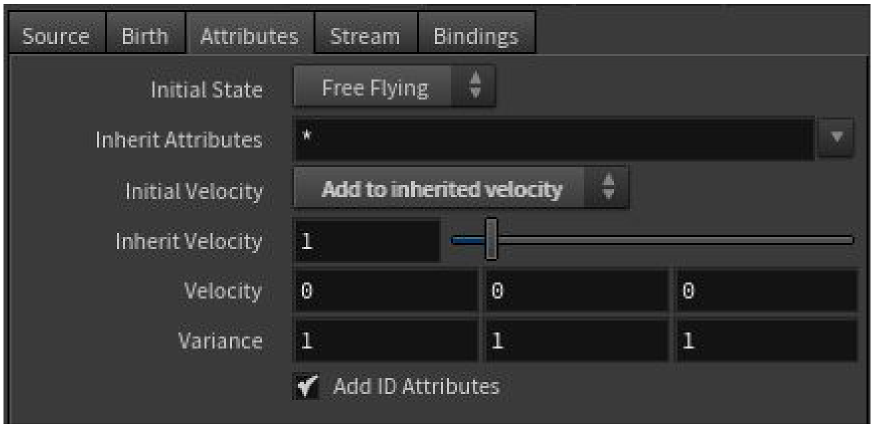
Life Expectancy How long each particle lives (in seconds).

Life Variance How much to vary the Life Expectancy value.

For example, if this was set to 5, the life expectancy would have a random number added to it ranging from -5 to 5.

Initial State

You can control the initial state of your particles via the Attributes tab…



Initial State	Free Flying		
Inherit Attributes	*		
Initial Velocity	Add to inherited velocity		
Inherit Velocity	1		
Velocity	0	0	0
Variance	1	1	1

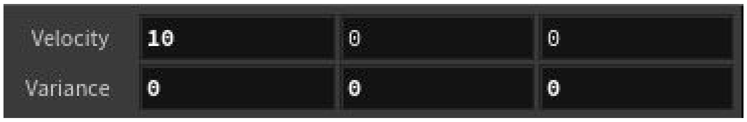
☑ Add ID Attributes

Velocity The initial velocity of the particles being generated

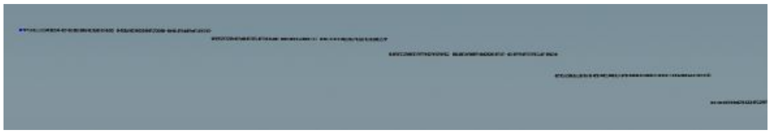
Variance How much to vary the velocity value.

> **NOTE**: What about the Initial/Inherit fields? These fields are only available if you're using a Source Particle Emitter (if you're generating your source from geometry). I think what it's for is if the geometry you're generating particles on is also a dynamics object (e.g. rigid body), its velocity and stuff will be mixed into the velocity of the particles being generated.

For example, if you had a velocity of (10, 0, 0) and a variance of (0, 0, 0), your particles would shoot down the X-axis like a straight line…

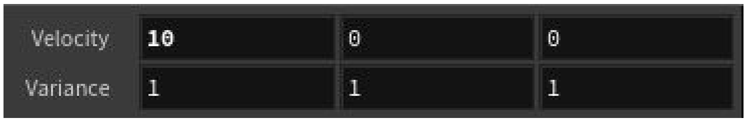


Velocity	10	0	0
Variance	0	0	0



> **NOTE**: Why is it dropping in steps instead of gradually? Probably because the FPS is higher than the physics iterations per second?

For example, if you had a velocity of (10, 0, 0) and a variance of (1, 1, 1), your particles would shoot down the X-axis like a straight line but each particle would vary its velocity in each axis by a random value from -1 to 1…



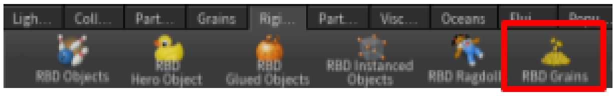
Velocity	10	0	0
Variance	1	1	1

Particle Collisions

Collisions w/ Dynamics

By default, collisions will interact with rigidbody objects and collision objects. They won't interact with each other though.

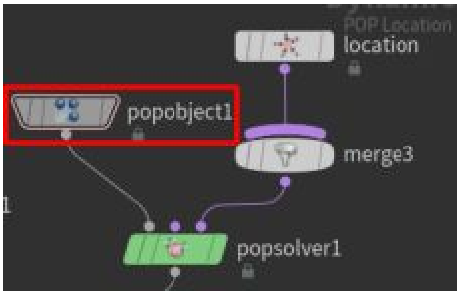
> **NOTE**: If you want particles to interact with each other, check out RBD Grains under the rigid body shelf…

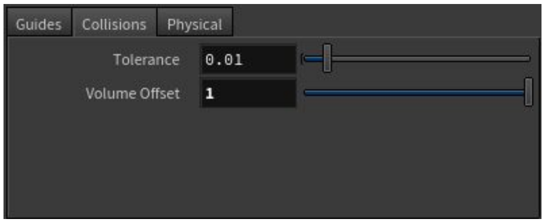


Particles don't seem to have a mass or volume associated with them. People on reddit say that the solver assigns some default mass/volume for each particle in order to calculate things like bounce and friction (I'm guessing mass/volume is required for these calculations).

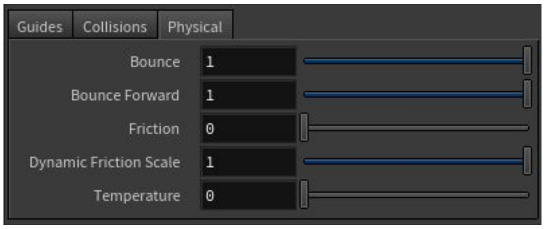
> **NOTE**: Remember that collision objects include: ground planes, static objects, etc.. -- basically anything from the Collisions shelf / anything being fed into a static solver.

The physical/collision details of the particles can be found under the popobject being fed into the particle solver in the AutoDopNetwork. If you select it, in the properties pane you'll find a Collisions tab and a Physical tab...





Tolerance 0.01
Volume Offset 1



Bounce 1
Bounce Forward 1
Friction 0
Dynamic Friction Scale 1
Temperature 0

Tolerance defines how close the particle has to be to the object to consider it collided? Since there is no volume for a particle (I think), it's calculated as a ray?

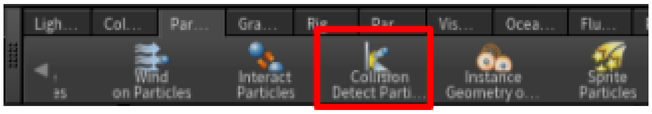
Volume Offset defines how much volumes are pushed out by a particle when they collide? Does volume here mean volumetrics (e.g. fog) or any object with a volume????

Bounce and Friction should be obvious. The rest I'm not too sure about.

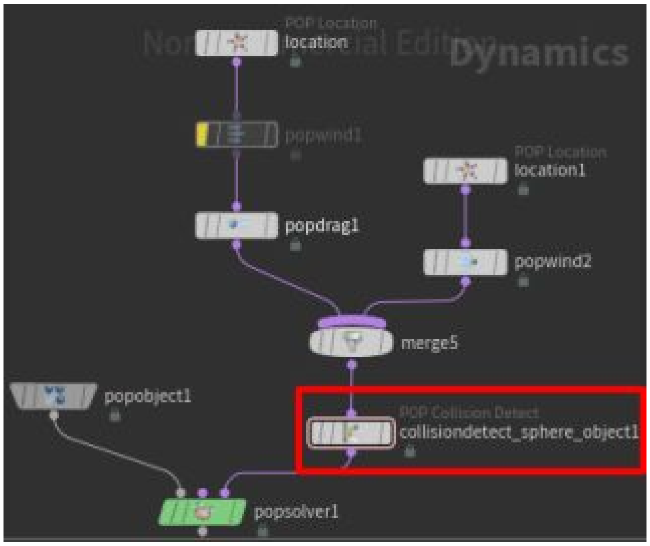
> **NOTE**: Popobject is where particles are generated. Check out the geometry spreadsheet when in the AutoDopNetwork and select popobject -> Geometry.

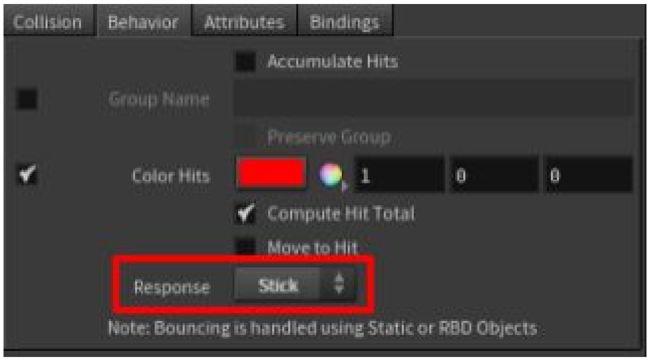
Collisions w/ Non-Dynamics

If you want to collide with a shape that isn't a dynamic (hasn't been added in as a rigid body or a static object or a ground plane or anything like that), you need to use the Collision Detect Particles item in the Particles shelf (it's near the end of the shelf)...



Once you do this, it adds POP Collision Detect node between your particles and the popsolver. In here you can adjust what happens on collision in the Collision tab via the Response dropdown (your options are slide/stick/die/stop -- all pretty self-explanatory).





☐ Accumulate Hits

☐ Group Name

☐ Preserve Group

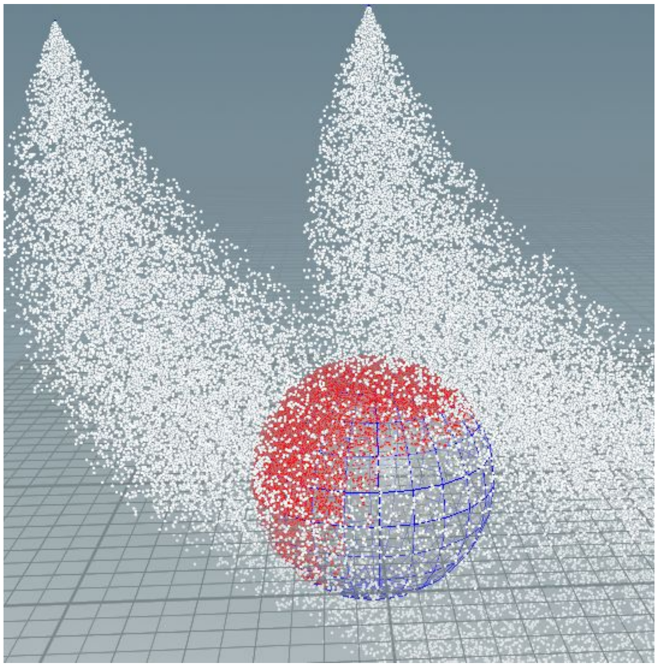
☑ Color Hits 🟥 1 0 0

☑ Compute Hit Total

☐ Move to Hit

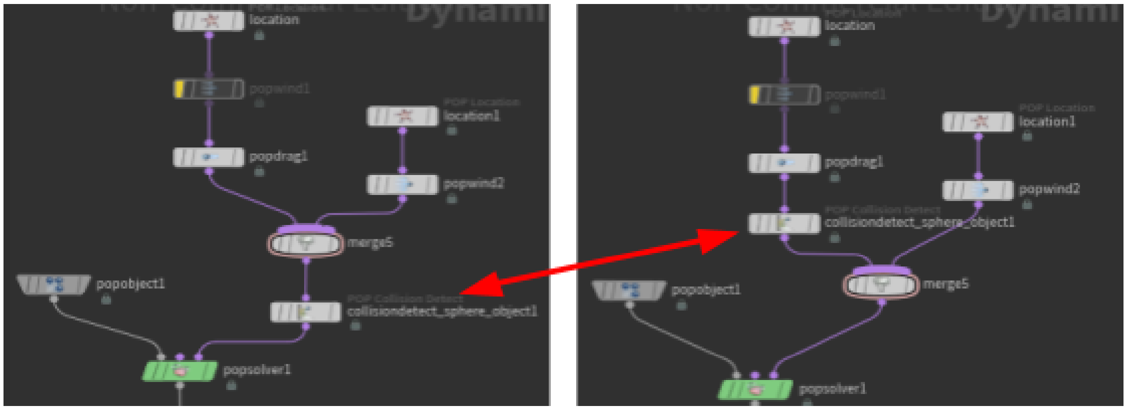
Response Stick

Note: Bouncing is handled using Static or RBD Objects



NOTE: The shape that the collision is being applied to is under the Collision tab.

NOTE: Notice where the POP Collision Detect node was generated. It's after the merge, which means all particles being fed into the popsolver will follow the collision detect rules added by this node. If you only want these rules only applied to particles emitted from certain emitters, move it so that it's in the path of those emitters (before the final merge).



Particle Forces

There a ton of different forces you can apply to particles. It seems like these forces ONLY effect particles and nothing else.

> **NOTE**: The gravity node is applied to particles as well. If you don't want it applied, you need to bypass it. Keep that in mind when applying these forces.

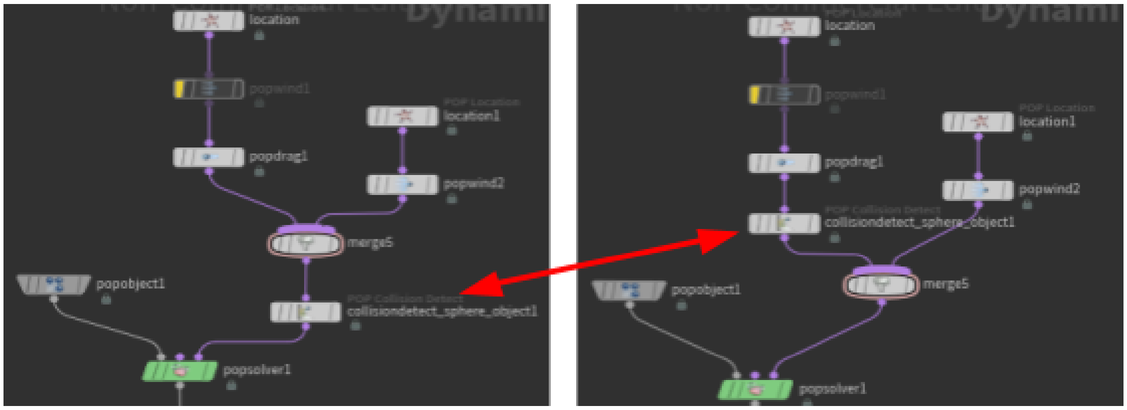
For example, rigid body objects are not affected by these objects. The particles may change direction as a result of the force and hit the rigid body differently (causing the rigid body to move differently), but the force itself won't move the rigid body.

Here are the types of forces you can apply…

- Axis Force → for spinny things like hurricanes/turbines/dust devils/etc..
- Wind/Drag → applies a constant force to particles regardless of location
- Fan → Like Wind/Drag but applied to a certain area
- Force → something akin to gravity? How's this different than Wind/Drag?
- Flock → flocking behaviour for particles
- Curve Force → creates forces generated from a curve????
- Point/Curve Attract → attracts particles to point or curve

> **NOTE**: See the section on Collision w/ Non-Dynamics… Just like how the placement of the POP Collision Detect node is important, the placement of your force node is also important. If it's after the final merge before the pop solver, it means all particles being fed into the popsolver will follow the force rules added by this node. If you only want these rules only applied to particles emitted from certain emitters, move it so that it's in

the path of those emitters (before the final merge).

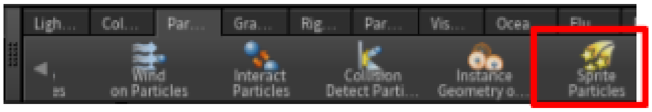


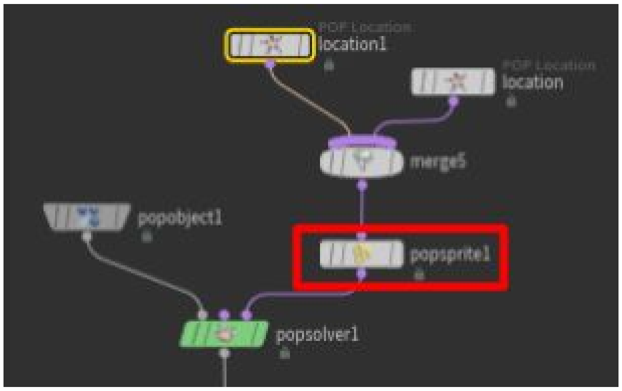
Particle Sprites

Sprites are textures that are attached to particles in billboard style.

> **NOTE**: Billboarding means that it's a flat 2d texture that always faces the camera. No matter how you orient the camera, you're always looking at the picture head-on. See https://gamedev.stackexchange.com/q/54871

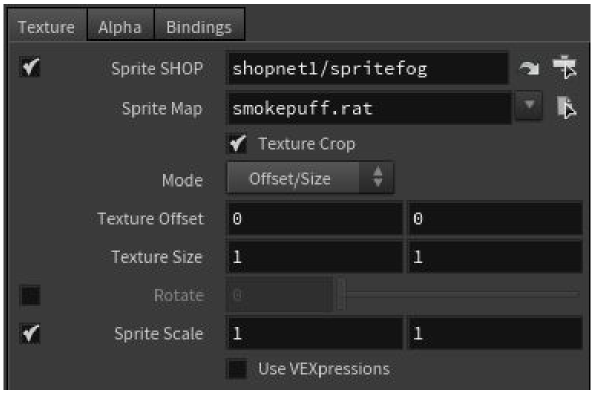
You can apply sprites to particles via Sprite Particles options of the Particles tab. Once you do, you'll get a POP Sprite node in your AutoDopNetwork.





> **NOTE**: See the section on Collision w/ Non-Dynamics… Just like how the placement of the POP Collision Detect node is important, the placement of your POP Sprite node is also important. If it's after the final merge before the popsolver, it means all particles being fed into the popsolver will have sprites applied. If you only want sprites applied to particles emitted from certain emitters, move it so that it's in the path of those emitters (before the final merge).

The two important tabs in the popsprite node’s properties are Texture and Alpha. Texture lets you define the particulars of the texture displayed, while Alpha lets you control how the sprite fades away (or fades in)...



☑ Sprite SHOP: shopnet1/spritefog

Sprite Map: smokepuff.rat

☑ Texture Crop

Mode: Offset/Size

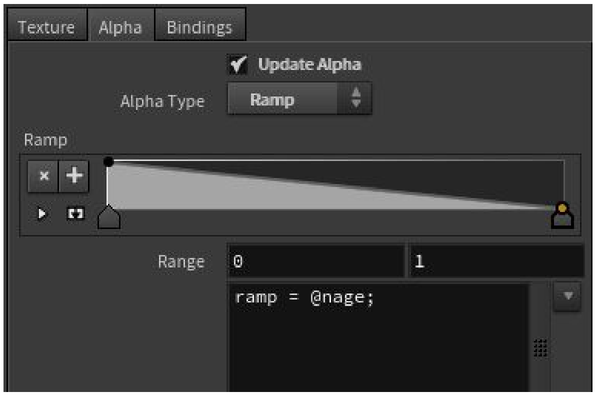
Texture Offset: 0 0

Texture Size: 1 1

□ Rotate: 0

☑ Sprite Scale: 1 1

□ Use VEXpressions



☑ Update Alpha

Alpha Type: Ramp

Ramp

Range: 0 1

```
ramp = @nage;
```

Everything here should be pretty self-explanatory. Sprite SHOP/Map refers to the material to apply to the sprite. Texture Offset/Size is for things like tilesets. Rotate/Scale is for rotation and scale. Alpha Type controls how much alpha is applied at which point.

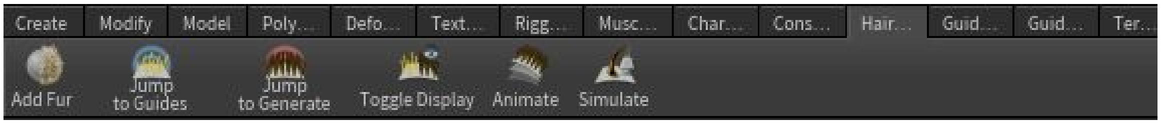
> **NOTE**: An important part of the lessons was using VEX to programmatically control some of these values. I’m going to skip this for now as it’ll probably be covered more in depth in later lessons.

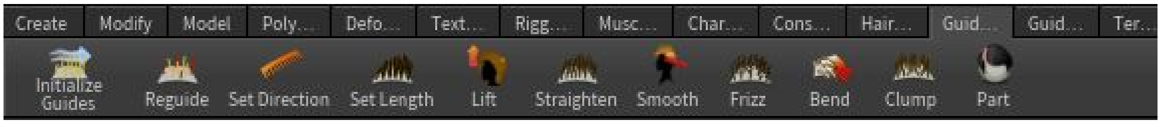
Houdini 16 Hair and Fur

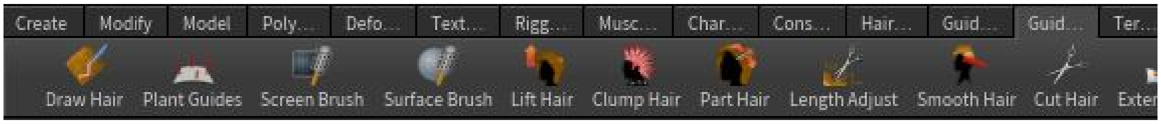
Introduction

Hair/fur in Houdini is done by through the concept of guide hairs. Guide hairs give detail on a particular cluster of hairs (e.g. length, shape, etc..), then when you actually do a render the real hairs show up modelled around those guide hairs.

All hair functionality is provided via 3 shelfs: Hair Utils, Guide Process, and Guide Brushes. You can find these in the shelves in the first shelf-set of the Build desktop…

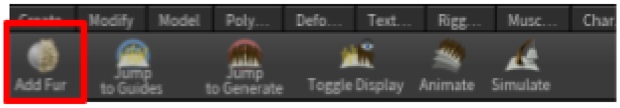






Creating Hair

To create hair, use Add Fur in the Hair Utils shelf.

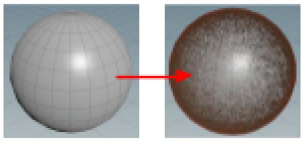


Once you do this…

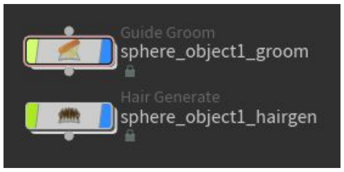
1. The scene view will prompt you to select an object and press Enter (only happens if nothing already selected when you clicked Add Fur).
2. The scene view will prompt you to select an object for “animated skin” and press Enter (you can just press enter without selecting anything -- don’t know what this is)

You’ll notice 2 things…

- Fur will show up on your entire object

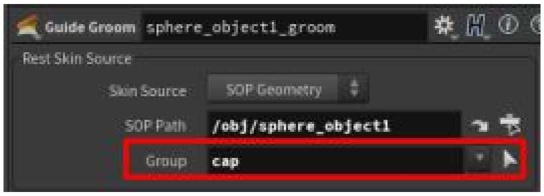


- There are 2 new nodes in /obj: a Guide Groom node and a Hair Generation node



The Guide Groom node is used for the hair guides and the Hair Generation node is what renders the actual hair. You do all your styling on the Guide Groom node, and the Hair Generation node will use those guides when it generates the hairs.

> **NOTE**: If you want to have only a specific group on your object have hair, select the Guide Groom node and choose the group you want to specifically target.



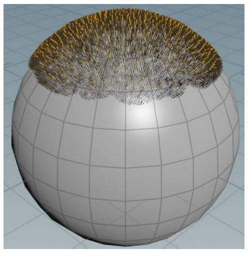
Guide Groom sphere_object1_groom

Rest Skin Source

Skin Source: SOP Geometry

SOP Path: /obj/sphere_object1

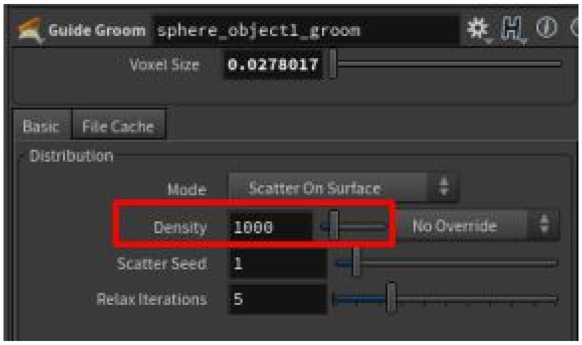
Group: cap



Remember that you don’t need to specifically target a group. You can use the little arrow next to the dropdown to select random polygons directly in the sceneview.

Hair Spread and Density

Hairs are distributed based on the guide hairs. You can set how many guide hairs you have via the Density property of the Guide Groom node (under the Basic tab)...



Guide Groom sphere_object1_groom

Voxel Size 0.0278017

Basic | File Cache

Distribution

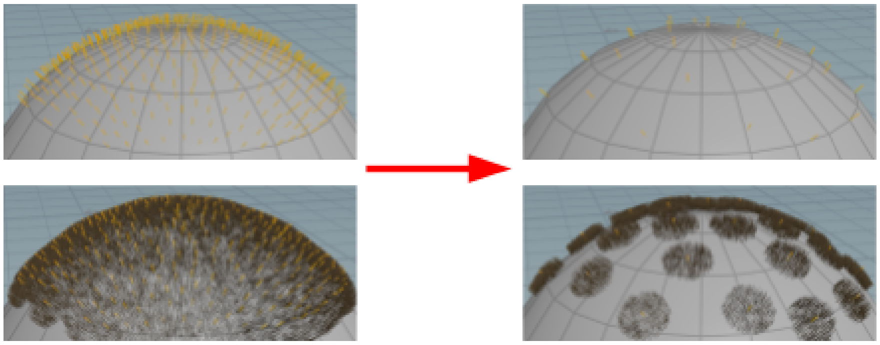
Mode: Scatter On Surface

Density: 1000 — No Override

Scatter Seed: 1

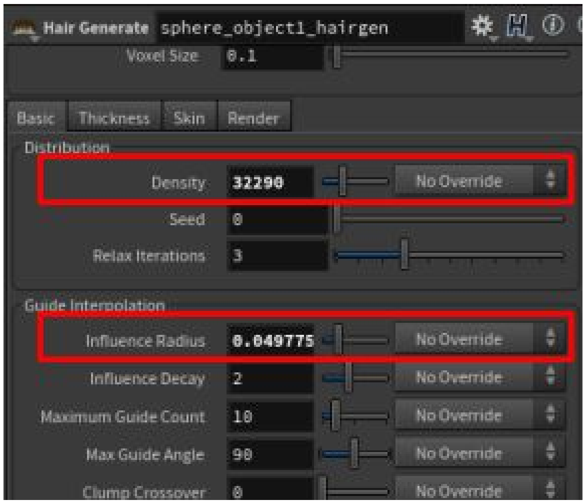
Relax Iterations: 5

Here’s an example of a density of 1000 vs 100…



Notice how it becomes super obvious how the hairs cluster around guide hairs once the hair density drops. We can have a low guide hair density if we want, we just need to go into the Hair

Generator node and jack up the Influence Radius and Density under the Basic tab…



sphere_object1_hairgen

Voxel Size 0.1

Distribution

Density 32290 No Override

Seed 0

Relax Iterations 3

Guide Interpolation

Influence Radius 0.049775 No Override

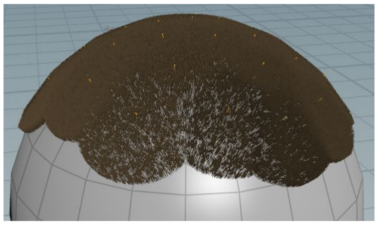
Influence Decay 2 No Override

Maximum Guide Count 10 No Override

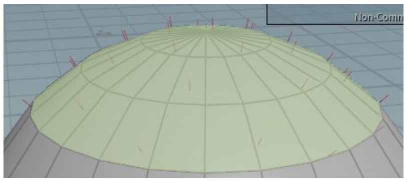
Max Guide Angle 90 No Override

Clump Crossover 0 No Override

For example, here’s a guide hair density of 100, but with a large influence radius (0.15) and a high density (132290)...



NOTE: The hairs are being generated on top of the sphere only because the groom node’s targeting the top polygons only…

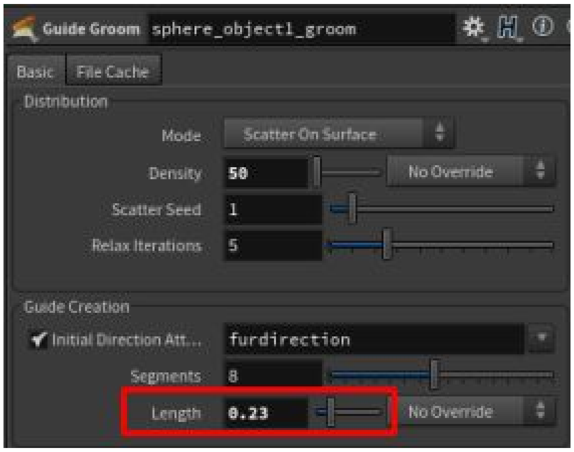


If you look closely at the images of hair above, you’ll see that the guide hairs (yellow lines) are on these polys but the actual rendered hair is bleeding out to the rest of the polys below it.

The only way to stop this is to explicitly DELETE all the polygons that hair shouldn’t be on. One trick is to simply create a copy of an object and only keep the polys that should have hair. Generate hair on that new object and hide the object from your scene. The hair will stay and it’ll be confined to those polys

Hair Length

You can control the length of your hair by going to the Length property of the Guide Groom node (under the Basic tab)...



sphere_object1_groom

Distribution

Mode: Scatter On Surface

Density: 50 — No Override

Scatter Seed: 1

Relax Iterations: 5

Guide Creation

✓ Initial Direction Att...: furdirection

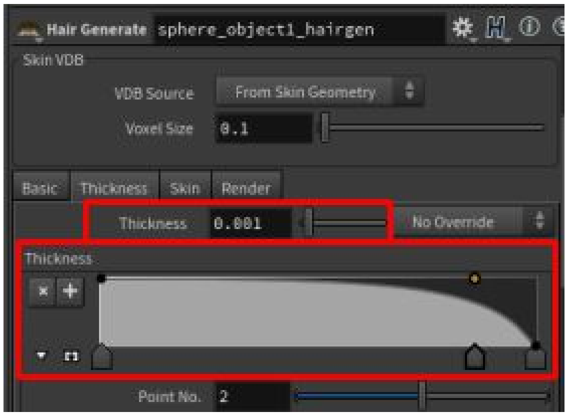
Segments: 8

Length: 0.23 — No Override

NOTE: This is the initial length of your hair. You'll likely either cut or extend it further down in your workflow as you stylize it. How to do that is discussed in later sections.

Hair Thickness

You can control how thick your hairs are (both overall and in certain areas) by going to the Thickness tab of the Hair Generator node…



sphere_object1_hairgen

Skin VDB

VDB Source: From Skin Geometry

Voxel Size: 0.1

Thickness: 0.001 — No Override

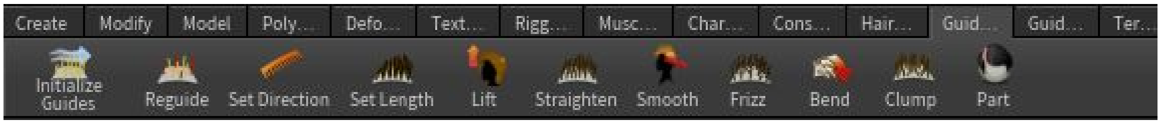
Thickness

Point No.: 2

It should be fairly straight forward what's happening here. The thickness is the overall thickness, and the ramp controls how that thickness grows/shrinks over the length of the hair.

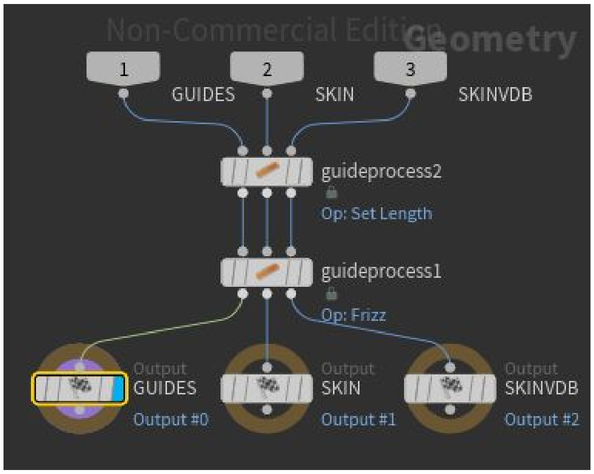
Procedural Styling

You can stylize your guide hairs procedurally by using the items in the Guide Processing shelf…



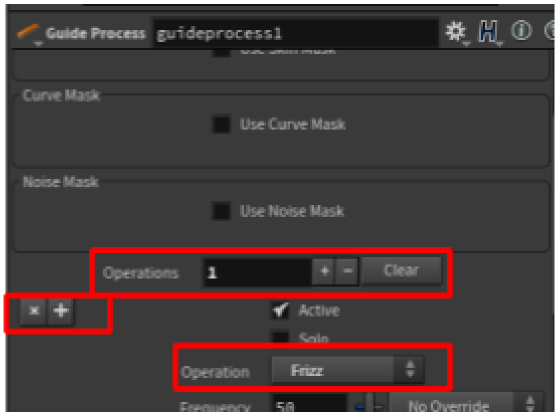
Most of what the items do should be self-explanatory. Remember that Houdini has a node based procedural workflow. As you use these items, they'll get added as nodes inside the Guide Groom node.

For example, here's what a Frizz + lengthen would look like inside the Guide Groom node…



You can do all the stuff you would normally do with nodes here: reorder, manually add more in, bypass, etc..

> **NOTE**: Most all the options in this Guide Processing shelf translate to a Guide Process node being added in (most… not all). Guide Process nodes have a Operation dropdown where you can select what you want them to do (the options in the shelf). You can even add more than one operation together here if you really wanted to…



Guide Process guideprocess1

Curve Mask

□ Use Curve Mask

Noise Mask

□ Use Noise Mask

Operations 1

✓ Active

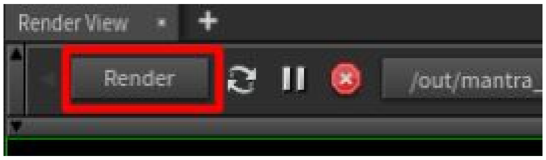
□ Solo

Operation Frizz

Frequency 58 No Override

> **NOTE**: If you're doing renders inside the renderview, as you add/change these nodes

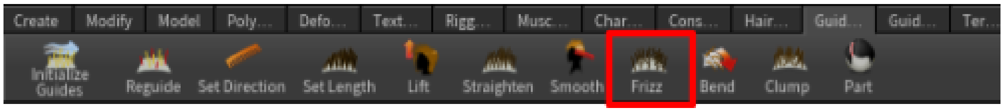
the refresh/auto-update won't pick up the new hair geometry. You need to explicitly hit the Render button if your hair geo changes…

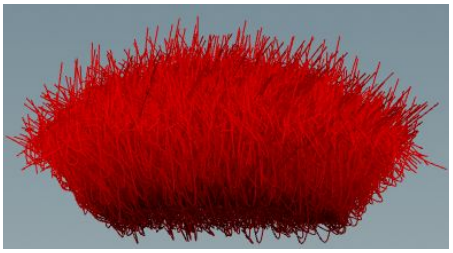


Not sure if something similar required if rendering to MPlay.

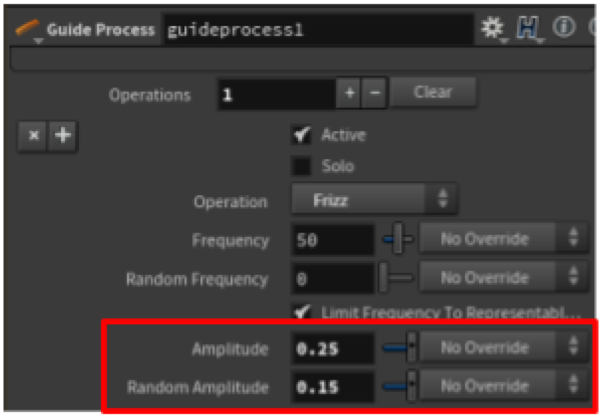
Frizz

You can add frizz to your guide hairs via Frizz…





The main property here is amplitude. The higher the amplitude is, the more noise (aka frizz) is added to the guide hair…



Guide Process guideprocess1

Operations 1 + − Clear

☑ Active

□ Solo

Operation Frizz

Frequency 50 No Override

Random Frequency 0 No Override

☑ Limit Frequency To Representabl…

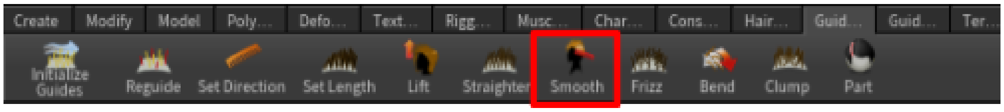
Amplitude 0.25 No Override

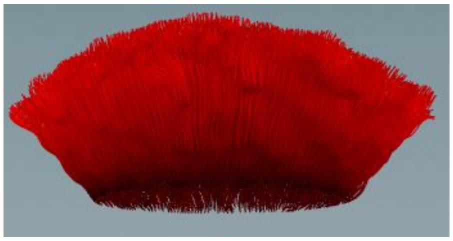
Random Amplitude 0.15 No Override

Remember that this operation is happening on the guide hair. If you have too few guide hairs, the entire thing will look off. It'll look like clusters/hunks of hair are bending randomly, as if someone stylized the hair to make it look like a “bedhead” look…

Smooth

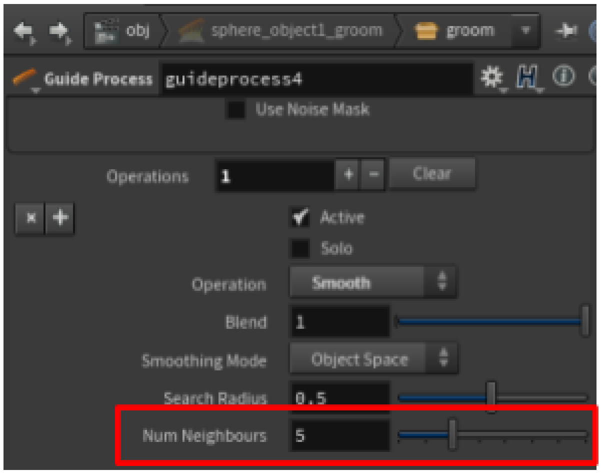
You can smooth your guide hairs via smooth…





> **NOTE**: The example above has frizz applied before smooth. Note that it’s almost back to being uniformly straight.

The main property here is number of neighbours. The higher this value is, the more neighbouring guide hairs are brought into the averaging calculations to create a smoothed hair (I think)…



obj > sphere_object1_groom > groom

Guide Process guideprocess4

□ Use Noise Mask

Operations 1 + − Clear

☑ Active

□ Solo

Operation: Smooth

Blend: 1

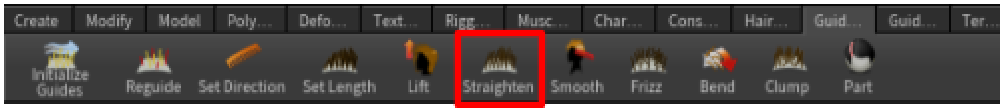
Smoothing Mode: Object Space

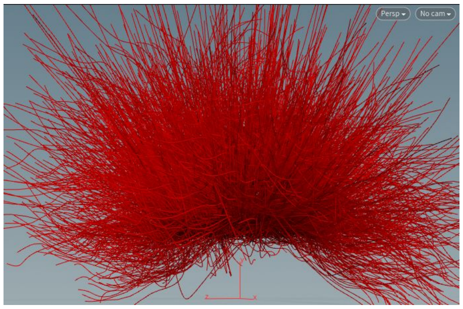
Search Radius: 0.5

Num Neighbours: 5

Straighten

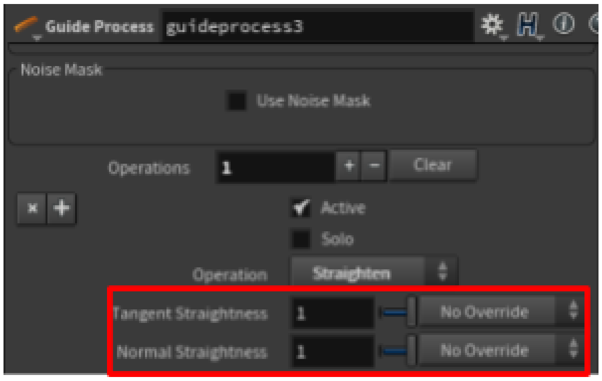
You can straighten your guide hairs via Straighten…





NOTE: The example above has frizz applied before straighten.

I honestly have no idea how this is suppose to work. There are 2 properties here: Normal Straighten and Tangential Straighten. No idea what they're suppose to do...



Guide Process guideprocess3

Noise Mask

□ Use Noise Mask

Operations 1 + − Clear

☑ Active

□ Solo

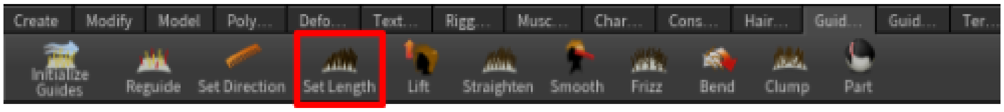
Operation Straighten

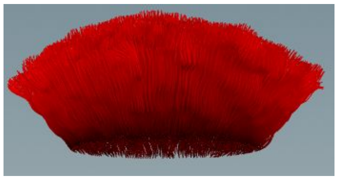
Tangent Straightness 1 No Override

Normal Straightness 1 No Override

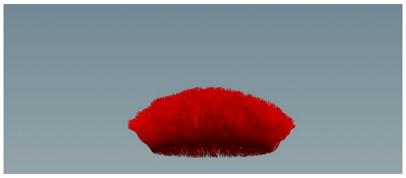
Set Length

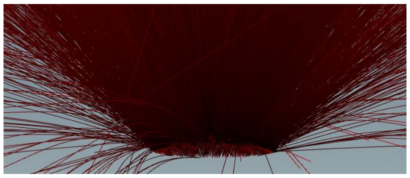
You can grow/shrink your guide hairs via Set Length…





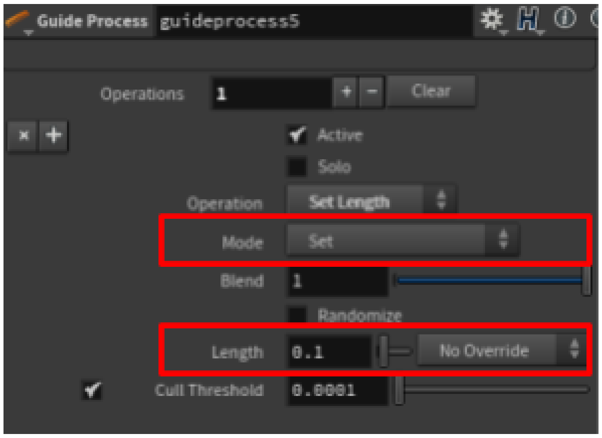
→





NOTE: The example above has frizz+smooth applied. Note that when we shorten the length, the hair pretty much truncates (as expected). But, when we grow the hair, it pretty much grows straight out in the direction it was before the growth.

The important property here is Length, which I think is the overall length of the hair. You can change the Mode dropdown if you want to do things like offset the hair length instead of setting it directly...



Guide Process guideprocess5

Operations 1 + − Clear

☑ Active

□ Solo

Operation: Set Length

Mode: Set

Blend: 1

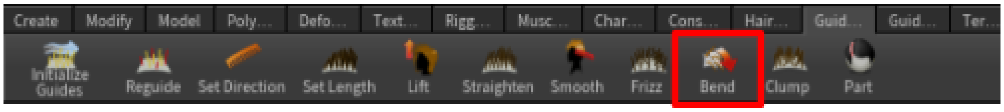
□ Randomize

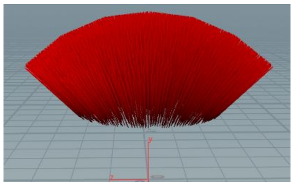
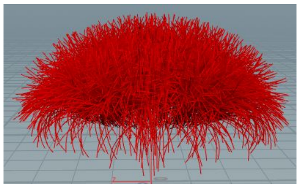
Length: 0.1 No Override

☑ Cull Threshold: 0.0001

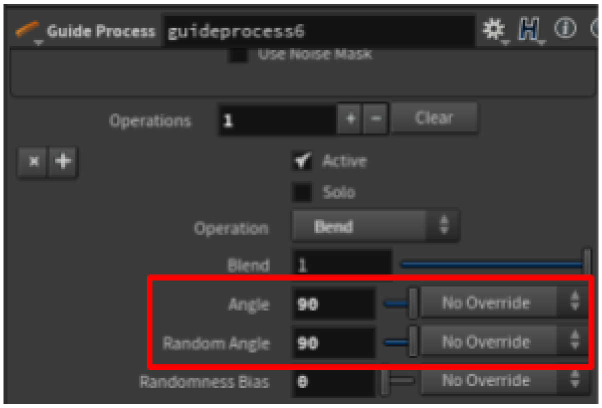
Bend

You can bend your guide hairs via Bend…




→


This bends the hair such that its new endpoint is x degrees different from the previous endpoint. The important property here is Angle and Random Angle, which controls how much of an angle you want to bend by + how much randomness you want in that angle for each guide hair.


Guide Process guideprocess6

Use Noise Mask

Operations 1 + − Clear

✓ Active

□ Solo

Operation Bend

Blend 1

Angle 90 No Override

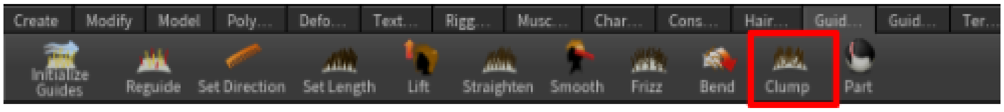
Random Angle 90 No Override

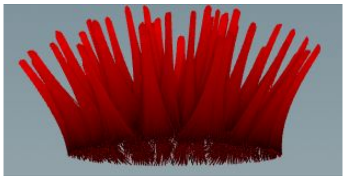
Randomness Bias 0 No Override

NOTE: It doesn't look like you can control the ultimate direction in which the bend is applied.

Clump

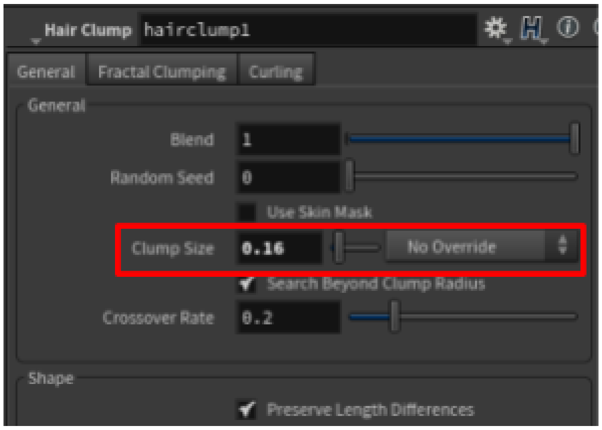
You can clump your guide hairs together via Clump…





Clump does not get added as a Guide Processing node, but as its own node type (Hair Clump).

The important property here is Clump Size, which technically will define how many clumps you'll have (based on the number of guide hairs you have). But, there are also a ton of other options here. For example, you can curl your clumps if you want to, or you can use a ramp to set the thickness of your clumps in different areas...



Hair Clump hairclump1

General

Blend 1

Random Seed 0

□ Use Skin Mask

Clump Size 0.16 No Override

☑ Search Beyond Clump Radius

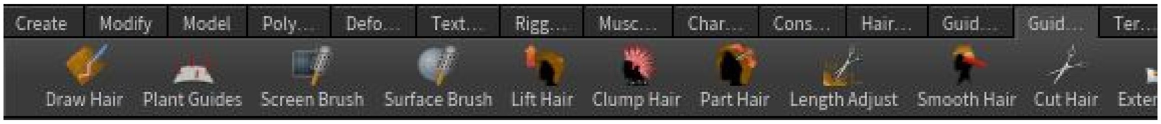
Crossover Rate 0.2

Shape

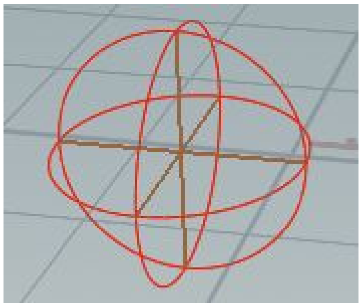
☑ Preserve Length Differences

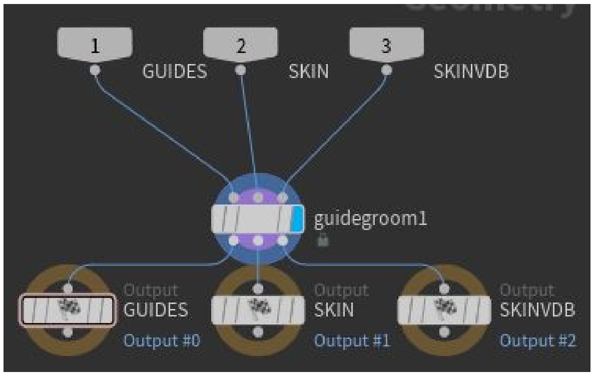
Manual Styling

You can stylize your guide hairs manually (via the scene view) by using the items in the Guide Brushes shelf…

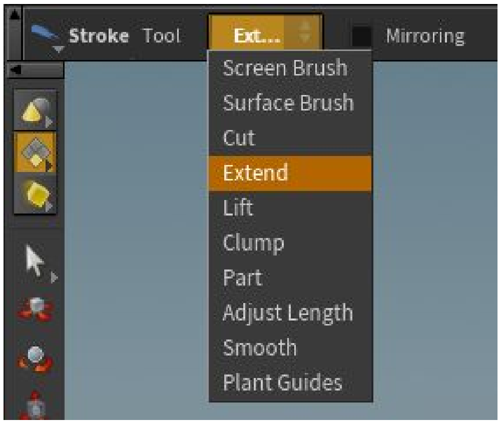


Once you select any of these (don't know about the first 2), your handle in the scene view should turn into a circular brush tool + you should get an internal Guide Groom node added to your top-level Guide Groom node.





Your scene view's toolbar should now also have a dropdown that'll let you specifically select which tool you want to use…

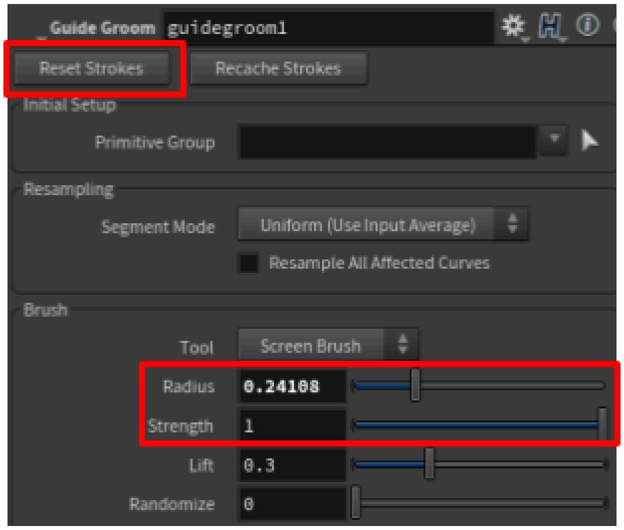


NOTE: These options are exactly what's in the Guide Brushes shelf. You can manually switch here as you're making changes to your hair.

From here you can use the brush tool to pretty much change anything however you see fit: cut pieces of hair, extened pieces of hair, clump, brush, etc..

NOTE: It may be better to do this in 4-panel orthographic view. Depending on how many guide hairs you have, it may be difficult to target an individual hair. You might have to zoom in a lot to isolate which kind of defeats the concept of brushing.

You can change the brush size, brush strength, and undo all your strokes in the properties of the Guide Groom node added for these changes. You may get more/less options depending on the Tool you're using (e.g. brush, extend, cut, clump, …)...



Guide Groom guidegroom1

Reset Strokes | Recache Strokes

Initial Setup

Primitive Group

Resampling

Segment Mode: Uniform (Use Input Average)

□ Resample All Affected Curves

Brush

Tool: Screen Brush

Radius: 0.24108

Strength: 1

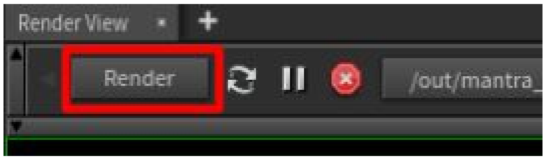
Lift: 0.3

Randomize: 0

NOTE: Remember that you can undo individual steps with Ctrl+Z. Also you can use the mouse wheel in the scene view to increase/decrease brush size.

NOTE: The lift parameter is also important. It controls an artificial buoyancy for the hair as you're manipulating it… such that if you were to brush it down, it wouldn't go exactly flat The hair would have some bounce back to it (unless maybe if you wanted to act as if it were being weighed down with something like hair gel).

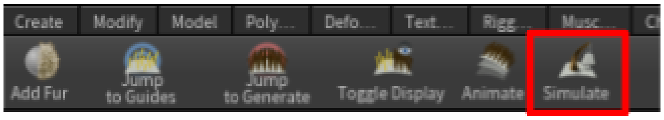
NOTE: If you're doing renders inside the renderview, as you add/change these nodes the refresh/auto-update won't pick up the new hair geometry. You need to explicitly hit the Render button if your hair geo changes…



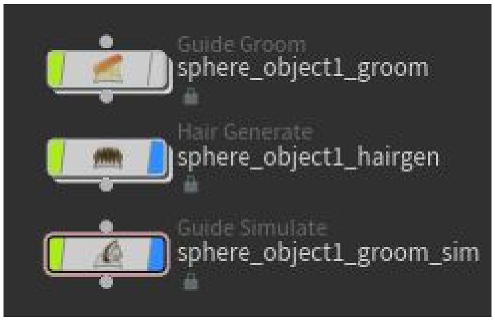
Not sure if something similar required if rendering to MPlay.

Dynamics/Physics

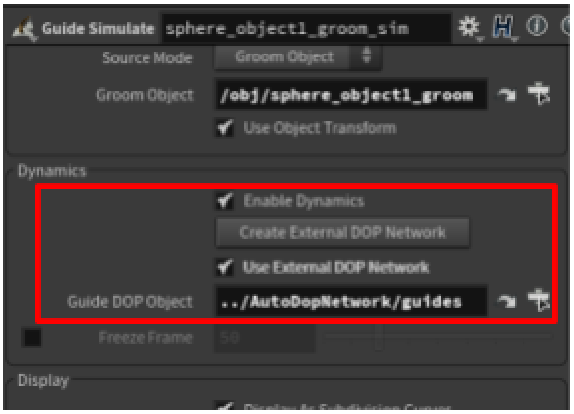
This section was not covered in the lessons, but it looks like you can enable physics for hair by using Simulate under the Hair Utils shelf…



Doing this does not do the same thing as creating a collision object (e.g. ground plane) or a rigid body. YOU WILL NOT GET STUFF ADDED TO AN AutoDopNetwork NODE. Instead what will happen is that you'll get a new Guide Simulate node…



In this Guide Simulate node's parameters, you'll get a button to create a DOP network for this piece of hair. If you choose the button, a new DOP network will be created and referenced. You can go into that DOP network and cut/paste the contents it to your own DOP network + update the Guide DOP Object path.



Guide Simulate sphere_object1_groom_sim

Source Mode: Groom Object

Groom Object: /obj/sphere_object1_groom

✓ Use Object Transform

Dynamics

✓ Enable Dynamics

Create External DOP Network

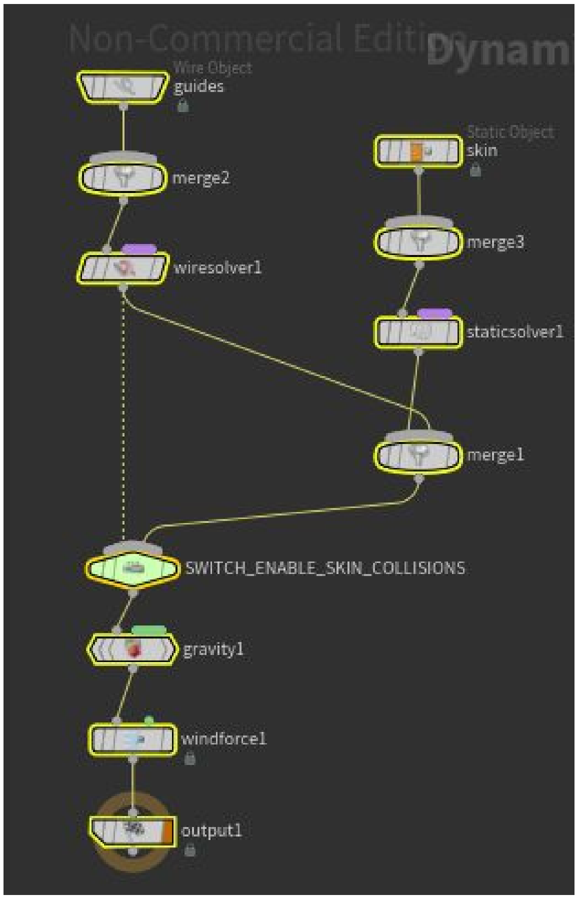
✓ Use External DOP Network

Guide DOP Object: ../AutoDopNetwork/guides

□ Freeze Frame 50

Display

✓ Display As Subdivision Curves

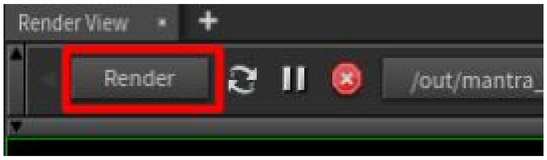


> **NOTE**: There's also a Simulation and Collision tab in the Guide Simulation node that defines how your guide hairs will react/collide with other stuff. There's too much to go into detail here and none of it was talked about in the lesson.

Once you do that, the hair should be able to interface with other dynamics in your system (particles, collisions, etc.. etc..).

Rendering

If you're doing renders inside the renderview, as you add/change these nodes the refresh/auto-update won't pick up the new hair geometry. You need to explicitly hit the Render button if your hair geo changes…

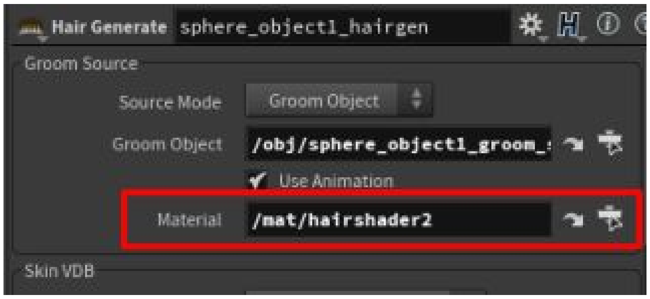


Not sure if something similar required if rendering to MPlay.

Hair Shader

> **NOTE**: If your hair geometry changes, you need to hit the Render button. But, if all you're doing is messing around with the hair shader, you don't need to do this. See the main Rendering section above if you're confused.

The Hair Geometry node contains a path reference to which material will be used to render whatever hair it's generating…



Hair Generate sphere_object1_hairgen

Groom Source

Source Mode: Groom Object

Groom Object: /obj/sphere_object1_groom_:

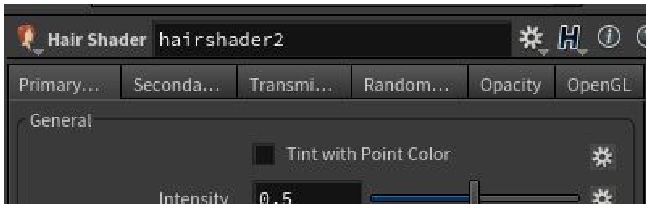
☑ Use Animation

Material: /mat/hairshader2

Skin VDB

> **NOTE**: You can use the second to last button (curved arrow) to go directly to the shader.

The shader here is of type Hair Shader, and you can pretty much modify the properties here to get your hair to look however you want it to look. There are multiple tabs here…



Hair Shader hairshader2

General

□ Tint with Point Color

Intensity: 0.5

- Primary Reflection

  This defines the primary color of the hair. You can set a tip color vs a root color and control how they transition with the provided ramp.

- Secondary Reflection

  This is like the specular. You'll notice that by default the hair shader gives off this horrible sheen as if the individual hairs hasve been coated in wax. This is where that sheen is coming from. You can turn off the entire thing via the Enable checkbox at the top of just lower the intensity.

- Transmission Color

  This is for "light passing through the hair". Unsure if this is the same thing as a transmittance color in Arnold, which is… "Like color, but tints the object more as the ray goes through the object. Thin parts of the object get less tint while thicker parts get more tint."

So maybe the hair thickness attribute has an effect on the output for this??? I tried playing with the values but I couldn't get it to show any difference.

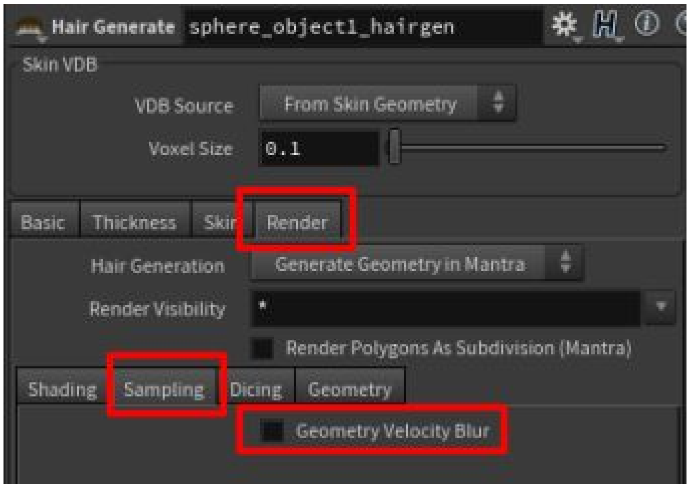
- Randomness

  Allows you to Randomize the HSV values of the diffuse (Primary Reflection) in addition to randomizing the specular. So the hair can technically be more varied in terms of brightness/darkness or color.

Motion Blur

By default, your Hair Geometry will not support motion blurring. If you want motion blurring to be enabled, go to your Hair Generate node. In the properties, there will be a Render tab. Under that tab there's tab called Sampling.

Turn on the Geometry Velocity Blur checkbox.



Hair Generate sphere_object1_hairgen

Skin VDB

VDB Source: From Skin Geometry

Voxel Size: 0.1

Hair Generation: Generate Geometry in Mantra

Render Visibility: *

□ Render Polygons As Subdivision (Mantra)

□ Geometry Velocity Blur

Render Quality

Once you start rendering, you'll notice areas with sparse amount of hairs (e.g. the tips of the hair where it tapers off) have a lot of noise. You need to bump the rendering pixel samples to get rid of the noise.

See the Render Quality section of the Mantra Rendering document to see how to do this.

Houdini 16 Mantra Renderer

Renders

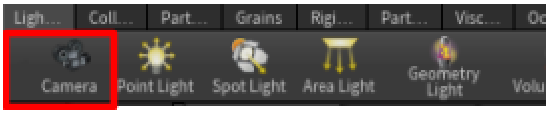
The renderer that comes with Houdini is called Mantra. It seems very similar to Maya’s Arnold renderer, but not as popular. Although Arnold does have a Houdini plugin, I think Houdini people prefer to avoid it when doing their FX work.

Setup

To setup the a render using Mantra, you need to create a render camera and add a Mantra render node in the /out context.

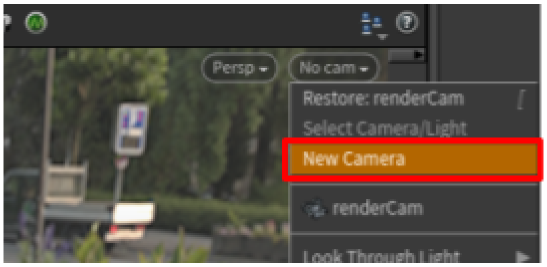
Create a Render Camera

Before you can render, you need to create a camera. To create a camera, you can either use the Camera item in the Lighting shelf…



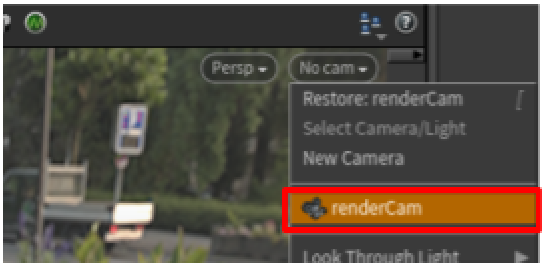
NOTE: Remember that if you hold Ctrl and click it'll dump it at (0,0,0) and select it upon creation.

Or, you can orient your scene to where you want your camera to be and in the upper right-most dropdown (just below the toolbar) in the scene view, choose New Camera…

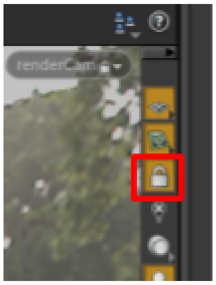


Once the camera is created, you can do a Maya "look-through object"-style orientation by…

1. Selecting it in that same dropdown (doesn't matter how you created it -- it'll exist in the dropdown)



2. Locking your view to the camera using the right toolbar



3. Orienting your view using the normal pan/tumble/zoom controls

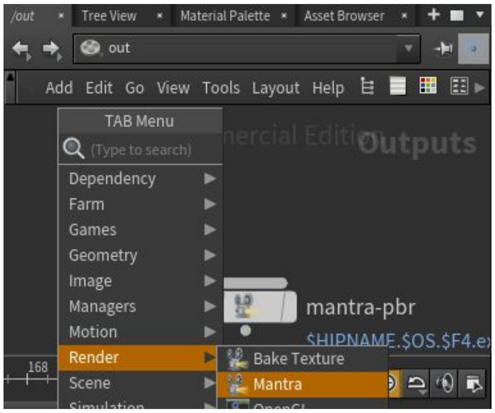
NOTE: Remember to unlock your view once you're finished orienting your camera.

NOTE: You can set things like aperture and f-stop directly on the camera you created. All those options will be in the properties pane.

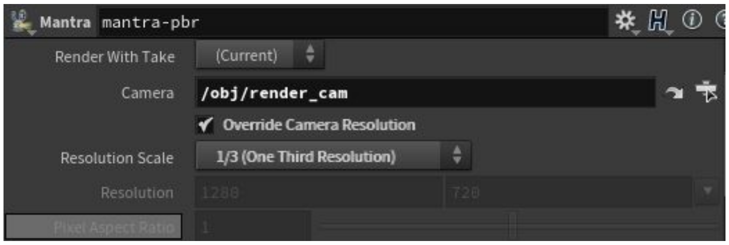
Create a Render Node

You can create render nodes in the /out context. Usually having 1 is fine, but you can create different render nodes if you want to have multiple different render setups (e.g. quick render vs production-quality render).

You can add a Mantra node by going to the network view, switching context to out, and inserting a Mantra node from the Tab menu…



After you set down the node, YOU MUST SET A CAMERA in the properties pane. That means your scene NEEDS to have at least 1 camera node in it...


Mantra mantra-pbr

Render With Take (Current)

Camera /obj/render_cam

☑ Override Camera Resolution

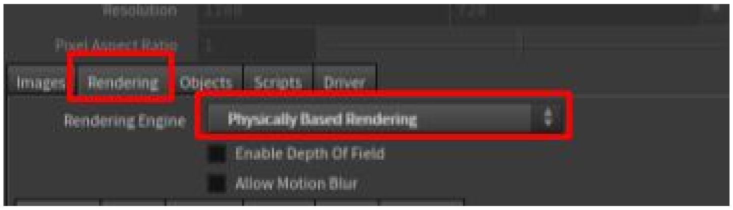
Resolution Scale 1/3 (One Third Resolution)

Resolution 1280 720

Pixel Aspect Ratio 1

NOTE: See the above section on how to create cameras if you don't have one created yet.

It's recommended by Houding to set the Rendering Engine to Physically Based Rendering. According to the lesson, this isn't the default but the other methods are legacy and PBR gives you a lot of features out-of-the-box for free without manual tuning.



Resolution 1280 720

Pixel Aspect Ratio 1

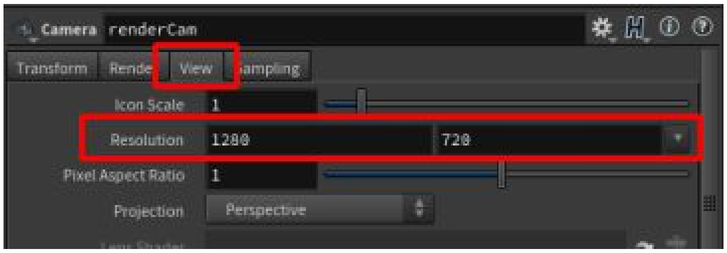
Rendering Engine: Physically Based Rendering

□ Enable Depth Of Field

□ Allow Motion Blur

Render Resolution

You can set the render resolution on the camera that your Mantra node is linked to. In the camera properties, under the View tab there should be a Resolution property…



Camera renderCam

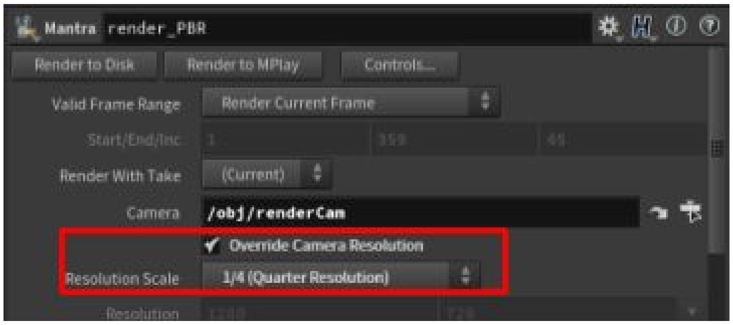
Icon Scale 1

Resolution 1280 720

Pixel Aspect Ratio 1

Projection: Perspective

You can scale down the rendering resolution of the camera in the Mantra node via the Resolution Scale in the property. To enable it, you need to check Override Camera Resolution property…



Mantra render_PBR

Valid Frame Range: Render Current Frame

Start/End/Inc 1 359 45

Render With Take: (Current)

Camera /obj/renderCam

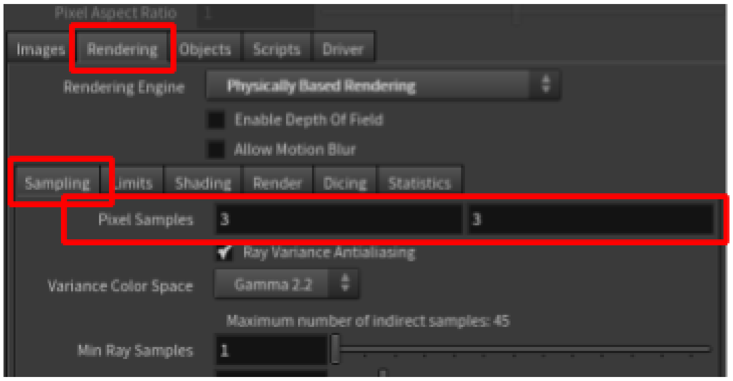
☑ Override Camera Resolution

Resolution Scale: 1/4 (Quarter Resolution)

Resolution 1280 720

Render Quality

You can set the samples for the render (just like you do with Arnold). You can do so under Rendering -> Sampling in the properties pane of the Mantra node...



Pixel Aspect Ratio 1

Rendering Engine: Physically Based Rendering

☐ Enable Depth Of Field

☐ Allow Motion Blur

Pixel Samples: 3 3

☑ Ray Variance Antialiasing

Variance Color Space: Gamma 2.2

Maximum number of indirect samples: 45

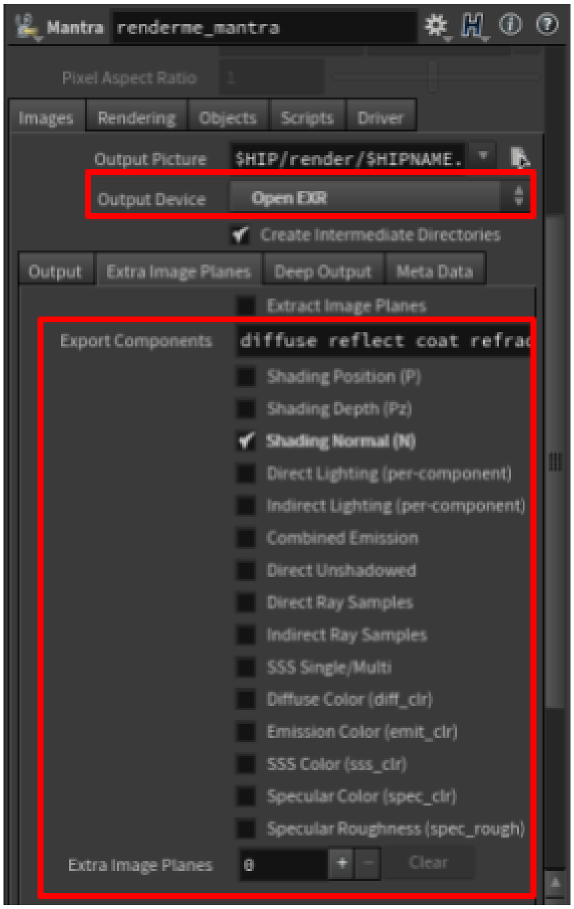
Min Ray Samples: 1

Pixel Samples seems to be the most important option here. For things like subsurface scattering, if you don't want a lot of noise you need to jack this value high. I set it to around 20 or 30 to get a okay quality subsurface scattering. There are also individual "quality" sliders below Pixel Samples for things like SSS and specular.

Image Planes (AOVs)

Mantra's equivalent of Arnold's AOVs is called image planes. You can define image planes in the Mantra node by selecting what you want via the properties under Images -> Extra Image Planes…

NOTE: Make sure you set your Output Device to OpenEXR.


Mantra renderme_mantra

Pixel Aspect Ratio 1

Output Picture $HIP/render/$HIPNAME.

Output Device Open EXR

☑ Create Intermediate Directories

□ Extract Image Planes

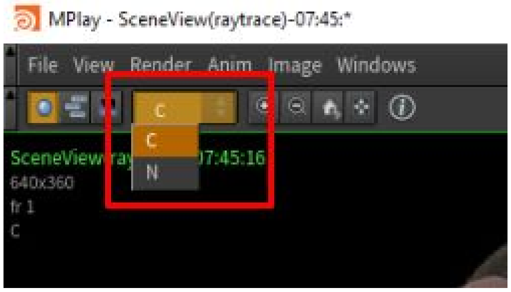
Export Components diffuse reflect coat refrac

□ Shading Position (P)
□ Shading Depth (Pz)
☑ Shading Normal (N)
□ Direct Lighting (per-component)
□ Indirect Lighting (per-component)
□ Combined Emission
□ Direct Unshadowed
□ Direct Ray Samples
□ Indirect Ray Samples
□ SSS Single/Multi
□ Diffuse Color (diff_clr)
□ Emission Color (emit_clr)
□ SSS Color (sss_clr)
□ Specular Color (spec_clr)
□ Specular Roughness (spec_rough)

Extra Image Planes 0

Once selected, YOU MUST RENDER TO MPLAY to see the AOVs… You can select which AOV is being shown via the dropdown in the toolbar.

> **NOTE**: C is the beauty render… To figure out what the rest of the codes are, look at the names of the checkboxes you selected for extra image planes.


MPlay - SceneView(raytrace)-07:45:*

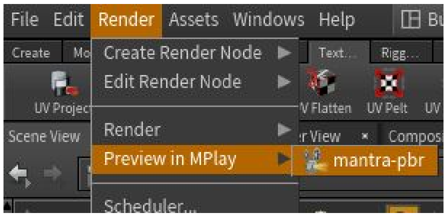
SceneView(raytrace)-07:45:16
640x360
fr 1
C

Standalone Render

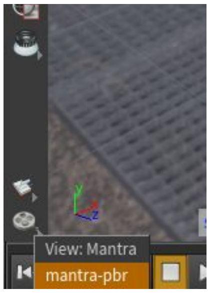
There are many ways to trigger a render…

1. In the main menu… Render -> Render -> (mantra node name)

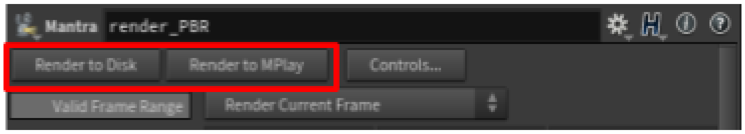
2. In the main menu… Render -> Preview in MPlay -> (mantra node name)



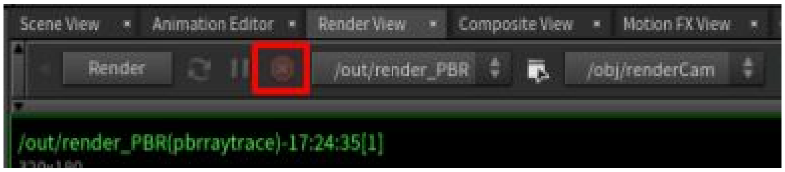
3. In the left toolbar of scene view… click the movie reel (last button in the toolbar) and choose the render node name



4. In the Mantra node itself, there are the following options available at the very top of the properties pane…



If you choose Render to Disk, you can kill the Render by going to the Render View panel and choosing the Stop button…



```
/out/render_PBR(pbrraytrace)-17:24:35[1]
```

NOTE: MPlay is Houdini’s version of Maya’s fcheck? Fcheck is the Maya image playback viewer checker thing. You need to render to MPlay if you have extra image planes being output (image planes are the equivalent of Arnold AOVs) -- this is detailed in further sections

NOTE: It turns out as your render’s happening, you can tell MPlay/Mantra to focus on a particular part first by clicking on it during the render. The tiles should start to focus on that area. This may only work if progressive rendering is turned OFF (double check this when you have a chance)

Interactive Render

Remember that IPR stands for Interactive Photorealistic Render. It's used to get feedback on how the render looks in real-time. It's works like downloading old school progressive scan JPEGs from the Internet back in the days of 56k modems.

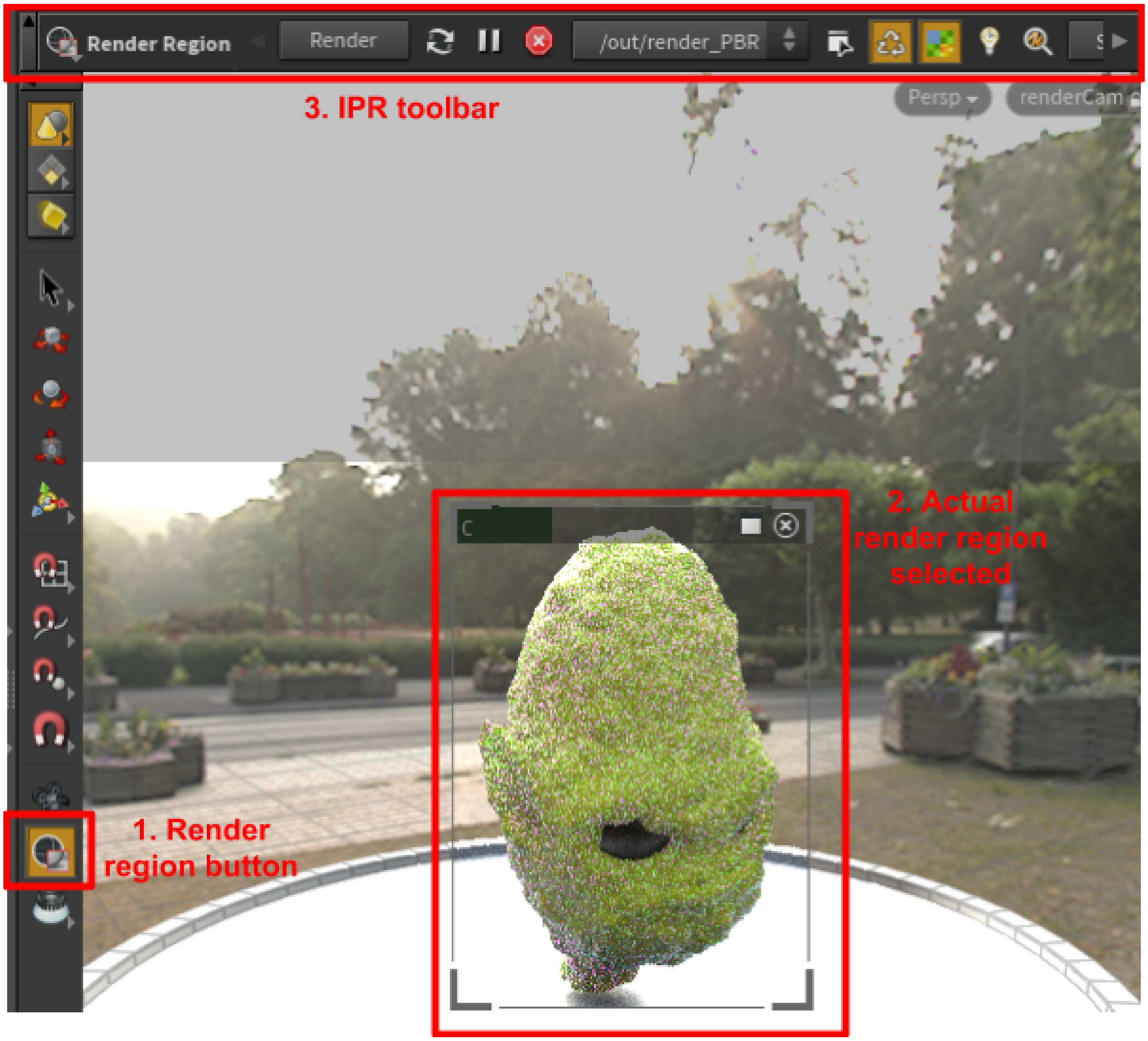
There are 2 ways to do IPR rendering: directly in the Scene View or via the Render View.

Scene View

You can do IPR on a select part of your scene through the scene view via the Render Region feature.

To activate a render region…

1. Select render region button on the left-hand toolbar (it's near the bottom).
2. Once selected, you can click-and-drag on a region in your scene view -- that region will render directly in the scene view.
3. Notice how you get the exact same toolbar showing as you do in the Render View



The render region has a little progress bar on top of it that fills up as the render completes. You can click the X in the righthand side of the progress bar to kill the render region.

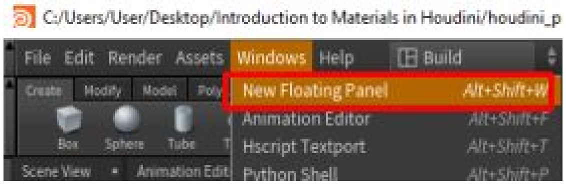
The toolbar will remain visible so long as the Render Region button is selected. If you want to know the particulars of the toolbar, check out the Render View section directly below (its the same toolbar).

> **NOTE**: Remember that if you can still manipulate your view while the Render Region is going. Use the shortcuts -- Space+LMB/MMB/RMB to rotate/pan/zoom.

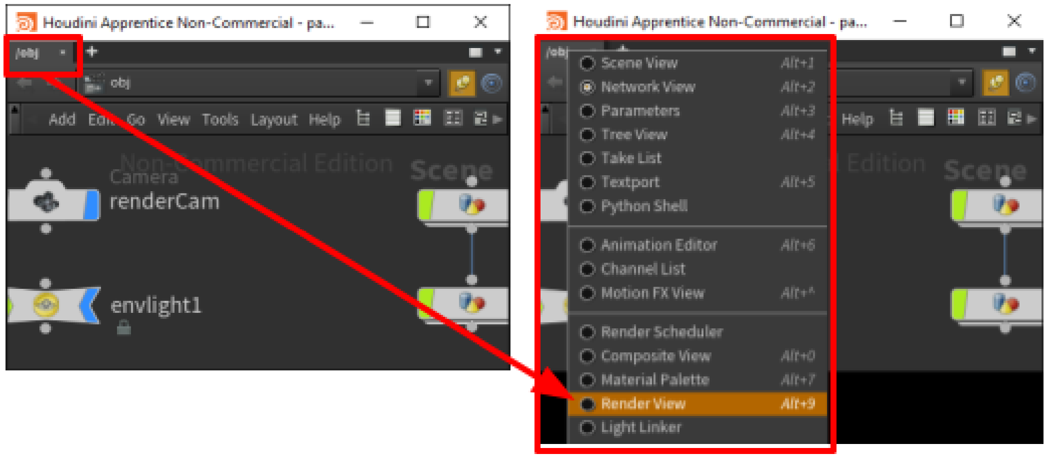
Render View

You can do IPR through the Render View. The best way to set this up is to create a new floating window and set it to be a Render View. You can do this by…

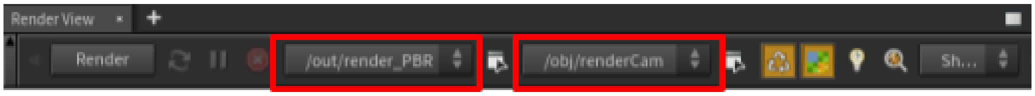
1. In the main menu, select Windows -> New Floating Panel


C:/Users/User/Desktop/Introduction to Materials in Houdini/houdini_p
File
Edit
Render
Assets
Windows
Help
Build
Create
Modify
Model
Poly
Box
Sphere
Tube
Scene View
Animation Edit
New Floating Panel Alt+Shift+W
Animation Editor Alt+Shift+F
Hscript Textport Alt+Shift+T
Python Shell Alt+Shift+P

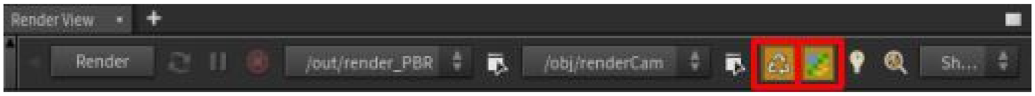
2. In the new floating panel, right-click the top-level tab (there should only be one) and set it to Render View



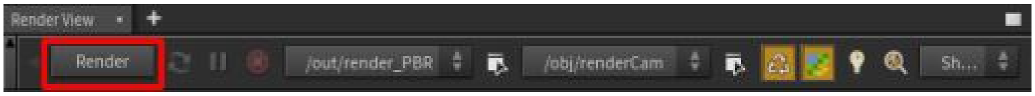
3. Make sure the proper Mantra node and Camera are selected in the toolbar



4. Make sure that Auto-Update and Progressive Rendering are select in the toolbar



5. Click the Render button

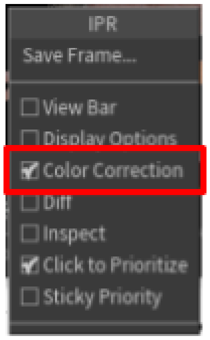


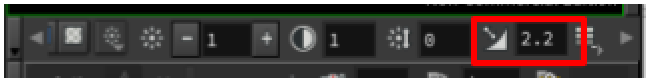
If at any point you want to STOP a render, hit the Stop button in the toolbar. Hitting this button will also stop renders from happening when something's updated.

If at any point you want to PAUSE a render, hit the Pause button in the toolbar. Pausing just pauses the render mid-stream -- you can continue it from the point it paused by hitting hte pause button again.

Gamma Correction

You can set gamma the same way both in the render view and the MPlay. Right click on the render and select Color Correction…





I don't know what's wrong with Houdini but it keeps defaulting the gamma to 1 instead of 2.2. If you click the icon it should automatically set to 2.2. It'll get reverted back the next time you render.

Materials

> **NOTE**: This section was adapted from the main document. It isn't worth adding further detail in here because most of the lessons focused on legacy shaders within Mantra/Houdini. Also, this isn't a renderer anyone uses for anything, so going in depth would be a waste of time. Much of the material stuff is pretty close to Blender/Arnold's material stuff.

Materials in Houdini 16 are handled in the /mat context (Materials) instead of the /shop context (SHader OPerations). The /shop context is still there but I'm not sure what it's used for.

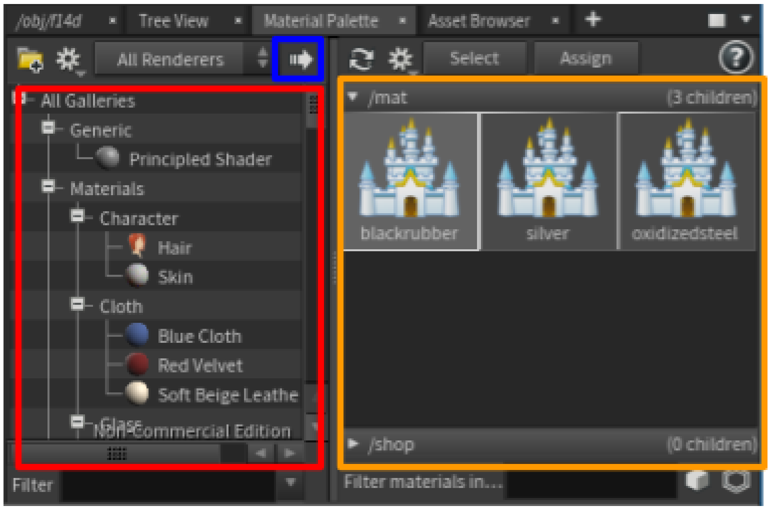
If you try to pull in any of the default materials via the Materials pane, they aren't allowed to go into a /shop context. But, if opening up legacy Houdini files all the materials are still in the /shop context.

Create Materials

You can create materials in the /mat context or via the Material's palette.

Materials Palette View

You can create materials by going to the Materials Palette view (in the same section as the network view when under the Build desktop).



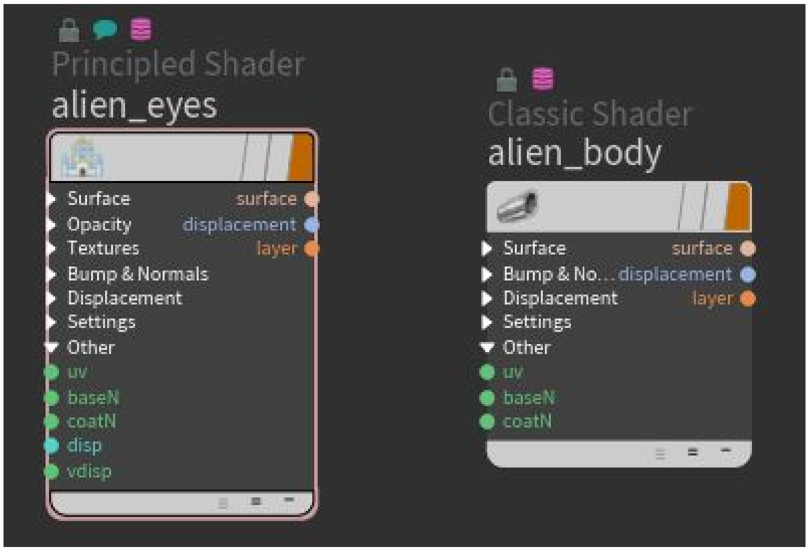
You can select from the left pane, and move over to the right pane by pressing the right arrow or by dragging-and-dropping over. These create the materials for you under the /mat namespace, which you can navigate to in the network view and tweak as you see fit.

> **NOTE:** If you look at bottom left-hand side of right pane, you can see /shop. But,the Materials Palette won't let you move stuff into the /shop namespace. I think the reason for this is that can be moved over is technically a Principled Shader, which can only appear under the /mat context.

A discussion on materials is outside the scope of this document, but these are all Principled Shader (can you load in custom shaders that aren't principled shaders???). Principled shaders are similar to Maya Arnold's shaders or Blender's principled shader: sub-surface scattering, specular, reflectivity, diffuse, etc.. -- all the stuff that makes it easy for artists to make materials.

Network View (/mat Context)

In the network view, you can switch to the /mat context and drop in Principled Shader nodes. If you want, you can also use the legacy Classic Shader here (called the Mantra Surface Shader in older versions of Houdini)...



If you notice, the interface here is very similar to Maya's hypershade instead of other networks inside of Houdini (e.g. dynamics or geometry networks). You can do things here like drop in perlin noise nodes and plug the output to the diffuse or specular.

> **NOTE**: Note the 3 small buttons on the bottom of the nodes… these are exactly the same as Maya's buttons to collapse/expand nodes.

Assign Materials

The following subsections describe how to assign materials.

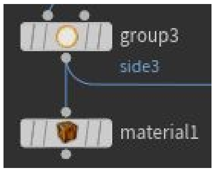
You can technically drag-and-drop the material from the Network View /mat context or the Material Palette view onto the object in the Scene view to assign a material, but that seems to not work a lot of the times.

If it does work, when you drop you'll be prompted with a dropdown that asks if you want to drop on the object or one of the groups within the object (if you dropped on a group?).

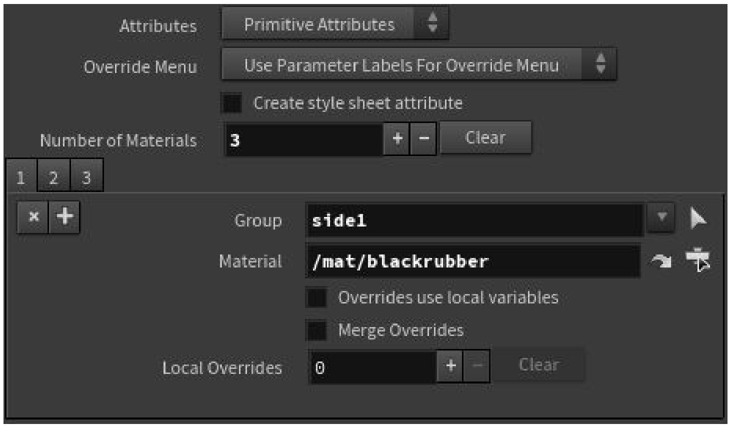
Partial Object (Groups within Object)

> **NOTE**: Need a refresher on groups? Check out the main Houdini document.

It's fairly straight-forward what to do with the Material node. Go into the Network view, into your geometry object, drop a Material node, and hook your geometry into it…



In the parameters pane, choose the components which you want the color to apply to. You can do so by clicking the arrow on the right of the Group. Once you've done that, select the material you want to use by clicking the right-most button next to Material (will open a selection menu of materials under /mat).



Attributes: Primitive Attributes

Override Menu: Use Parameter Labels For Override Menu

□ Create style sheet attribute

Number of Materials: 3

Group: side1

Material: /mat/blackrubber

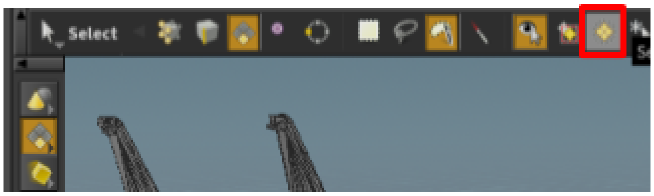
□ Overrides use local variables

□ Merge Overrides

Local Overrides: 0

Note that you aren't limited to one material. You can add multiple materials for different parts of the geo. In the example above, we have 3 different materials being added to 3 different parts of the geo.

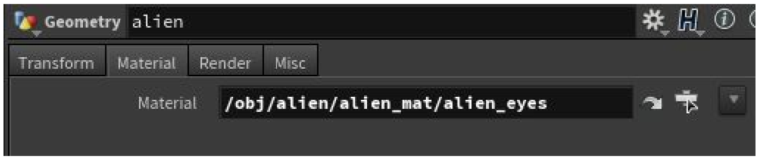
> **NOTE**: You typically would have these groups pre-defined via group nodes (especially for materials). See the Group Nodes section for more information.
>
> **NOTE**: If you click the arrow never to Group, it'll actually ask for a grouping rather than let you put in arbitrary components. You can turn this off by disabling the group selection toggle. You need to do this every time for Material nodes…



Full Object

In the network view (in the top-level /obj context), select the geometry object you want to apply the material to.

Simply go to the Material tab in the properties pane and select the material…



Geometry: alien

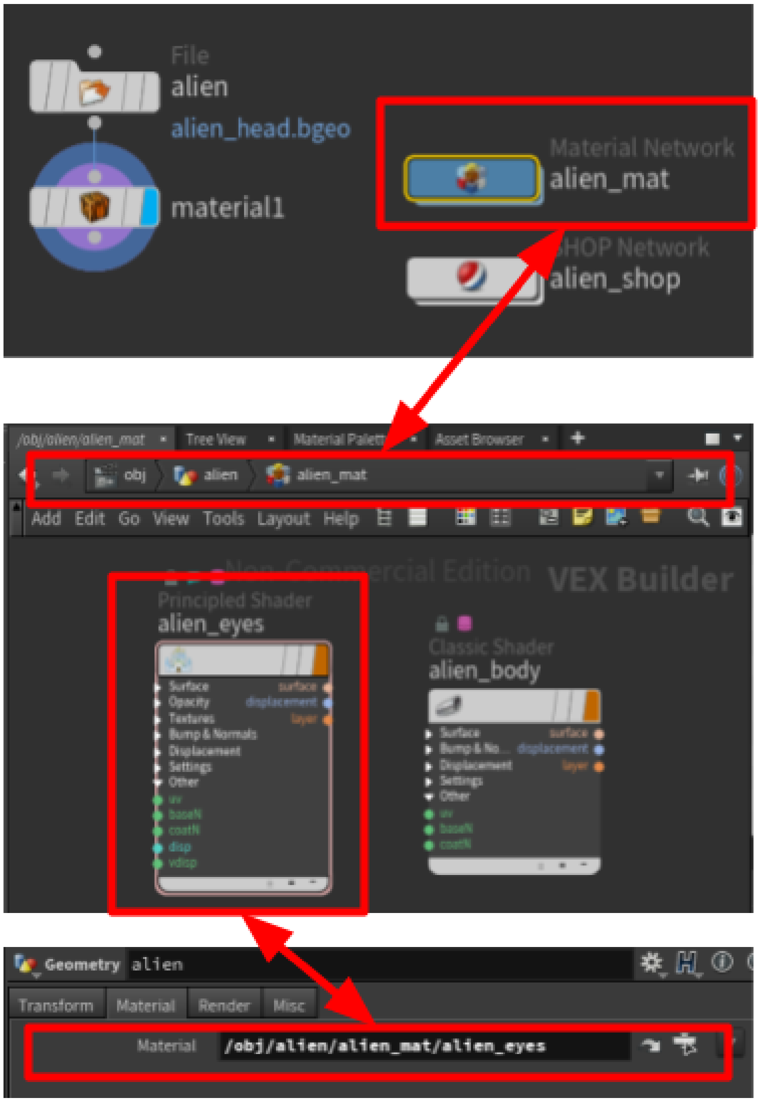
Material: /obj/alien/alien_mat/alien_eyes

NOTE: Applying materials at this level will ONLY APPLY TO PARTS OF THE OBJECT THAT DON'T HAVE MATERIALS ALREADY APPLIED. That means that if you've applied materials to certain groups inside the model (via the Material node), those polygons will keep those materials.

Internal Material Networks

You can set up a /mat or /shop context INSIDE your geometry node. There are a lot of reasons why you'd want to do this, but the most important is that you want you bundle your materials and your models together if you're working within a pipeline.

To do so, simply create a Material Network node (or a Shader Network node) inside your geometry node. Once you do that, you can dive into that node and create materials, then reference those materials from your geometry…



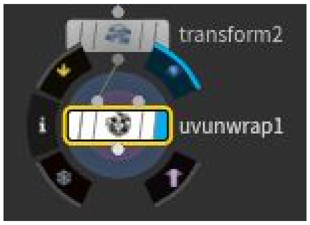
Material /obj/alien/alien_mat/alien_eyes

NOTE: Once you create an internal Material Network, it'll be viewable as a section along with /mat in the Material Palette view aswell.

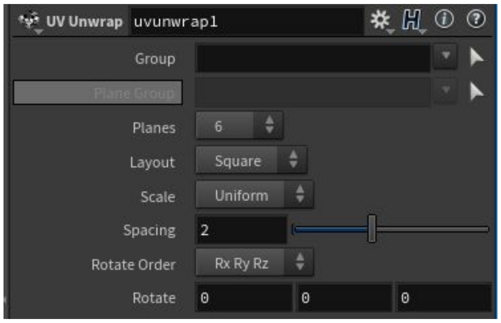
UV Unwrap

Attempts to generate UVs for the input.

You can add/fix UVs in the network view. Drop a UV unwrap node and connect whateer geometry you want to it…



You can choose how the UVs get applied…



UV Unwrap uvunwrap1

Group

Plane Group

Planes 6

Layout Square

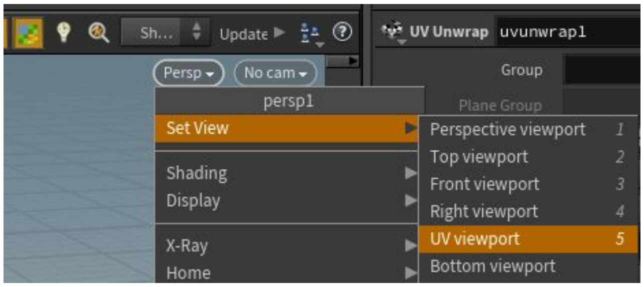
Scale Uniform

Spacing 2

Rotate Order Rx Ry Rz

Rotate 0 0 0

NOTE: To actually see the generated UVs, you can use the Scene view and switch your view from perspective to UV (shortcut is Space+5)…

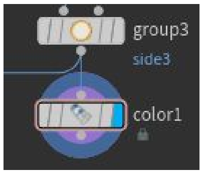


You can also switch to the geometry spreadsheet to see the UV attributes added to your points/vertices/faces/whatever.

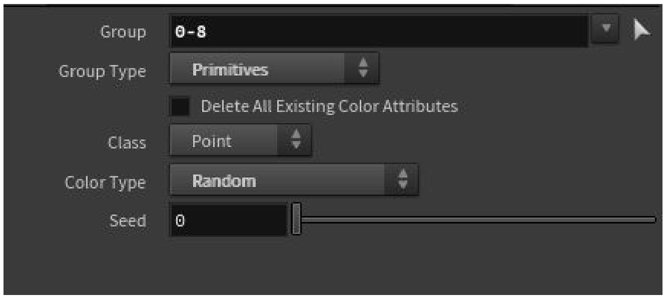
Color Nodes

You can directly apply color(s) to geometry components without going through materials or shaders. When you do this, it mixes the color you select with whatever material/shader is being applied to your geometry.

It’s fairly straight-forward what to do with a Color node. Go into your geometry node in the Network view and drop a Color node, then hook your geometry into it…



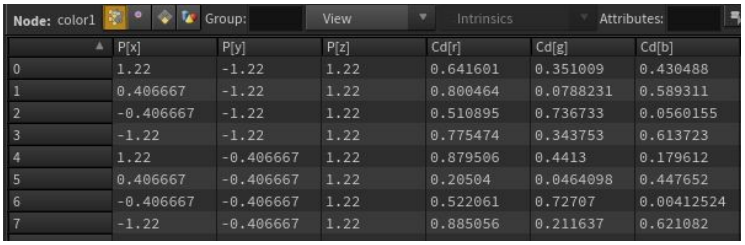
In the parameters pane, choose the components which you want the color to apply to. You can do so by clicking the arrow on the right of the Group.



Group 0-8
Group Type Primitives
□ Delete All Existing Color Attributes
Class Point
Color Type Random
Seed 0

The other parameters here should be straightforward. Class determines what component type to apply the geometry to (e.g. faces or points or edges or whatever). Color type can be set to Constant or Random (or a few others). Everything after that point will be dependent on what color type was set to.

> **NOTE**: If you go into the geometry spreadsheet, you can see the color show up under attributes for whatever component type you used for Class (Cd is the name of the attribute).



	P[x]	P[y]	P[z]	Cd[r]	Cd[g]	Cd[b]
0	1.22	-1.22	1.22	0.641601	0.351009	0.430488
1	0.406667	-1.22	1.22	0.800464	0.0788231	0.589311
2	-0.406667	-1.22	1.22	0.510895	0.736733	0.0560155
3	-1.22	-1.22	1.22	0.775474	0.343753	0.613723
4	1.22	-0.406667	1.22	0.879506	0.4413	0.179612
5	0.406667	-0.406667	1.22	0.20504	0.0464098	0.447652
6	-0.406667	-0.406667	1.22	0.522061	0.72707	0.00412524
7	-1.22	-0.406667	1.22	0.885056	0.211637	0.621082

Lights

> **NOTE**: This section was copied over from the main document. There’s nothing more worth adding here because this isn’t a renderer anyone uses for anything.

Houdini (mantra) provides a bunch of lights that are very similar to Arnold and Maya lights…

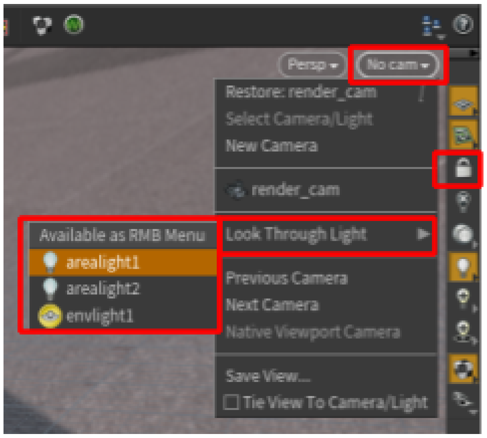
- Point Light → emits light equally in all directions
- Area Light → a rectangle/disk/whatever that emits light
- Geometry Light → similar to having a polygon mesh in Arnold that you set to emit light

- Distant Light → parallel rays, as if the sun were shining
- Environment Light → provide an HDR image and get light emitted based on that -- similar to mentalray's image based lighting or Arnold's skydome light
- Sky Light → how is this different from the environment light above???

Add them to your scene and move them around just like any other scene node. You can find them in the Light shelf (2nd shelf set in the build desktop) or via the tab menu.

An easy to position a light is through the scene view. You can make it so as if you're looking through the light, and orient it as you move around your scene. To do this…

1. Select the light in the camera dropdown on the top-right of the scene view (just below the top toolbar)
2. Toggle the lock camera/light lock button on
3. Orient your view around, just as normally would
4. Toggle the lock camera/light lock button off



Almost all lights come with many properties that are similar to Arnold. They probably work just like Arnold does. The properties panel has sections for increasing the light sampling (if stuff comes out too grainy) and setting intensity/exposure property and a bunch of other stuff.

Intensity and exposure seem to be the main control properties here. Here's the description of it from my Arnold notes…

Intensity Intensity of your light. This works the same way as a regular Maya light. The higher your intensity is, the more light will be given off.

Exposure

Intensity of your light. This is different from the above setting in that it’s measured in f-stops (just like real photography). It’s designed to help you interact with real cinematographers. For example, the cinematographer may come and ask the artist to increase something by half a stop -- you can do that directly here in this setting.

This is exponential… each time you go up by 1, it doubles the intensity of your light.

NOTE: This is linked to the intensity in that the exposure uses the current intensity value as the basis for its setup.

Houdini 16 VEX

Introduction

VEX is a proprietary programming system that's used for a bunch of different things within Houdini: shaders, deforming geometry, particle movements, etc..

Some facts about VEX…

- VEX is inspired by RenderMan's shading language
- VEX has its own compiler (called vcc)
- VEX can be "written" with nodes through VOPs (Vex OPerators)
- Prior to VEX, Houdini provided a scripting system called Hscript

This document will focus on using VEX through the VOPs interface in Houdini. That means that we'll be programming by dragging-and-dropping nodes and connecting them up. We won't writing programs in VEX.

For more information on writing VEX, see the VEX language reference…
http://www.sidefx.com/docs/houdini/vex/lang

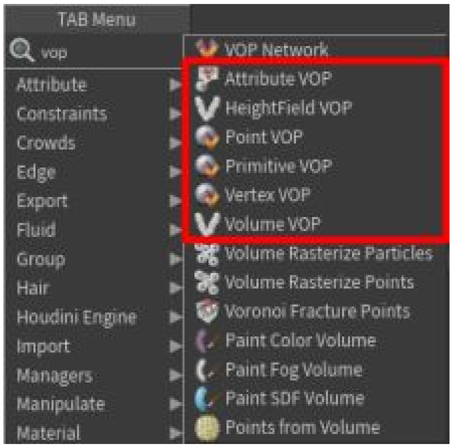
For more of an introduction on VEX/VOPs, see http://www.tokeru.com/cgwiki/index.php?title=Houdini_Vops&oldid=2317

Create

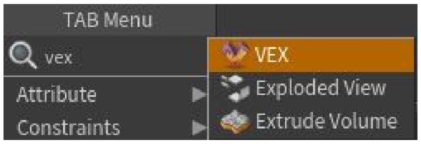
It looks like you can create VOP nodes in pretty much any context other than directly in /obj…

In the network view, go to the Tab Menu and search for “VOP”. You’ll get a list of VOP node types that you can use.

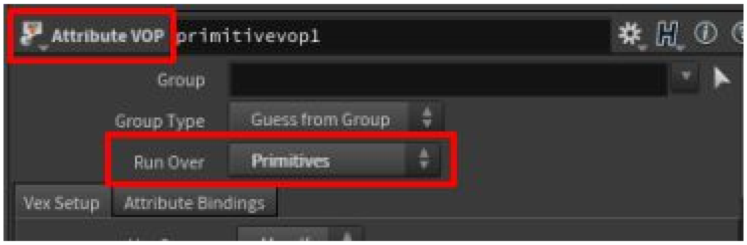
For example, if you go inside one of your geometry nodes and use the Tab Menu to search for VOP, here’s what you’d see…



NOTE: You can also add in a node where you directly type in VEX script via the VEX node…



Pretty much all the VOP nodes listed above either boil down to being Volume VOP or Attribute VOP. For example, if you choose Primitive VOP, you’ll actually get an Attribute VOP node but the Run Over property will automatically be set to Primitives.


Attribute VOP primitivevop1

Group

Group Type Guess from Group

Run Over Primitives

NOTE: Remember that "primitive" is Houdini's term for polygons/faces.

In the case of Attribute VOP, the Run Over property defines what level of geometry that the VEX operations will be processing. Your VEX operations will update/add/delete attributes on that geometry based on however you defined your logic/operations.

Remember that most of Houdini is about attributes being updated/added/deleted as they flow through the system. For example, if you're working with points, your points may have an attribute called…

- P → a vector that describes their location in space
- N → a vector that describes their normal (if they have one)
- Cd → a 3D vector used for RGB coloring (if they're colored)
- uv → a 2D vector that describes the UV coordinate for the point (if mapped to UV)
- etc..

NOTE: See http://www.sidefx.com/docs/houdini/model/attributes for more common attributes

In addition to these common attributes, you may have attributes that get used by solvers to do computations on the points (e.g. an attribute called v for velocity) or custom attributes that you add yourself (e.g. maybe the artists need attributes to do something or some other Houdini person will use them to do some additional calculations).

Export

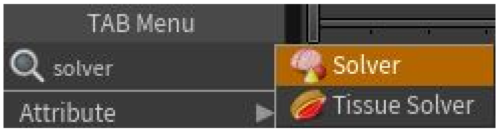
You can export your Attribute VOP node as a Houdini Digital Asset by first putting it in its own Subnetwork and then exporting that subnetwork as a Houdini Digital Asset.

See the Subnetwork section of the main Houdini document to see how to do this.

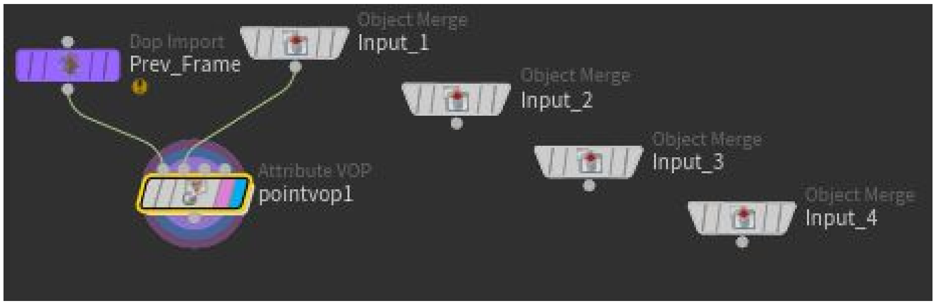
NOTE: It looks like previous versions of Houdini let you do this straight from the VOP node (instead of having to create a subnetwork). The option seems to be missing now.

Solve

In certain cases, you will want the computation happening in your Attribute VOP node to make use of previous frame's geometry. You can do this by putting your Attribute VOP inside of a Solver node…



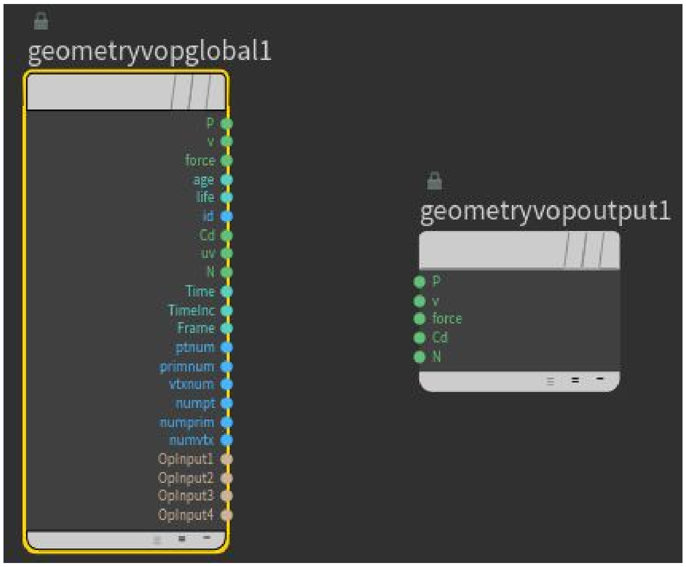
Inside of your Solver, you'll have access to the previous frame's geometry as well as any inputs you fed into the solver node. Simply feed your previous geometry and your current geometry into your VOP node and do whatever computations you need to...



Inputs and Outputs

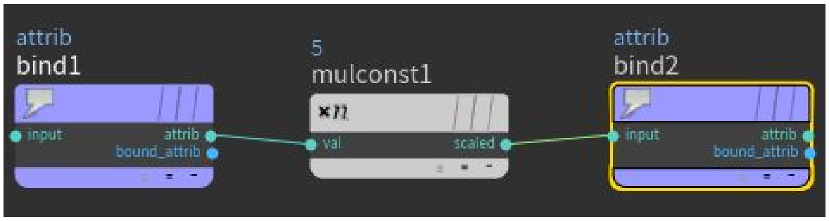
Attributes

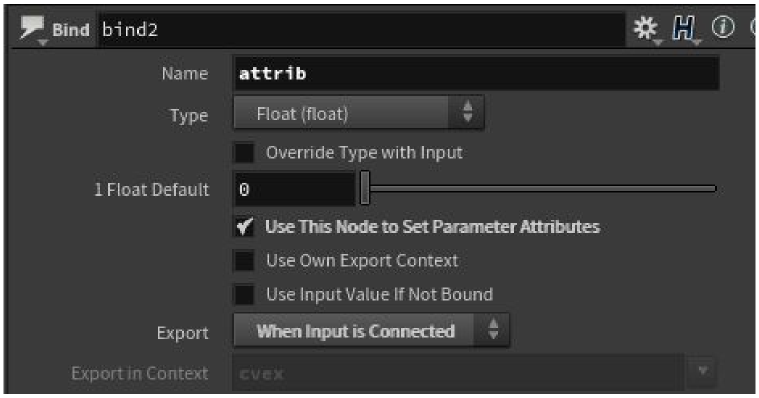
By default, when you create an Attribute VOP node, you'll get 2 nodes that will let you import/export common attributes...



You can drop in a Bind node for any additional attributes that you want to import/export. Just make sure that you…

1. set the Name property to the name of the attribute
2. set the Type property to the correct type the attribute is set to
3. set the Export property to When Input is Connected (the Bind node will gain an input)





Bind bind2

Name attrib

Type Float (float)

☐ Override Type with Input

1 Float Default 0

☑ Use This Node to Set Parameter Attributes

☐ Use Own Export Context

☐ Use Input Value If Not Bound

Export When Input is Connected

Export in Context cvex

NOTE: Notice how we dropped 2 Bind nodes for the same attribute (named attrib in the example). If you have multiple Bind nodes pointing to the same attribute, you can only set the parameter on the first Bind node. All other Bind nodes for the attribute will copy those parameters.

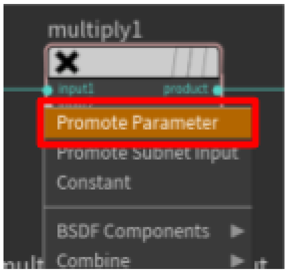
If you want to take the parameters from a Bind node that isn't the first for the attribute, check the 'Use This Node to Set Parameter Attributes' checkbox.

NOTE: If you set the attribute to one that's already being imported/exported by the 2 default nodes, your properties are going to be disabled. That's because the 2 default nodes are internally doing the same thing to give you access to these attributes, and they already have the properties set.

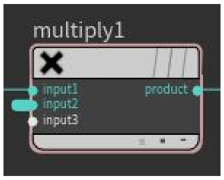
Parameters

You can expose the inputs of a node my MMB as a Parameter on the Attribute VOP node that it sits in.

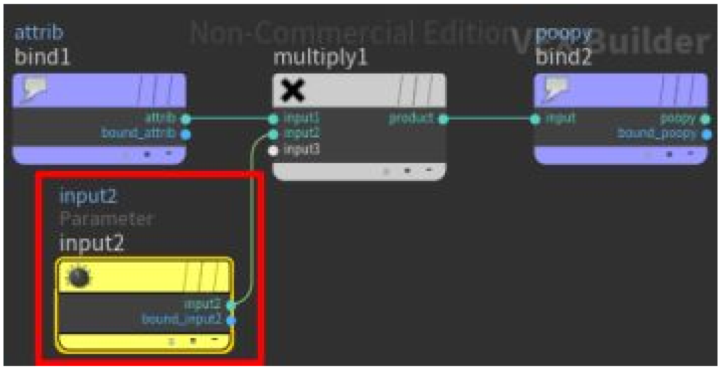
To do this, begin by MMB clicking on the input and choosing Promote Parameter. When you do this, a little nubby will appear for that input…



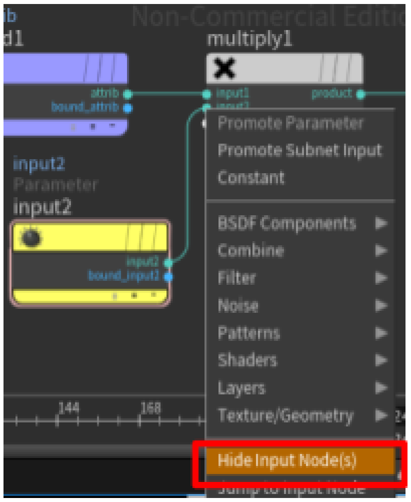
→



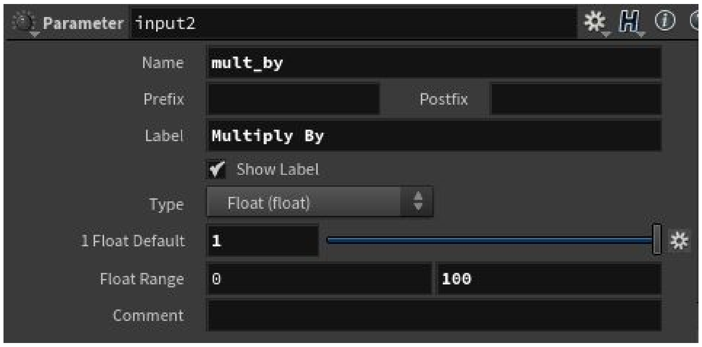
Double-click on the nub to expand out the hidden Parameter node.



NOTE: If you want to turn this back into a nub, MMB click on the input and choose Hide Input Nodes...



In the Parameter node, you can set the name of the parameter, the label that shows up for the parameter, the default value of the parameter, and the range of inputs for the parameter (will show up as a slider)...



Parameter input2

Name: mult_by

Prefix: Postfix:

Label: Multiply By

☑ Show Label

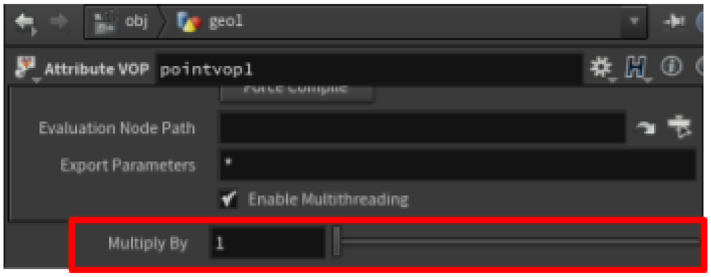
Type: Float (float)

1 Float Default: 1

Float Range: 0 100

Comment:

If you go back out to your Attribute VOP node, you'll see your new parameter show up in the parameter window…



obj geo1

Attribute VOP pointvop1

Force Compile

Evaluation Node Path:

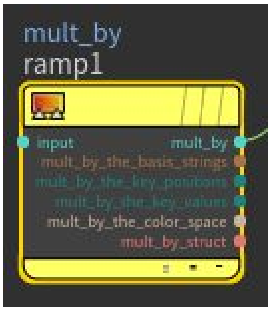
Export Parameters: *

☑ Enable Multithreading

Multiply By: 1

You can re-organize your Attribute VOP node's parameters via the Edit Parameter Interface. For more information on this, see the Subnetworks section in the main Houdini document.

NOTE: We dealt with a basic parameter node, but you also have the option of using a Ramp Parameter node. A Ramp Parameter node makes in some input and changes it so that it fits some ramp property on your Attribute VOP node (it'll show up as an actual ramp vs just a field).

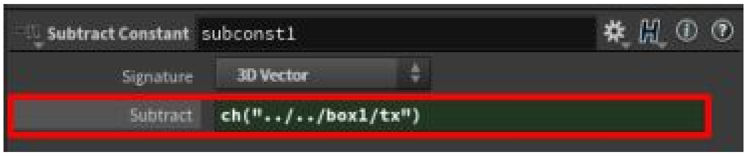


A couple of things to be aware of…

1. the Ramp Type option should almost always be set to Spline Ramp.
2. your input has to be normalized to be between 0 and 1 (I think you can use a Fit Range node to normalize to a 0 to 1 range before feeding in)

Relative References

You can make a relative reference to the node of any other parameter, even if it's outside of the Attribute VOP node that you're working in…



Subtract Constant subconst1

Signature 3D Vector

Subtract `ch("../../box1/tx")`

In the example above, we're going outside of our Attribute VOP node and instead accessing the tx parameter of a node called box1. This is perfectly legal to do.

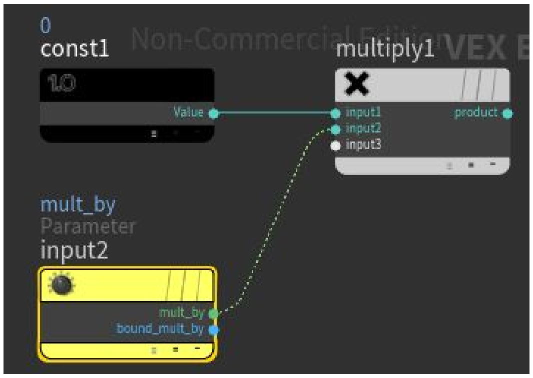
For more information on setting relative references, see the Parameter View section in the main Houdini document.

NOTE: If you want to reference an attribute in the expression, you can do so by prefixing the attribute's name with a @. See https://houdinitricks.com/quicktip-new-attribute-syntax-in-houdini-15/

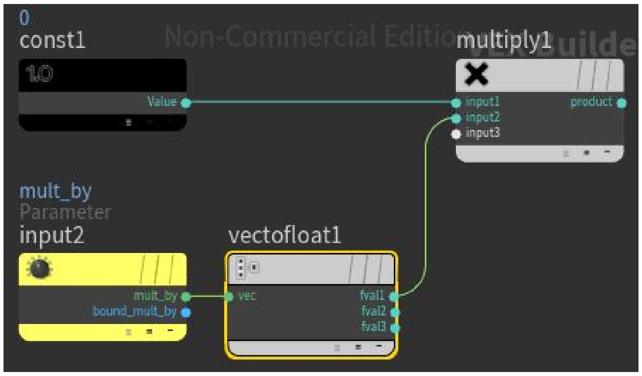
Connections

If the connect is a solid line, you're fine.

If the connection shows up as a dotted line, it means that there's some implicit conversion happening between the output and the input. For example, if the node you're working with expects a float but you're giving it a vector of floats, the line will show up as dotted…

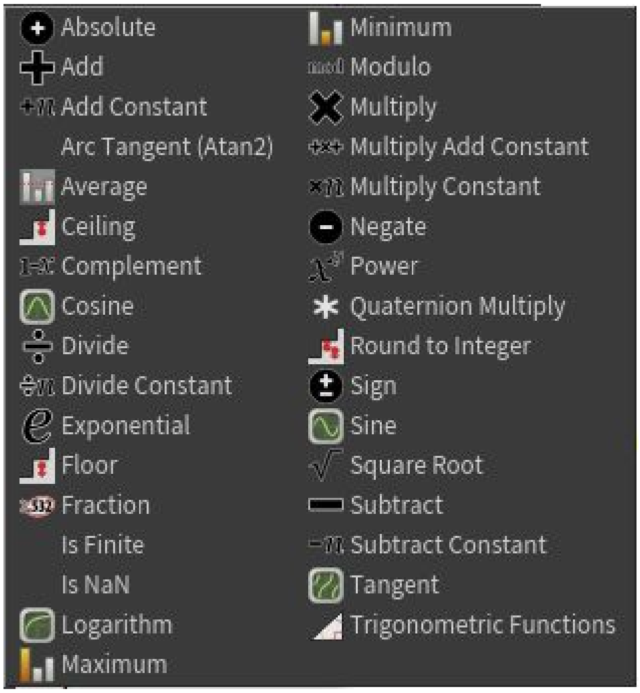


It's best to explicitly convert the data to the type expected rather than relying on these implicit conversions. In the example above, we can convert our vector to a float using a Vector to Float node…



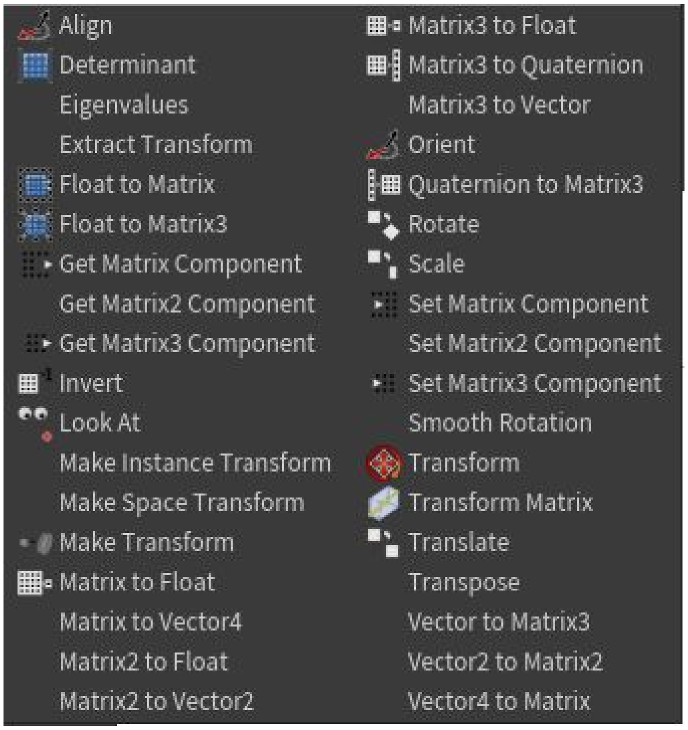
Common Nodes

Arithmetic/Aggregation



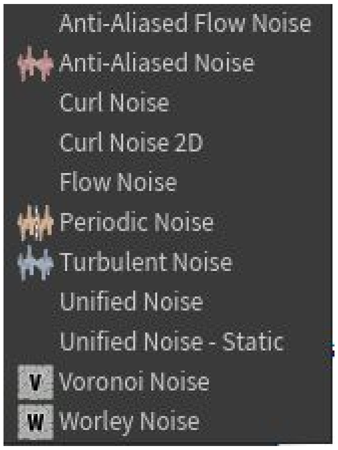
Transform

Important nodes here are Look At, Translate, Scale, Rotate

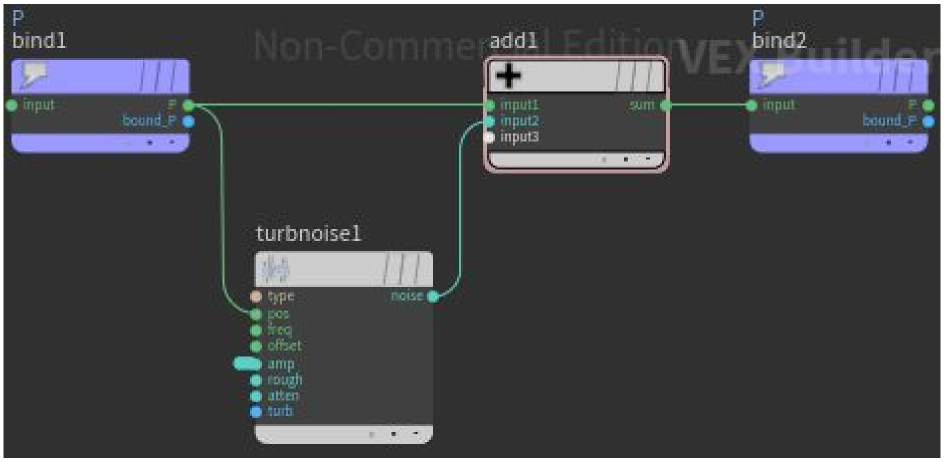


Noise

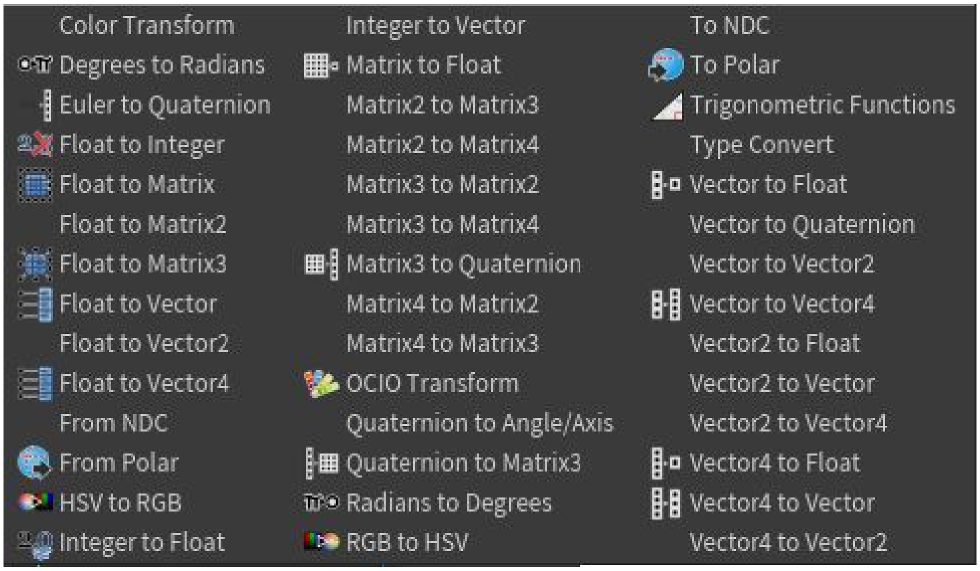
Import node here is Turbulent Noise.



Make sure you look at the documentation for the noise before using it. If you're feeding in positions, the output is the position offset -- you have to add the offset to the original value to get the final result…



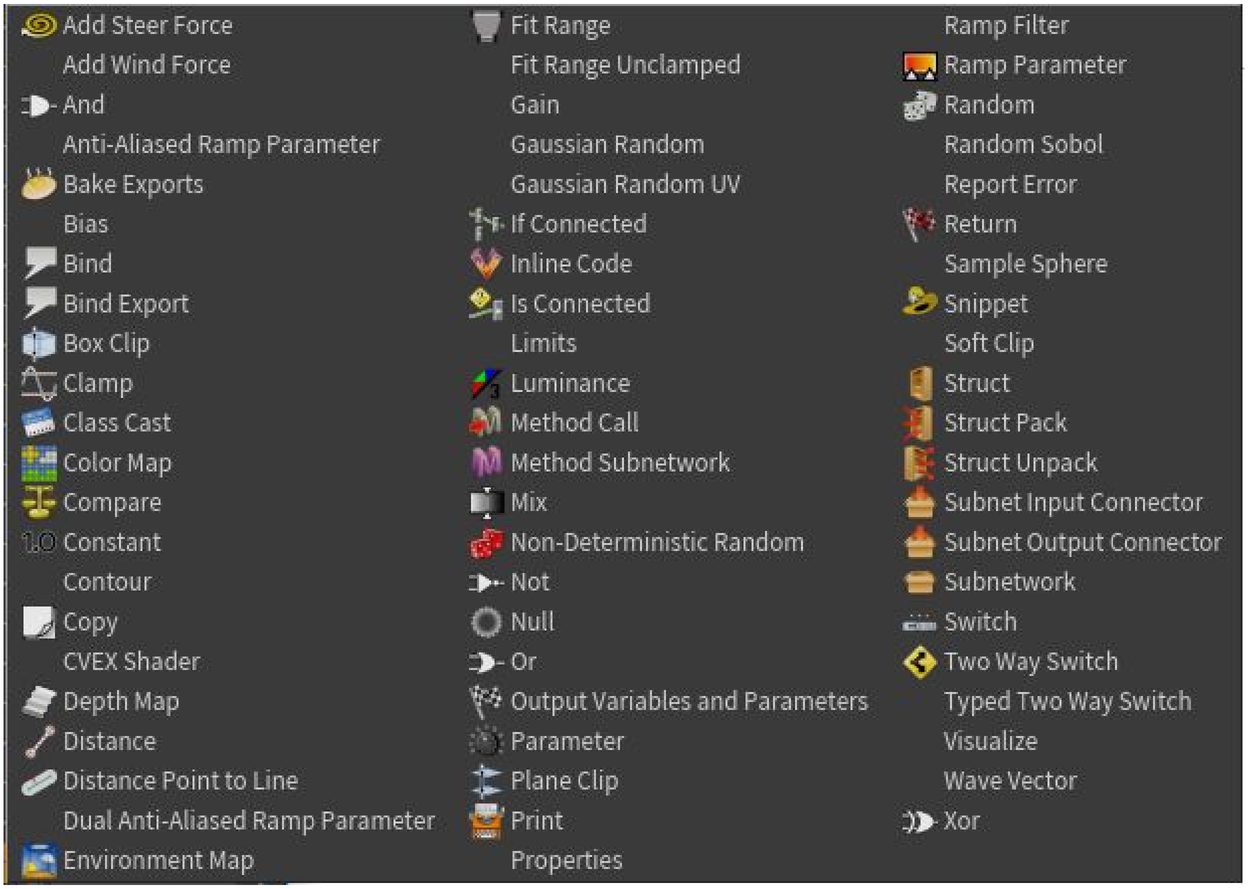
Conversions



Utility

Important nodes here are the ones used for comparison and fitting (shrinking down values to some range)…

- Comparison: Switch, Compare
- Fitting: Fit Range, Fit Range Unclamped


Add Steer Force
Add Wind Force
And
Anti-Aliased Ramp Parameter
Bake Exports
Bias
Bind
Bind Export
Box Clip
Clamp
Class Cast
Color Map
Compare
Constant
Contour
Copy
CVEX Shader
Depth Map
Distance
Distance Point to Line
Dual Anti-Aliased Ramp Parameter
Environment Map
Fit Range
Fit Range Unclamped
Gain
Gaussian Random
Gaussian Random UV
If Connected
Inline Code
Is Connected
Limits
Luminance
Method Call
Method Subnetwork
Mix
Non-Deterministic Random
Not
Null
Or
Output Variables and Parameters
Parameter
Plane Clip
Print
Properties
Ramp Filter
Ramp Parameter
Random
Random Sobol
Report Error
Return
Sample Sphere
Snippet
Soft Clip
Struct
Struct Pack
Struct Unpack
Subnet Input Connector
Subnet Output Connector
Subnetwork
Switch
Two Way Switch
Typed Two Way Switch
Visualize
Wave Vector
Xor