

# Houdini 16

## [Basic Interface](#)

[Toolbar/Shelf Offscreen Icons](#)

[Toolbar/Shelf/Menu Manipulation](#)

[Panel Manipulation](#)

[Desktops](#)

[Reloading Desktops](#)

[Saving Desktops](#)

[Projects](#)

## [Contexts](#)

### [Scene View](#)

[Multiple Viewports](#)

[Viewport Orientation](#)

[Creating Objects](#)

[Manipulating Objects and Components](#)

[Selecting Components](#)

[Selecting Visible Geometry Only](#)

[Selecting by Group](#)

[Converting Selection between Component Types](#)

[Growing/Expanding Selection](#)

[Handle Alignment](#)

[Handle Size](#)

[Manipulating Camera](#)

[Frame up on Selected Object](#)

[Frame up on All Objects](#)

[Revert to Home Orientation](#)

[Changing Viewport Object Ghosting](#)

[Changing Viewport Lighting/Shading](#)

[Display Options](#)

## [Network View](#)

[Manipulating View](#)

[Nodes](#)

[Node Flags](#)

[Node Information](#)

[Coloring Nodes](#)

[Positioning Nodes](#)

## [Stepping Into Nodes](#)

[Creating Netboxes](#)

[Creating Notes](#)

[Creating Nodes](#)

## [Parameter View](#)

[Value Ladder](#)

[Setting Relative References](#)

[Setting Dynamic References](#)

## [Geometry Spreadsheet View \(Attributes\)](#)

[Manipulating Attributes / Moving Around Attributes](#)

## [Scene Scale](#)

## [Subnetworks](#)

[Custom Inputs / Parameters](#)

[Custom Outputs](#)

[Exporting \(Digital Assets\)](#)

## [Importing/Exporting Assets](#)

[Correcting Imported Geometry](#)

## [Component Selection and Manipulation](#)

[Selecting Primitives \(Faces\)](#)

[Selecting Edges](#)

[Selecting Points](#)

[Converting between Primitive/Edge/Point Selections](#)

[Scale/Rotate/Translate Tools](#)

## [Group Nodes](#)

[Group \(Create component groups\)](#)

[Group Combine](#)

## [Creating Geometry](#)

[Interactive Modeling \(Scene View\)](#)

[Non-Interactive Modeling \(Network View\)](#)

[Creating Primitives](#)

[Importing Geometry](#)

## [Geometry Nodes](#)

[PolyExtrude \(Extrude faces/edge\)](#)

[PolyFill / PolyCap \(Fill holes\)](#)

[PolyBridge \(Bridge between edges\)](#)

[Edge Loop](#)

[Blast \(Delete faces and points\)](#)

[Dissolve \(Delete edges\)](#)

[Subdivision](#)

[Normal \(Generate or correct normals\)](#)

[Utility Nodes](#)

[Merge \(Combine geometry/outputs\)](#)

[Copy and Transform](#)

[Material Nodes](#)

[Color](#)

[Material](#)

[UV Unwrap \(Generate or correct UVs\)](#)

[Light and Camera Nodes](#)

[All Light Nodes](#)

[Camera](#)

[Render Nodes](#)

[Mantra](#)

[Keyframe Animation](#)

[Setting Keyframes](#)

[Animation Editor](#)

[Scripting References](#)

[Physics Animation \(Dynamics\)](#)

[Scene Scale](#)

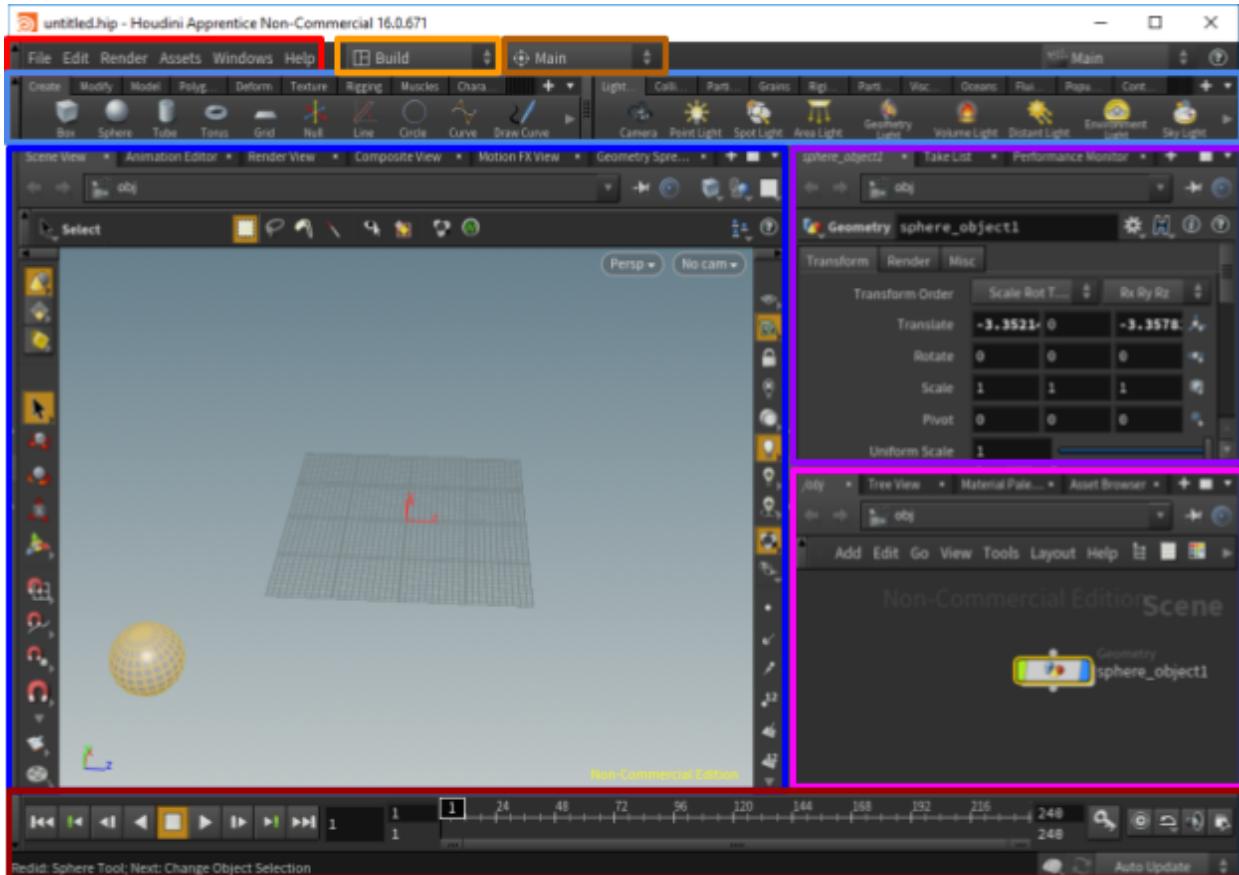
[Creating Dynamics for Objects](#)

[Setting Gravity](#)

[Setting Physics Properties for Objects](#)

[Collision Geometry](#)

# Basic Interface



**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.

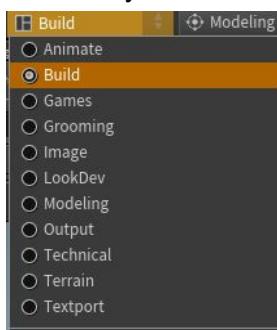
## Main Menu

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

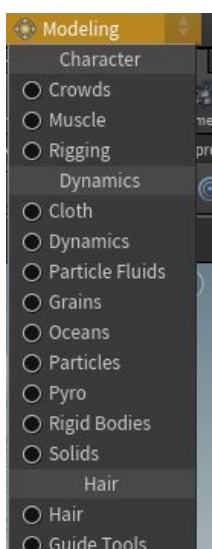
Desktops are essentially layouts. They're similar to workspaces in Maya 2017. The Build layout seems to be the default -- this is probably similar

to the Maya Classic workspace.



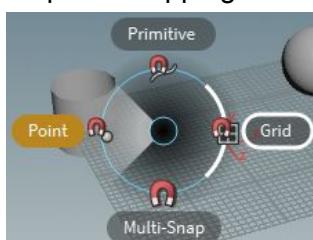
## Radial Menu

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...

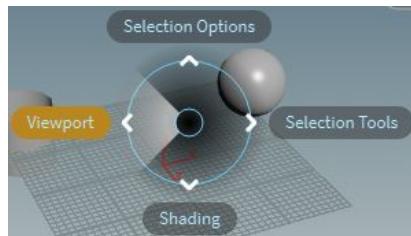
X opens snapping functions



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 it is you have selected in the viewport / network.

### Network View

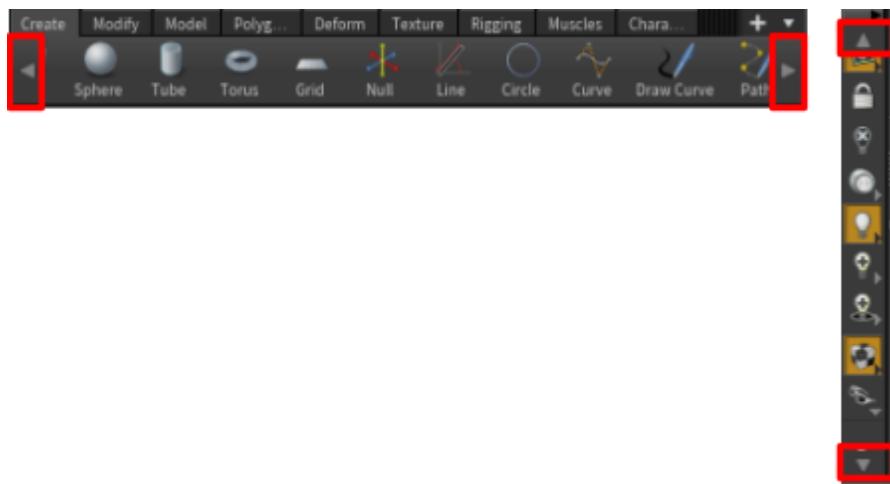
This is like Maya's node editor. When you select something in here, it'll also get selected in the viewport (and vice versa).

### 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...

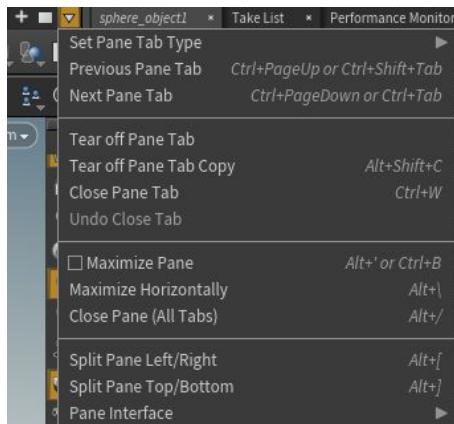


The middle portion of the divider bar (the dotted portion) can be click+dragged to expand/contract the panel. You can also just click it (instead of click+drag) to swap the positions of the panels.

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).

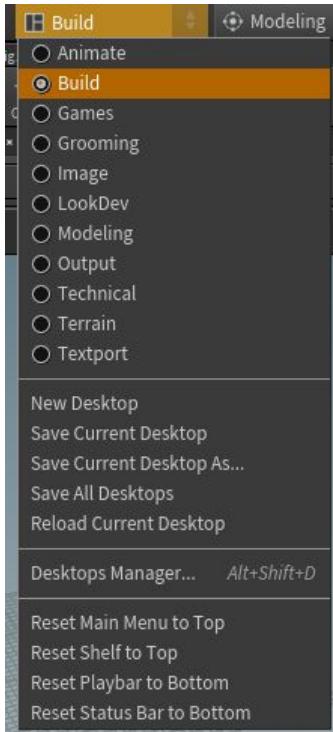
The top-right of a panel contains 3 buttons.

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.



## 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.

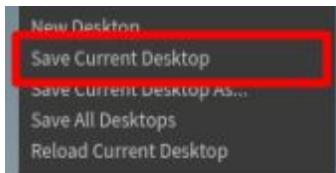


## Saving Desktops

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

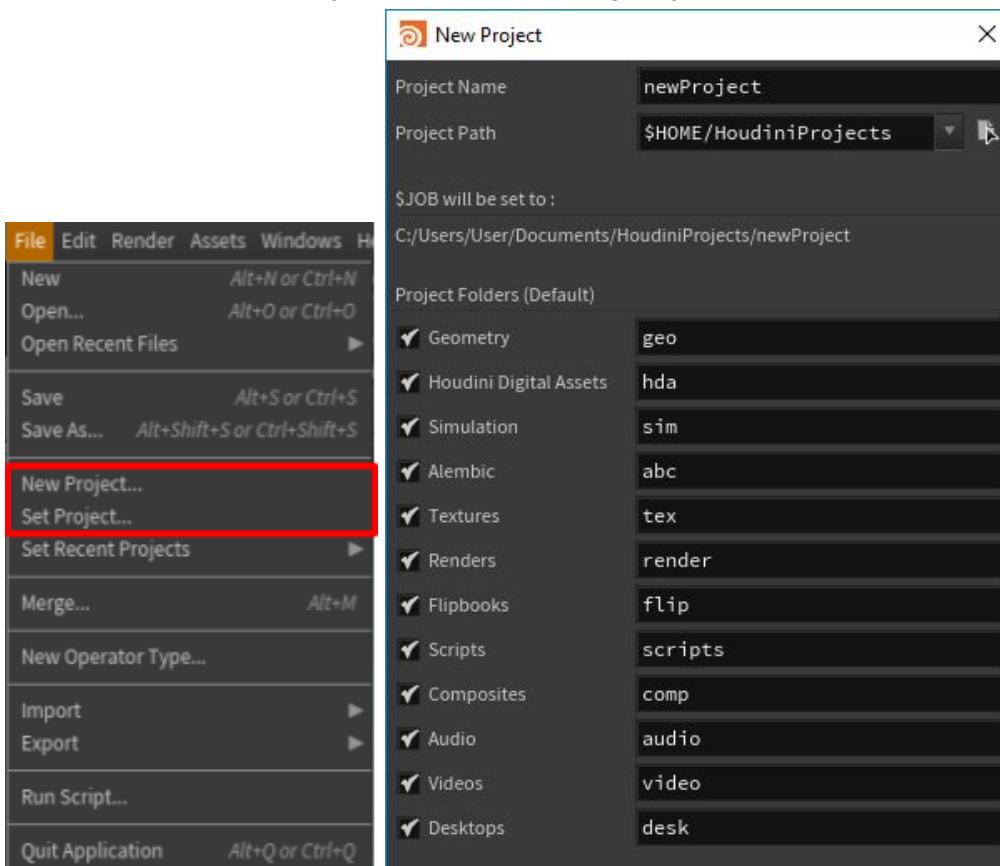
blowing away all your preferences..



## 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...



## 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...

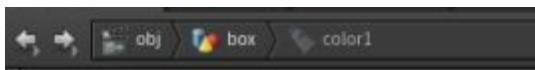


- ch / chops  
Channel Operators
  - Used for manipulation of raw data (e.g. movement of objects) and some audio stuff.
- img / cops  
Compositing Operators
  - Houdini offers an almost Nuke-like compositing package. All of that is container here.
- obj  
????????????
  - This is where your scene / shapes 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...

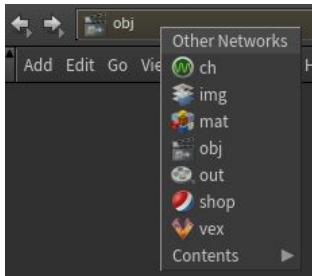
- Some contexts aren't listed here. For example, if you're working with particles, those are called pops (particle operators). Another one is dops: dynamic operators.
- Certain contexts are independent of each other, while other contexts are crossed together (information can flow between them).

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

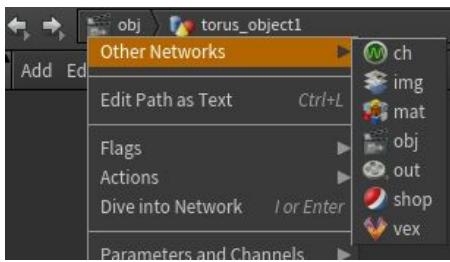


Depending on the view that you're in, you'll be able to switch to different contexts. Some views may only support a subset of contexts (e.g. scene view), while others may support all contexts (e.g. network view).

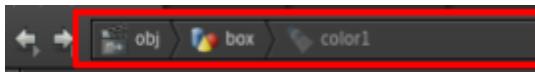
If you're at the top-level, you can switch to contexts by left-clicking on the top breadcrumb...



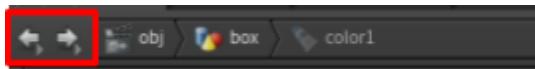
Another way to switch between contexts is to right-click on the top breadcrumb (doesn't matter if you're further down in the hierarchy) and go to Other Networks...



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

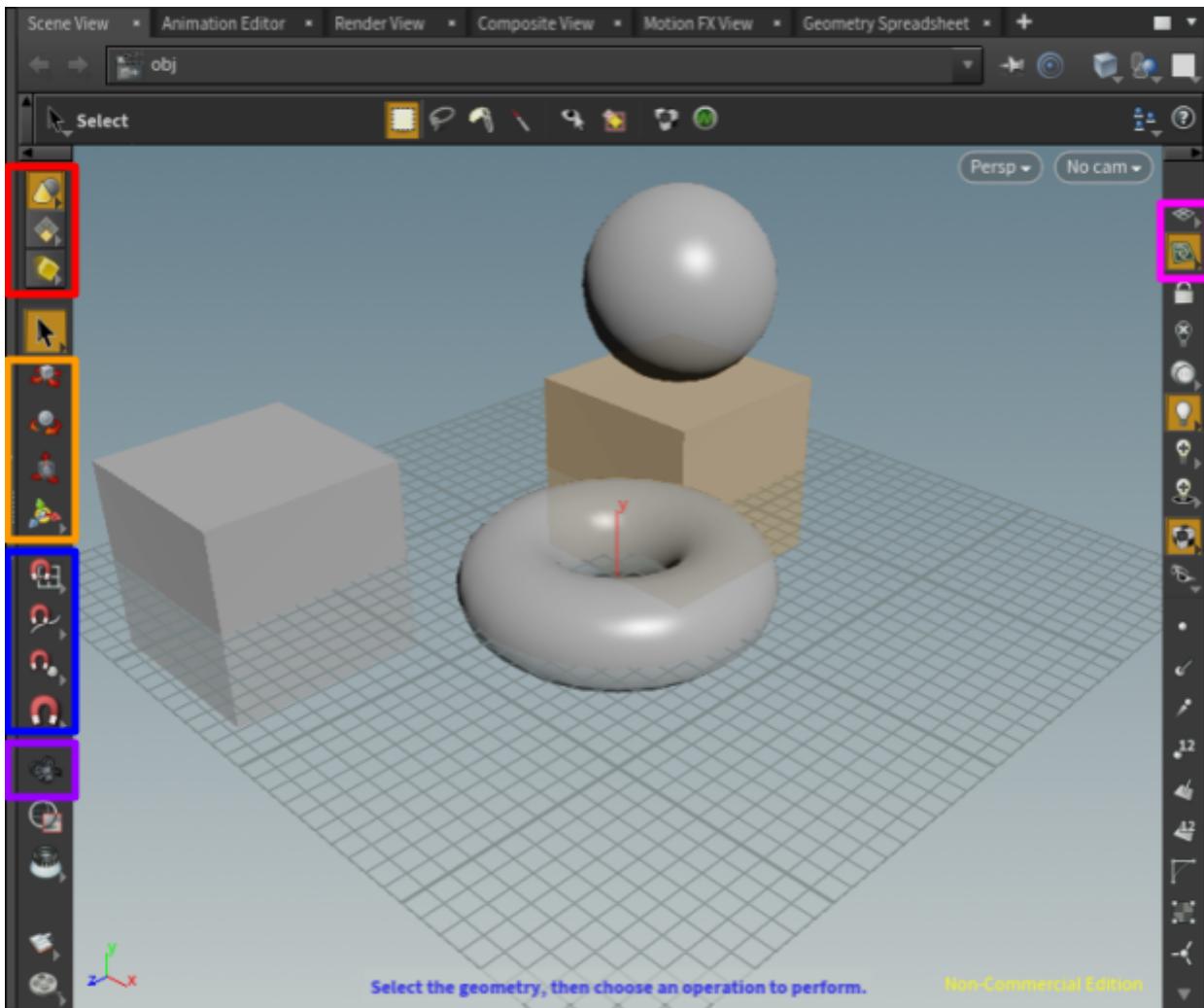


And... just like a web browser, you can also use the back/forward buttons to move between your history of context switches...



## Scene View

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



**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.

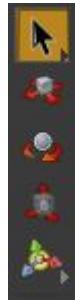
Button 2 is component selection (object must be selected first).

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

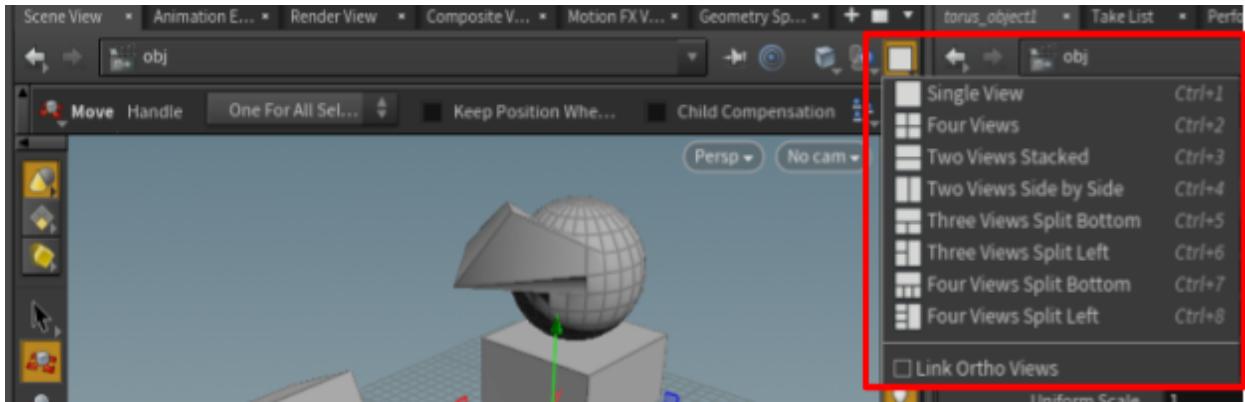
The first button shows the construction plane. It's good to have this on because when you're dropping objects on the viewport they'll be aligned to this plane.

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

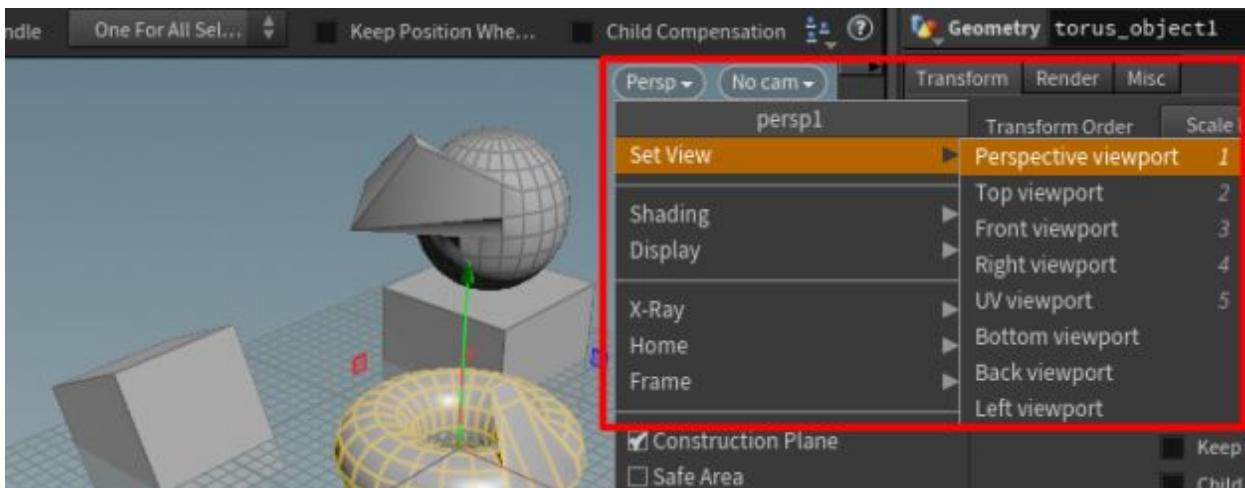
If you want multiple viewports in your scene view (just like you have in Maya), you can use the the viewport dropdown on the upper-right of the panel...



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...



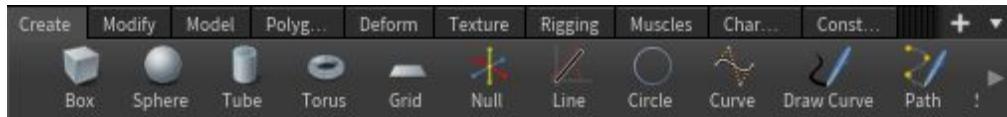
Just like Maya, top/front/right views are orthographic, meaning you won't be able to tumble them.

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

# Creating Objects

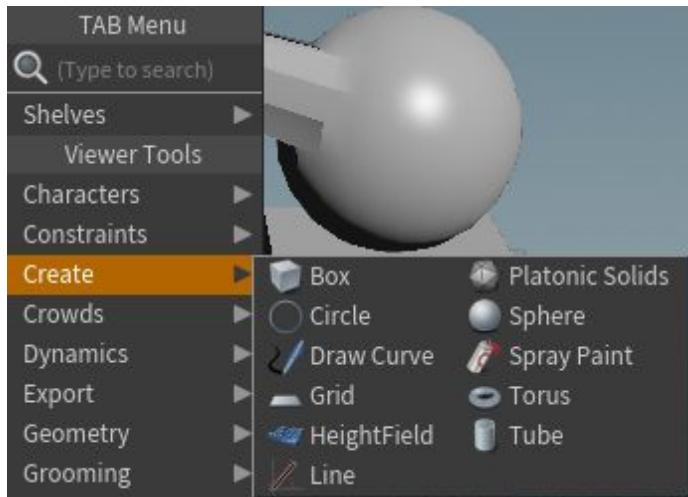
You can create objects either through the...

1. create shelf

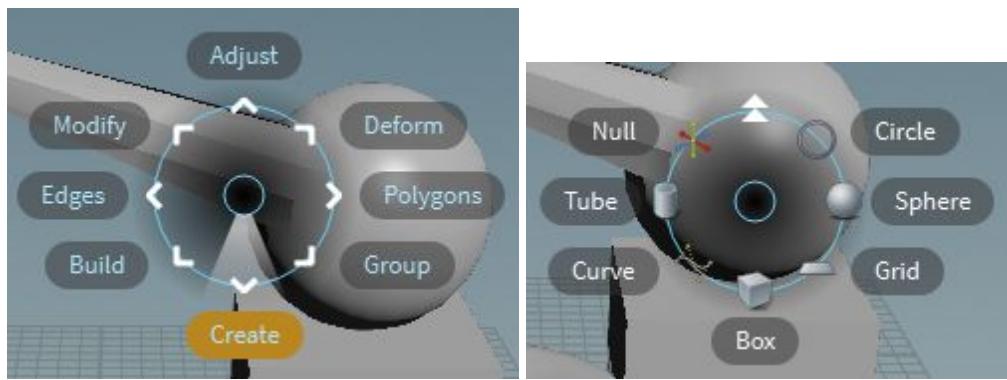


**HINT:** Hold CTRL when clicking to have it show up at origin vs interactively placing it in the viewport

2. tab menu



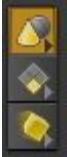
3. radial menu (if radial menu is set to Main or Modeling) via Create



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...

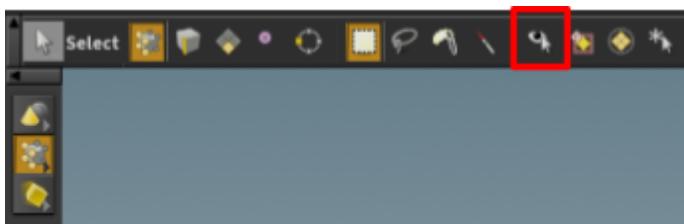


Button 1 is object selection.

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

**HINT:** When you're doing marquee selections, be mindful of the 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.

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.

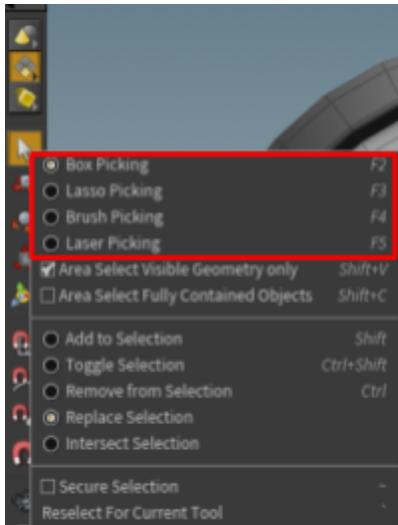
## Selecting Components

You can select components just like you do in Maya.

Hold Ctrl and click to remove a selection.

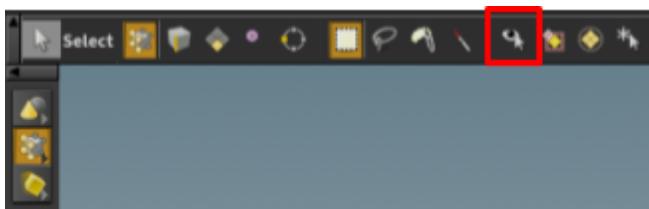
Hold Shift and click to add a selection.

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.



### Selecting Visible Geometry Only

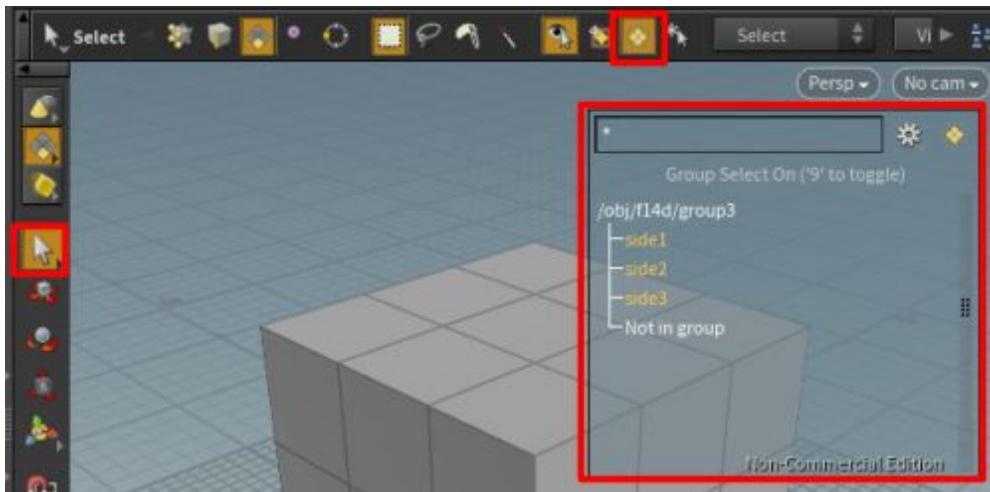
Regardless of which selection tool you use, be mindful of the Select Visible Geometry Only option. When this is enabled, you'll only select what you can see. So if you're in shaded mode, everything hidden won't be selected. But, if you're in wireframe mode, nothing is hidden so everything will get selected.



## Selecting by Group

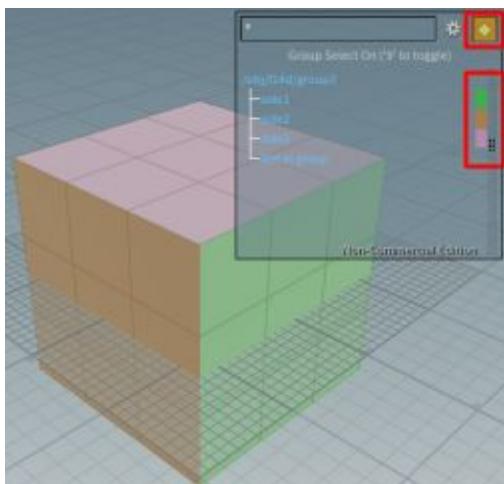
**NOTE:** Don't know what groups are used for? Check out the Group nodes section for more information.

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...



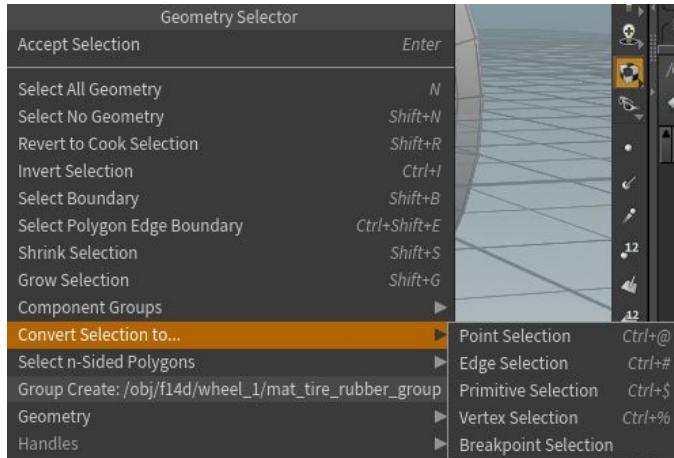
When you do this, a popup will show up on the lefthand side and you'll only be able to select by groups. You can select groups as a whole by clicking on the actual model causing it to highlight in the popup pane (and vice versa).

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



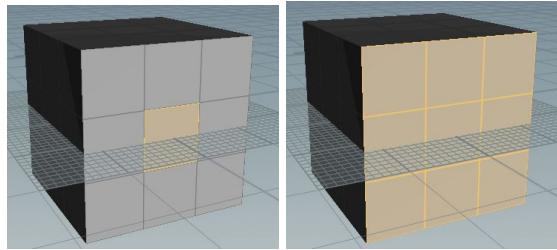
## Converting Selection between Component Types

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

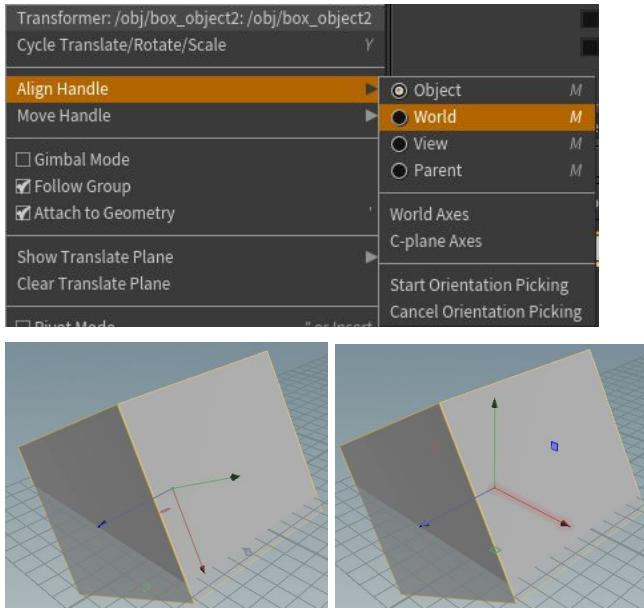
You can grow a selection outwards by hitting Ctrl+G (or right-clicking on the selection and choosing Grow Selection). Note that the shortcut doesn't always work -- I think if you move your mouse out of scene view and then back in, it won't work anymore unless you make a new selection.



## 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.

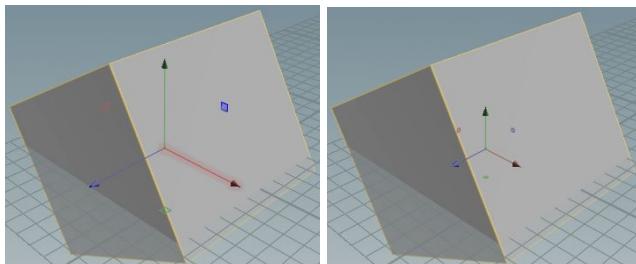
To fix this, hover over the handles and right-click. Then, go to Align Handle -> World...



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

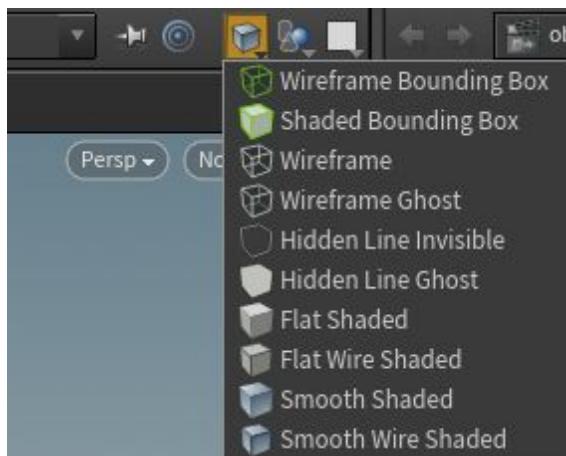
You can choose how objects appear based on your depth in the hierarchy by going to the ghosting type in the top-right of the viewport...



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)...

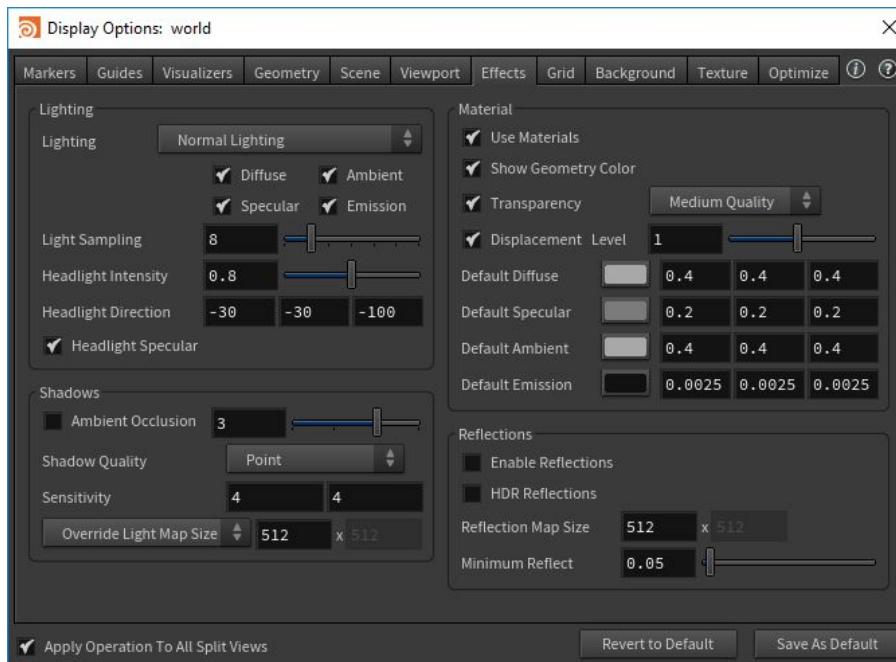
1. Disable lighting (flat shade?)
2. Headlight only
3. Normal lighting
4. High-quality lighting
5. High-quality lighting w/ shadows
6. Display materials on objects

## 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.

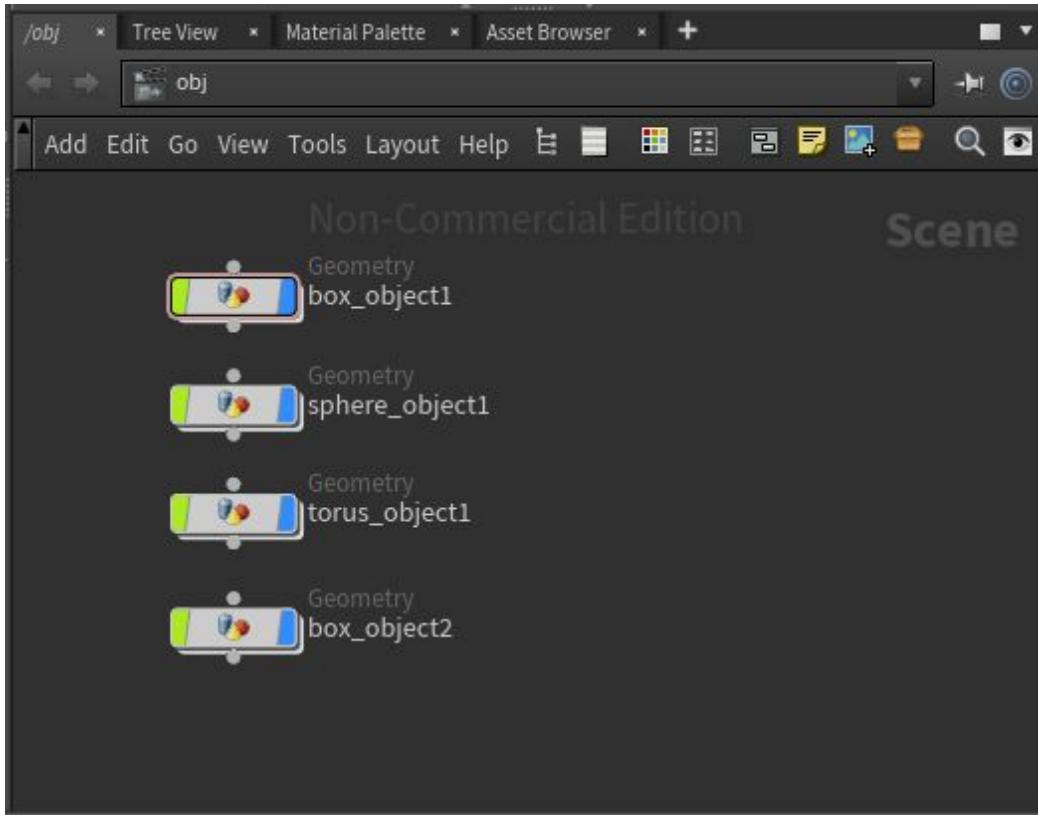


Scene will let you turn up AA samples and Effects will let you turn on special rendering options in the scene view (ambient occlusion, reflections, etc..). Depending on your workstation, your rig

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...

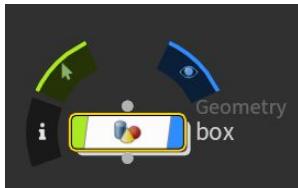
- MMB pans
- RMB zooms
- H brings everything into view
- G brings selection into view

**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...

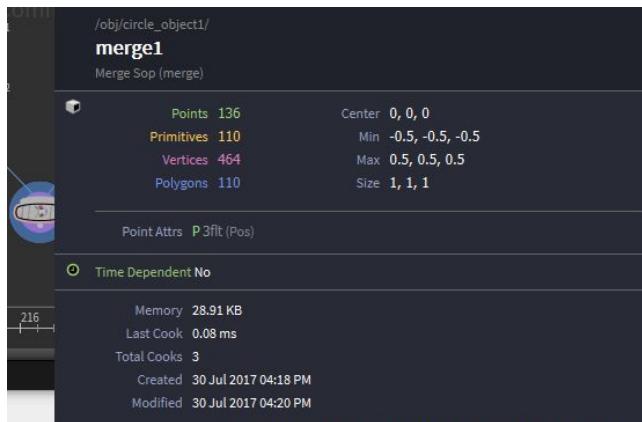
1. Bypass - Bypasses a node so that it doesn't get evaluated in the node tree.
2. Lock - ???
3. Template>Selectable Template - ???

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.

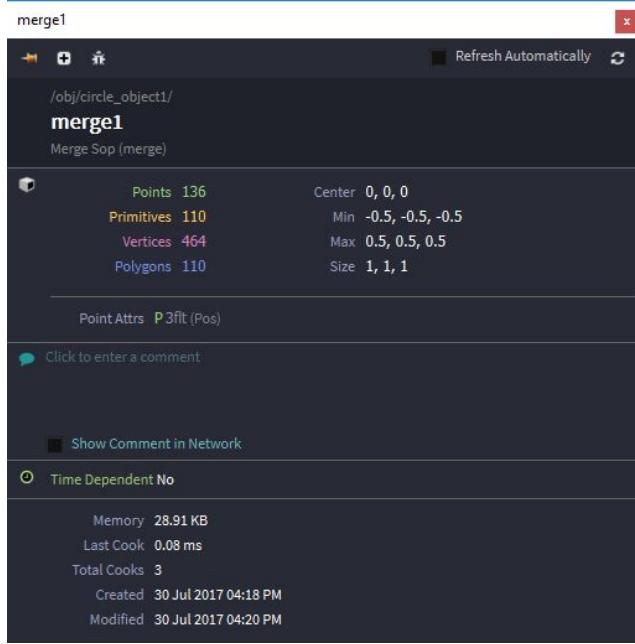
## Node Information

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...



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

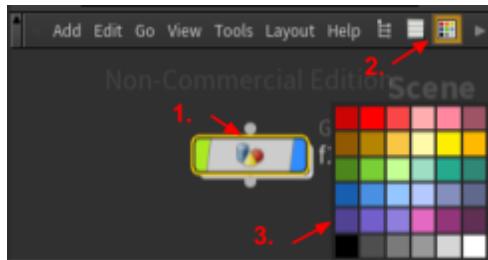




## Coloring Nodes

You can assign custom colors to nodes.

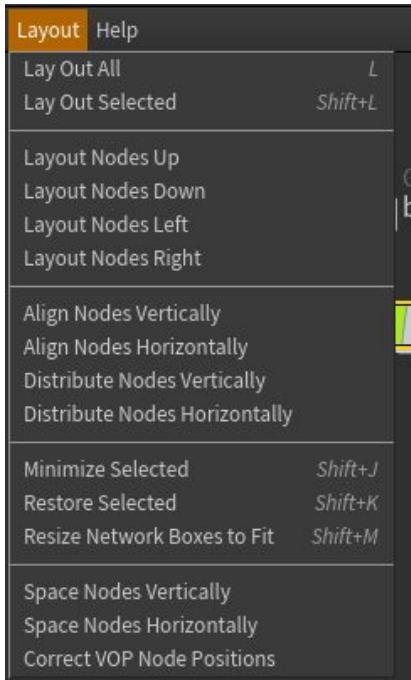
Select the nodes you want to color, then click the color palette toggle in the toolbar and choose the color you want.



## 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.



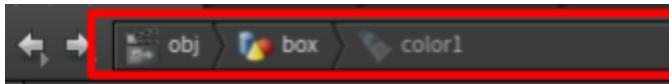
**NOTE:** L is the auto-layout shortcut. If your node placement is messy, just hit L to organize it.

## 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...



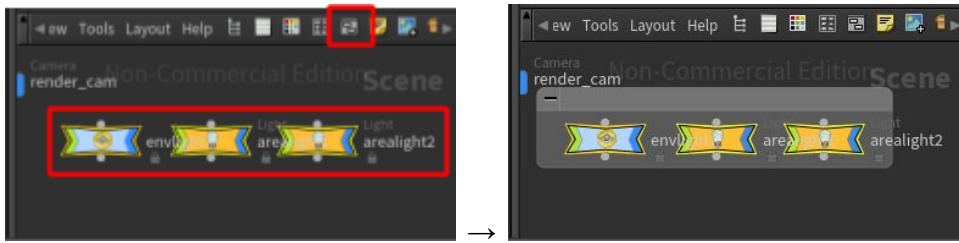
Just like a web browser, you can also use the back/forward buttons to move between your history of context switches...



## 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 ...

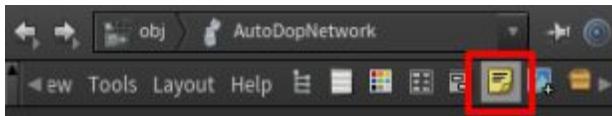
- move a netbox, it shifts everything inside of it with it.
- delete a netbox, it only removes the netbox, not the stuff inside it

**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).

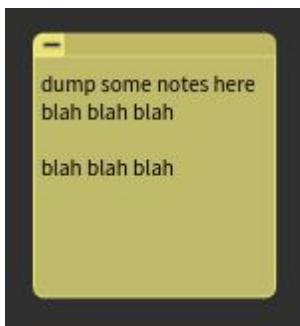
## Creating Notes

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...

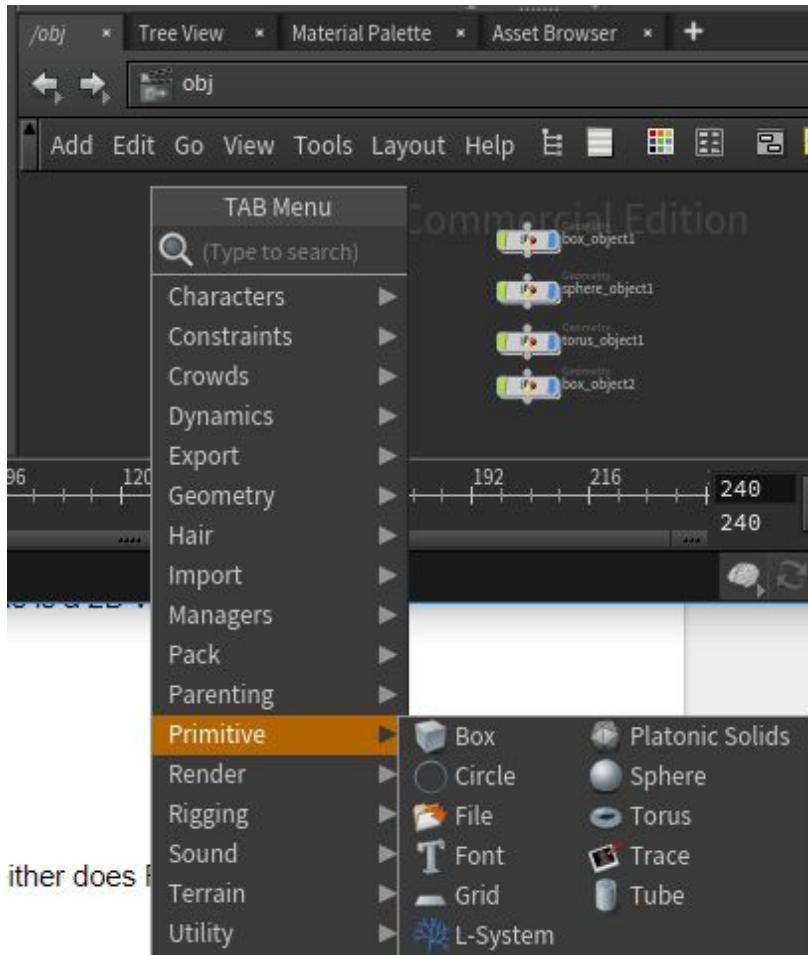


A little yellow box will show up and you can click inside to fill it in. Click the minus sign in the upper right corner to collapse it.



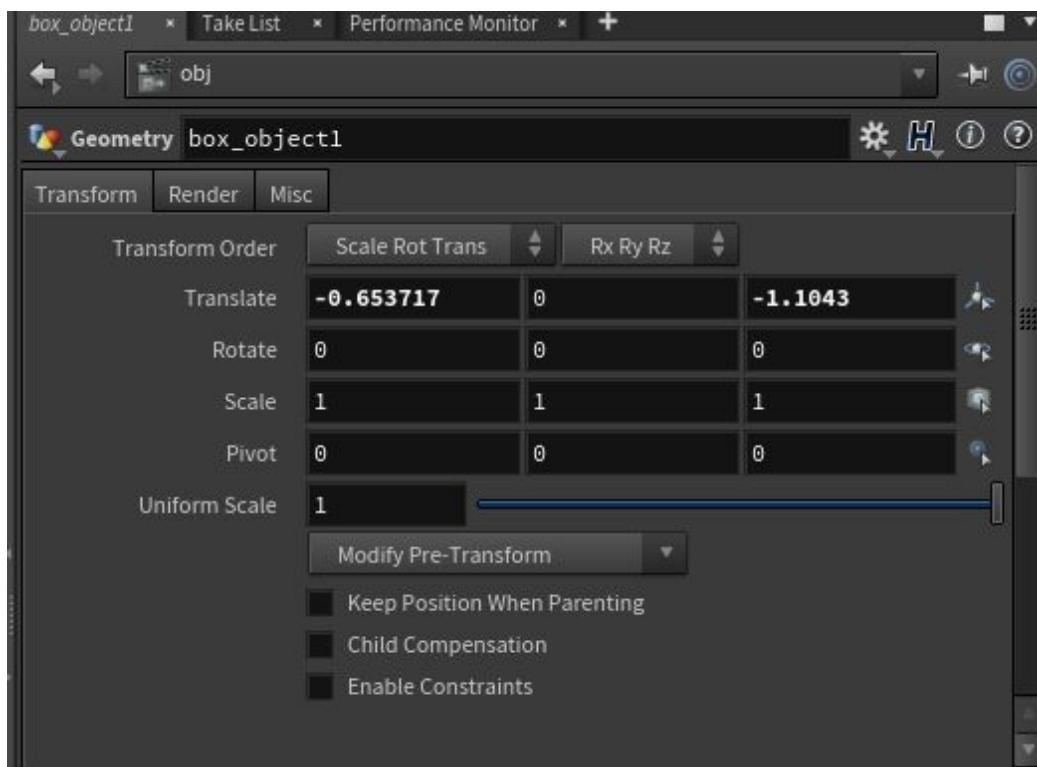
## Creating Nodes

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.



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.



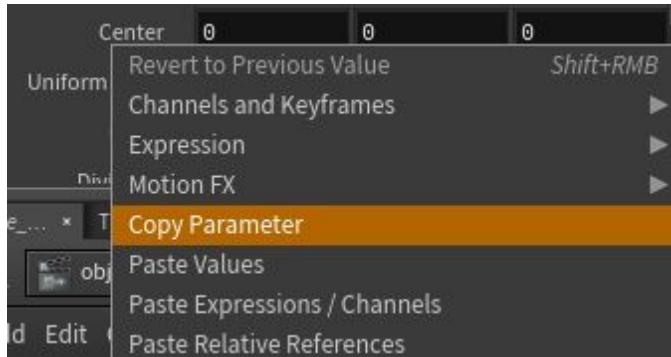
The number in the middle (0 in the example above) is the current value. Move your mouse pointer up/down to the scale you wish to adjust by, and move right to ADD and left to SUBTRACT.

## Setting Relative References

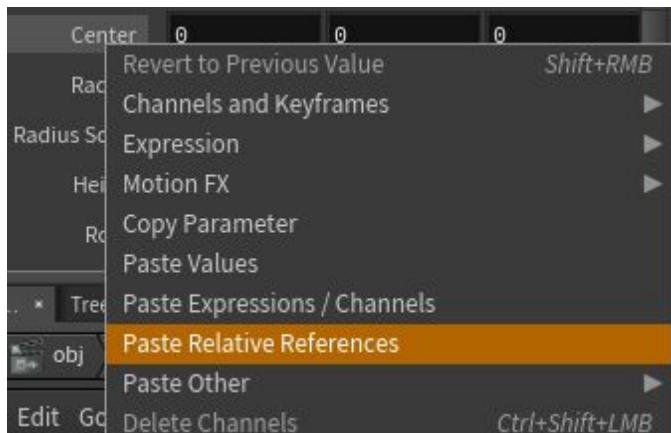
Sometimes, you want to bind certain parameters to the parameters of some other node. For example, you may want your object to be centered around another object.

To create a reference...

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



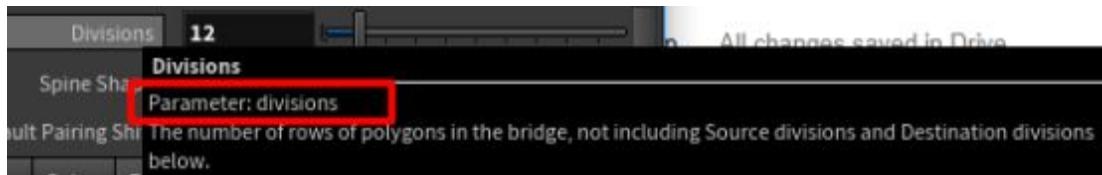
2. Go to the parameter(s) to bind to and right-click and choose Paste Relative References...



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...



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



## 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

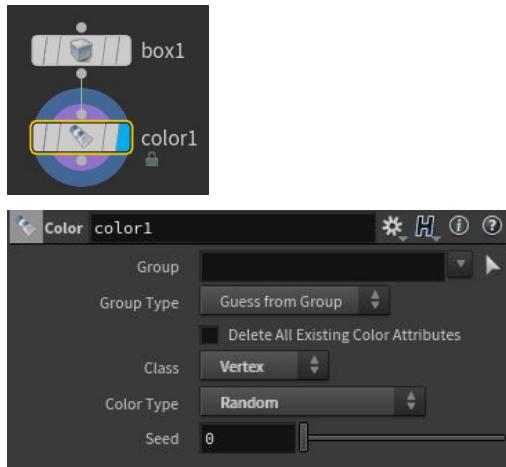
## Geometry Spreadsheet View (Attributes)

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...



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:	Group:	View	Intrinsics	Attributes:
0:0	Point Num	Cd[r]	Cd[g]	Cd[b]
0:1	1	0.641601	0.556797	0.965223
0:2	4	0.800464	0.593685	0.0728977
0:3	3	0.510895	0.716574	0.631335
1:0	1	0.775474	0.14131	0.413666
1:1	7	0.894847	0.340292	0.0923847

**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...



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

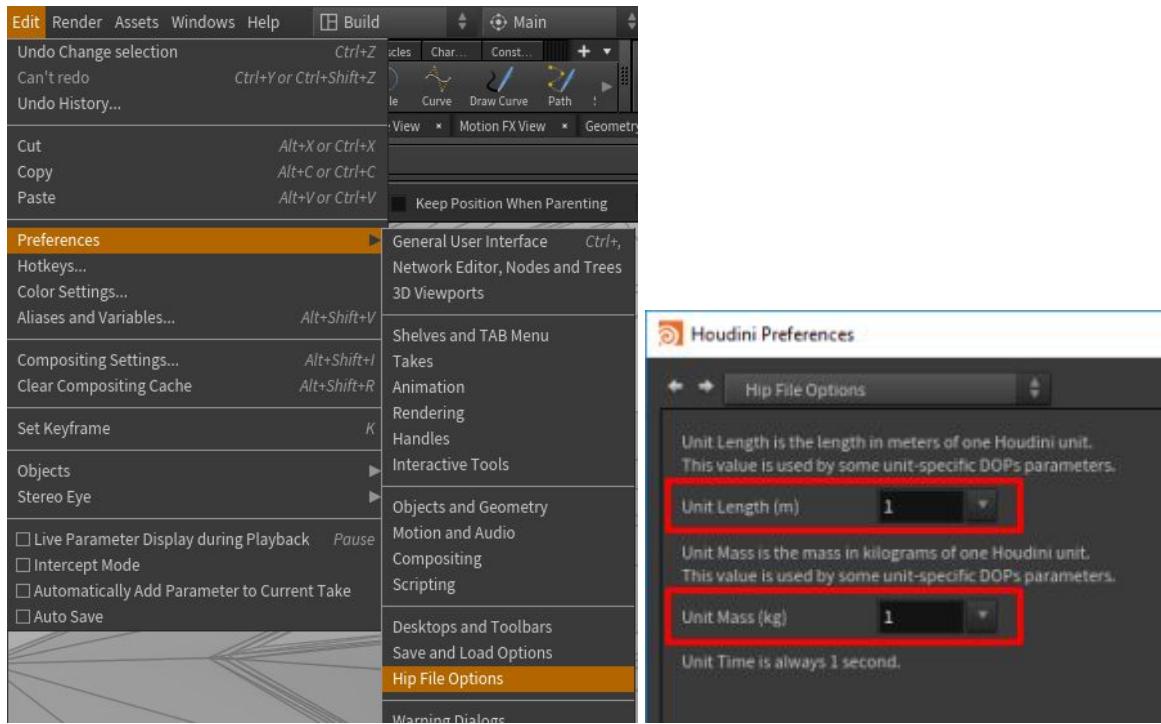
It's super important to remember that the attributes MUST BE APPLIED AT THE SAME LEVEL. Having UVs for the 1st input on points while having them for the 2nd input on vertex IS A MISMATCH and needs to be corrected (use Attribute Promote for this).

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...



**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..

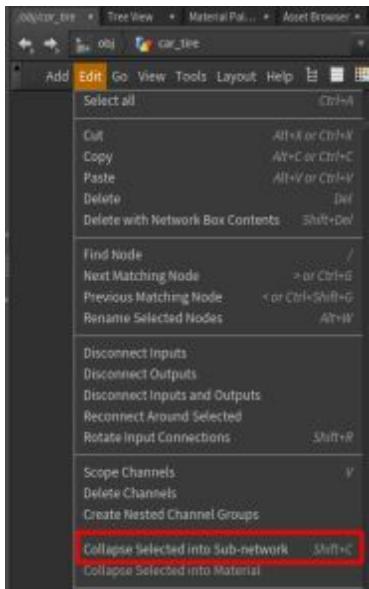
## Subnetworks

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

To create a subnetwork...

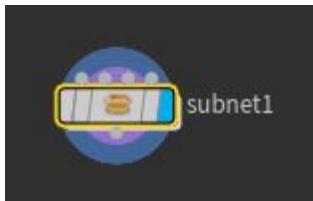
1. Go to the Network View
2. Select the nodes you want to package up

3. Go to Edit -> Collapse Selected Into Subnetwork



**HINT:** Ctrl+A to select all

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



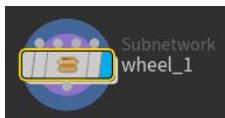
## Custom Inputs / Parameters

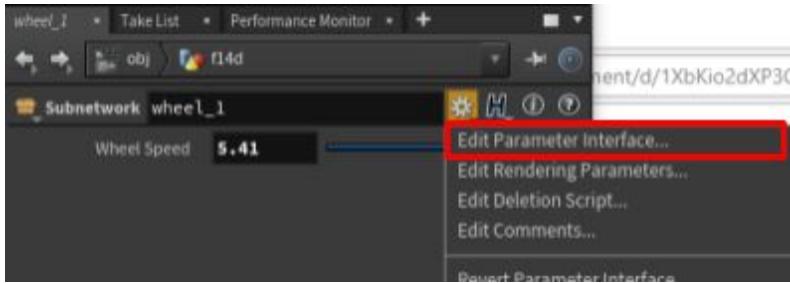
**NOTE:** Subnetworks will always have 4 baked in inputs. There's no way to get rid of these or add more. The input connectors will always show up. But, you can add in custom parameters in the parameter view. See the following links for more info...

<http://forums.odforce.net/topic/23798-subnetwork-can-i-remove-inputs/>

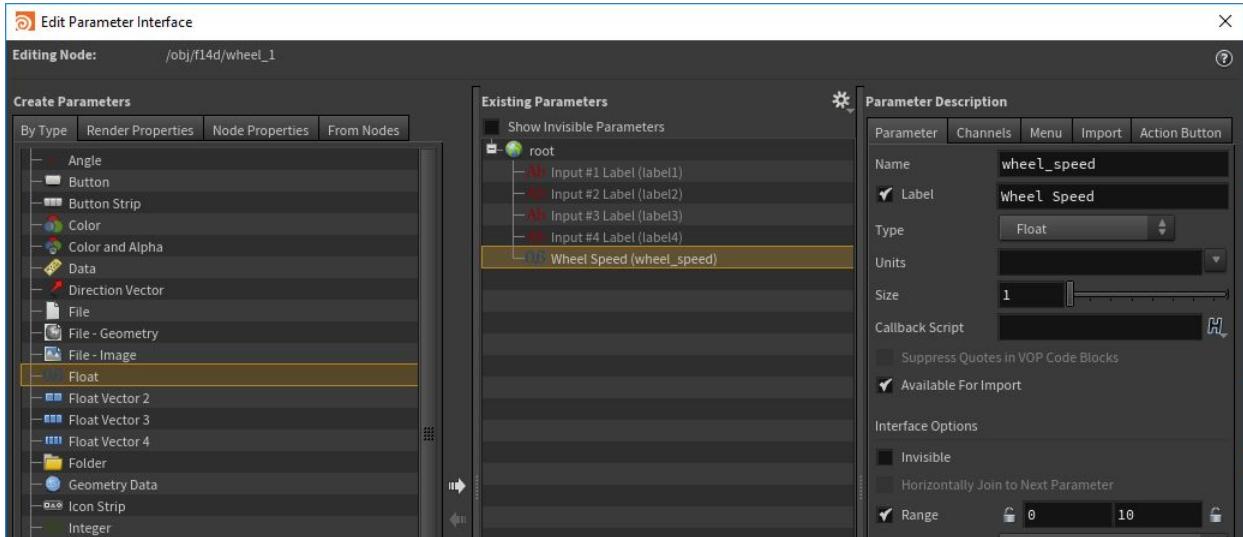
<http://forums.odforce.net/topic/10387-more-than-4-inputs-in-a-subnet/>

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.





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



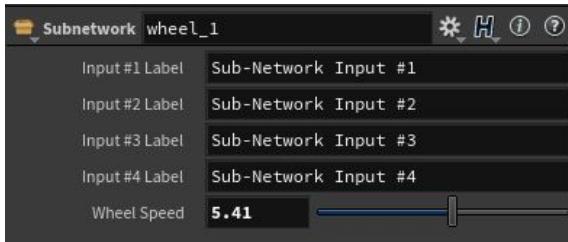
The window is split into 3 sections...

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

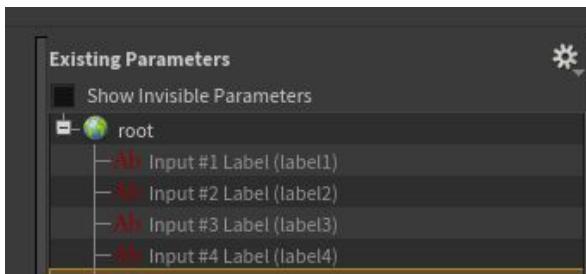
To create a new parameter, either drag-and-drop from 1 to 2 or select in one and hit the right arrow in between the two sections....



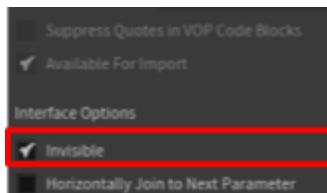
Once created, set a name and label in 3. You'll see your new parameter show up in the Parameter View when you select...



**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 dialog is open and on top)....

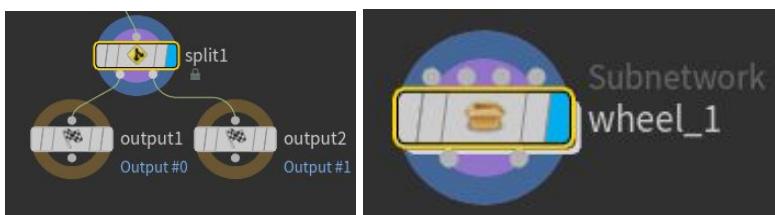


**NOTE:** You can stop the 4 baked inputs (e.g. Input #1 Label) from showing up in the parameter view if you choose them in the Edit Parameter Interface dialog and select to hide them. This only hides them in the parameter view... the input connectors on the subnetwork node will still be there.



## 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)

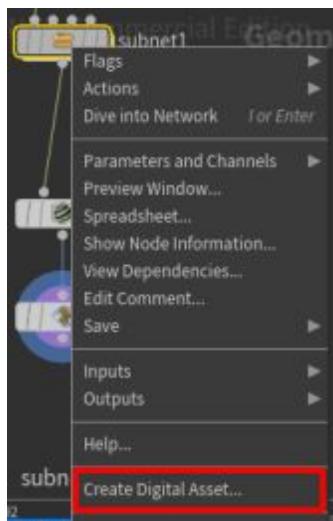
Houdini lets you export your subnetwork as a Houdini Digital Asset. When you do this, you can have it available for reuse in other houdini scenes that you work on / make it available for other artists by going to it in the Tab menu (it'll be just like any of the other hardcoded nodes that Houdini provides).

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.

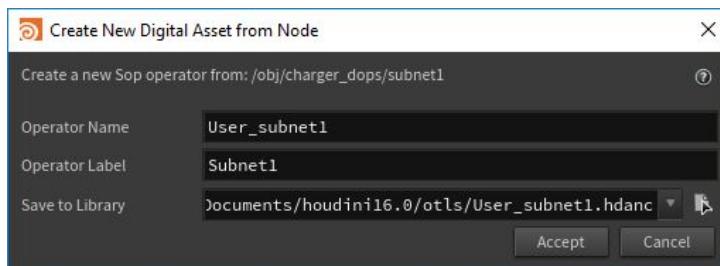
**NOTE:** Houdini Engine doesn't have an apprentice edition so I couldn't try this out. But, the video showed how it worked with Maya. You just have to load the plugin and then a Houdini menu shows up in the main menu that you can use to load it up and assign stuff to the inputs/properties/etc...

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...

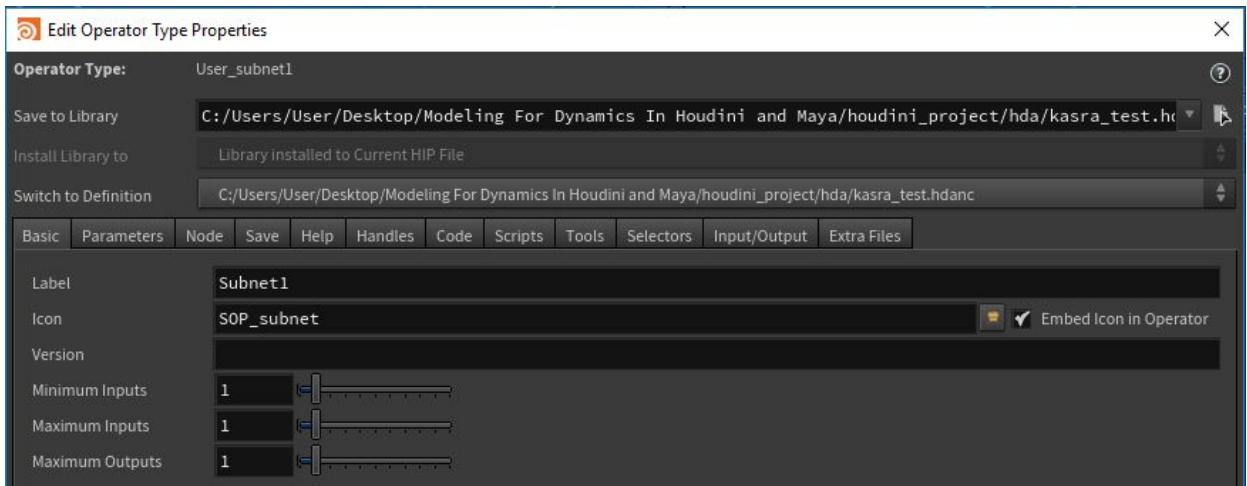


Operator Name → what shows up in the tab menu  
Operator Label → the default name once placed  
Save to Library → disk location to save to

**NOTE:** By default the save location will be the otl's 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).

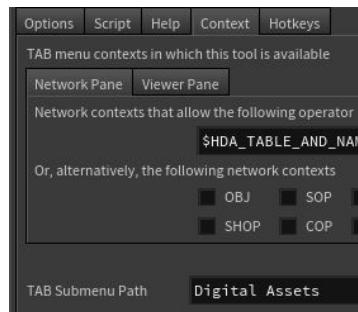
**NOTE:** otl's stands for Operator Type LibrarieS.

3. You'll be presented with a dialog to provide the operator properties...



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

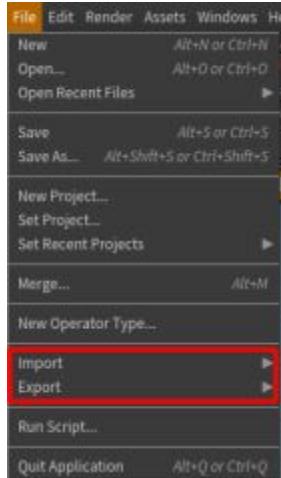
- Basic should be pretty straight forward.
- Help is just a textbox where you document what your node does.
- Tools is where you can set what Sub-menu of the Tab menu your Digital Asset pops-up under -- TAB Submenu Path under the Context tab.



- 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...

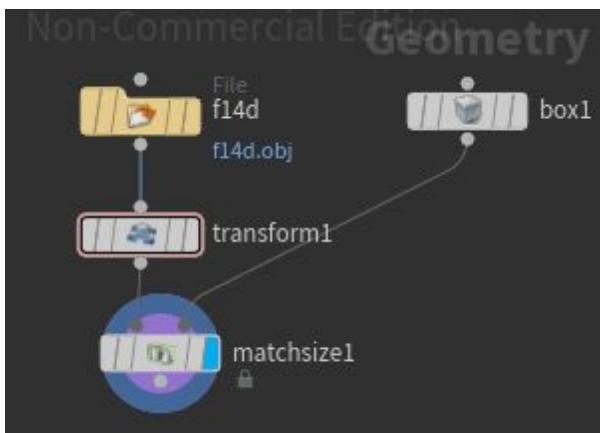


If you're importing assets, make sure that you place your assets inside of your project folder in the **CORRECT** subfolder. For example, all imported geometry should sit in /geo. It looks like the import isn't a one-time thing -- the file being imported is referenced continually.

## 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.

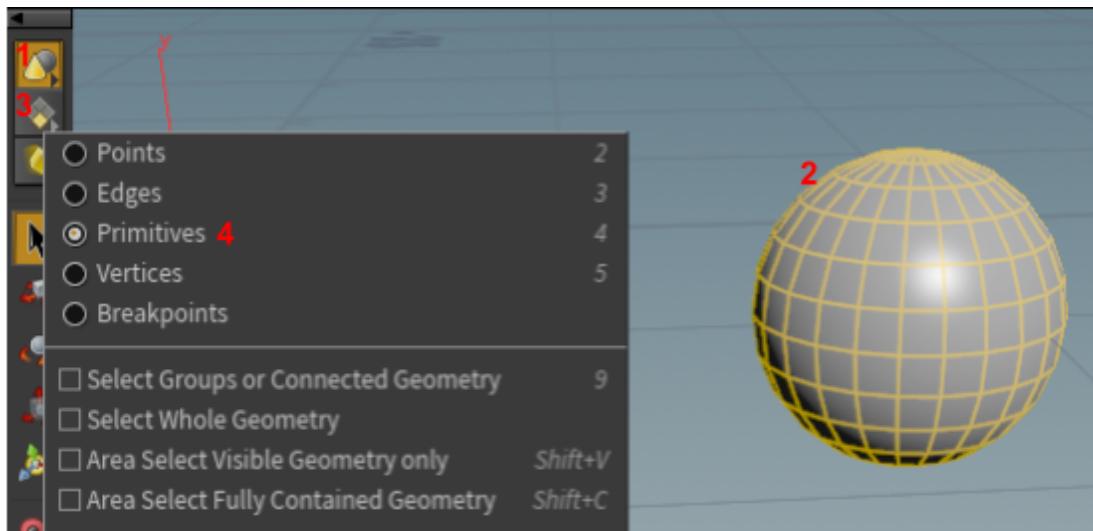


**HINT:** For all selections (even when you're doing marquee selections). Hold Shift to add to your existing selection / Ctrl to remove from existing selections. The mouse cursor should change to show a mini + or - symbol.

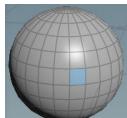
**NOTE:** There's more on selections in the Scene view notes above.

### Selecting Primitives (Faces)

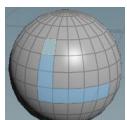
1. Select object selection button
2. Select your object
3. Right-click component selection button (or left-click and hold)
4. Choose Primitives in the pop-up



- Click to select a face



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

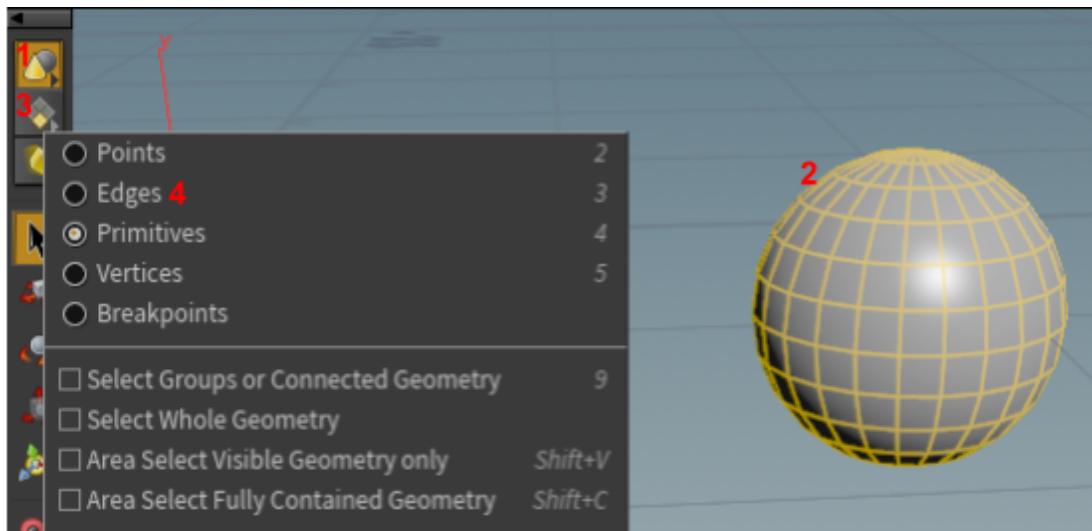


- Double-click to select all faces

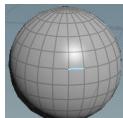


## Selecting Edges

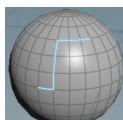
1. Select object selection button
2. Select your object
3. Right-click component selection button (or left-click and hold)
4. Choose Edges in the pop-up



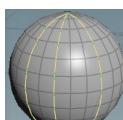
- Click to select an edge



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



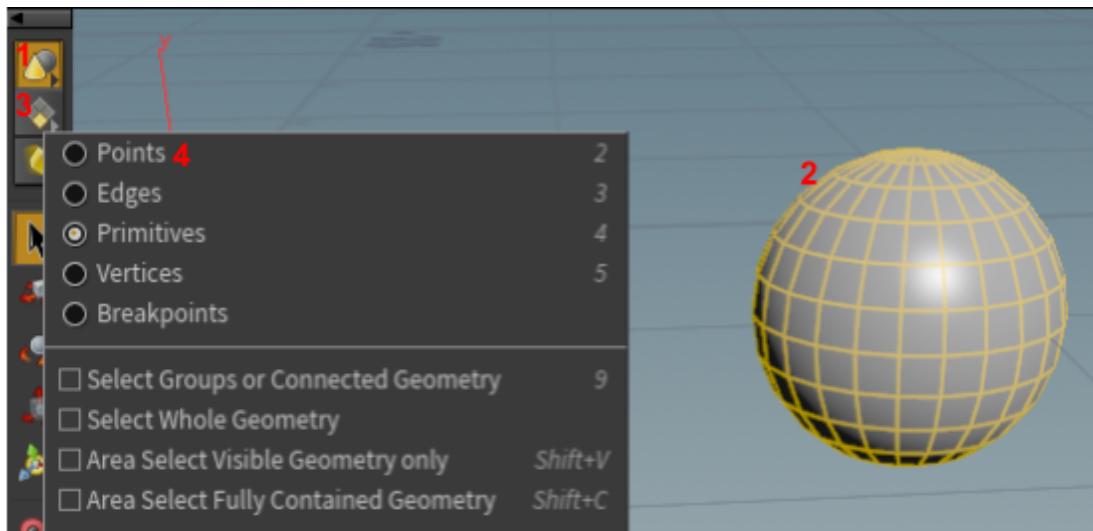
- Double-click to select an edge loop



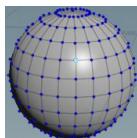
## Selecting Points

**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.

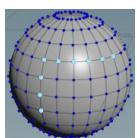
1. Select object selection button
2. Select your object
3. Right-click component selection button (or left-click and hold)
4. Choose Points in the pop-up



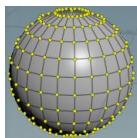
- Click to select a point



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

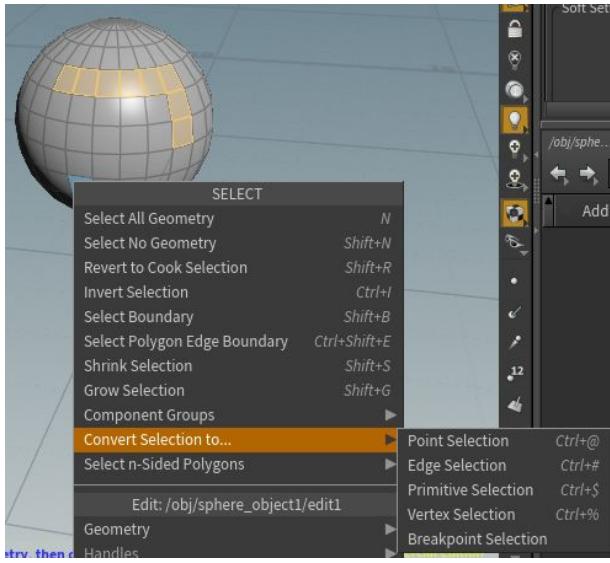


- Double-click to select all points



## 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

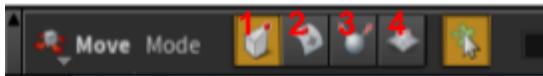
The scale/rotate/translate tools work the way they normally do. Select via the left toolbar...



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

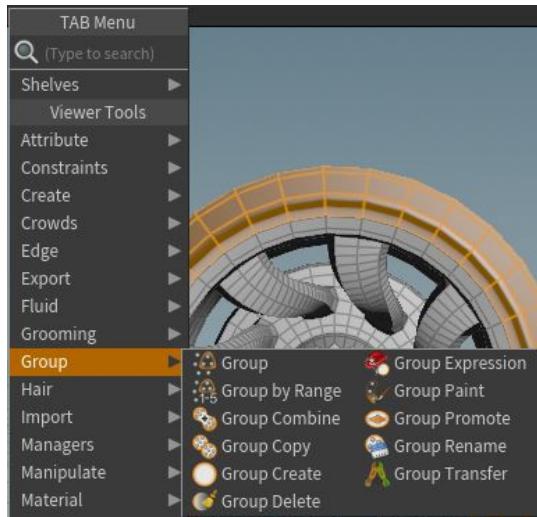
## Group Nodes

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.



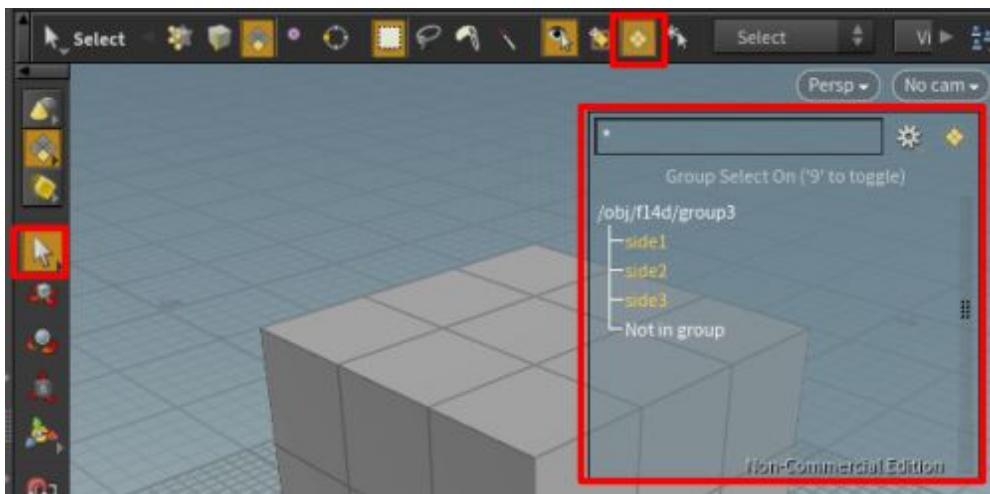
Once your group is available, you'll be able to see it in the geometry spreadsheet as an attribute. If the component is included in the group, you'll see a 1 for the attribute that that group is for. Otherwise you'll see a 0.

Node:	mat_tire_rubber_group	Group:	View
860	group:mat_tire	0	
861		0	
862		0	
863		0	
864		1	
865		1	
866		1	

You'll also be 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)...



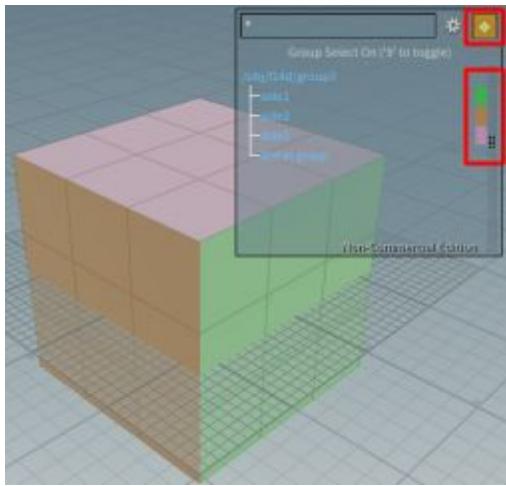
In addition to both of these, you can view / highlight the various groups in the scene view by choosing the select tool and enabling 'Select Groups' in the top toolbar...



When you do this, a pop-up 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...



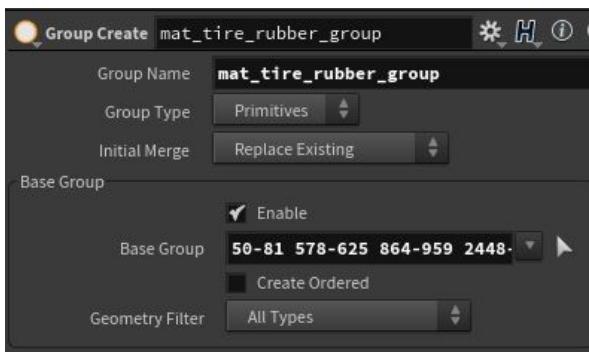
## Group (Create component groups)

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.



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.

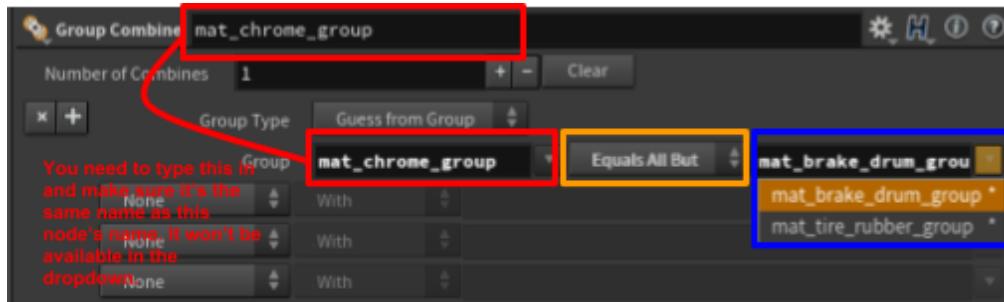
**NOTE:** There are lots of different ways to group stuff by in the properties pane: keep by normals/edges/etc... I don't know what all of these actually do, but I think manually selecting the faces is the main way this node is used.

**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.

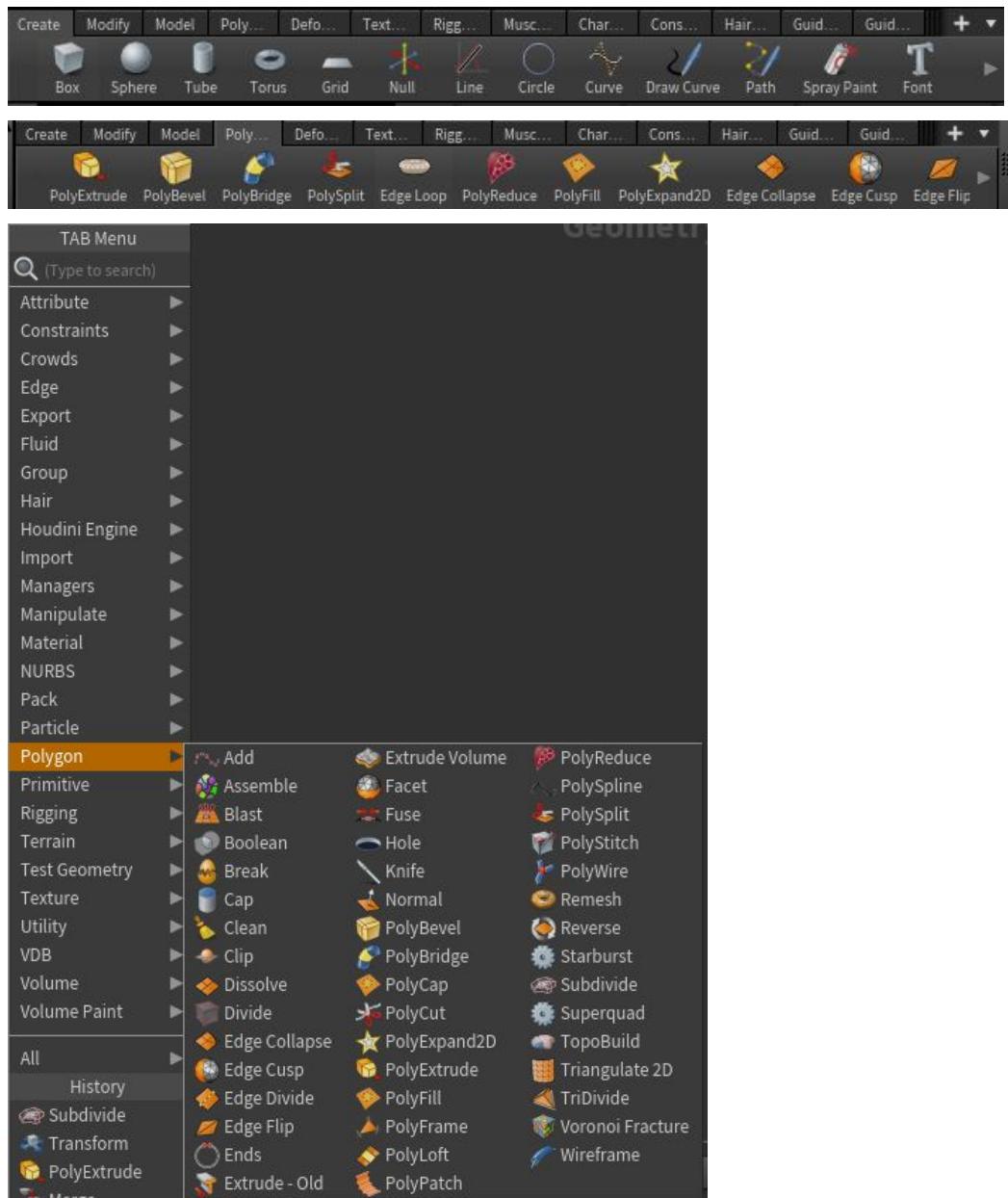
One important use case for this is if you want to take all remaining geometry (geo that isn't part of a group) and stick it into some final group. Here's what the properties dialog looks like...



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 prompt 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...

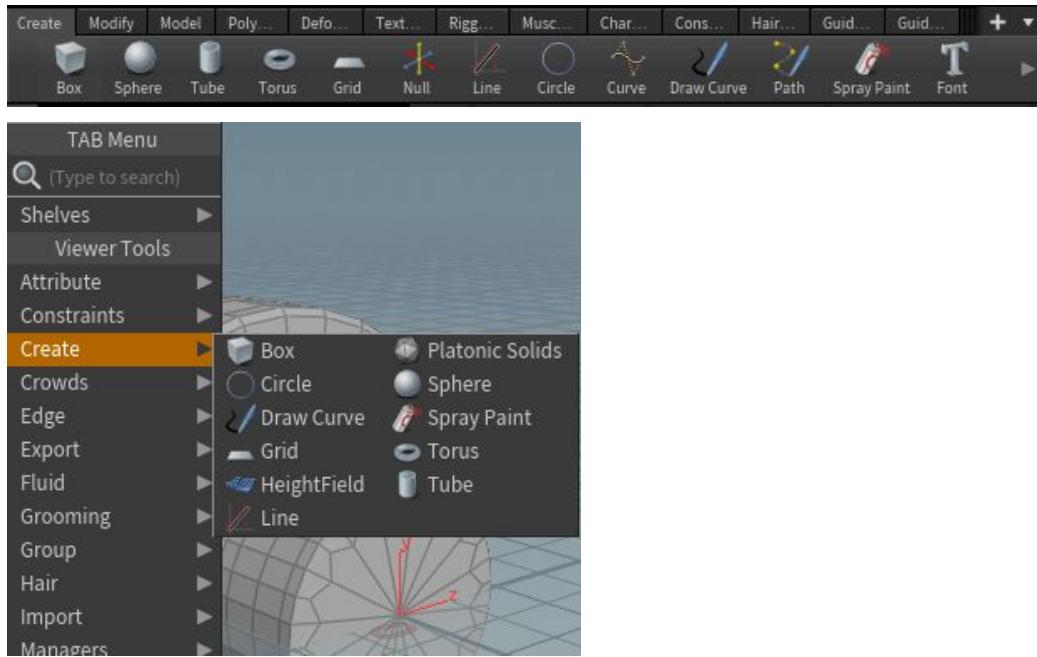


**NOTE:** Some tools won't have this. For example, trying to specify edge loops through the parameter window would be a nightmare -- use the shelf to interactively place those instead.

**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...



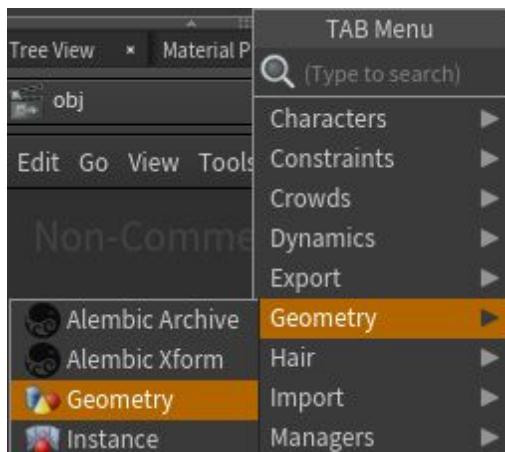
## Importing Geometry

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.

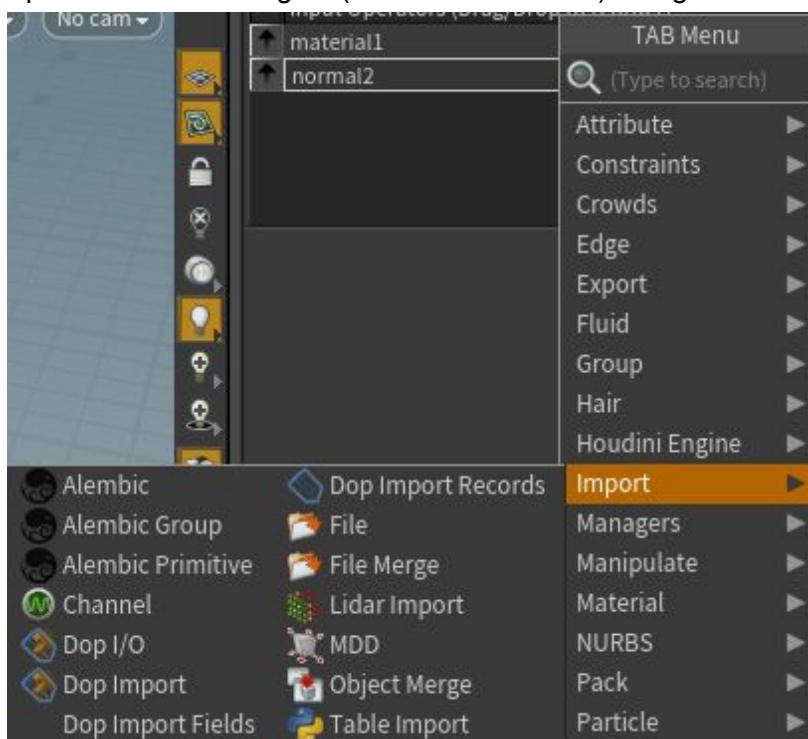


In the same vein, if you want to import some geometry, you can dump a basic Geometry node and add import nodes in there instead of nodes to create some primitive. To do so...

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

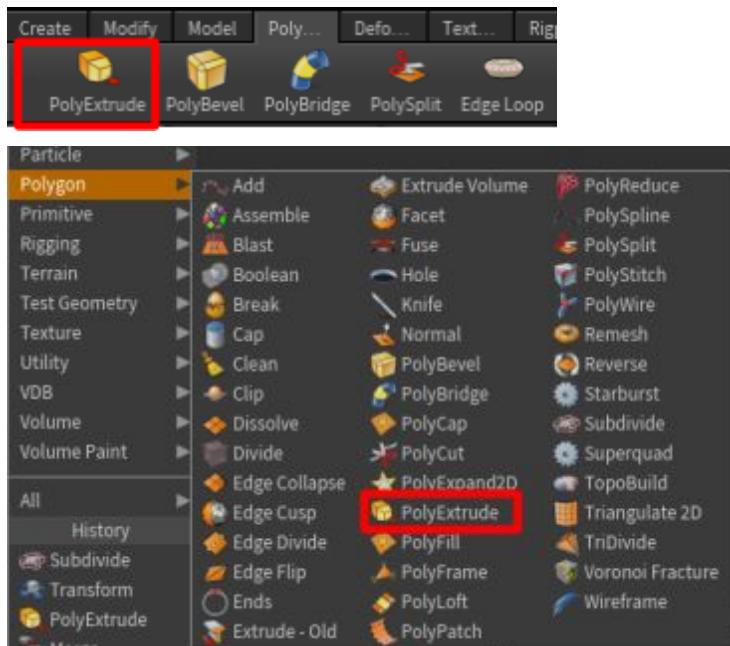
# Geometry Nodes

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.

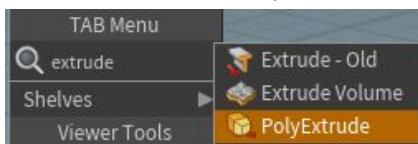
These nodes will ONLY be available when you traverse into the geometry node!

## PolyExtrude (Extrude faces/edge)

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

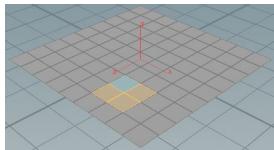


**NOTE:** Do not choose the other Extrude tools by accident. Those tools are totally different from the PolyExtrude tool.



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

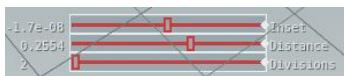
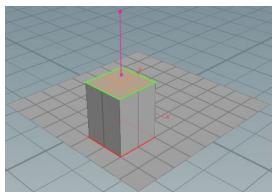
1. Choosing the components (e.g. faces) you want to extrude



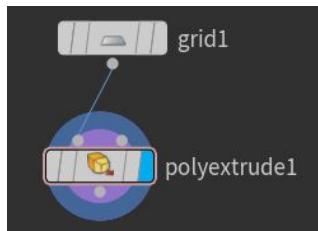
2. Clicking the PolyExtrude tool in the shelf



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



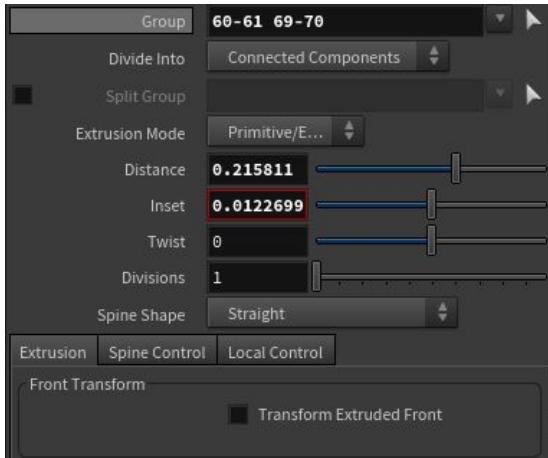
If you're working in non-interactive mode (via the network view) you can extrude polygons by connecting your geometry to the first input of a PolyExtrude node...



**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...



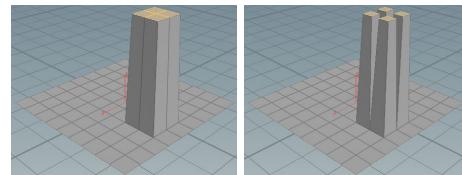
Here's an outline of what each of these means...

**Group** Components being extruded.

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

If empty, it extrudes everything.

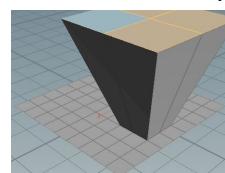
**Divide Into** If you want your extrusion to affect the selected components as a whole...



**Exclusion Mode** ???

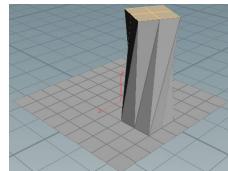
**Distance** How far to extrude.

**Inset** How much to expand/contract the components.

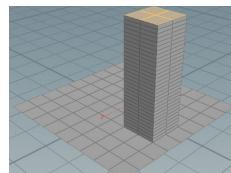


**Twist**

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...

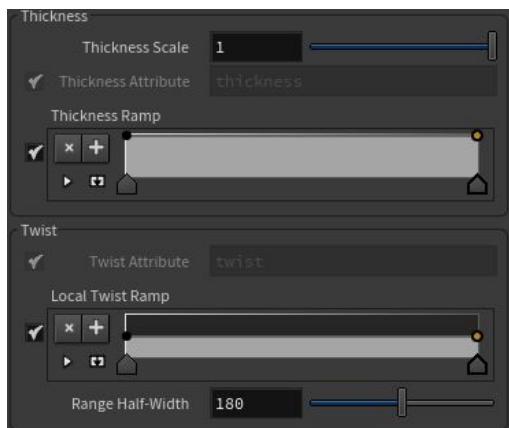
- \* straight (linear)
- \* curved (you define properties of the curve in the spline tab)
- \* curved from 2nd input (you feed in a NURBs curve into the 2nd input of the node)

**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.

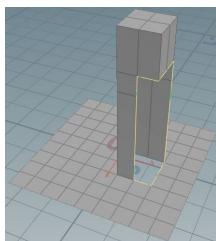


## PolyFill / PolyCap (Fill holes)

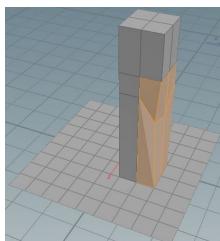
The PolyFill tool (called PolyCap in Houdini 15) lets you create faces between edges in your geometry. So if you have a hole in your geometry, you can use the PolyFill tool to create face(s) to fill in that hole.

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

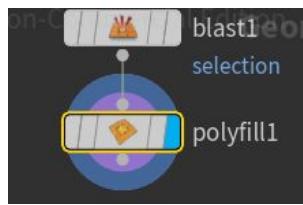
1. Selecting the edges you want filled in



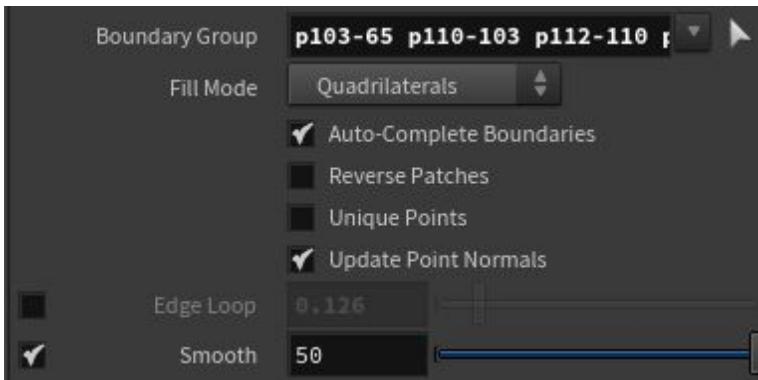
2. Clicking the PolyFill tool in the shelf



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** 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.

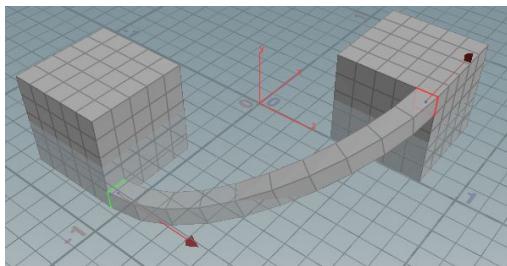
**Auto-Complete Bound** If your boundary only contains only part of a hole, this option tries to get the edges for the rest of the hole.

**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.

<b>Unique Points</b>	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?
<b>Update Point Norma</b>	The normals of the points that are getting filled will be updated to include the new faces?
<b>Edge Loop</b>	Seems to add an extra edge around the geo that is created???
<b>Smooth</b>	Seems to shift around how the fill geo that is created???

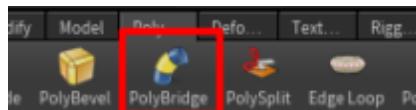
## PolyBridge (Bridge between edges)

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**

The number of divisions you want your bridge to have.

**Spine Shape**

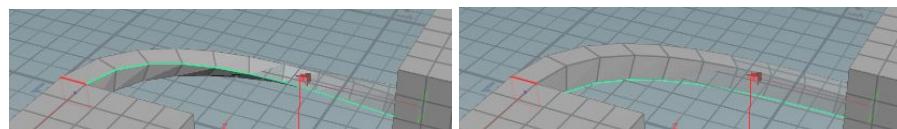
Controls how the bridge curve. Depending on the option you choose, this can either be...

- \* 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)

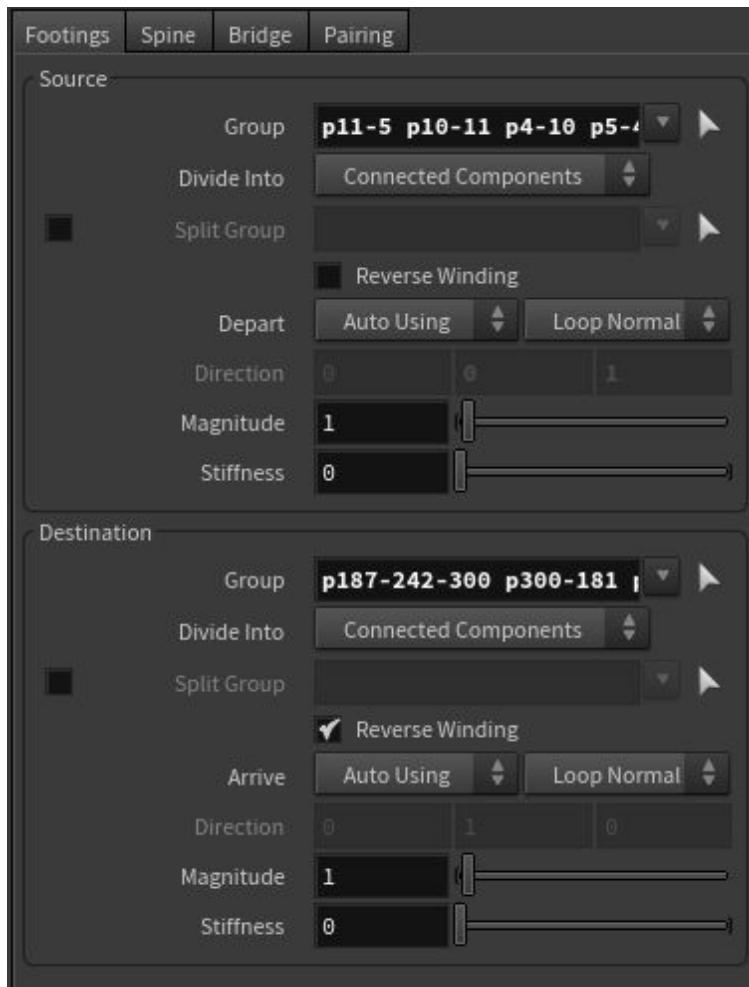
**NOTE:** This seems to be very similar to Maya's extrusion.

**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.



#### Group

The source/destination edges for the bridge.

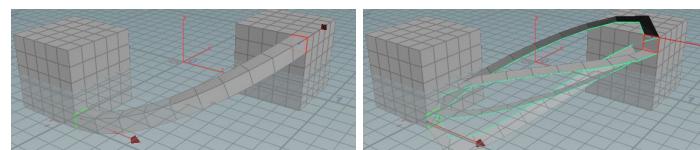
Click arrow to select interactively in scene view. Follow the prompts in the scene view.

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 exploded by edge.

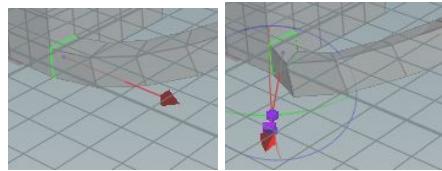


#### Reverse Winding

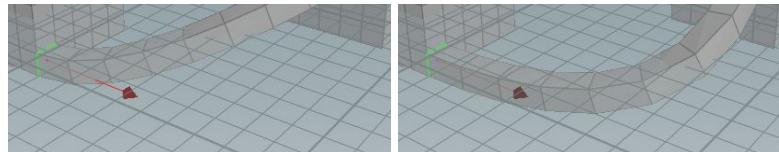
Try this if the bridge is coming out twisted or out of place.

**Depart/Arrive**

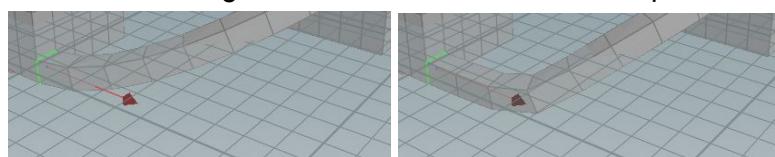
Controls the direction that the bridge this starts/stops from.

**Magnitude**

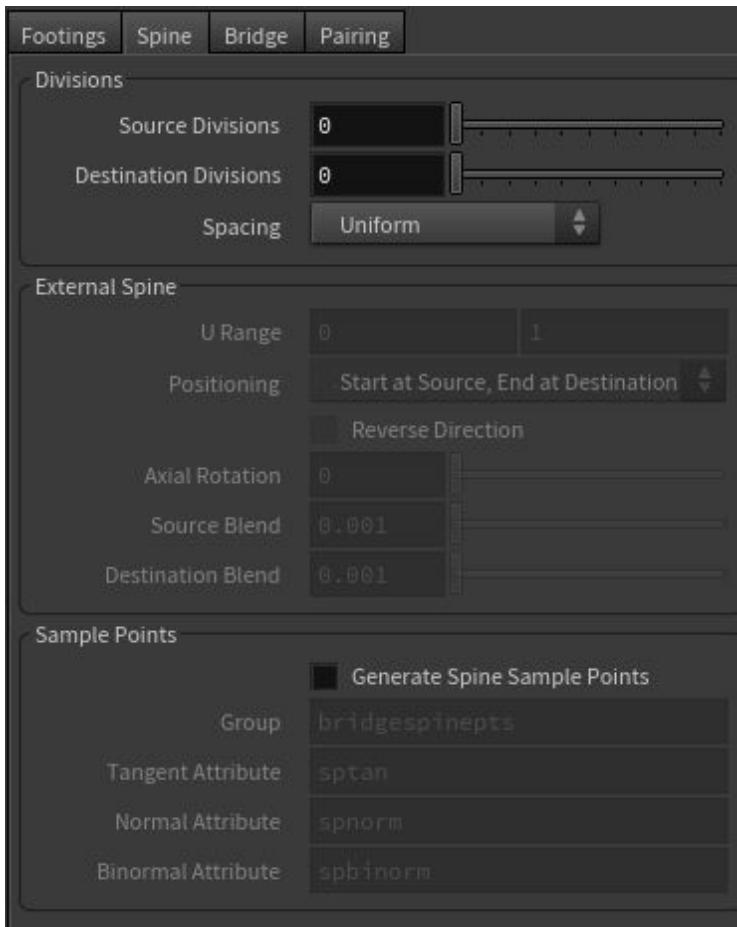
Blasts out (scales) the bridge from the source/destination position.

**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.



**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.

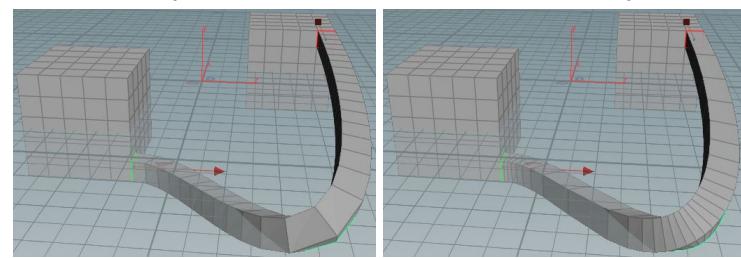
**Destination Divisions** Adds divisions “to the end of the bridge.” I don’t know how this works, the divisions don’t seem to be bound the the start or the end.

**Spacing** There are 2 options here...

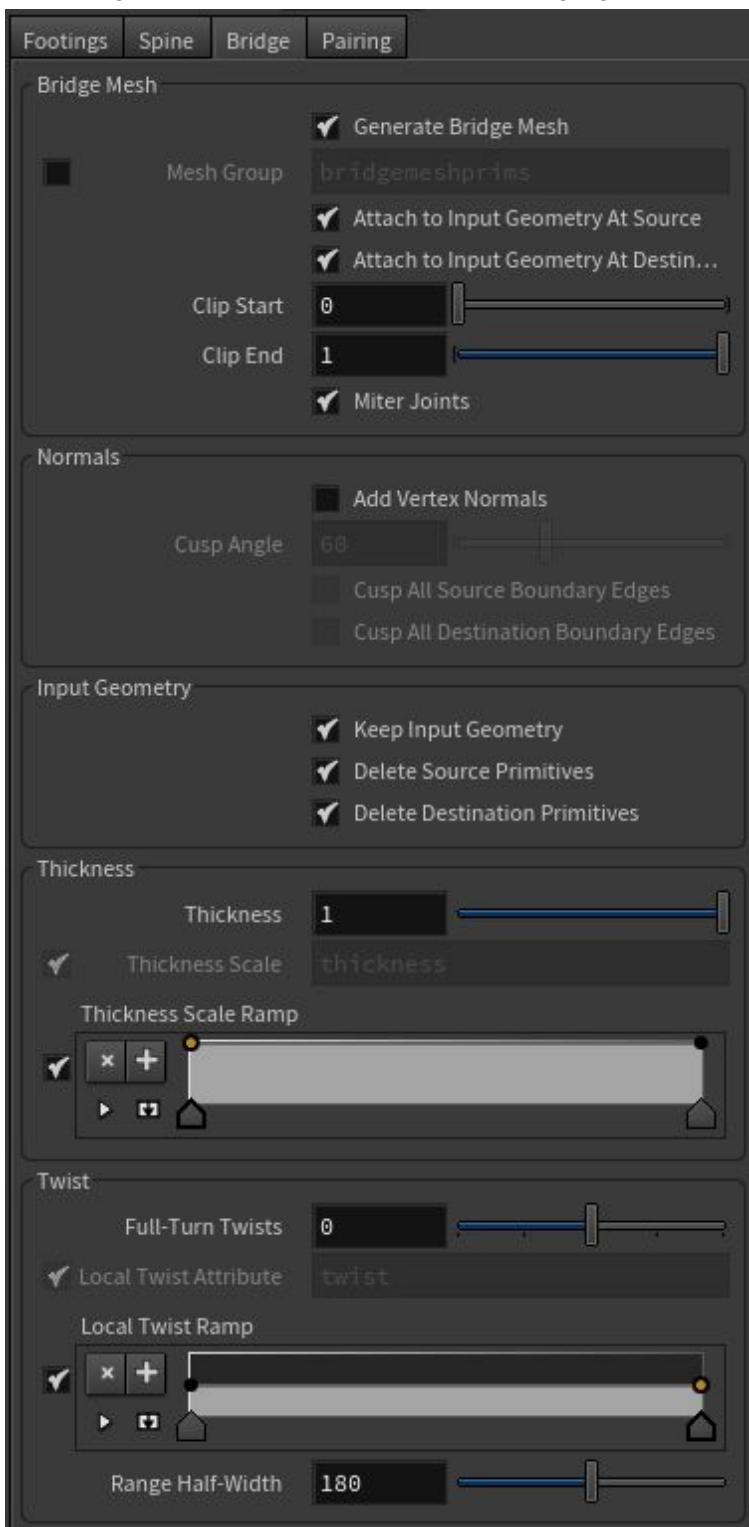
- \* uniform
- \* curve sensitive

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.

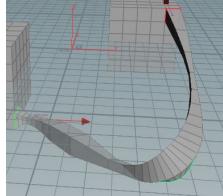


The Bridge tab controls how the actual bridge gets created

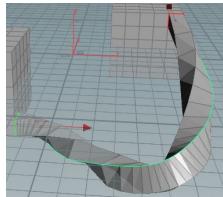


#### Clip Start/End

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.

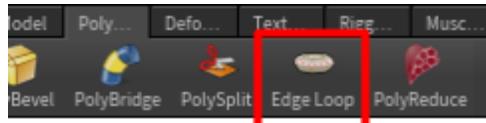
<b>Normals</b>	If normals should be added for your bridge???
<b>Delete Primitives</b>	Delete the faces/edges which these bridges were generated from???
<b>Thickness</b>	How thick you want to be the bridge. You can change thickness in multiple places.
	

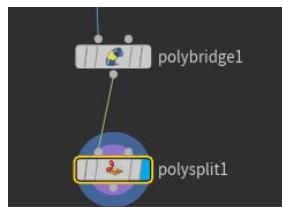
<b>Twist</b>	How you want your bridge to twist. You can change the twist in multiple places.
	

## Edge Loop

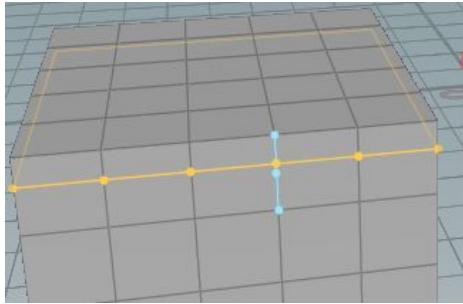
You can create an edge loop by going to the shelf and selecting Edge Loop...



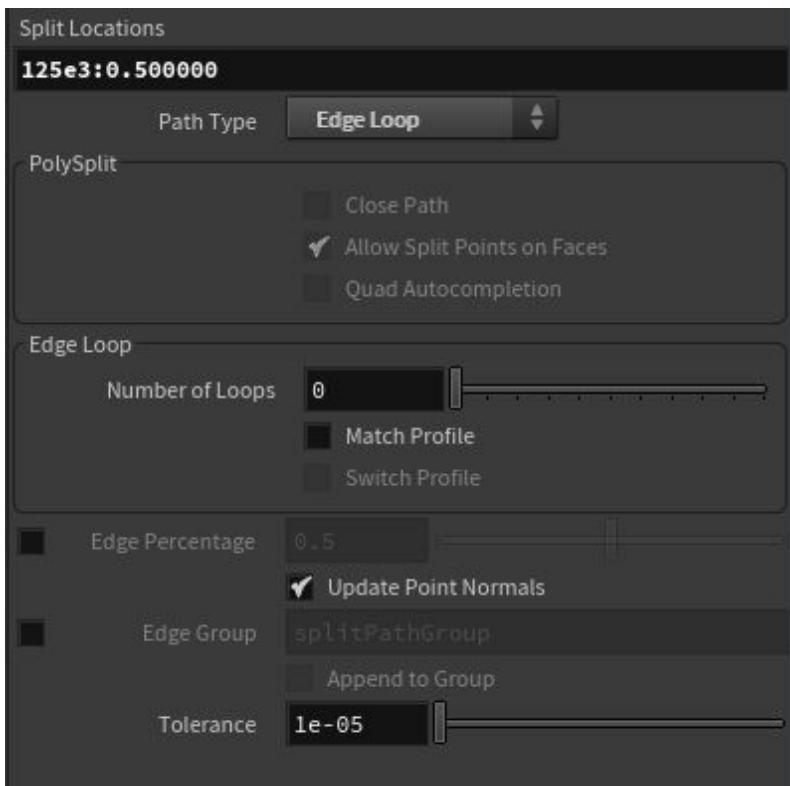
Or by dropping a PolySplit node in the network view and hooking your geometry into it...



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...

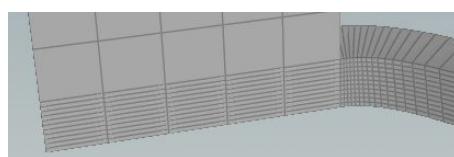


#### Split Locations

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

#### Number of Loops

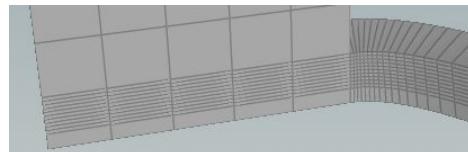
If this is set to > 0, the number in split location (the one after the colon) is ignored and instead n new edge loops are created equidistant from each other.



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

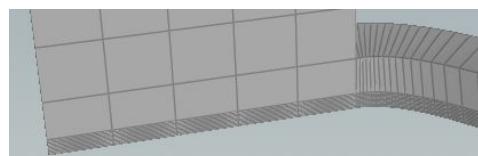
#### Match Profile

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.



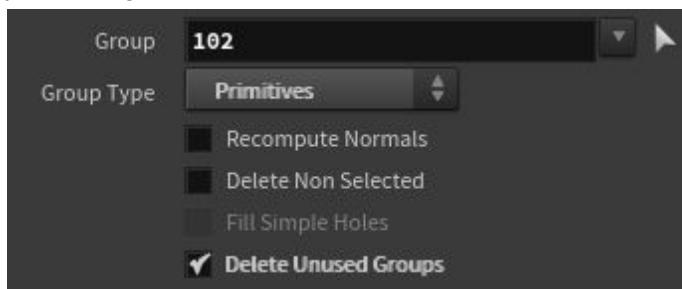
## Blast (Delete faces and points)

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

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.
<b>Group Type</b>	If you're dealing with faces or points.
<b>Recompute Normals</b>	Recomputes the normals for whatever that was a sibling of whatever got deleted.
<b>Delete Non Selected</b>	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)

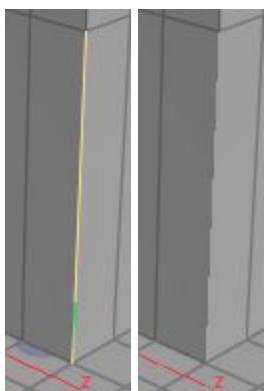
The dissolve tool lets you remove edges.

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

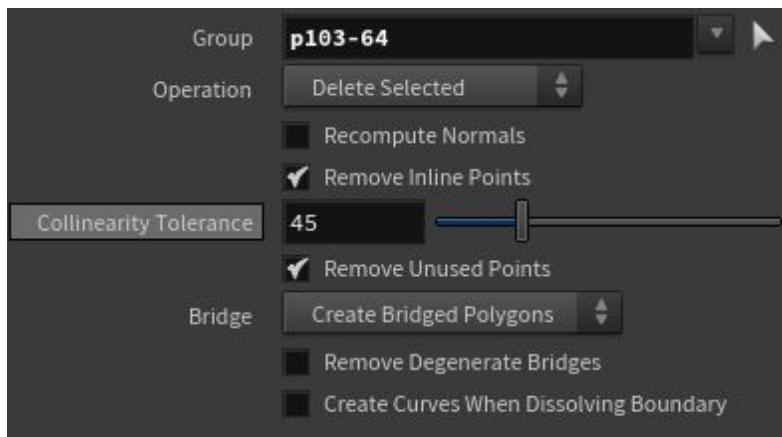
To non-interactively blast faces (via network view), connect your geo to a dissolve node.

**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** 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)...

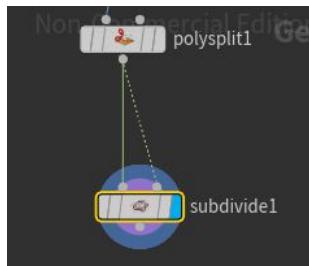
1. Choose Subdivide from the shelf



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

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

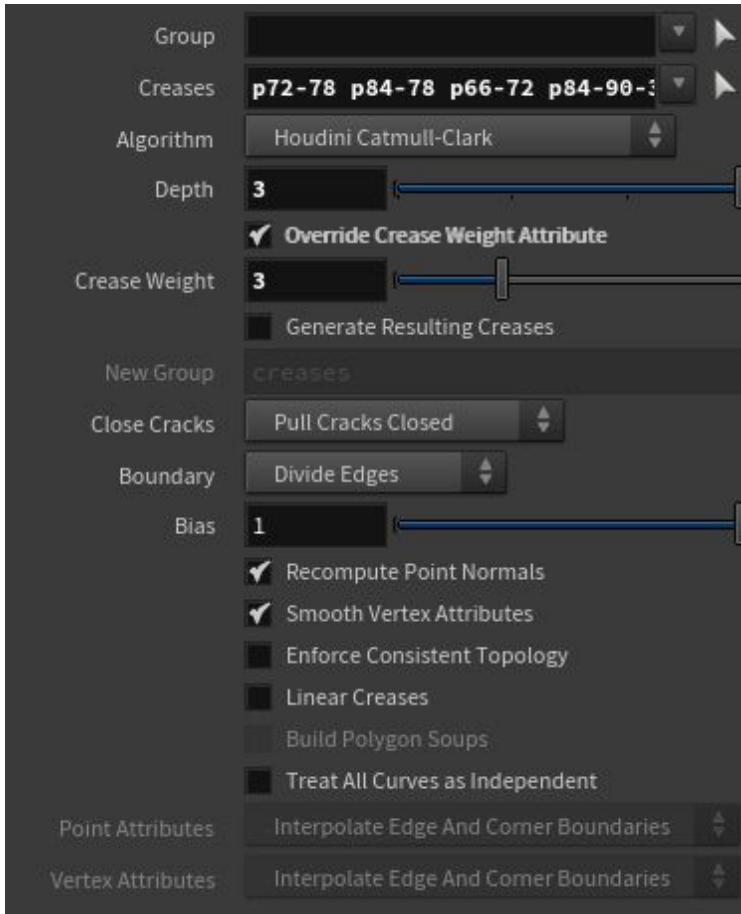
1. Connect your geo to a subdivide node



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** 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.

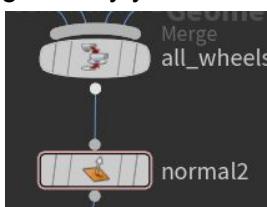
**NOTE:** The second input is for specifying the creases. If you don't have a second input, you can ignore the creases property. You can almost always feed the same geo to both inputs.

<b>Algorithm</b>	Stick to Houdini Catmull-Clark. Otherwise you won't get support for creases.
<b>Depth</b>	How many iterations of subdivision to apply.
<b>Crease Weight</b>	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.

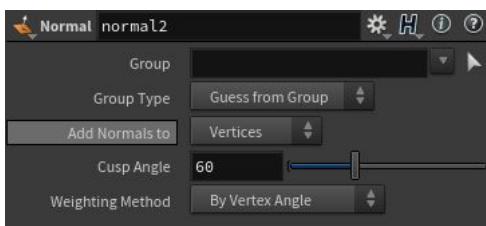
## Normal (Generate or correct normals)

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...



**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

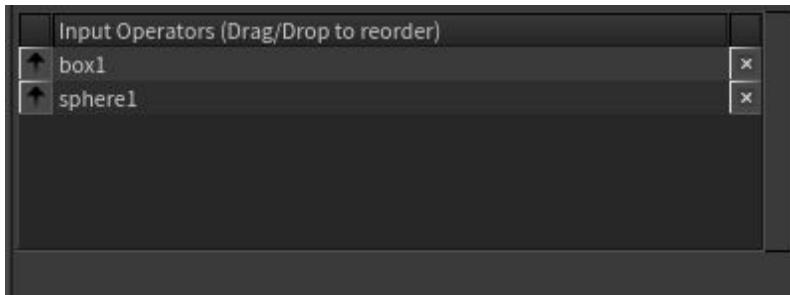
## Merge (Combine geometry/outputs)

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...

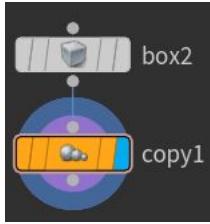


**NOTE:** Is this node giving you a warning? Check the attributes of your inputs at various levels via the geometry spreadsheet. For example, does your first input have normals on primitives (faces) but your other input have normals on vertices? Mismatches and missing attributes may cause warnings to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

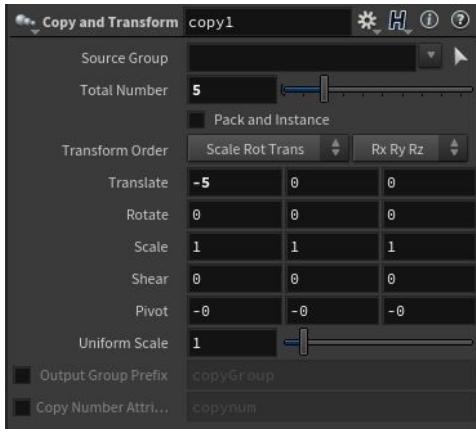
## Copy and Transform

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.

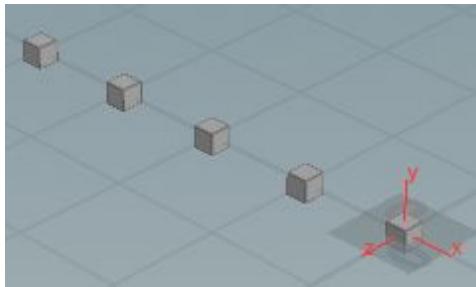
You can copy nodes in the network view. Drop a Copy and Transform node and connect whatever geometry you want to it...



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



**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.

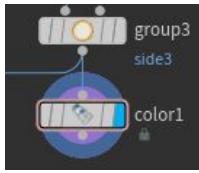


## Material Nodes

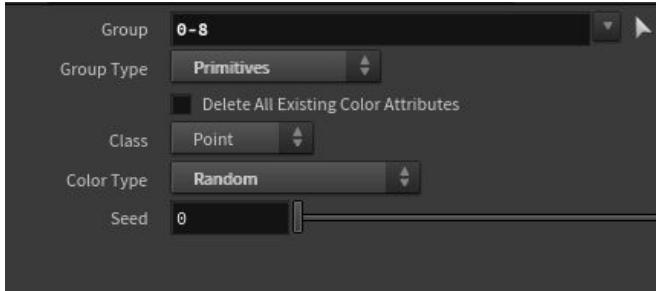
### Color

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.



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.

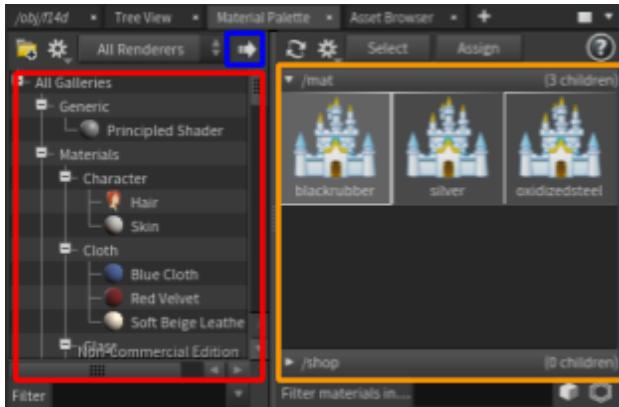
**NOTE:** If you go into the geometry spreadsheet, you can see the color show up under attributes for whatever component type you used for Class (Cd is the name of the attribute).

Node: color1	Group:	View	Intrinsics	Attributes:
0	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).

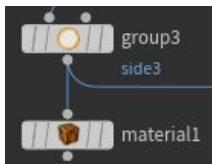


You can select from the **left pane**, and move over to the **right pane** by pressing the **right arrow** or by dragging-and-dropping over. These create the materials for you under the `/mat` namespace (or the `/shop` namespace which is collapsed below -- look at bottom lefthand side of **right pane**),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

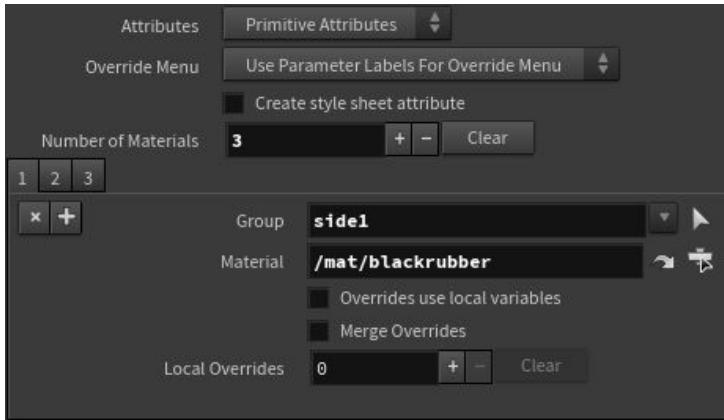
A discussion on materials is outside the scope of this document, but if you switch your network to `/mat` you can change the properties of these materials. It's very similar to Arnold shaders: sub-surface scattering, specular, reflectivity, diffuse, etc..

**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...



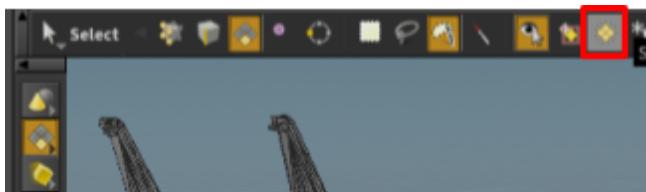
In the parameters pane, choose the components which you want the color to apply to. You can do so by clicking the arrow on the right of the Group. Once you've done that, select the material you want to use by clicking the right-most button next to Material (will open a selection menu of materials under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



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 next 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...



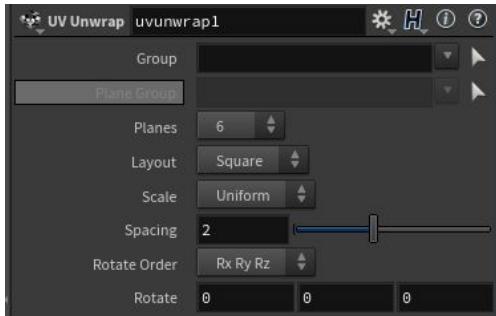
## UV Unwrap (Generate or correct UVs)

Attempts to generate UVs for the input.

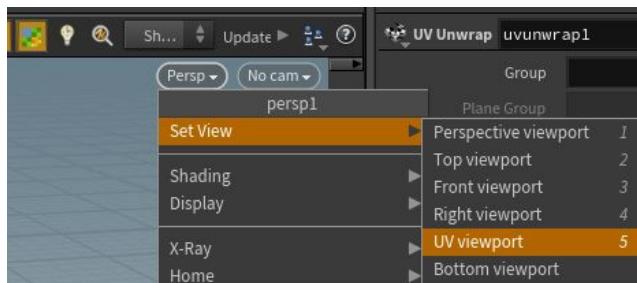
You can add/fix UVs in the network view. Drop a UV unwrap node and connect whatever geometry you want to it...



You can choose how the UVs get applied...



**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.

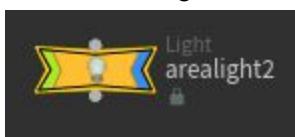
## Light and Camera Nodes

### All Light Nodes

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

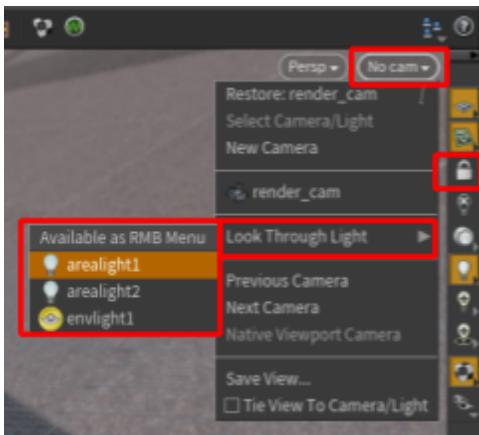
- Point Light → emits light equally in all directions
- Area Light → a rectangle/disk/whatever that emits light
- Geometry Light → similar to having a polygon mesh in Arnold that you set to emit light
- Distant Light → parallel rays, as if the sun were shining
- Environment Light → provide an HDR image 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 way 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

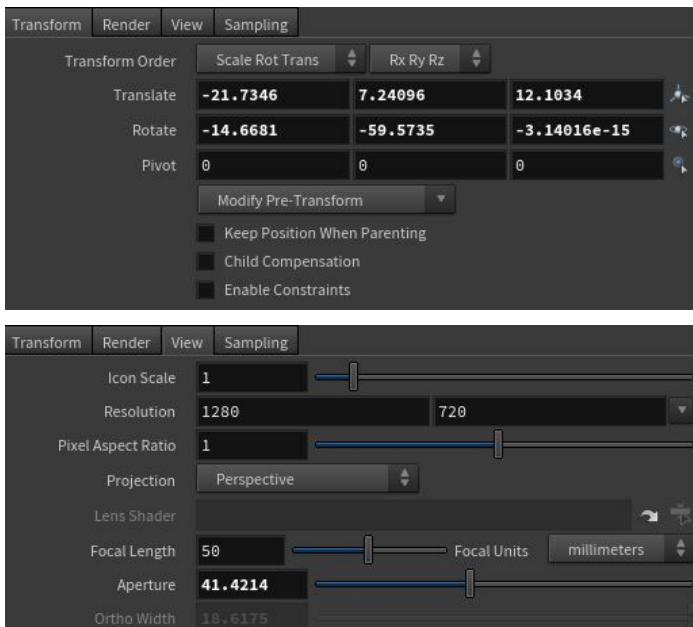
Adds a camera to the scene, which a Mantra node (render node) can use to render from.

**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.

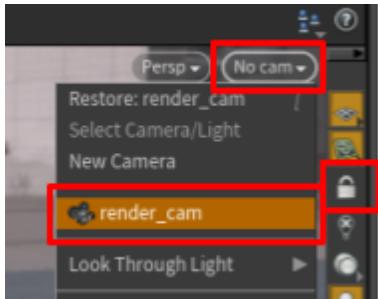


The camera properties contain the basics: positions, focal length, aperture, near clip, far clip, etc... Set these parameters however you see fit. Most of them are self-explanatory...



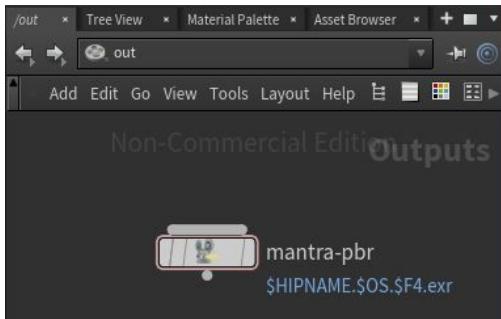
An easy way to position the camera is through the scene view. You can look through the camera and orient as if you're looking through it. To do this...

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.

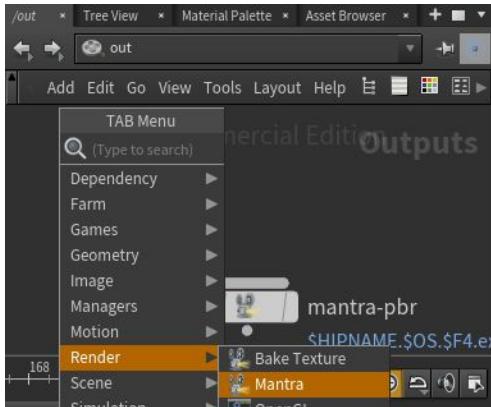
**NOTE:** Keep in mind that different renderers likely need customized materials and lights. You probably can't use Mantra's principal shader with Arnold and vice-versa.

**NOTE:** Pushing Houdini jobs to cuebot would be done through here via a custom node.

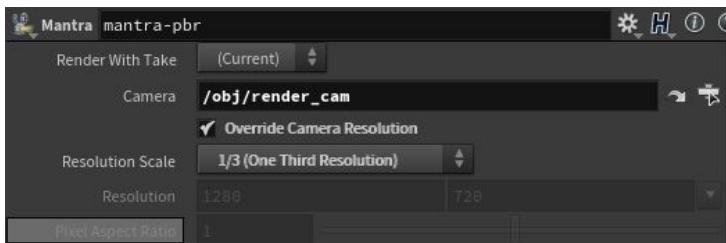
## Mantra

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

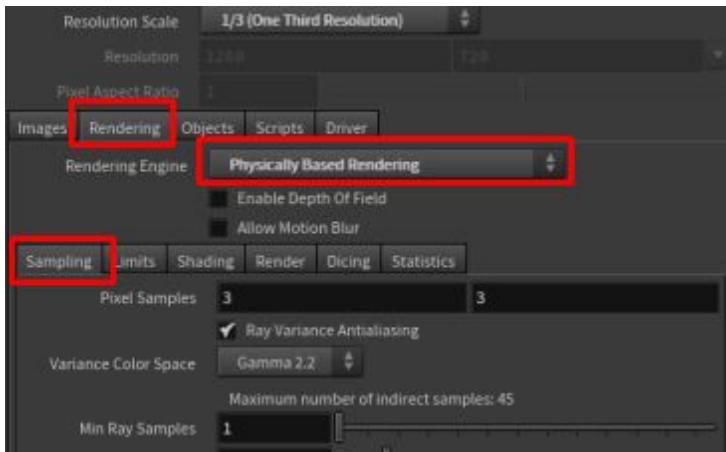
You can add a Mantra node by going to the network view, switching context to out, and inserting a Mantra node from the Tab menu...



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...

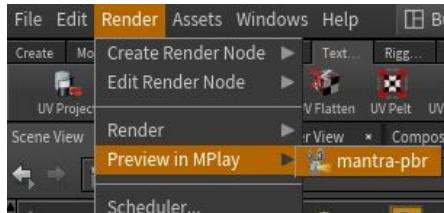


You can set the samples for the render (just like you do with Arnold) under Rendering -> Samples in the properties pane. You should probably set your rendering engine option to Physically Based Rendering...



**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)



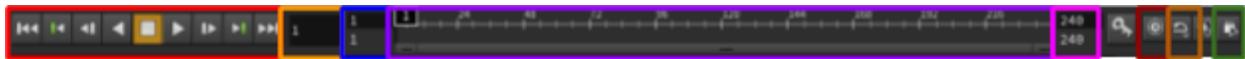
3. In the left toolbar of scene view... click the movie reel (last button in the toolbar) and choose the render node name



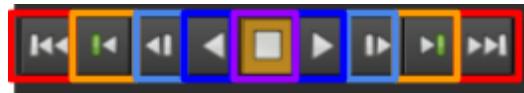
**NOTE:** MPlay is Houdini's version of fcheck (Maya's image playback thing)?????

## 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.

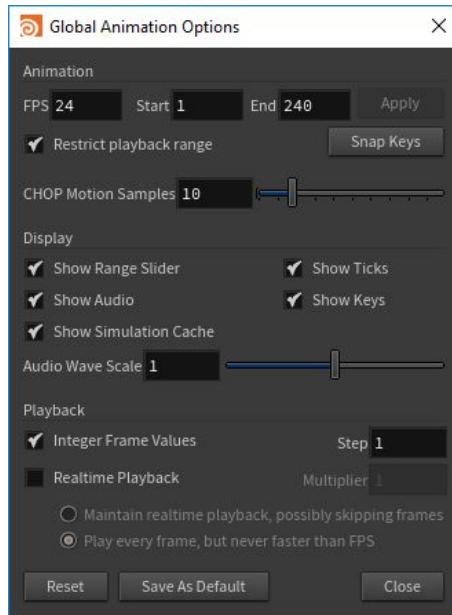
**Realtime** If enabled, will playback animation at however many FPS the animation is set to.

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)



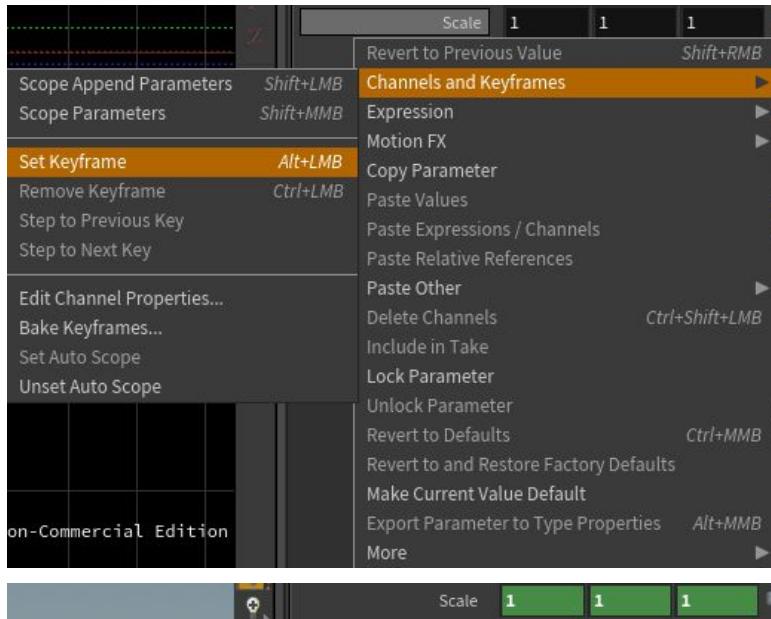
## Setting Keyframes

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.

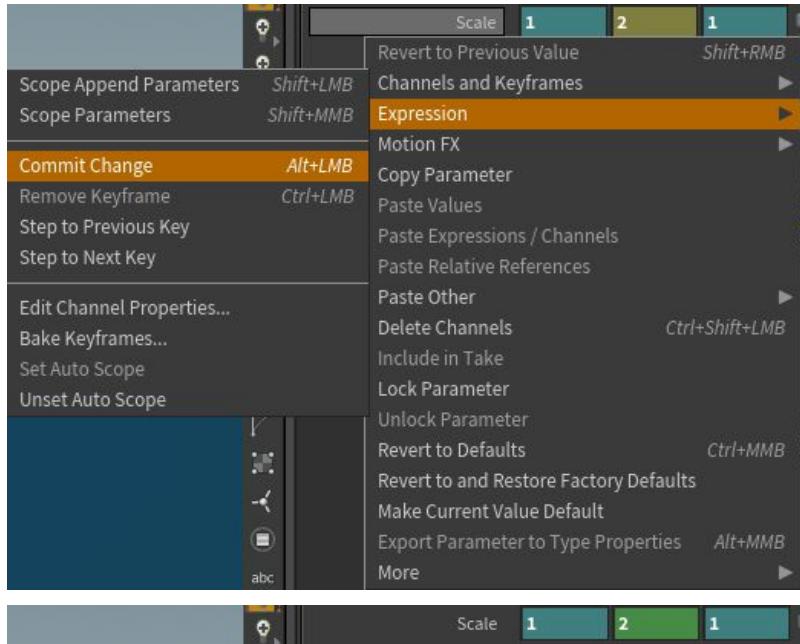
The property will turn green.



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.





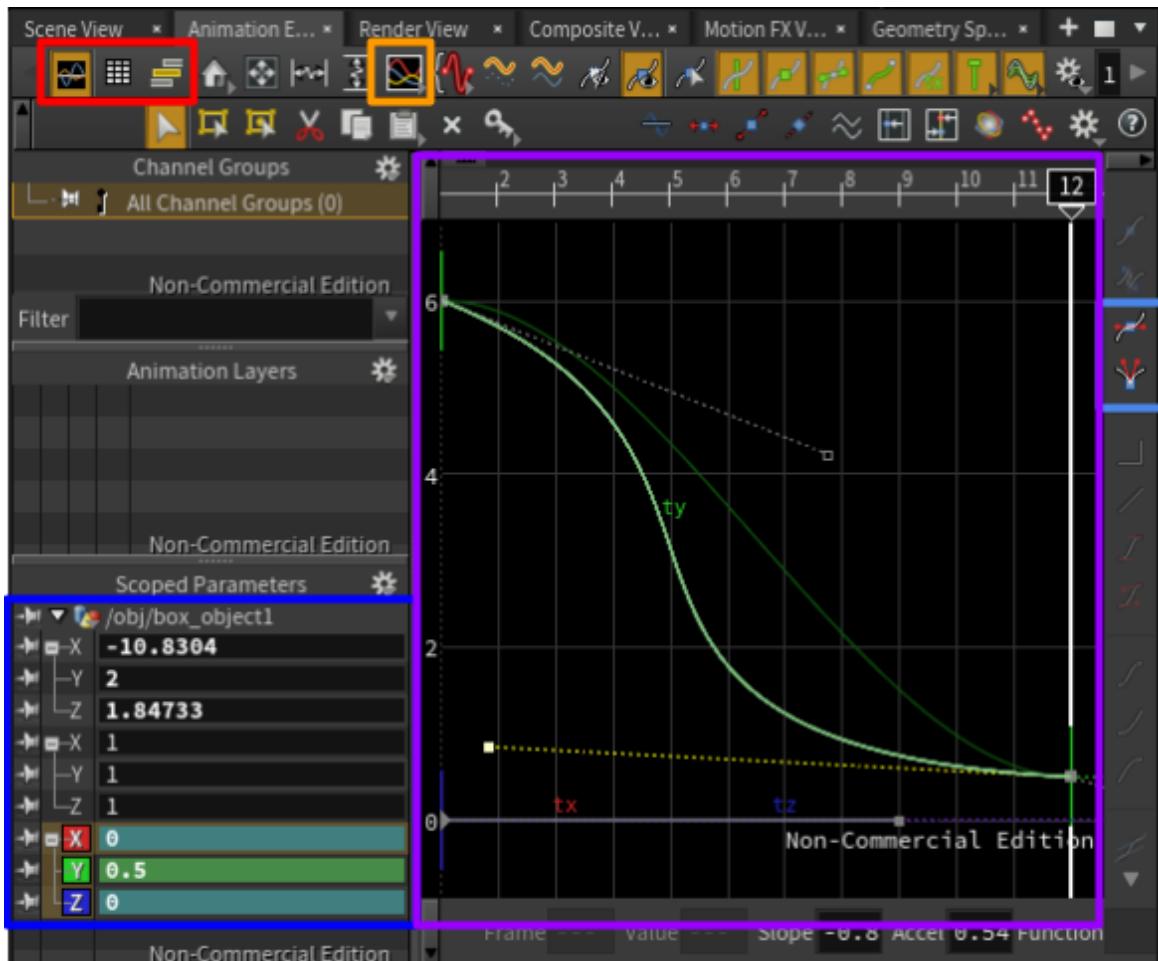
**NOTE:** Want to remove the keyframe-ability of a property?? Right click and select Delete Channel.

**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.

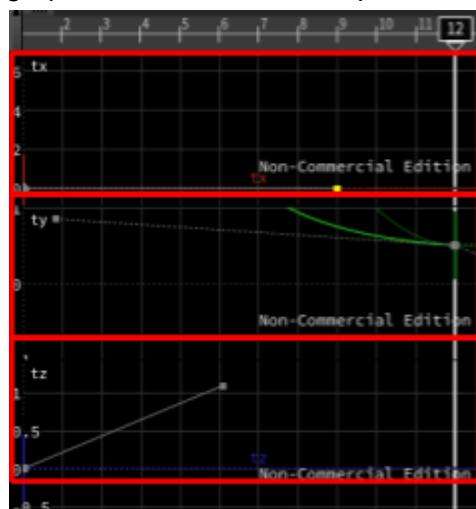


#### Mode

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

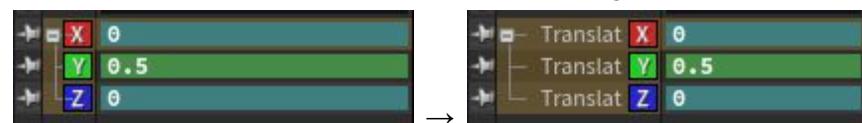
Similar to Maya's feature where you can break up the bezier handles. You can use this so to make it easier to have abrupt transitions between keyframes.



## 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....



## 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
Scale	1	1
Shear	0	0
Pivot	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.



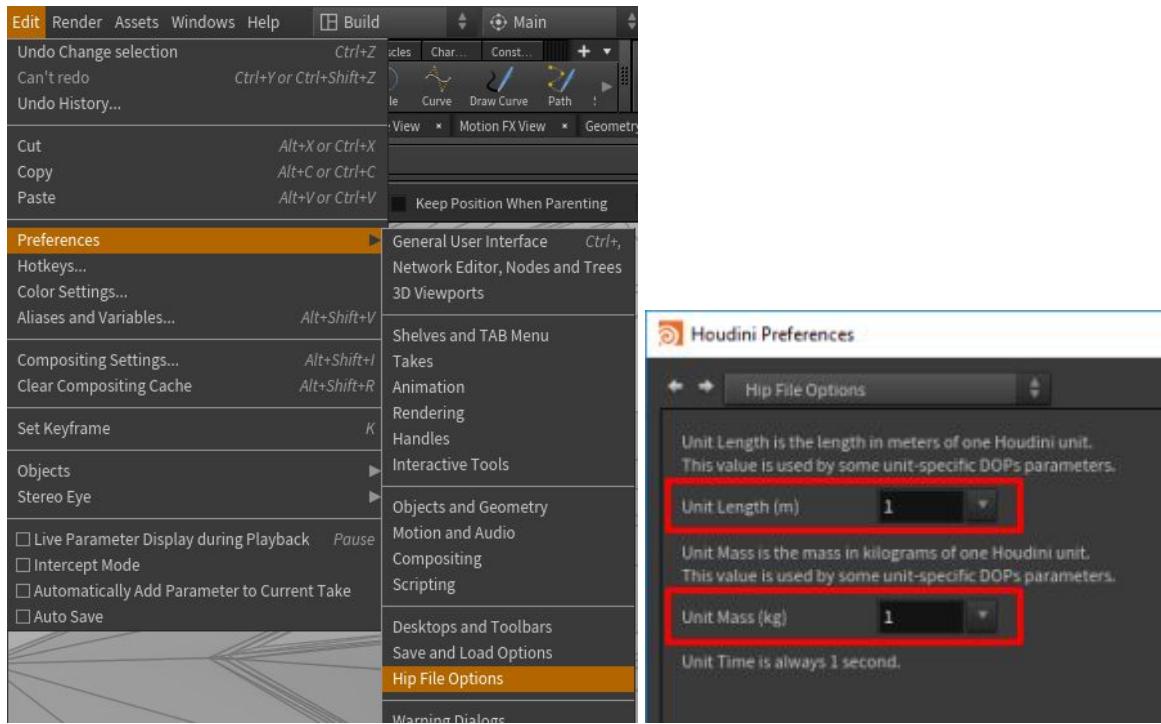
**NOTE:** Keyframe animations and animations from dynamics don't seem to be mutually exclusive? You can have some geo keyframed and others animated dynamically.

## 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...



**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.

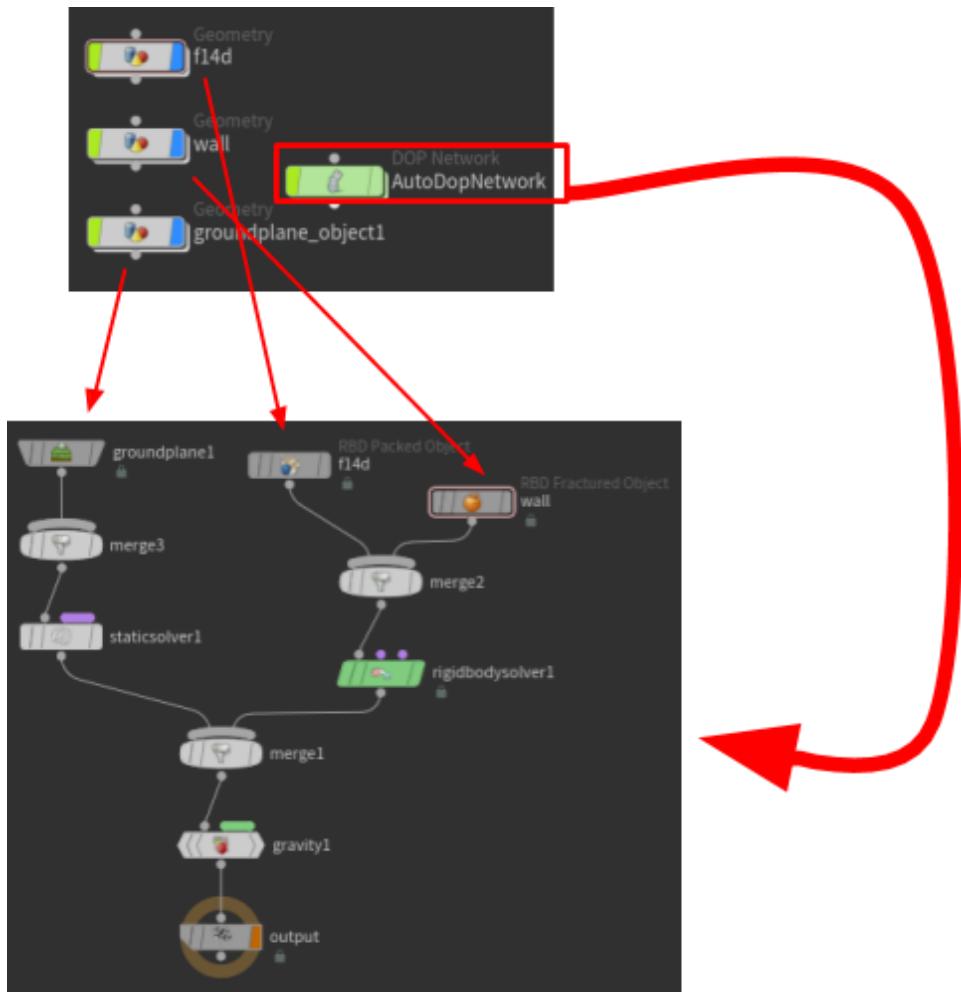
There are multiple types of rigid bodies available:

- 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

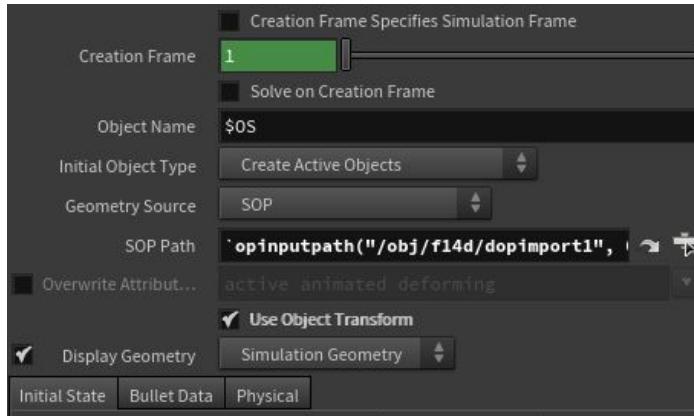
**NOTE:** Those are 4 we'll be discussing, but there's a ton of shit here and you should probably go over another lesson on it instead of messing around with it.

**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...



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	
<input type="checkbox"/> Unique Data Name		

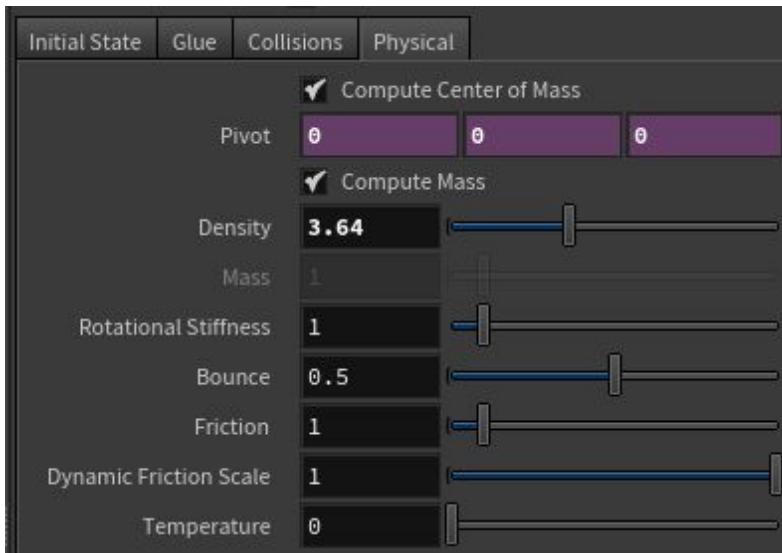
The important property here is Force, and by default it's set to the correct value (-9.8 meters in the Y-axis). If you want to change it, you can.

## 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.

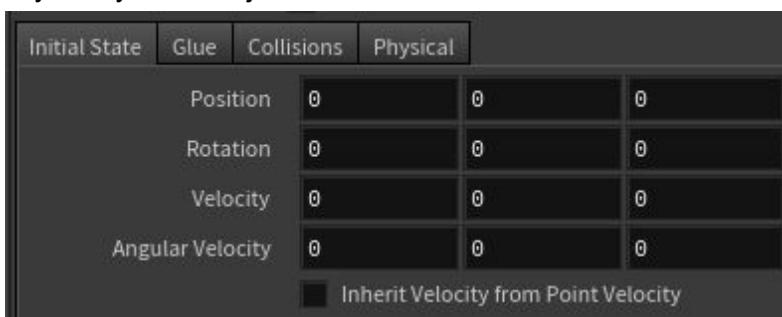
You can find them in tabs within the last section.

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



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...



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.

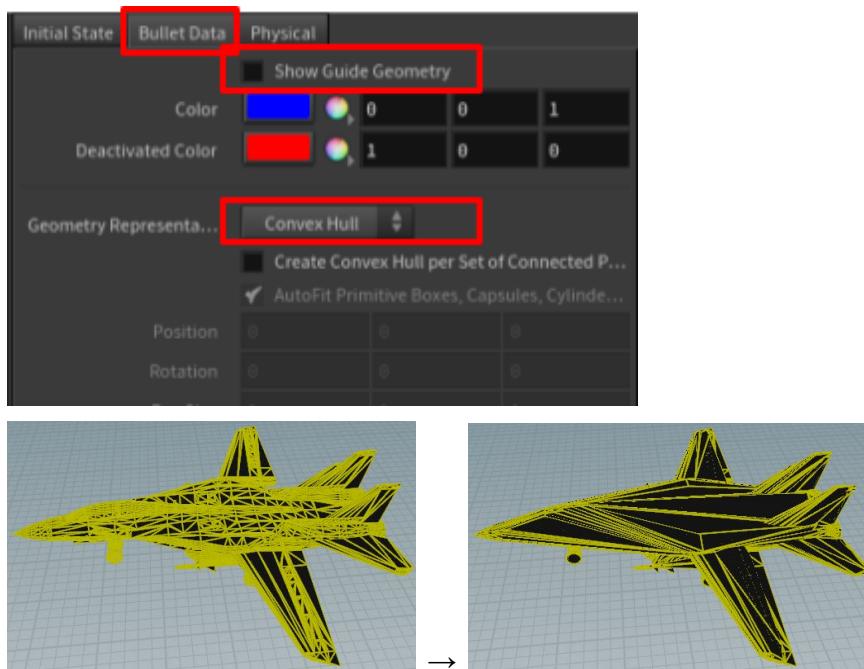
## Collision Geometry

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)



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.



