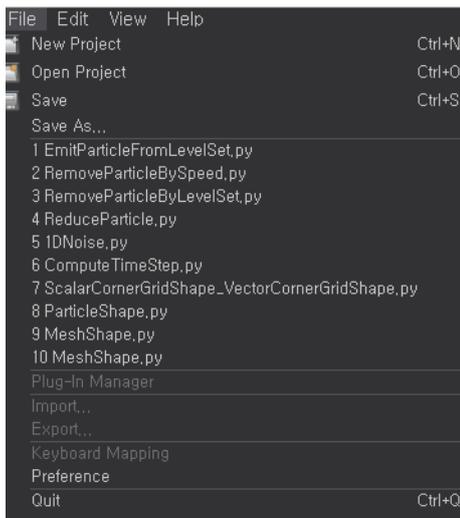


# User Interface

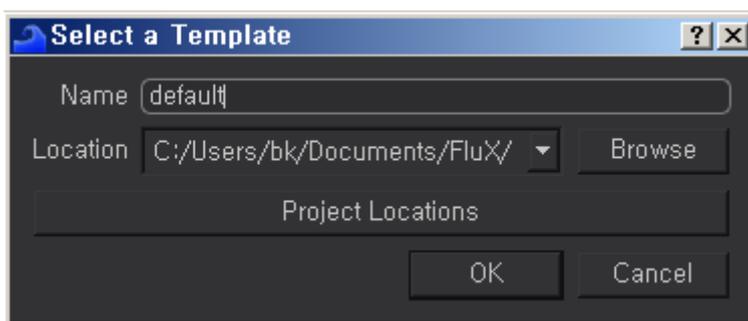
# Menu

In the menu bar, the overall functionality can be utilized such as creating a new project, loading saved projects, displaying each panel which relates to the operation of "Flux".

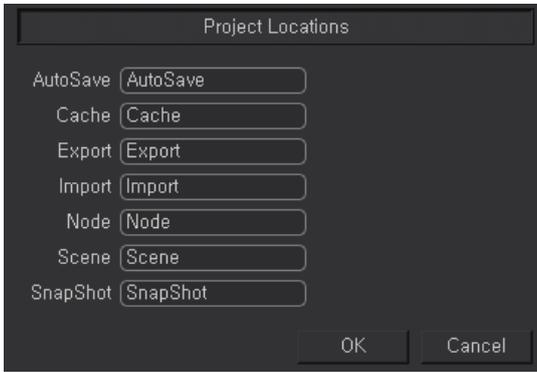
**File** : The functionality of managing overall project file is gathered.



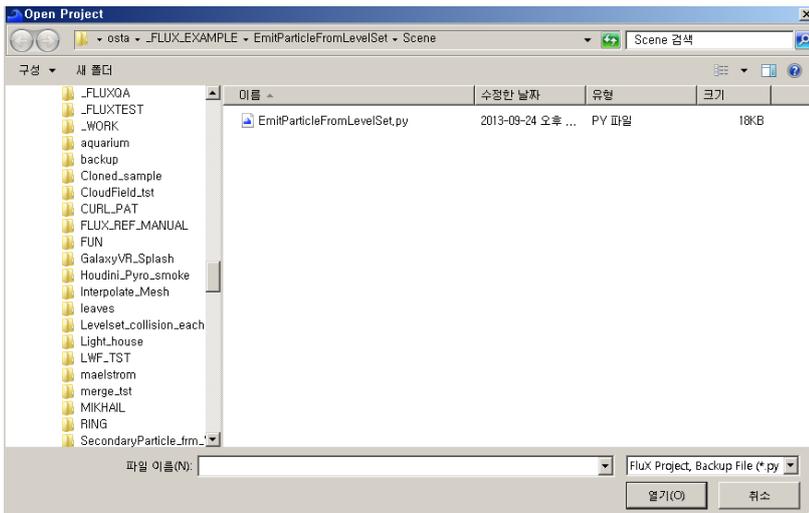
**New Project** : Creates a new project. You can specify the name of a new project and the path. The new project is created under the named directory in the specified location.



It is possible to specify the sub-path of the project as desired by clicking on the "Project Locations" button.



OpenProject : Opens the explorer window to load an existing project.

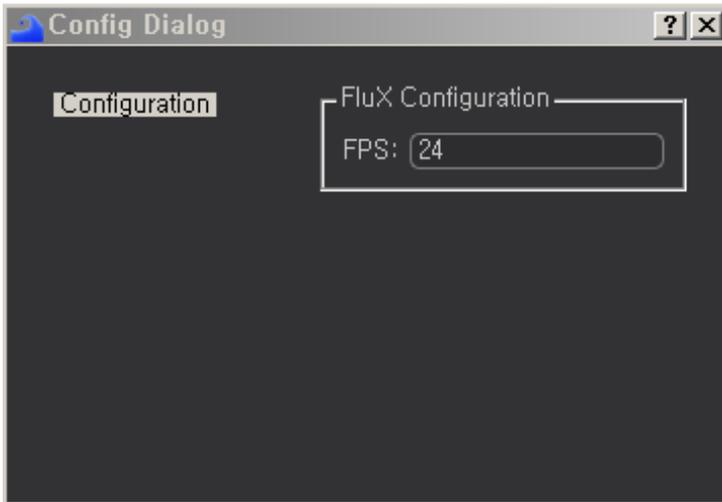


Save/SaveAs... : Saves the current project. The explorer window appears if the project has not been saved before or if the "Save As" menu is used to save in different path with different name.

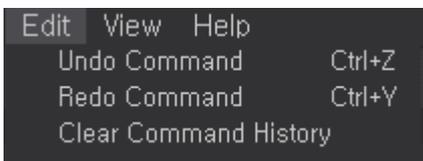
History : Shows the history of existing projects. You can open an existing project right away by selecting it from the "History" list.



Preference : Displays "ConFig Dialog" which the default setting of "Flux" can be set.



**Edit** : Undo performed command or Redo canceled command.

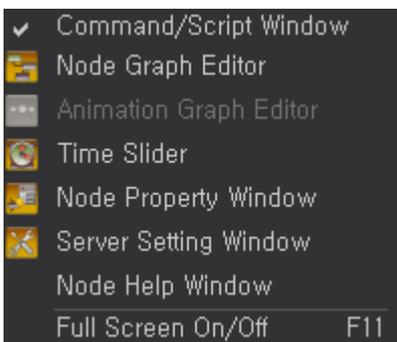


Undo Command : Undo performed command

Redo Command : Redoes canceled command.

Clear Command History : Deletes the command history. If you delete the command history, Undo/ Redo command does not work to the previously performed commands.

**View** : Shows or hides each interface that configures "Flux".



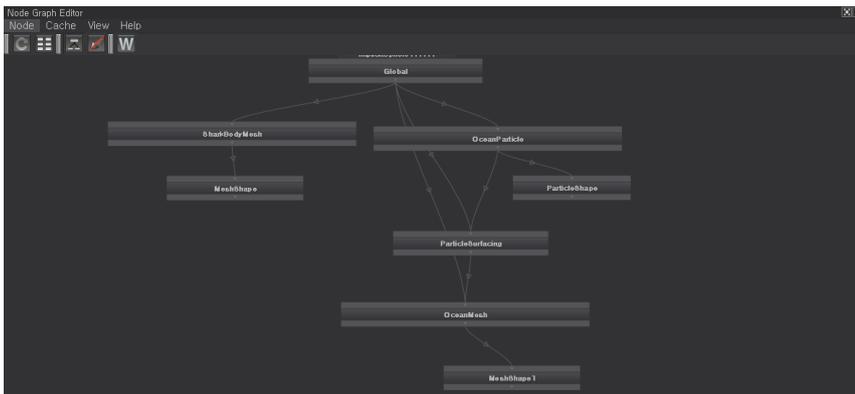
Command/Script Window : Shows or hides "Command/Script Window". In the command/Script Window, the input logs of "Flux" can be verified, and by entering the general "Python" & the exclusive "Python" command, In the command/Script Window can be executed.

```

Command/Script Window
File Edit History Command Help
Command History Window Find
Command History Window
Global.undoCmd()
# Empty Do History
Python1.py- EmitParticleFromLevelSet.py EmitParticleFromLevelSet.py SharkMesh.py
1 Global.cleanup();
2
3 Global.doCmd "CreateNode Cache SharkBodyMesh";
4 Global.doCmd "CreateNode MeshShape MeshShape";
5 Global.doCmd "CreateNode Cache OceanParticle";
6 Global.doCmd "CreateNode ParticleSurfacing ParticleSurfacing";
7 Global.doCmd "CreateNode Cache OceanMesh";
8 Global.doCmd "CreateNode ParticleShape ParticleShape";
9 Global.doCmd "CreateNode MeshShape MeshShape1";
10
11 Global.doCmd "SetSingletonData Global nodePosition -1,00000
    0,0,0";
12 Global.doCmd "SetSingletonData Global enable true";
13 Global.doCmd "SetSingletonData Global CurrentFrame 100 USER";
14 Global.doCmd "ConnectPort Global CurrentFrame SharkBodyMesh
    CurrentFrame";
15 Global.doCmd "ConnectPort Global CurrentFrame OceanParticle
    CurrentFrame";
16 Global.doCmd "ConnectPort Global CurrentFrame OceanMesh
    CurrentFrame";
17 Global.doCmd "SetSingletonData Global NextFrame 100 USER";
18 Global.doCmd "ConnectPort Global NextFrame SharkBodyMesh
    NextFrame";
19 Global.doCmd "ConnectPort Global NextFrame OceanParticle
    NextFrame";

```

Node Graph Editor : Shows or hides "Node Graph Editor". "Node Graph Editor" is the main workspace in "Flux" where the simulation logic can be designed through the connections of available nodes.

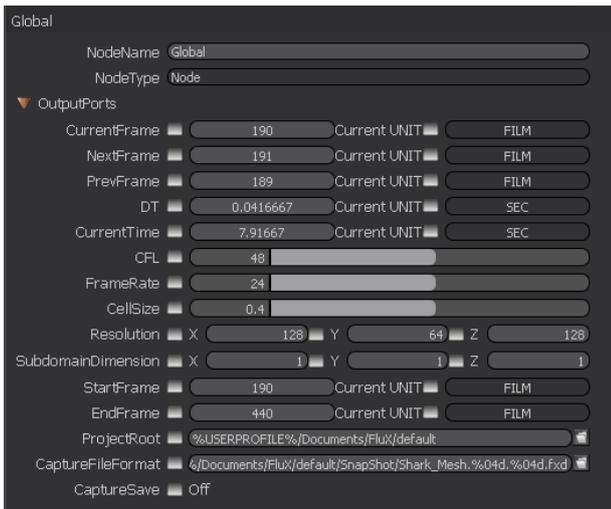


Animation Graph Editor : The parameters of node can be controlled by an animation curve. This feature will be added in the future.

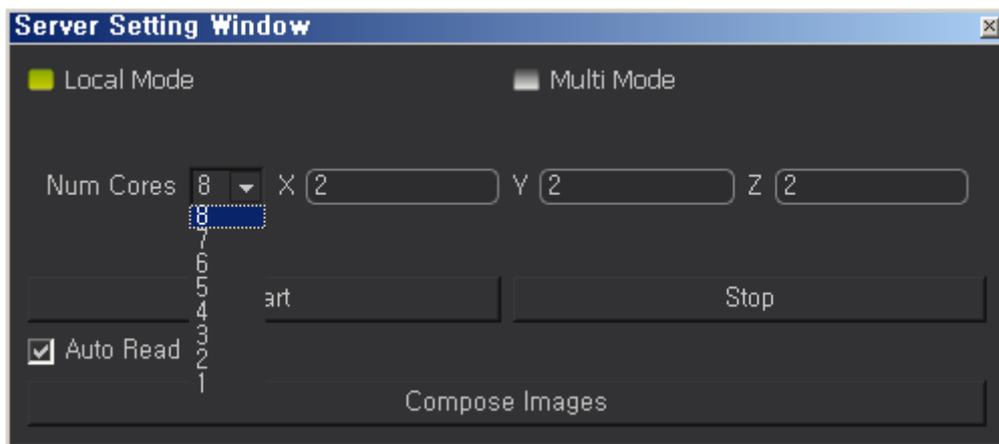
Time Slider : Shows or hides "Time Slider". The beginning and end of the simulation, frame rate, simulation progress, etc can be controlled in "Time Slider".



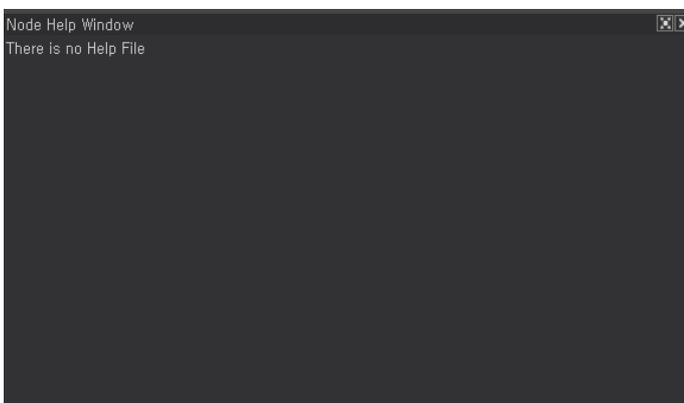
Node Property Window : Shows or hides "Node Property Window". The simulation aspect can be adjusted by entering the parameter details of node in "Node Property Window".



Server Setting Window : Shows or hides "Server Setting Window". The distributed simulation can be set and controlled by using the multiple cores or multiple computers over the network in "Server Setting Window".



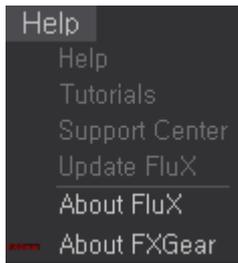
Node Help Window : Shows or hides "Node Help Window". A brief help information can be viewed on the selected node in "Node Help Window".



Full Screen On/Off : Maximizes the Flux to Full Screen or restores the screen back to window mode.

**Help** : Provides overall help and the information about the software and the development company and updates

new functions of "Flux",.



Help : Provides overall help.

Tutorials : Provides tutorial guide documents and exercise files.

Support Center : Shares the know-how information of "Flux", and you can also get help from "Support Center".

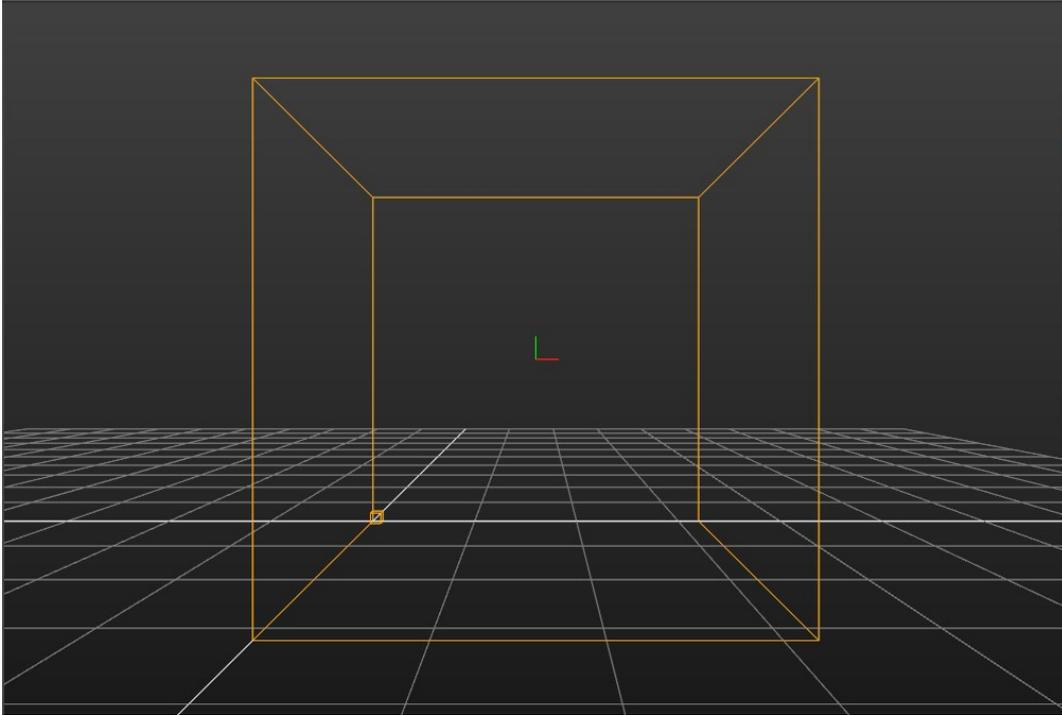
Update Flux : Checks online updates to download and update if there is a new version.

About Flux : Shows the basic information about "Flux".

About FXGear : shows the information about the development company of "Flux", "FXGear".

# Viewport

The imported geometries of scene and the simulation geometries are displayed and can be verified in "Viewport".



## Navigation :

You can hold down the "Alt" key and drag the right mouse button to zoom in or out the workspace. (This also works by scrolling the wheel mouse button.)

1. You can hold down the "Alt" key and drag the middle mouse button to pan (parallel translation – up, down, left, right) the 3D Scene.
2. You can hold down the "Alt" key and drag the left mouse button to rotate the workspace in all directions.

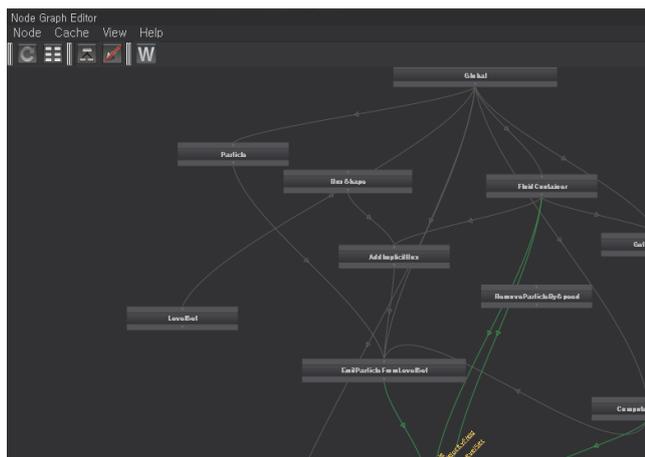
**Shortcut keys :** Manipulates the display status of "Viewport". The shortcuts will only work when "Viewport" is activated by clicking on an empty space on "Viewport"

1. X : The view aligns in X-axis direction
2. Y : The view aligns in Y-axis direction
3. Z : The view aligns in Z-axis direction
4. P : The view returns to Perspective or Orthographic mode.
5. , : Turns on/off the X, Y, Z coordinates reference axis.

6. [ : Turns on/off the X, Z axis guide line on the reference grid.
7. B : Turns on/off the Domain boundary box that displays the simulation area.
8. G : Turns on/off the reference grid.
9. F : Frames automatically the main area of the scene (the domain boundary box) in viewport.

## Node Graph Editor

The simulation logic can be designed through the connections of available nodes in "Node Graph Editor". "Node Graph Editor" is the main workspace in "Flux".



**Select** : Selects a specific node or a connection between ports. You can delete the selected nodes or connections by the "Delete" key on the keyboard.

1. Click : You can click a node or a connection to activate it
2. Drag : The quadrangle selection area will be appeared by dragging the left mouse button. The nodes that cross or are inside of the selection area are selected.

### Navigation :

1. You can drag the middle mouse button to move the workspace to up, down, left and right. (This also works by holding down the "Alt" key and dragging the left mouse button.)
2. You can hold down the "Alt" key and drag the right mouse button to zoom in or out the workspace. (This also works by scrolling the wheel mouse button.)

### Shortcut keys:

A : Frames all node graphs in the workspace. This is convenient when you want to see the whole node network.

E : Shows or hides all connections between the ports of nodes.

R : Shows or hides all names of ports and inputs & outputs of nodes. (Normally, only the names and ports of the selected nodes are displayed.)

G & F : Frames only the selected nodes in the nodegraph.

**Create node :** You can click on a node that you want to create from "Node List" at the bottom of "Node Graph Editor" to create a relevant node.

Primitive   Field   Particle   Mesh   Converter   Memory   File   Channel   Shape   Misc			
AdvectFoamParticle	AdvectParticle	AgeParticle	CollideParticleToContainer
CollideParticleToLevelSet	CollideParticleToMesh	DragParticle	EmitBubbleFromSplash
EmitFoamFromBubble	EmitFoamFromSplash	EmitParticleFromBox	EmitParticleFromLevelSet
EmitParticleOnMesh	EmitSecondaryParticle	EmitSecondaryParticleFLIP	EmitSecondaryParticlePLS
EmitSplashFromMarker	FLIPSolver	MergeParticle	MergeParticleCloned
ReduceParticle	RemoveFoamParticle	RemoveParticleByAge	RemoveParticleByLevelSet
RemoveParticleBySpeed	RemoveSplashParticle	SPHSolver	SolveBubbleParticle
SolveFoamParticle	SolveFreeParticle	SolveSecondaryParticle	SolveSplashParticle
UpdateParticleVelocity	VARSolver		

### Connection of node :

#### 1. Auto connection :

Most ports of some nodes are automatically connected when you create nodes. This is a preset connection to not do cumbersome work. If you need a direct control of connection, you can delete the existing connection and connect it with the port you want.

#### 2. Manual connection :

- A. When you hover your mouse over a node, connectable ports are displayed. It will also show the connectable ports when you move your mouse to another node that will be connected after clicking on the port you wish to connect. (At the same time, the dashed shape of the connection line is displayed.)
- B. If you click on the port of another node you want to connect, these two nodes will be connected.

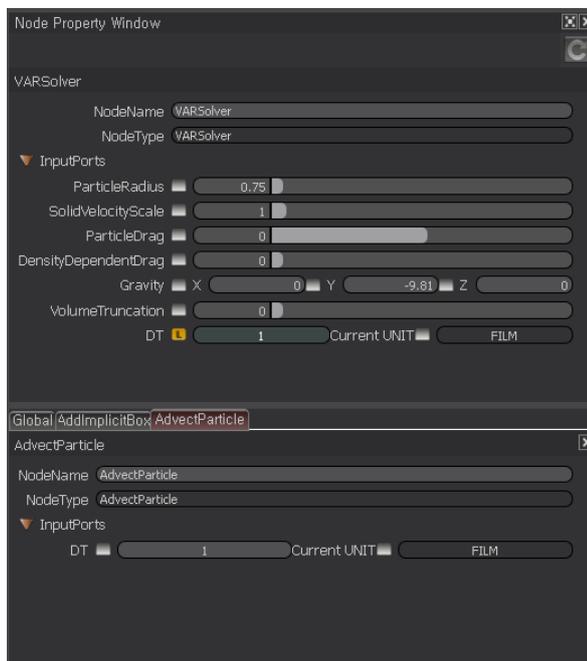


3. Delete connection : Click the connection between the ports correctly then press the "Delete" key to remove it.



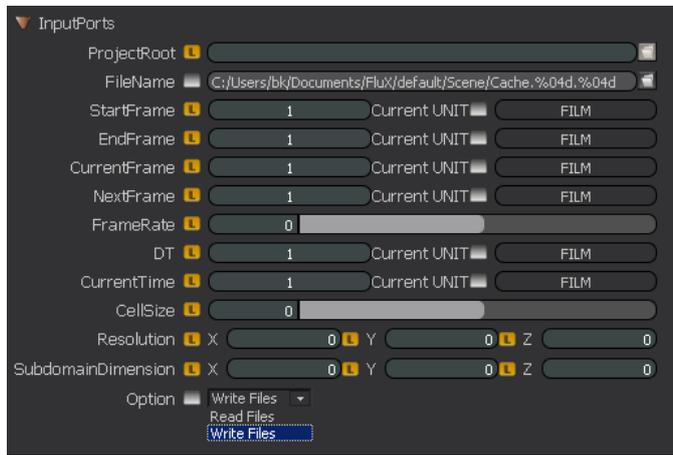
## Node Property Window

In "Node Property Window", the parameters of each node created from "Node Graph Editor" can be adjusted in detail.



### Adjusting parameters:

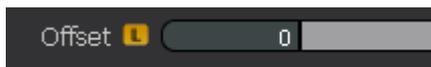
1. The parameter value, characters, and options can be entered through the input box next to the name of each parameter.



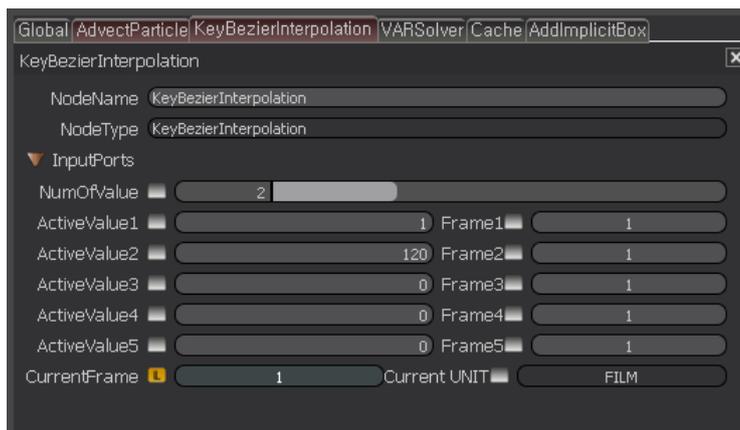
2. If there is a check box or slider bar next to the name or input box of each parameter, the parameter can be entered by adjusting the element.



3. If the parameter is already connected, it cannot be adjusted through "Node Property Window", and the box next to the parameter name changes to "[L]" sign.



**Node History** : Even if you finish editing with one node and then start editing the other node, the previously edited parameter of the node can be accessed quickly through the "Node History" tab at the straight bottom of "Node Property Window".



# Time Slider

From "Time Slider", functions related to the simulation progress/play can be controlled.



  : Sets the entire time range of the scene.



: Sets the start/end frames of the simulation/play.



: Displays the current frame. You can move forward/backward frame-by-frame by clicking the +/- button.



: Sets the FPS (Frames Per Second) of the simulation.



: Initialize the simulation state.



: A common play button combination. From left to right - go to the start frame, play backwards, play forwards, go to the end frame, loop/once play toggle buttons.

## Command Line, Command/Script Window

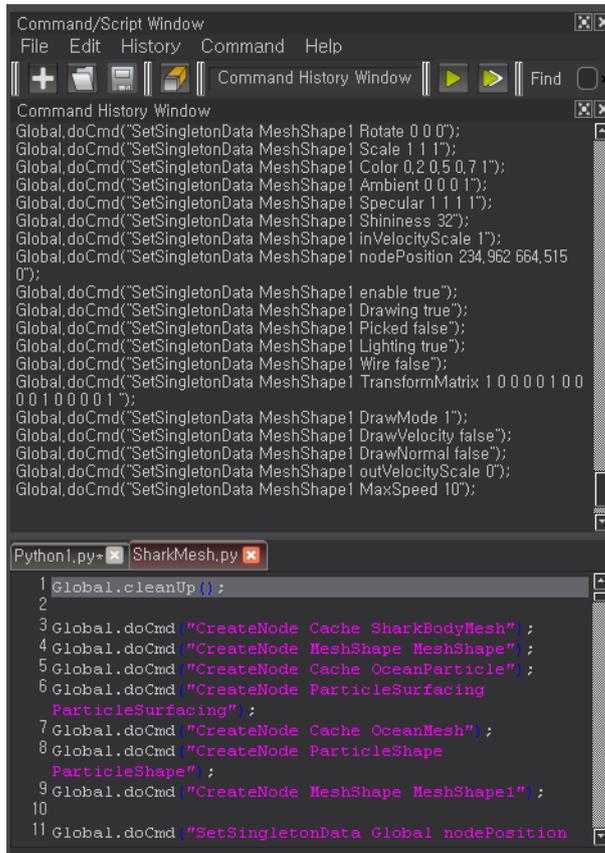
From Command Line, Command/Script Window, you can control "Flux" directly by typing the python command which "Flux" supports. In addition, general python commands can also be used.

Command Line : Python commands can be entered in the command line on the left, the output of "Flux" can be verified in the output line on the right. (If the command is not an exclusive command of "Flux", the output may not work properly. In this case, it is recommended to use "Command/Script Window".)



Command/Script Window : You can activate "Command/Script Window" by clicking its button on the right side of the "Time Slider". The simulation logic of "Flux" is saved as "Python Script". Most of the functionality of "Flux" (file

management, simulation, logic configuration, parameter editing, etc.) can be utilized through "Command / Script Window" without going through "GUI".



The image shows a screenshot of a software interface with two windows. The top window is titled "Command/Script Window" and has a menu bar with "File", "Edit", "History", "Command", and "Help". Below the menu bar is a toolbar with icons for adding, deleting, and running commands, along with a "Command History Window" button and a "Find" search box. The main area of this window displays a list of commands, all starting with "Global.doCmd". The bottom window is titled "Python1.py" and "SharkMesh.py" and shows a Python script with 11 lines of code. The code includes cleanup, node creation for SharkBodyMesh, MeshShape, OceanParticle, ParticleSurfacing, OceanMesh, ParticleShape, and MeshShape, and setting singleton data for nodePosition.

```
Global.doCmd("SetSingletonData MeshShape1 Rotate 0 0 0");
Global.doCmd("SetSingletonData MeshShape1 Scale 1 1 1");
Global.doCmd("SetSingletonData MeshShape1 Color 0.2 0.5 0.7 1");
Global.doCmd("SetSingletonData MeshShape1 Ambient 0 0 0 1");
Global.doCmd("SetSingletonData MeshShape1 Specular 1 1 1 1");
Global.doCmd("SetSingletonData MeshShape1 Shininess 32");
Global.doCmd("SetSingletonData MeshShape1 inVelocityScale 1");
Global.doCmd("SetSingletonData MeshShape1 nodePosition 234,962 664,515 0");
Global.doCmd("SetSingletonData MeshShape1 enable true");
Global.doCmd("SetSingletonData MeshShape1 Drawing true");
Global.doCmd("SetSingletonData MeshShape1 Picked false");
Global.doCmd("SetSingletonData MeshShape1 Lighting true");
Global.doCmd("SetSingletonData MeshShape1 Wire false");
Global.doCmd("SetSingletonData MeshShape1 TransformMatrix 1 0 0 0 1 0 0 0 1 0 0 0 1");
Global.doCmd("SetSingletonData MeshShape1 DrawMode 1");
Global.doCmd("SetSingletonData MeshShape1 DrawVelocity false");
Global.doCmd("SetSingletonData MeshShape1 DrawNormal false");
Global.doCmd("SetSingletonData MeshShape1 outVelocityScale 0");
Global.doCmd("SetSingletonData MeshShape1 MaxSpeed 10");

1 Global.cleanup();
2
3 Global.doCmd "CreateNode Cache SharkBodyMesh";
4 Global.doCmd "CreateNode MeshShape MeshShape";
5 Global.doCmd "CreateNode Cache OceanParticle";
6 Global.doCmd "CreateNode ParticleSurfacing
  ParticleSurfacing";
7 Global.doCmd "CreateNode Cache OceanMesh";
8 Global.doCmd "CreateNode ParticleShape
  ParticleShape";
9 Global.doCmd "CreateNode MeshShape MeshShape1";
10
11 Global.doCmd "SetSingletonData Global nodePosition
```

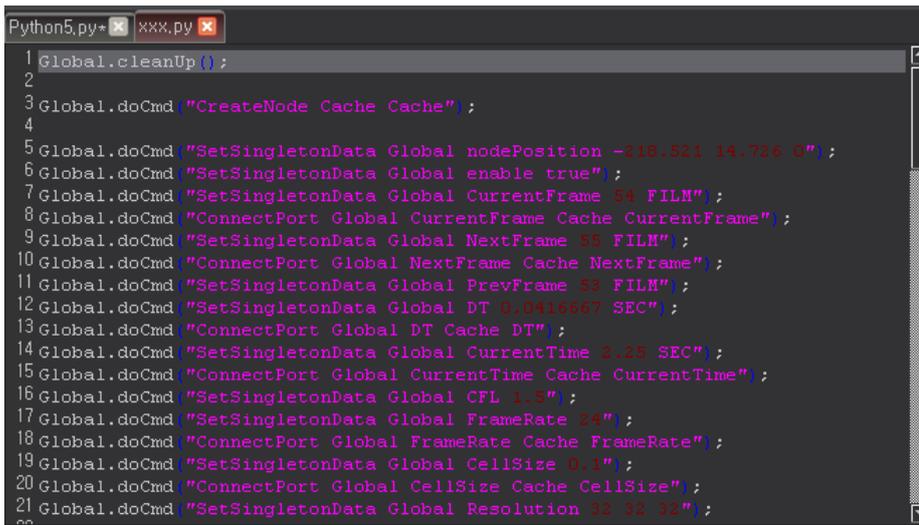
**Command History Window** : Displays each command entered and the history of output of FluX when a command is entered.

#### Menu :

1. File : Creates, imports and saves script files. Using a tab interface can manage multiple "Python Scripts", and remember the history of the files those were used.
2. Edit : Performs the script editing functions (cut, paste, copy, find, replace, etc.). In addition, it also has a function of organizing the "Script" and "History".
3. History : Toggles "Command History Window".
4. Command : Runs the entered or imported script, and manages the tab interface.

Script Window : Is a text editor that runs or edit the entered or imported script. If you use multiple script, separate

each script by tabs.



```
Python5.py* x xxx.py x
1 Global.cleanUp();
2
3 Global.doCmd "CreateNode Cache Cache";
4
5 Global.doCmd "SetSingletonData Global nodePosition -218.521 14.726 0";
6 Global.doCmd "SetSingletonData Global enable true";
7 Global.doCmd "SetSingletonData Global CurrentFrame 54 FILM";
8 Global.doCmd "ConnectPort Global CurrentFrame Cache CurrentFrame";
9 Global.doCmd "SetSingletonData Global NextFrame 55 FILM";
10 Global.doCmd "ConnectPort Global NextFrame Cache NextFrame";
11 Global.doCmd "SetSingletonData Global PrevFrame 53 FILM";
12 Global.doCmd "SetSingletonData Global DT 0.0416667 SEC";
13 Global.doCmd "ConnectPort Global DT Cache DT";
14 Global.doCmd "SetSingletonData Global CurrentTime 3.25 SEC";
15 Global.doCmd "ConnectPort Global CurrentTime Cache CurrentTime";
16 Global.doCmd "SetSingletonData Global CFL 1.5";
17 Global.doCmd "SetSingletonData Global FrameRate 24";
18 Global.doCmd "ConnectPort Global FrameRate Cache FrameRate";
19 Global.doCmd "SetSingletonData Global CellSize 0.1";
20 Global.doCmd "ConnectPort Global CellSize Cache CellSize";
21 Global.doCmd "SetSingletonData Global Resolution 32 32 32";
~
```

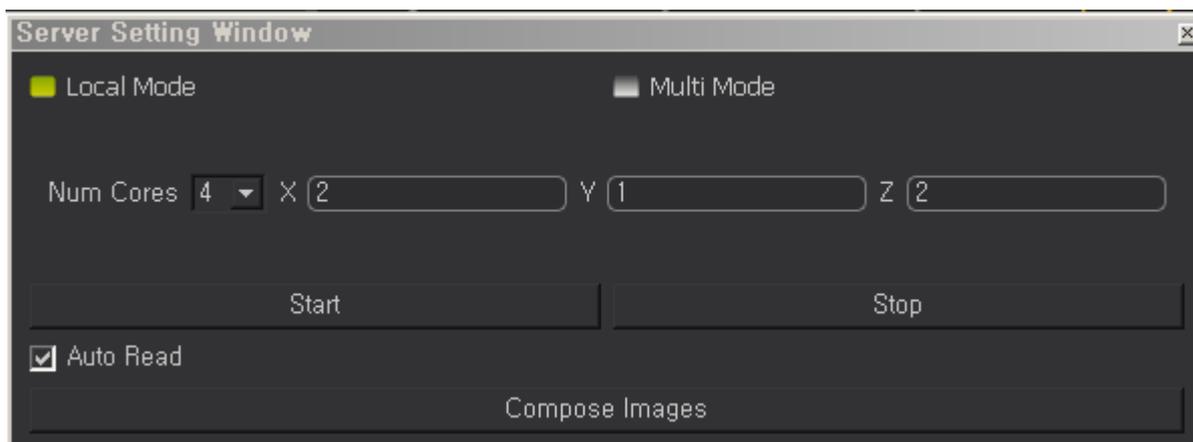
You can control the functions of "Flux" with exclusive "Flux" commands and also the standard Python commands and scripts those are not related to "Flux" can be controlled.(Python 2.7.7 based)

## Server Setting Window

Local and network MPI simulation can be set from the Server Setting Window.

Using MPI simulations performs outstanding speed improvement than the existing single CPU simulation using the play button.

### Local MPI Setting Window



Local Mode CheckBox / Multi Mode Checkbox : Is switchable "MPI Setting Window" between Local and Multi mode.

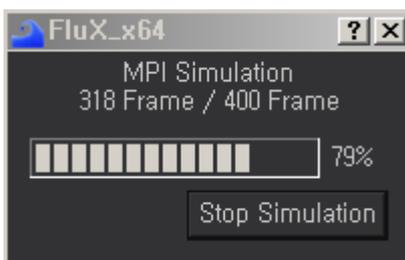
Num Cores : shows number of usable "physical" cores. The number of cores which will be used for the simulation can be intentionally assigned by selecting from the drop-down menu.

X/Y/Z Domain : If you proceed with MPI simulation, the number of cores allocated must be divided properly to match the simulation domain area. (The initial value is adjusted automatically to fit in core number) At this time, mark the divisions of each axis of coordinate direction. The division value can be adjusted according to user's intention, however, the multiplication of each division must match with number of cores allocated in simulation.

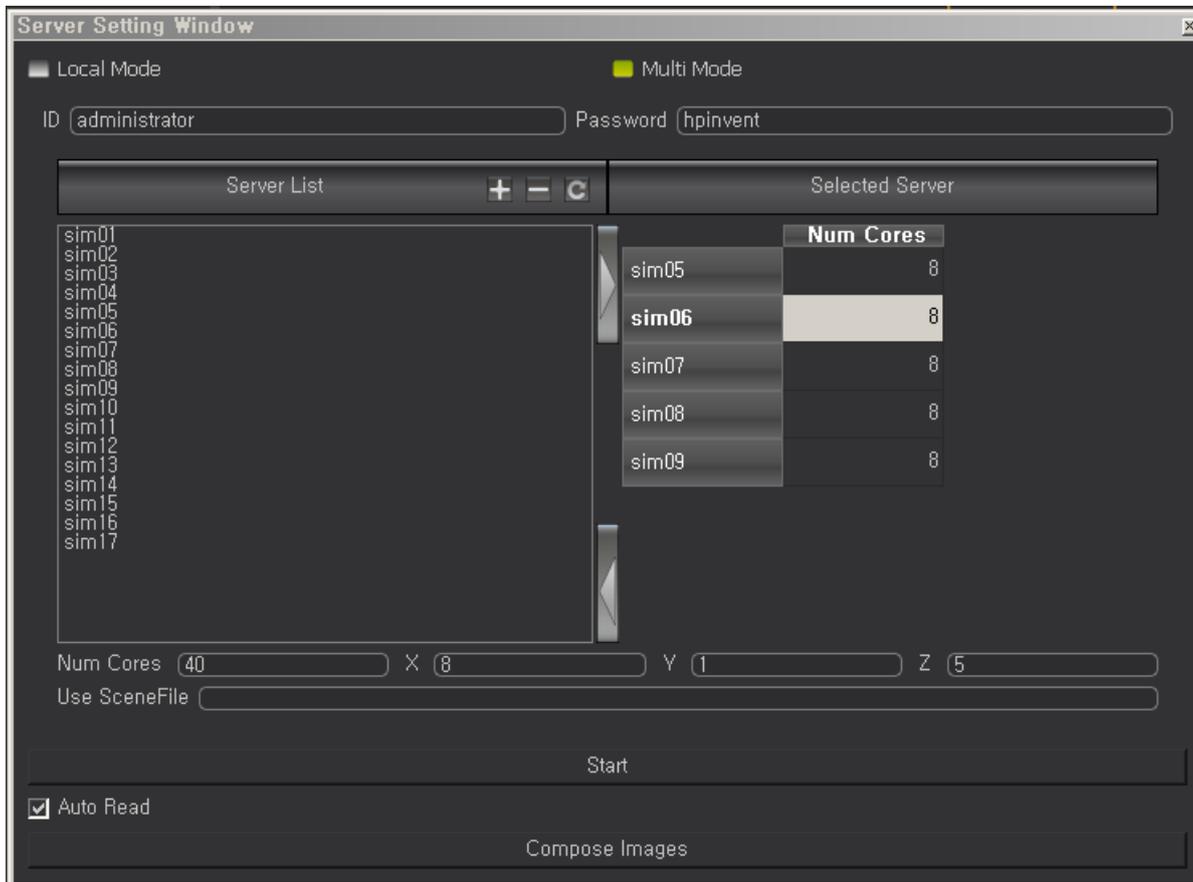
(TIP: X/Y/Z Resolution of the Global node must be divided into the division value of each direction of MPI Setting Window. For example, if the Resolution of the Global node is X: 100, Y: 33, Z: 100, the division value of MPI Setting Window must be 2x1x2, 10x3x10. (If not, an error may occur.)

Start/Stop Button : Starts or ends MPI simulation. The separate progress monitor window is displayed when you start MPI simulation (The figure below). In this window, you can monitor the progress of the simulation, and also the simulation can be stopped in progress.

Auto Read CheckBox : Brings the simulation result and updates every frame in the viewport. If you leave this option turned on, the simulation becomes slow. This is recommended to use if you need to check every frame for the intermediate results.



Network MPI Setting Window



Multi Mode Checkbox / Local Mode CheckBox : Are switchable MPI Setting Window between Local and Multi mode.

ID Field : The user ID is set to access Flux Simulation Server on the network.

Password Field : The user password is set to access Flux Simulation Slave on the network.

Server List : The available servers can be added/deleted by using the + / - buttons. If MPI simulation is run once, the added server is saved and also it is automatically included in the list. The list on the left which is already registered displays the list of the available servers. You can select several servers from the left / right list and add / delete by using ">", "<" buttons. Num Cores field of the right list (selected servers list) displays the number of assigned cores on each server. Depending on your needs, you can modify the number of cores.

Num Cores : Num Cores at the bottom of the list displays the sum of assigned cores from selected servers.

X/Y/Z Domain Number : Acts as same as the X/Y/Z Domain of local mode MPI.

Use SceneFile : Displays the current scene file that will proceed with the MPI simulation. If you proceed with the

network MPI simulation, each simulation server must be able to access the scene file with same path. In addition, the simulation works properly with using the universal path.

(e.g.: z:\FluxTest\Scene\test.py -> //nas-01/sim/FluxTest/Scene/test.py)

Start Button : Starts the network MPI simulation. The simulation monitor window is also displayed when you start the network MPI simulation. (Caution! When you need to stop the MPI simulation, press the "Stop Simulation" button to stop the behavior of MPI simulation server. Otherwise, it is possible that the simulation engine of the server can be still running.

