# Python Commands

82

Tutorials

# Python in Flux

FluX supports Python script. This chapter introduces Python commands for controlling FluX. Python commands work in the input window of "Command/Script Window".



Click the button above the play button in the bottom right corner to bring up the "Command/Script Window".



# Command format

Global.doCmd("command option option ...")

# Script Commands

#### Command: "ConnectPort"

Syntax: "ConnectPort NodeA Port1 NodeB Port2" Function: Connects NodeA's Port1 and NodeB's Port2. Example: Global.doCmd("ConnectPort VARSolver inParticle Particle Particle")

#### Command: "CreateNode"

Syntax: "CreateNode NodeType (NodeName)" Function: Creates a node that matches with the inputted node type as parameter. The second parameter will be the name of node. (Optional) Example: Global.doCmd("CreateNode Particle MyParticle")

#### Command: "DeleteNode"

Syntax: "DeleteNode NodeName" Function: Deletes the node that has the matching name with Nodename Example: Global.doCmd("DeleteNode ImplicitSphere1")

#### Command: "DisconnectPort"

Syntax: "DisconnectPort NodeA Port1 NodeB Port2" Function: Deletes the link between NodeA's Port1 and NodeB's Port2. Example: Global.doCmd("DisconnectPort Particle Particle DragParticle inParticle")

#### Command: "GetConnectedPorts"

Syntax: "GetConnectedPorts NodeA Port1" Function: Returns the nodes and ports linked to Port1 of NodeA in Node.Port format. Example: Global.doCmd("GetConnectedPorts FluidContainer CellSize")

#### Command: "GetInputPorts"

Syntax: GetInputPorts NodeA Function: Returns the list of NodeA's all input ports. Example: Global.doCmd("GetInputPorts VARSolver")

#### Command: "GetNodeBoundingBox"

Syntax: "GetNodeBoundingBox NodeA"

Function: Returns the bounding box of geometries that NodeA displays. Only works with displayable nodes. Example: Global.doCmd("GetNodeBoundingBox BoxShape")

## Command: "GetNodeHelpFileName"

Syntax: "GetNodeHelpFileName NodeA" Function: Displays the name of the Help file related to NodeA. Example: Global.doCmd("GetNodeHelpFileName Global")

## Command: "GetNodeQuery";

Syntax: "GetNodeQuery NodeA" Function: Returns all the information of the node name, catergory, summary, all input/output ports and the data type list of each port etc. Example: Global.doCmd("GetNodeQuery Root")

# Command: "GetNodeType";

Syntax: "GetNodeType NodeA" Function: Returns the type of NodeA. Example: Global.doCmd("GetNodeType MeshShape")

## Command: "GetOutputPorts";

Syntax: "GetOutputPorts NodeA" Function: Returns the output port list of NodeA. Example: Global.doCmd("GetOutputPorts MeshShape")

## Command: "GetPortList"

Syntax: "GetPortList NodeA" Function: Returns the list of all input/output ports of NodeA. Example: Global.doCmd("GetPortList ParticleShape")

# Command: "GetPortType";

Syntax: "GetPortType NodeA Port1" Function: Outputs the type ID of Port1 port of NodeA. Example: Global.doCmd("GetPortType ParticleShape Color")

#### Command: "GetSingletonData"

Syntax: "GetSingletonData NodeA Port1" Function: Returns the Port1 value of NodeA when it is one-dimensional variable. Example: Global.doCmd("GetSingletonData Global CFL")

#### Command: "IsInputPort";

Syntax: IsInputPort NodeA Port1 Function: Checks if Port1 of NodeA is an input port and returns 'True' if it is, 'False' if not. Example: Global.doCmd("IsInputPort Camera Pivot")

#### Command: "IsSceneNode";

Syntax: IsSceneNode NodeA Function: Checks if NodeA is SceneNode and returns 'True' if it is, 'False' if not. Example: Global.doCmd("IsSceneNode MeshShape")

#### Command: "RenameNode";

Syntax: RenameNode NodeA NodeB Function: Changes the node name that is named as NodeA to NodeB Example: Global.doCmd("RenameNode Cache1 HoHo")

#### Command: "SetSingletonData";

Syntax: "SetSingletonData NodeA Port1 ValueSet" Function: Sets the Port1 value of NodeA' as the value of ValueSet. Example: Global.doCmd("SetSingletonData VARSolver Gravity 0 -9 0")

#### Command: "TogCacheMode";

Syntax: "TogCacheMode true/false" Function: Changes the Cache mode to : true=Wirte->Read, false=Read->Write Example: Global.doCmd("TogCacheMode false")

# Using MPI Simulation

- A. Basic Setting
  - i. Create a new project in the public storage where the simulation farm is accessible. (Menu>File>New Project)
    - 1. Select the browser.

🐴 Select :	a Template	? X
Name	default	
Location	C:/Users/bk/Documents/FluX/ 💌	Browse
Template	None O Liquid Simulation O Gas S	Simulation
	Project Locations	
	ОК	Cancel

i. Create a folder of public storage which is accessible from the farm and press the OK button.



ii. Type the name of the project and press OK.



iii. Set up the Global node.



iv. Double click on the Global node to activate the parameter window.

Global							
NodeName	Glo	obal					
NodeType	(No	de					
Show Hidden	-	Off					
🔻 OutputPorts							
CurrentFrame	-	1		Current UNIT 🔲 (		FILM	
NextFrame	-	2		Current UNIT 🗖 (		FILM	
PrevFrame	-	0		Current UNIT 🗖 (		FILM	
DT	-	0.0416667		Current UNIT 🔲 (		SEC	
CurrentTime	-	0.0416667		Current UNIT 🗖 (		SEC	
CFL	-	1.5					
FrameRate	-	24					
CellSize	-	0.1		_			
Resolution	-	х 🤇 👘	32 🗖 Y 🤇		32 Z 🤇		32
SubdomainDimension	-	х 🤇 👘	1 Y 🤇		1) Z 🤇		1)
StartFrame	-	1		Current UNIT 🗖 (		FILM	
EndFrame	-	120		Current UNIT 🔲 (		FILM	
ProjectRoot	-	Z:/osta/MPI_doc_test/defa	ult				
CaptureFileFormat	-	Z:/osta/MPI_doc_test/defa	ult/SnapShot/	default.%04d.%04d	l.fxd		
CaptureSave	-	Off					

- v. Change the path to the network (universal) path. Z:/osta/MPI\_doc\_test/default -> //nas-01/new/osta/MPI\_doc\_test/default
- vi. Besides the main path of the Global node, it is recommended to use the relative path based on the root directory of the project

The project directory structure

🖃 🍌 MPI_doc_test
🖃 🌗 default
🌗 AutoSave
퉬 Cache
🎉 Export
📙 Import
UND NODE
Scene
🏨 ShapShot
"default" - ProjectRoot
"AutoSave" - AutoSave path
"Cache" – Simulation Cache path
"Export" – Saving path of gathered result
"Import" - External geometry file path for simulation
"Scene" – Saving path of simulation scene file
"SnapShot" – Saving path of snapshot image

vii. The following setting is the changed setting of the global node.

Global							
NodeName	Glo	bal					
NodeType	No	de					
Show Hidden		Off					
🔻 OutputPorts							
CurrentFrame	-	1		Current UNIT		FILM	
NextFrame	-	2		Current UNIT		FILM	
PrevFrame		0		Current UNIT		FILM	
DT	-	0.0416667		Current UNIT		SEC	
CurrentTime	-	0.0416667		Current UNIT		SEC	
CFL	-	1.5					
FrameRate	-	24					
CellSize	-	0.1					
Resolution	-	х 🦳 👘	32 Y (		32 Z		32
SubdomainDimension	-	х 🤇 👘	1 Y (		1) Z 🤇		1
StartFrame	-	1		Current UNIT		FILM	
EndFrame	-	120		Current UNIT		FILM	
ProjectRoot	-	//nas-01/new/osta/MPI_d	oc_test/defa	ult			
CaptureFileFormat	-	SnapShot/default.%04d.9	%04d.fxd				
CaptureSave		Off					

viii. Set up the simulation network.



ii. Set up the Cache node for saving.



iii. Set up the path of particle cache of the Cache node. (Verify if the mode is in Write mode when you proceed with the simulation.)

Cache					
NodeName	Ca	che			
NodeType	Ca	che			
Show Hidden	-	Off			
🔻 InputPorts					
ProjectRoot	L	//nas-01/new/osta/MPI_do	oc_test/default		
FileName	-	Cache/particle_Cache.%0	4d.%04d.fxp		
StartFrame	L	1	Current UNIT	FILM	
EndFrame	L	120	Current UNIT	FILM	
CurrentFrame	L	37	Current UNIT	FILM	
NextFrame	L	38	Current UNIT	FILM	
FrameRate	L	24			
DT	L	0.0111875	Current UNIT	SEC	
CurrentTime	L	1.54167	Current UNIT	SEC	
CellSize	L	0.1			
Resolution	L	Х (З	D L Y	30 U Z	30
SubdomainDimension	L	x 🤇 👘	1 U Y	1 U Z 🤇	1)
Option		Write Files 🛛 🔻			
V OutputPorts					

## B. MPI setting

i. Bring up Server Setting Window by clicking on the Server Setting Window button.



# ii. Server Setting Window

1. Local mode

Server Setting Window					×
📒 Local Mode			🔲 Multi Mode		
Num Cores 🛛 🖛 X 2		) Y (1		) Z (2	
	Start			Stop	
🗹 Auto Read					
		Compos	e Images		

#### 2. Multi mode

Server Setting Wind	0 <b>₩</b>			X
Local Mode		📒 Multi	Mode	
ID (administrator		Password (h	pinvent	
	Server List	+ - c	Selected Server	
sim01 sim02 sim04 sim05 sim05 sim07 sim08 sim09 sim10		Num	n Cores	
Num Cores () Use SceneFile (	X ()		( <u> </u>	<u> </u>
		Charle		
		Start		
🗹 Auto Read				
		Compose Images		

- iii. MPI setting local mode
  - 1. Setting the number of cores Set the number of cores of the workstation (the current version does not support the multi thread. Therefore, set the number of physical cores as default.)

Server Setting Window					X
📒 Local Mode			🔜 Multi Mode		
Num Cores 🛛 💌 X (2		) Y (1		) Z (2)	
	Start			Stop	
🗹 Auto Read					
		Compos	e Images		

- Adjust the simulation area to suit the situation
  X: 2, Y: 1, Z: 2
- iv. MPI Setting distributed processing mode
  - 1. Registering the server register the server that will be used for distributed processing

Server Setting Window	×
🔜 Local Mode	🛑 Multi Mode
ID (administrator Pas	sword (hpinvent
Server List 🛛 🛨 🗖 C	Selected Server
sim01 sim02 sim03 sim04 sim05 sim06 sim07 sim08 sim09 sim10	Num Cores
Num Cores (0 X (0	Y 0 Z 0
Use SceneFile (	
St	art
🗹 Auto Read	
Compose	e Images

2. Manually enter the address of the server, or type the name of server to register.

	? X
Add	Cancel
	? X
Add	Cancel
	Add Add

3. ID/Password setting – for all the servers those will be used must have the same ID and Password.

ID (	administrator	Password	(hpinvent )

4. Selecting the server - once the registration is completed, select the servers to use for the simulation.

Serv	er Setting Window				×
<b>–</b> L	ocal Mode		🛑 Multi Mode		
ID	administrator	) Pas	sword (hpinvent		
-					-
	Server List		Sele	cted Server	
	sim01			Num Cores	
	sim02 sim03 sim04		sim01		
	sim05 sim06		sim02		
	sim07 sim08 sim08		sim03		
	simu9 sim10		sim04		
			sim05		
N	lum Cores (40	X 8	Y (1	Z ( <u>5</u>	)
U	lse SceneFile (				)
		Sta	art		
	Nuto Read				
		Compose	Images		

5. Core setting – enter the number of physical core of each server.

Server	r Setting Window				×
🔳 Lo	cal Mode		📒 Multi Mode		
ID (a	administrator	Pas:	sword (hpinvent		
	Server List	<u>+</u> = c	Sele	cted Server	1
si	m01			Num Cores	-
SI Si	mU2 m03 m04		sim01	4	
si si	m05 m06		sim02	4	
si  si	m07 m08 		sim03	4	
si	m10		sim04	8	
			sim05		
Nu	m Cores (28)	X (7	☐ Y (1	Z (4	2
Us	e SceneFile (				ר
		Sta			
☑ Auto Read					
Compose Images					

6. Saving the server setting – after setting the simulation server, press the Start button. This will save the server list as "list.txt" in FluX installation directory.

📄 list,txt	// list.txt - 메모장	
船 machineid,exe	파일(F) 편집(E) 서식(O) 보기(V) 도움말(H)	
📄 pwd.txt	simO1 simO2 simO3 simO4 simO5 simO6 simO7 simO8 simO9 sim10	<u></u>
🚳 python27, dll		
🚳 PythonQt, dll		
🚳 QtCore4,dll		
🚳 QtGui4, dll		
🚳 QtOpenGL4,dll		
		-
	4	

7. Subdomain setting – set up the appropriate subdomain to suit the form of the scene

Num Cores (28) X (7 Use SceneFile (	Y 1 Z 4
	Start
🗹 Auto Read	
	Compose Images

- C. Scene file setting
  - i. Set the name of the scene file that will use MPI to proceed with the simulation.
  - ii. You must use the universal path.

Server Setting Windov	v				×
🔲 Local Mode			📕 Multi Mode		
ID (administrator		Pass	word hpinvent		
	Server List	- c	_	Selected Server	
sim01 sim02				Num Cores	
sim03  sim04			simU1	4	
sim05 sim0 <u>6</u>			sim02	4	
sim07  sim08  sim09			sim03	4	
sim10			sim04	8	
			sim05	8	
Num Cores (28	X (7		) Y (1	Z (4	
Use SceneFile ( <u>//na</u>	is-01/new/osta/MPI_doc_test/de	efault/Scene/	default, py		
		Star	t		
🗹 Auto Read					
		Compose	Images		

iii. Auto Read option – Auto Read also updates Viewport as the simulation proceeds. It is good to see the result but the simulation will run slower so it is recommended that you turn it off if not necessary.



- iv. Proceeding with MPI simulation When all settings are completed, press the Start button to proceed with (MPI) distributed simulation
  - 1. Local mode

Server Setting Window	×
📒 Local Mode	🗖 Multi Mode
Num Cores 4 💌 X 2	1 Z (2)
Start	Stop
🗹 Auto Read	
Cc	mpose Images

## 2. Multi mode

Server Setting Window			
🔜 Local Mode	📒 Multi Mode		
ID (administrator	Password (hpinvent		
Server List 🗧 🗧	C Selected Server		
sim01 sim02 sim03 sim04	sim01 4		
sim05 sim06	sim02 4		
sim07 sim08 sim09	sim03 4		
sim05 sim10	sim04 8		
	sim05 8		
Num Cores (28 X (7	Y (1 Z (4		
Use Scenerile (//nas-ut/new/osta/MPI_doc_test/defaut/	/scene/detault,py		
	Start		
☑ Auto Read			
Cc	ompose Images		

# D. Gathering the result

i. The result can be gathered by selecting the Read option. (the file extension of the particle = fxp and the mesh = tri)



ii. When gathering the result, set the subdomain list as if 8 cores = 0-7, 28 cores = 0-27.

#### E. Export

- i. Connect the Cache node to the gathered result and set the saving path, then press the Play button to start exporting as a single file.
- ii. When exporting the particle or mesh, using the Realflow format bin is recommended.
- F. Things you need to beware of
  - 1. Save the scene file before proceeding with the MPI simulation

2. If you need to stop the MPI simulation for any reason, you must use the Stop MPI button on the MPI simulation window.



3. With the Cache node, the Write mode switches to Read mode after the simulation starts, so setting the Read/Write mode of each Cache when working on the simulation requires careful attention.

# Resimulation

Flux Supports resimulation.

When simulation is interrupted or simulation range is expanded, it simply can continue with existing data from the former process with resimulation. Starts simulation from the beginning is not necessary. (Example file is available) Also, it can be used when another node to be set up in the middle of simulation process.

#### Simulation



First, sets simple scene and process simulation. The example simulation file only lasts 40 frames. (Example reference Resimulation\_Sample.py)

#### Resimulation

Processes resimulation after the end of the simulation frame. (Reference Resimulation\_Sample.resimpy)

Opens node property window of particle node and address existing simulation data routes at FileName field.



1. Checks ReSimulation at node property window of particle node.



2. Sets time slider start frame as one frame before resimulation's starting frame.



3. Processes simulation with play button or MPI window.

