

# Phase Flow - Water Fall — SimScale Platform Documentation documentation

## Step-by-Step

The project already has an uploaded mesh, which was created in a normal manner without any special assignments.

So, the tutorial starts with creating a new simulation set-up for multiphase flow. The two different phases are defined and assigned in the simulation setup to follow.

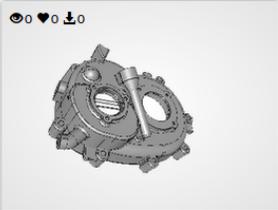
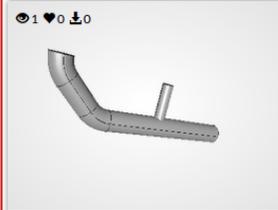
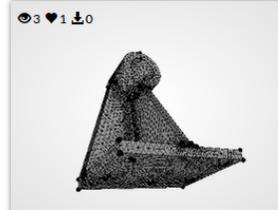
### Import tutorial project

- To start this tutorial, you have to import the tutorial project into your **'Dashboard'** via the link above.
- Alternatively, you can also add the tutorial project from the *'Public Projects'* library by searching for "tutorial" name.

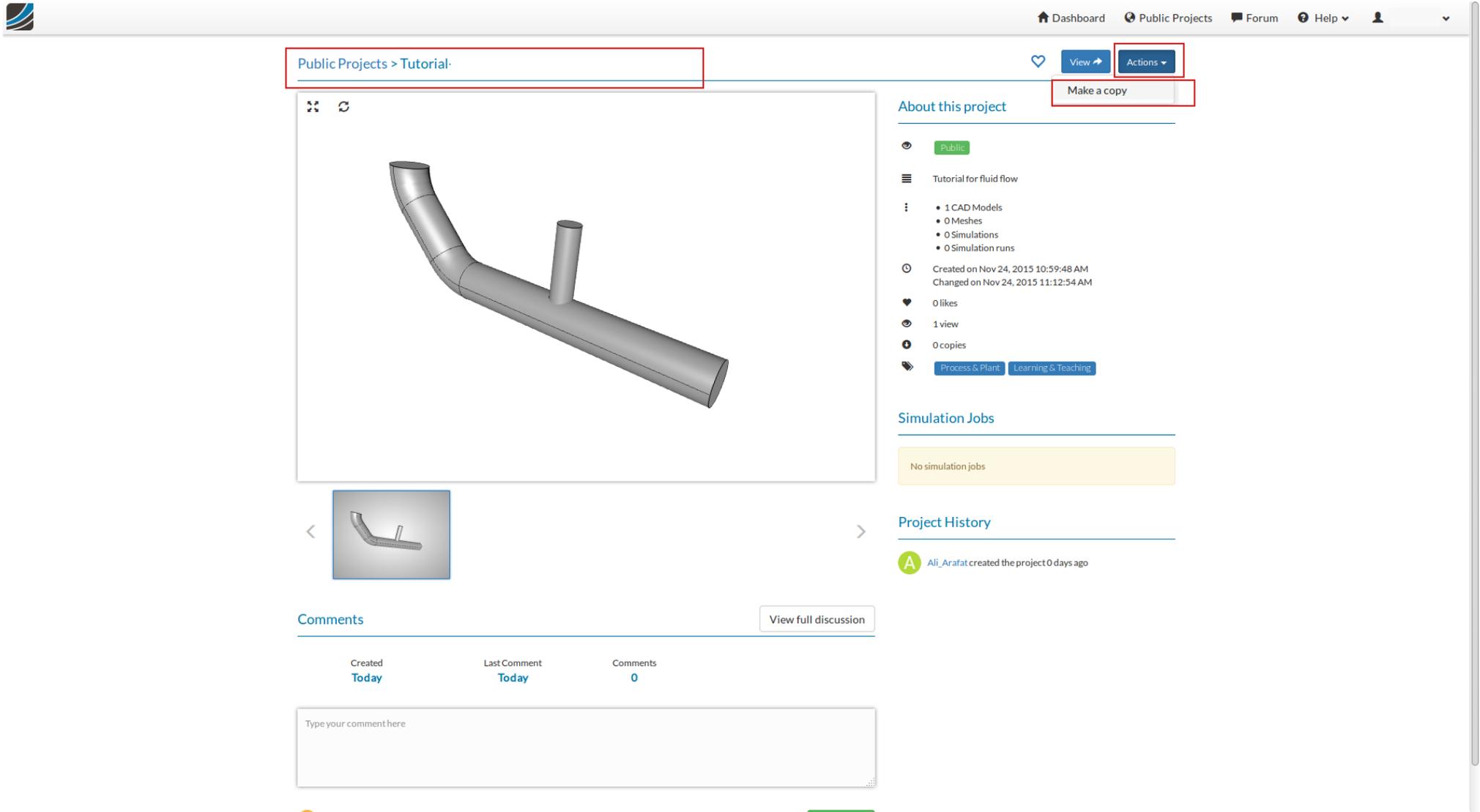


## Public Projects



 <p>0 0 0</p> <p><b>Tutorial-03: Differential casing...</b> 1 CAD model 0 Meshes 0 Simulations <i>Created today</i> Ali_Arafat</p>	 <p>1 0 0</p> <p><b>Tutorial-01: Connecting rod stress...</b> 1 CAD model 0 Meshes 0 Simulations <i>Created today</i> Ali_Arafat</p>	 <p>1 0 0</p> <p><b>Tutorial-</b> 1 CAD model 0 Meshes 0 Simulations <i>Created today</i> Ali_Arafat</p>	 <p>3 1 0</p> <p><b>Another SimScale logo</b> 0 CAD models 1 Mesh 0 Simulations <i>Created 5 days ago</i> cklein</p>
---	--	---	---

- Clicking on the project, then clicking on 'Actions' and 'make a copy' option to add it to your 'Dashboard'. This process is illustrated by the figures below.



- Once the project is in your '**Dashboard**', simply move the mouse over to the upper right corner click on the blue icon to open it in your workbench as shown in figure below.



## Dashboard

[New Project](#)[View all](#)

## My Latest Forum Activity

Public



**Tutorial-03: Differential casing...**  
1 CAD model  
0 Meshes  
0 Simulations  
Created today  
Ali\_Arafat

Public



**Tutorial-01: Connecting rod...**  
1 CAD model  
0 Meshes  
0 Simulations  
Created today  
Ali\_Arafat

Public



**Tutorial-**  
1 CAD model  
0 Meshes  
0 Simulations  
Created today  
Ali\_Arafat

[Open in your workbench](#)

You created new topic 'Tutorial-03: Differential casing thermal analysis' simulation project by Ali\_Arafat



You created new topic 'Tutorial-01: Connecting rod stress analysis' simulation project by Ali\_Arafat



You created new topic 'Tutorial 02: Fluid flow analysis of a pipe junction' simulation project by Ali\_Arafat



You created new topic 'Pipe Junction Flow' simulation project by Ali\_Arafat



You received a new private message from mmazo

## My Account

Current plan: **Community**

[Upgrade my plan](#)

Core Hours Used: 0 / 30000 hours

Left: 100%

Storage Used: 27mb / 5368709mb

Left: 100%

## My Latest Simulation Jobs

No simulation jobs

## Latest Public Projects

[View all](#)

## Create a multiphase simulation

1. To create a new simulation, switch to the Simulation Designer tab and click on "New simulation". Enter a name for the simulation e.g 'Multi-Phase\_Flow' and click OK.
2. From the analysis type choose: Fluid Dynamics section of the analysis types, then Multiphase and setup the properties as shown in the figure below and click on 'save'. After saving a new tree will be automatically generated in the left panel with all the parameters and settings that are

necessary to completely specify such an analysis.

All parts that are completed are highlighted with a green check. Parts that need to be specified have a red circle. While, the blue circle indicates an optional settings that does not need to be filled out

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor<sup>BETA</sup>

Dashboard Public Projects Forum Help Aii\_Arafat

+New simulation Import simulation

Simulations Multi-Phase\_Flow

- Analysis Type
- Domain
- Geometry Primitives
- Model
- Materials
- Initial Conditions
  - modified pressure
  - velocity
  - phase fraction
- Boundary Conditions
- Advanced Concepts
- Numerics
- Simulation Control
- Result Control
- Simulation Runs

Analysis Type

Physics Perspective Solver Perspective

- Fluid dynamics
- Solid mechanics
- Therostructural analysis
- Acoustics
- Particles

- Incompressible
- Compressible
- Natural convective heat transfer
- Passive scalar transport
- Multiphase**
- Discrete phase model
- Potential flow

Properties

Local time-stepping off

Turbulence model Laminar

Steady-state or transient Transient

Save

Click here to save your settings

History Job Status

Name	Status
no entities available	

### Multiphase

The analysis type **Multiphase** enables you to simulate the time dependent behavior of fluid mixtures, for example air and water. The analysis is carried out using the VoF (Volume of fluid) method which is a standard approach for the computation of multiphase systems. Multiphase simulations are performed using the OpenFOAM® solver *interFoam*.

Filling of a fuel tank

U Magnitude 7.8 6 4 2 0

Find more detailed information on this analysis type in the [documentation](#).

#### Disclaimer

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks. OPENFOAM® is a registered trade mark of OpenCFD Limited, the producer of the OpenFOAM software

Selecting a domain

1. Now, click on the “Domain” entry in the tree and select the mesh “waterfall-hex-dominant-mesh” to assign it to this simulation from the options panel. Clicking on ‘save’ will automatically load the selected mesh in the viewer.

The screenshot displays the Multi-Phase Flow software interface. On the left is a navigation tree with categories like Simulations, Model, and Numerics. The 'Domain' entry is selected. The main panel shows the 'Domain' configuration with a list of meshes, including 'waterfall-hex-dominant-mesh'. Below the list are buttons for 'Save', 'Upload file', and 'Switch to Mesh Creator'. The right side features a 3D viewer showing a rectangular mesh. The viewer's toolbar includes options for Orientation, surfaces with edges, Selection, Filter, and Create set. A scene panel on the right lists 'waterfall-hex-dominant-mesh' and 'volumeOnGeoVolumes\_0'. At the bottom left, a 'Job Status' table is shown with the text 'no entities available'.

Name	Status
no entities available	

1. So click on “Topological Entity Sets” from the tree. Also for easier working with the viewer we switch from ‘Surfaces with Edges’ to the ‘Surfaces’ view via the menu on top of the viewer. Then click in “pick faces” icon from the tool-bar above the viewer and select all faces except “inlet” and “Outlet” from the viewer. change the entity type filter to ‘face sets’ and click on “Create set from viewer selection” to create set named ‘walls’.

The screenshot displays the Multi-Phase Flow software interface. On the left, a sidebar contains a tree view with 'Topological Entity Sets' selected and highlighted with a red box and the number '1'. The main panel, titled 'Topological Entity Sets', features a filter dropdown set to 'face sets' (numbered '4') and an empty table with columns 'Name', '# Entities', and 'Actions'. Below the table is a button labeled '+ Create entity set from viewer selection' (numbered '5') with a tooltip that reads 'Create a new topological entity set from selection'. The right side of the interface shows a 3D view of a red rectangular block with 'Inlet' and 'Outlet' labels. A label '3-Walls ( all except Intel and Outlet)' points to the side surfaces. The toolbar at the top includes a 'Create set' button (numbered '2') and a 'Scene' dropdown. A coordinate system (X, Y, Z) is visible in the bottom left of the 3D view.

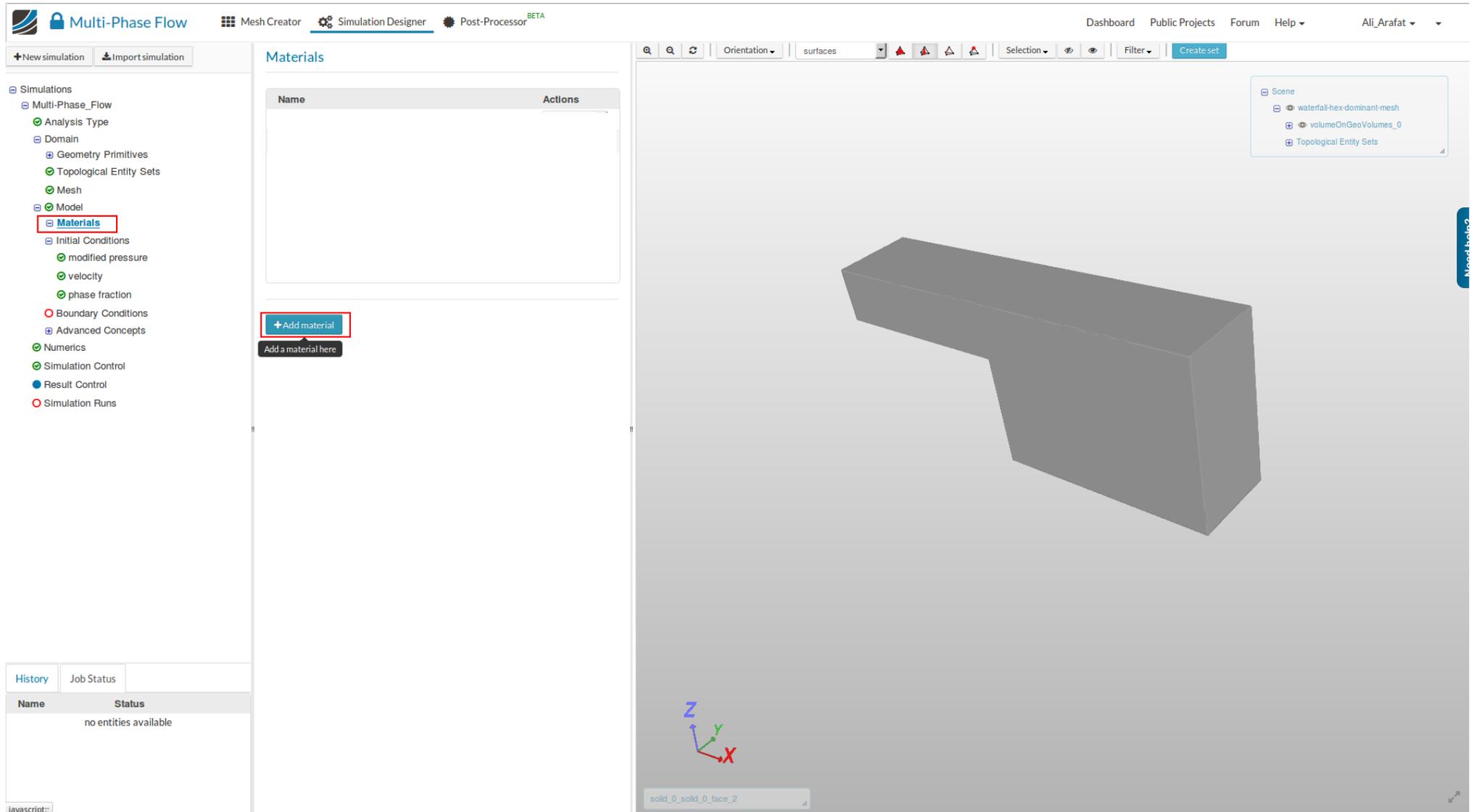
1. Now, create 2 more sets for “Inlet” and “outlet” one by one in similar way to have a total of 3 sets.

## Setting up the Model:

### Adding materials to the domain and assigning phases

1. After, creating the sets move to the “model” entry in the tree where global variables of the simulation are defined. Here we will add the materials

from the 'Material Library', assign a 'Phase' to each and assign the materials to the simulation domain or mesh. This would then determine the materials that will exist in the flow domain and be used for the simulation. First, we start with clicking on sub-tree "materials". Click on "Add material" from the options panel as shown.



1. Re-name the material, e.g Air. We do not need to enter the material properties as we will import pre-defined ones by clicking on "Import from material library" at the bottom.

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor <sup>BETA</sup> Dashboard Public Projects Forum Help Ali\_Arafat

+New simulation Import simulation

Simulations  
Multi-Phase\_Flow  
Analysis Type  
Domain  
Geometry Primitives  
Topological Entity Sets  
Mesh  
Model  
Materials  
Air  
Initial Conditions  
modified pressure  
velocity  
phase fraction  
Boundary Conditions  
Advanced Concepts  
Numerics  
Simulation Control  
Result Control  
Simulation Runs

### Material

Name: Air

Associated phase: Phase 1

Viscosity model: Newtonian

**Details**

Kinematic viscosity [m<sup>2</sup>/s]: 0.000001

Density [kg/m<sup>3</sup>]: 1000

### Topological Mapping

Filter for entity types: volumes

Assigned	Name
<input type="checkbox"/>	volumeOnGeoVolumes_0

+ Add selection from viewer → Select assignment Clear assignments

Save Import from material library

Scene  
waterfall-hex-dominant-mesh  
volumeOnGeoVolumes\_0  
Topological Entity Sets

solid\_0\_solid\_0\_face\_2

Need help?

This pops-up a 'Material Library' from which we select "Air" and click on save. This will then automatically load the standard properties for air.

The screenshot shows the 'Material Library' dialog box in a simulation software. The 'Materials' list includes:
 

- Engine oil SAE 30 - 120C
- Gasoline
- Sulfur dioxide
- Carbon dioxide
- Seawater 3.5pc saline
- Air** (highlighted with a red box)
- Water
- Gaseous R-134a
- Engine oil SAE 30 - 20C
- Liquid R-134a

 The 'Material properties' section displays the following values:
 

- As: 0.0000146
- Specific heat (Cp) [J/(kg K)]: 1004
- Specific heat (Cv) [J/(kg K)]: 717
- Heat of formation [J/kg]: 0
- Prandtl number [-]: 0.713
- Fluid constant [J/(kg K)]: 287.058
- Standard entropy [J/kg]: 202.499
- Ts: 110.4
- Density [kg/m³]: 1.28
- Kappa [W/(m K)]: 2.436
- Molar weight [kg/kmol]: 28.97
- Kinematic viscosity [m²/s]: 0.0000106

- As our domain/mesh will have two fluids for this simulation, we must select their “Associated phase” and assign them to a volume mesh region.
- So, we change the “Associated phase” for air to ‘Phase 0’ to set it as the primary fluid phase and assign it to the mesh volume called ‘volumeOnGeoVolumes\_0’ (volume of the chosen mesh) and click ‘save’.

The screenshot displays the Multi-Phase Flow software interface. The top navigation bar includes 'Multi-Phase Flow', 'Mesh Creator', 'Simulation Designer', and 'Post-Processor BETA'. The main window is divided into several sections:

- Left Panel (Simulations):** A tree view showing the simulation setup. The 'Materials' section is expanded, and 'Air' is selected. Other sections include 'Analysis Type', 'Domain', 'Model', 'Initial Conditions', 'Boundary Conditions', 'Advanced Concepts', 'Numerics', 'Simulation Control', and 'Simulation Runs'.
- Material Configuration Panel (Center):**
  - Name:** Air
  - Associated phase:** Phase 0 (highlighted with a red box and labeled '1')
  - Viscosity model:** Newtonian
  - Details:** Kinematic viscosity [m<sup>2</sup>/s] is 0.0000106.
  - Density [kg/m<sup>3</sup>]:** 1.28
  - Topological Mapping:** Filter for entity types is set to 'volumes'.
  - Assigned Table:** A table with columns 'Assigned' and 'Name'. The entry 'volumeOnGeoVolumes\_0' is checked (highlighted with a red box and labeled '2').
  - Buttons:** '+ Add selection from viewer', '→ Select assignment', and 'Clear assignments'.
  - Bottom Buttons:** 'Save' (highlighted with a red box and labeled '3') and 'Import from material library'.
- 3D Viewport (Right):** Shows a gray 3D model of a T-shaped part. A coordinate system (X, Y, Z) is visible at the bottom left. A 'Scene' panel on the right lists 'waterfall-hex-dominant-mesh', 'volumeOnGeoVolumes\_0', and 'Topological Entity Sets'.

## Important

The associated phase of 'Phase 0' means that the fluid material will be the primary fluid phase. This is then represented by a 'Phase fraction' of value 0 that corresponding to 100% of this fluid. Further, Every fluid material must be assigned at least 1 volume (and vice-versa).

## Important

At this stage, we are only defining the properties of the two fluid materials that will be present in the domain/mesh and assigning them to the corresponding mesh volume (in this case the same one). As for the initial distribution of the fluid materials, it will be defined later under “Initial Conditions” sub-tree entry (under “Model”) with the help of “Geometry Primitives”.

1. Similarly, now add the secondary fluid material named ‘water’ by clicking “Add material” as before. Click on “Import from material library” at the bottom and add “Water”.

The screenshot displays the ANSYS Fluent interface with the 'Material Library' dialog box open. The 'Water' material is selected in the 'Materials' list. The 'Material properties' section shows the following values:

Property	Value
As	
Specific heat (Cp) [J/(kg K)]	4180
Specific heat (Cv) [J/(kg K)]	4180
Heat of formation [J/kg]	-286000
Prandtl number [-]	6.5241
Fluid constant [J/(kg K)]	69.95
Standard entropy [J/kg]	69.95
Ts	
Density [kg/m³]	1000
Kappa [W/(m K)]	0.6029
Molar weight [kg/kmol]	18
Kinematic viscosity [m²/s]	0.000941

The background shows the 'Material' panel with the following settings:

- Name: Water
- Associated phase: Phase 1
- Viscosity model: Newtonian
- Details: Kinematic viscosity [m²/s] = 0.000001
- Density [kg/m³] = 1000
- Topological Mapping: Filter for entity types = volumes

- Now select the options as indicated in the figure below, assign it ‘Phase 1’ and ‘save’ it.

The screenshot displays the Multi-Phase Flow software interface. The top navigation bar includes 'Multi-Phase Flow', 'Mesh Creator', 'Simulation Designer', and 'Post-Processor BETA'. The main window is divided into a left sidebar, a central configuration panel, and a right-side 3D viewer.

**Left Sidebar:** A tree view under 'Simulations' shows 'Multi-Phase\_Flow' expanded. Under 'Domain', 'Materials' is selected, and 'Water' is highlighted with a red box and the number '1'.

**Material Configuration Panel:** The 'Material' panel is titled 'Water'. The 'Associated phase' dropdown is set to 'Phase 1' and is highlighted with a red box and the number '2'. Under 'Details', 'Kinematic viscosity [m<sup>2</sup>/s]' is 0.000941 and 'Density [kg/m<sup>3</sup>]' is 1000. The 'Topological Mapping' section has a filter for 'volumes'. A table shows one assigned entity: 'volumeOnGeoVolumes\_0', which is highlighted with a red box and the number '3'. At the bottom, the 'Save' button is highlighted with a red box and the number '4'. Below the table is a 'History' section with a table showing 'no entities available'.

**3D Viewer:** The central 3D viewer shows a grey L-shaped geometry. A coordinate system with X, Y, and Z axes is visible at the bottom left. A 'Scene' panel on the right lists 'waterfall-hex-dominant-mesh', 'volumeOnGeoVolumes\_0', and 'Topological Entity Sets'.

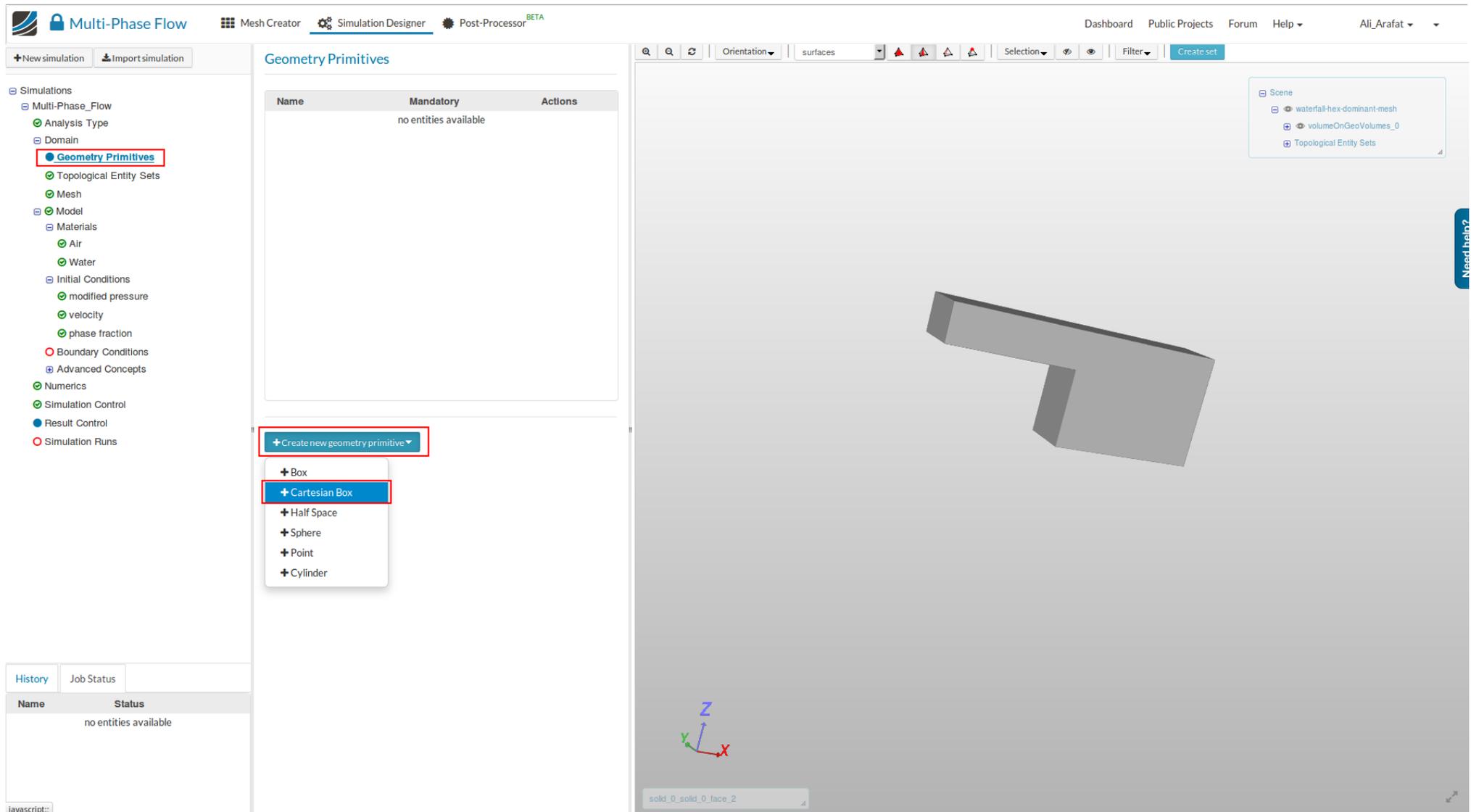
## Important

“So now as water is assigned ‘Phase 1’, it will be represented by a phase fraction value of 1”.

## Creating a geometry Primitive:

1. As the initial distribution of the fluid materials is defined with the help of “geometry Primitives”. So we first click on “Geometry Primitives” under

“Domain” and click on ” Create new geometry primitive” from the options panel to select a type “Cartesian Box”. This will be used to define the initial distribution of the column of water in the channel.



1. Re-name the cartesian box to a meaningful name and specify the values as shown in the figure below.

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor <sup>BETA</sup> Dashboard Public Projects Forum Help Ali\_Arafat

+New simulation Import simulation

Simulations

- Multi-Phase\_Flow
  - Analysis Type
  - Domain
    - Geometry Primitives
      - Initial-State-Water**
    - Topological Entity Sets
  - Mesh
  - Model
  - Materials
    - Air
    - Water
  - Initial Conditions
    - modified pressure
    - velocity
    - phase fraction
  - Boundary Conditions
  - Advanced Concepts
- Numerics
- Simulation Control
- Result Control
- Simulation Runs

Cartesian Box

1	Name	Initial-State-Water
	Min. Point (x)	-0.5
	Min. Point (y)	-2
2	Min. Point (z)	0
	Max. Point (x)	8.5
	Max. Point (y)	2
	Max. Point (z)	1.7

Save 3

Click here to save your settings

Scene

- waterfall-hex-dominant-mesh
- volumeOnGeoVolumes\_0
- Topological Entity Sets
- Geometry Primitives
  - Initial-State-Water

Created geometry primitive

History Job Status

Name	Status
no entities available	

javascript;

solid\_0\_solid\_0\_face\_6

## Important

Remember: “Geometry primitives are used to specify the initial distribution (state) of the fluid materials”.

### Initial Conditions (Defining initial flow variables and distribution for fluid phases):

1. The next tree item “Initial conditions” allows to define the initial velocity, pressure and “phase fraction” (in other words, initial phase distribution) for

the fluids. For pressure and velocity we keep the default values. Then click on 'Phase fraction', change type to 'sub-domain-based', keep "Default phase fraction value" of 0 ( that is air ) and click save. This means for now all the volume is filled with air.

The screenshot shows the 'Initial Condition' configuration panel in the Multi-Phase Flow software. The 'Type' is set to 'Subdomain-based'. Under 'Details', the 'Default phase fraction value' is set to 0. The 'Subdomains' section is currently empty. A 3D model of a T-shaped object is visible in the background. The interface includes a sidebar with simulation settings, a top navigation bar, and a bottom status bar.

1. Initial Conditions  
2. phase fraction  
3. Subdomain-based  
4. Default phase fraction value [-] 0  
5. Save  
6. Add subdomain

Name	Status
no entities available	

1. Now click on "Add subdomain" at the bottom under 'Subdomains' to add the water domain as shown in figure above. Enter the settings as shown in the figure below (phase fraction value of 1 means water phase) and assign the previously created geometry primitive to this sub-domain and save.

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor BETA

Dashboard Public Projects Forum Help Ali\_Arafat

+New simulation Import simulation

Subdomain ?

1 Name Water-Initial-Phase

2 Type Phase fraction subdomain

3 Details

Phase fraction value inside the subdomain [-] 1

Assigned Geometry Primitives

4

Assigned	Name	Mandatory	Actions
<input checked="" type="checkbox"/>	Initial-State-Water	false	...

+ Create new geometry primitive Clear assignments

5 Save

Scene

- waterfall-hex-dominant-mesh
- volumeOnGeoVolumes\_0
- Topological Entity Sets
- Geometry Primitives
  - Initial-State-Water

History Job Status

Name	Status
no entities available	

solid\_0\_solid\_0\_face\_2

Need help?

- So, the common region of the geometry primitive and the volume mesh will now define the initial distribution of the water phase.

### Important

Remember: “Under Initial Conditions the initial distribution (before start of simulation) of the fluid materials is defined (by using geometry primitives)”.

## Boundary Conditions:

Now, we come to define the boundary conditions using the topological entity sets created earlier.

1. Click on “Boundary conditions” in the tree and click “Add boundary condition” button to create a new entry as shown below.

The screenshot displays the Multi-Phase Flow software interface. The top navigation bar includes 'Multi-Phase Flow', 'Mesh Creator', 'Simulation Designer', and 'Post-Processor BETA'. The main window is divided into several sections:

- Left Panel (Tree View):** Shows a hierarchical tree of simulation components. 'Boundary Conditions' is highlighted with a red box. Other visible items include 'Simulations', 'Multi-Phase\_Flow', 'Analysis Type', 'Domain', 'Geometry Primitives', 'Initial-State-Water', 'Topological Entity Sets', 'Mesh', 'Model', 'Materials', 'Air', 'Water', 'Initial Conditions', 'modified pressure', 'velocity', 'phase fraction', 'Water-Initial-Phase', 'Advanced Concepts', 'Numerics', 'Simulation Control', 'Result Control', and 'Simulation Runs'.
- Boundary Conditions Panel:** A table with columns 'Name', 'Type', '# Assigned Entit.', and 'Actions'. It currently shows 'no entities available'. Below the table is a red-bordered button labeled '+ Add boundary condition' and a tooltip that says 'Add a new boundary condition here'.
- 3D Viewport:** Displays a 3D model of a T-junction geometry. The top horizontal part is grey, and the bottom vertical part is blue. A coordinate system with X, Y, and Z axes is visible in the bottom left corner. The text 'solid\_0\_solid\_0\_face\_2' is shown at the bottom of the viewport.
- Right Panel (Scene):** Shows a list of scene objects: 'waterfall-hex-dominant-mesh', 'volumeOnGeoVolumes\_0', 'Topological Entity Sets', 'Geometry Primitives', and 'Initial-State-Water'.

1. Re-name the entry as “water-Inlet” and specify the values shown in the figure below and click on save at the bottom. Note: you do not need to

select from the viewer in this case.

The screenshot displays the Multi-Phase Flow software interface. The left sidebar shows a tree view of simulation settings, with 'boundary condition 1' highlighted under 'Boundary Conditions'. The main panel is divided into two sections: 'Boundary Condition' and 'Topological Mapping'.

**Boundary Condition Configuration:**

- Name: Water-Inlet
- Type: Velocity Inlet
- Velocity: Fixed value
- Input Type: Value or Function
- x value [m/s]: 2
- y value [m/s]: 0
- z value [m/s]: 0
- Phase fraction: Fixed value
- Phase fraction value [-]: 1

**Topological Mapping:**

Filter for entity types: face sets

Assigned	Name
<input checked="" type="checkbox"/>	Inlet
<input type="checkbox"/>	walls
<input type="checkbox"/>	outlet

The 3D viewer on the right shows a grey L-shaped geometry with a red face on the left side, labeled 'Inlet' with a blue arrow. A coordinate system (X, Y, Z) is visible in the bottom left of the viewer.

1. Add another boundary condition ( as before ), re-name it as “outlet” and specify the settings as shown in figure below and save it.

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor <sup>BETA</sup> Dashboard Public Projects Forum Help Ali\_Arafat

+New simulation Import simulation

Boundary Condition

Name: Outlet

Type: Custom

Details

Velocity: Pressure-inlet-outlet velocity

Pressure: Total pressure

Details

Total pressure [Pa]: 0

Gamma [-]: 0

Phase fraction: Fixed value

Details

Phase fraction value [-]: 0

Topological Mapping

Filter for entity types: face sets

Assigned	Name
<input checked="" type="checkbox"/>	outlet
<input type="checkbox"/>	walls
<input type="checkbox"/>	Inlet

+ Add selection from viewer Select assignment Clear assignments

History Job Status

Name	Status
no entities available	

Scene

- waterfall-hex-dominant-mesh
- volumeOnGeoVolumes\_0
- Topological Entity Sets

Outlet

solid\_0\_solid\_0\_face\_2

1. Lastly, add the 3rd boundary condition, re-name it to “walls” and specify the settings as shown in the following figure. Note: you do not need to select from the viewer in this case.

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor <sup>BETA</sup> Dashboard Public Projects Forum Help Ali\_Arafat

+New simulation Import simulation

Simulations

- Multi-Phase\_Flow
  - Analysis Type
  - Domain
    - Geometry Primitives
      - Initial-State-Water
    - Topological Entity Sets
    - Mesh
  - Model
    - Materials
      - Air
      - Water
    - Initial Conditions
      - modified pressure
      - velocity
      - phase fraction
        - Water-Initial-Phase
    - Boundary Conditions
      - Water-Inlet
      - Outlet
      - boundary condition 3**
    - Advanced Concepts
  - Numerics
  - Simulation Control
  - Result Control
  - Simulation Runs

**Boundary Condition**

Name: Walls

Type: Wall

Details

- Velocity: No-slip
- Phase fraction: Set gradient to zero

Topological Mapping

Filter for entity types: face sets

Assigned	Name
<input checked="" type="checkbox"/>	walls
<input type="checkbox"/>	Inlet
<input type="checkbox"/>	outlet

+ Add selection from viewer Select assignment Clear assignments

Save

History Job Status

Name	Status
no entities available	

Orientation surfaces Selection Filter Create set

Scene

Need help?

solid\_0\_solid\_0\_face\_2

## Numerics:

- Based on the type of problem, we now modify some of the numerics as is illustrated in the figure below. This includes specifying the number of solver iterations per time step in "Number of correctors" and selecting the solvers based on the flow variables. These changes will help in better stability and convergence on the simulation.

Multi-Phase Flow Mesh Creator Simulation Designer Post-Processor <sup>BETA</sup> Dashboard Public Projects Forum Help Ali\_Arafat

+New simulation Import simulation

Simulations

- Multi-Phase\_Flow
  - Analysis Type
  - Domain
    - Geometry Primitives
      - Initial-State-Water
    - Topological Entity Sets
    - Mesh
  - Model
  - Materials
    - Air
    - Water
  - Initial Conditions
    - modified pressure
    - velocity
    - phase fraction
      - Water-Initial-Phase
  - Boundary Conditions
    - Water-Inlet
    - Outlet
    - Walls
  - Advanced Concepts
    - Numerics**
    - Simulation Control
    - Result Control
    - Simulation Runs

**Properties**

Momentum predictor On

Number of correctors 1

Number of non-orthogonal correctors 2

Pressure reference cell 0

Pressure reference value 0

**Solver**

[alpha.phase1] phase fraction Solver MULES

**Details**

Alpha correctors 1

Alpha sub-cycles 4

Compression coefficient 1

Isotropic compression coefficient 0.25

Semi-implicit MULES no

[p\_rgh] modified pressure solver GAMG

**Details**

[p\_rgh] modified pressure final solver GAMG

**Details**

[pcorr] pressure correction Solver Smooth solver

**Details**

[U] velocity solver Smooth solver

**Details**

[U] velocity final solver Smooth solver

**Details**

Orientation surfaces Selection Filter Create set

Scene

- waterfall-hex-dominant-mesh
- volumeOnGeoVolumes\_0
- Topological Entity Sets
- Geometry Primitives
  - Initial-State-Water

solid\_0\_solid\_0\_face\_2

## Simulation Control:

- Next, in simulation control we define some main control settings such as start and end times, time step size, auto time-stepping and number of processors for this simulation run. Follow the figure below to set up as shown and click 'save'.

The screenshot displays the 'Simulation Control' panel in the Multi-Phase Flow software. The panel is divided into several sections with various input fields and dropdown menus. A red box highlights the 'Simulation Control' section, which includes settings for time, timestep, and Courant number. Below this, there are settings for write control, computing cores, and runtime. A 'Save' button is highlighted with a tooltip that says 'Click here to save your settings'. The main view shows a 3D model of a T-shaped object with a coordinate system (X, Y, Z) at the bottom left.

1. Finally click on “Simulation Runs” and click on “Create New Run” from the options panel. Then click “Start” to start the simulation run. That’s it ...!

## Results:

1. Once the run finishes, the results can be post-processed by clicking on “Post-processor” tab and loading the results by clicking on “Solution fields” as shown in figure below.

Download Get ParaView®

alpha.phase1 [1] Surface 3 Rescale Viewport Tools Configuration

- Meshes
  - waterfall-hex-dominant-mesh
- Simulations
  - Multi-Phase\_Flow
    - MP-Run1
      - Solution fields** 2
      - Convergence plot
- Result Evaluation
  - Screenshots
    - Post-Processor Screenshot
  - Saved States
  - Reports

Save State Add Filter Delete Filter

MP-Run1

**Slice1** 3

Property panel

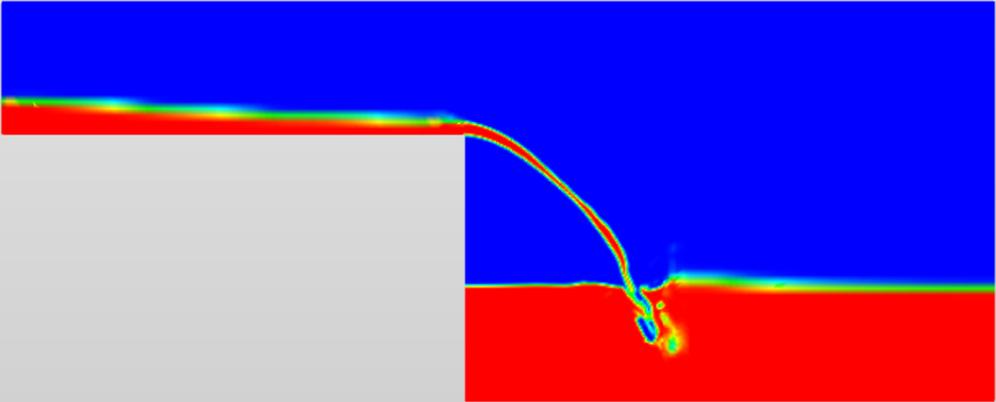
Slice Type: Plane

Triangulate the slice:

Origin: 0 0 0

Normal: 0 1 0

Offset: 0



History Job Status

Name	Status
no entities available	

Need help?