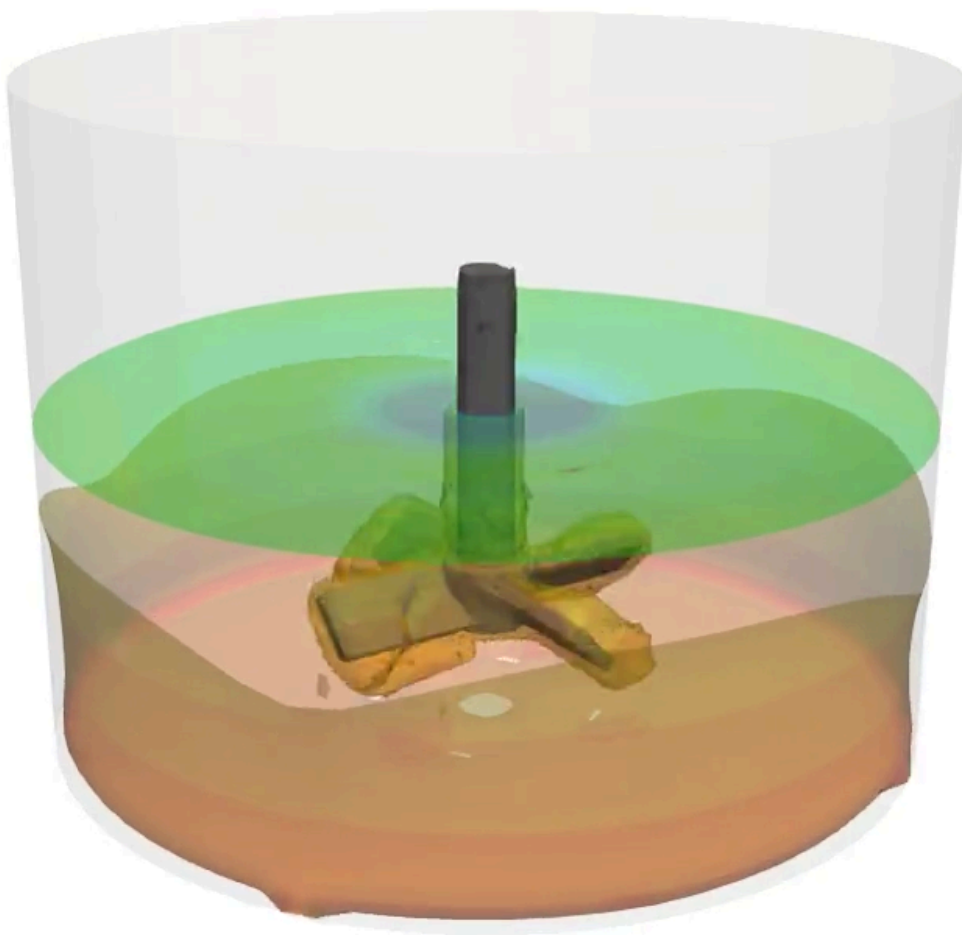




# Mixing Tank - CFD Simulation

CFD Software Tutorial

[Start](#)[Geometry](#)[Mesh](#)[Setup](#)[Boundary  
Conditions](#)[Run](#)[Postprocessing](#)

## 1. Introduction



In this tutorial, we will demonstrate how to use Dynamic Mesh to model rotating components. We will simulate the rotation of an impeller in a cylindrical tank partially filled with water. The impeller is located within the inner cylinder mesh zone and rotates inside the tank's larger domain. At the sliding mesh interface, which links the rotating and stationary mesh sections, data interpolation occurs. This tutorial uses a 2-phase Volume of Fluid (VoF) simulation to indicate a free surface vertex resulting from the rotation of the impeller.

## 2. Download SimFlow

---

SimFlow is a general purpose [CFD Software](#)

To follow this tutorial, you will need SimFlow free version, you may download it via the following link:

[Download SimFlow](#)

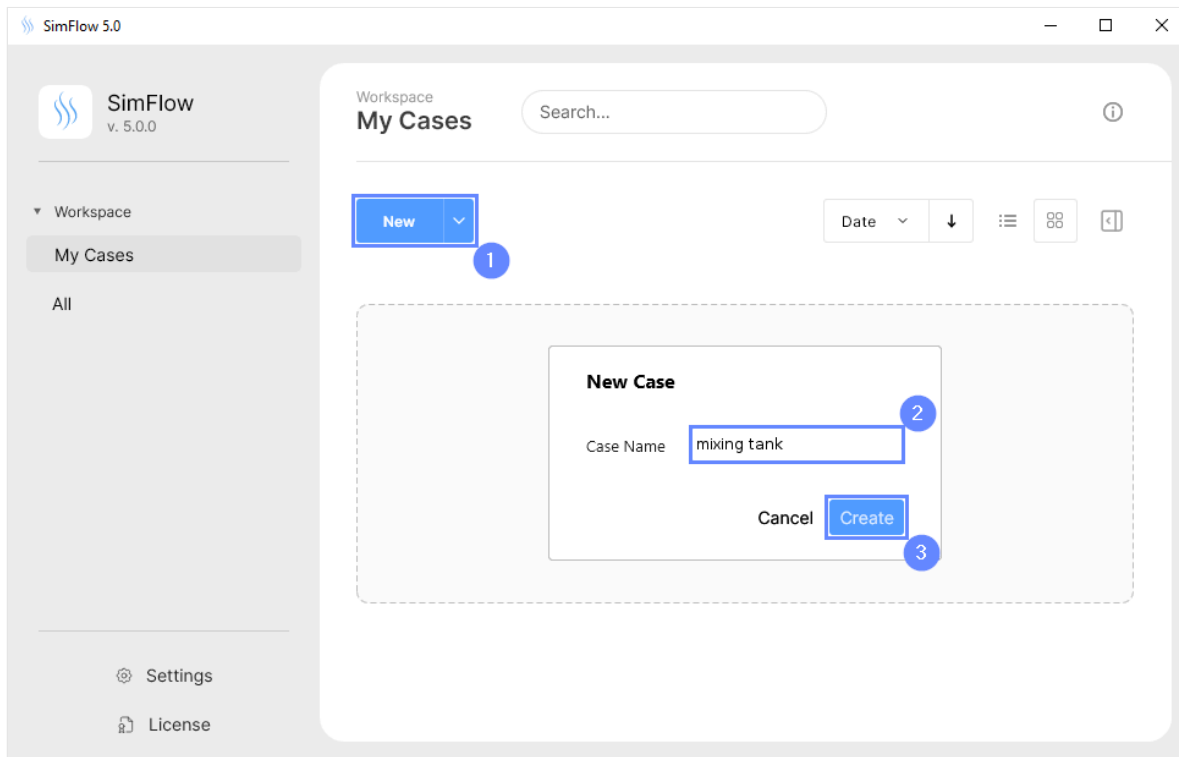
## 3. Create Case

---

Open SimFlow and create a new case named *mixing tank*

- 1 Click
- 2 Provide name **mixing tank**
- 3 Click  to open a new case



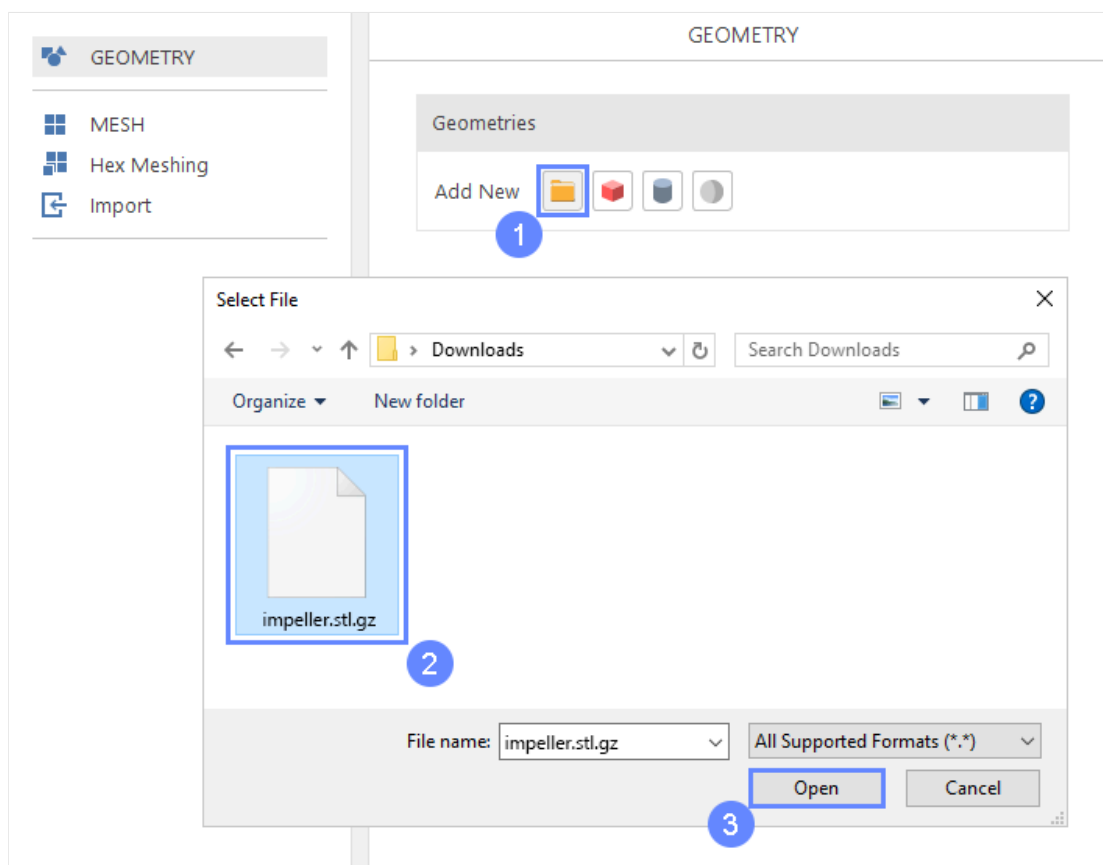


## 4. Import Geometry

After creating case [Download Geometry | Impeller](#) 

- 1 Click
- 2 Select geometry file `impeller.stl.gz`
- 3 Click





## 5. Imported Geometry Units

In a next step we need to select the unit in which the model was exported. The STL format does not contain the unit information which are defined during the geometry export. The *Geometry size* displays the overall size of the model what allows to select the suitable unit. In this case, the default unit meter is correct.

- 1 To confirm default unit **meter**, press



### Imported Geometry Units

Choose units in which the geometry was created  
(This operation is required only for geometry in STL format)

Geometry created in

mm

cm

m

in

ft

Geometry size

x

0.5 [m]

y

0.5 [m]

z

0.09 [m]

1

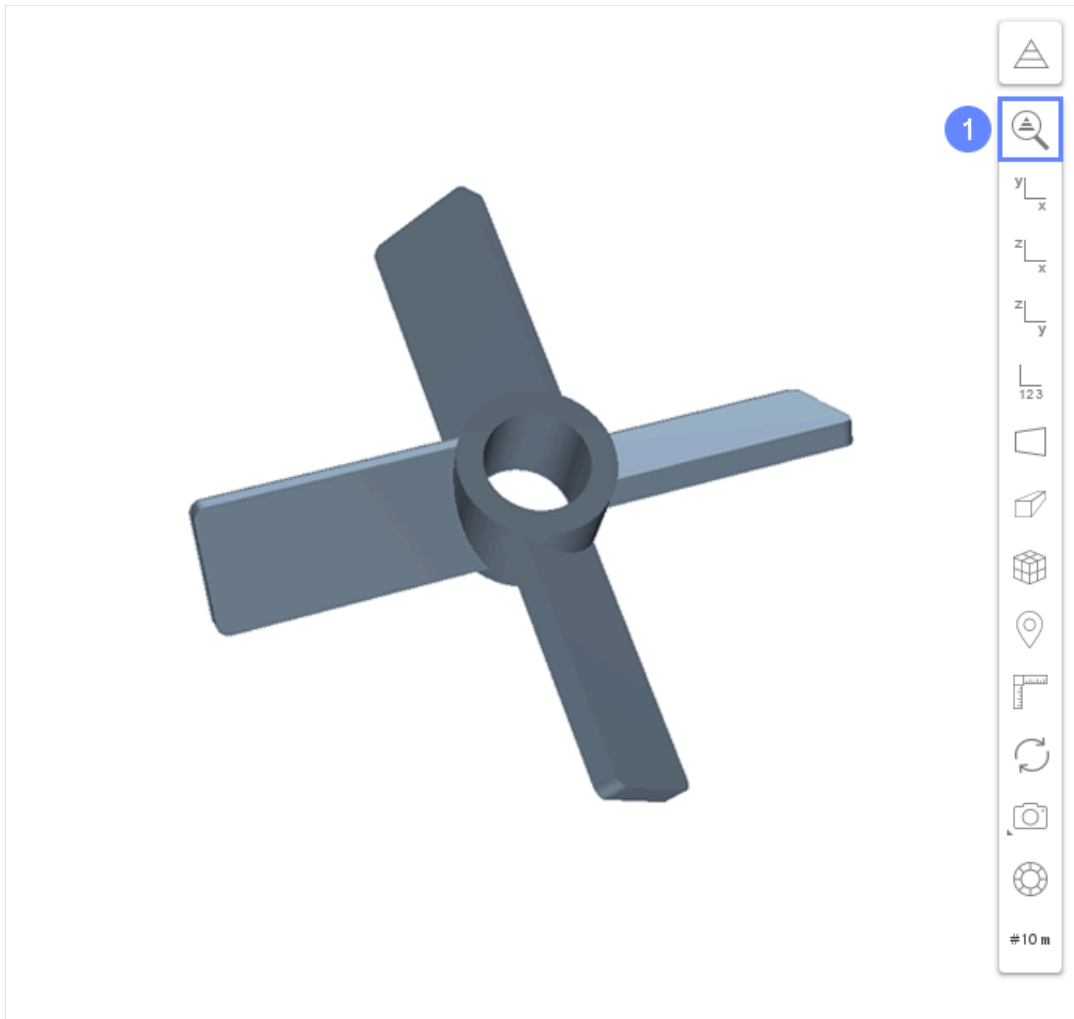
OK

## 6. Geometry - Impeller

After importing geometry, it will appear in the 3D window.

- 1 Click **Fit View** to zoom the geometry





## 7. Create Geometry - Shaft

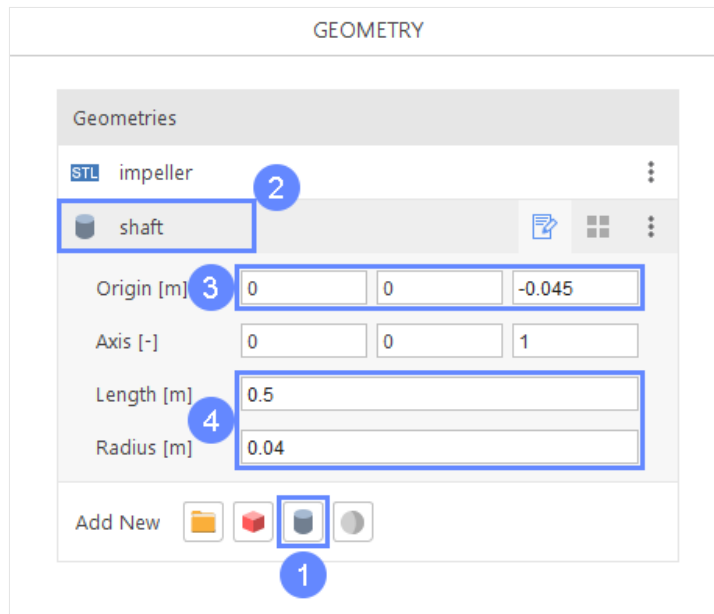
The original impeller geometry consists of only blades. To make the impeller complete we will add a shaft using a primitive geometry.

- 1 Click **Create Cylinder**
- 2 Rename **cylinder\_1** to **shaft**  
(double click on the geometry name to start rename)
- 3 4 Define cylinder origin, length, and radius

**Origin [m]**    0    0    -0.045

**Length [m]**    0.5

**Radius [m]**    0.04



## 8. Create Geometry - Rotating Zone

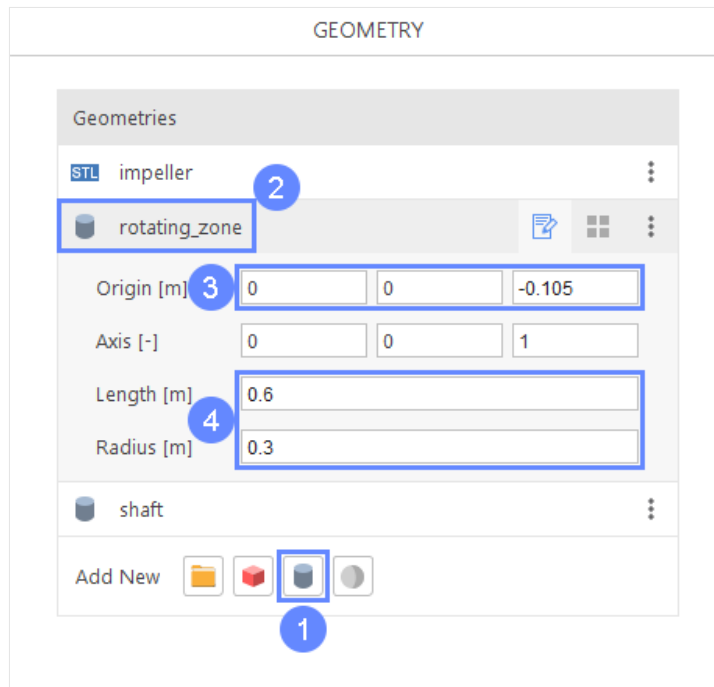
We want our impeller to be rotating. We will create a cylindrical zone that will be used to divide the mesh into rotating and stationary part.

- 1 Click **Create Cylinder**
- 2 Rename **cylinder\_1** to **rotating\_zone**
- 3 4 Define cylinder origin, length, and radius

**Origin [m]**    0    0    -0.105

**Length [m]**    0.6

**Radius [m]**    0.3



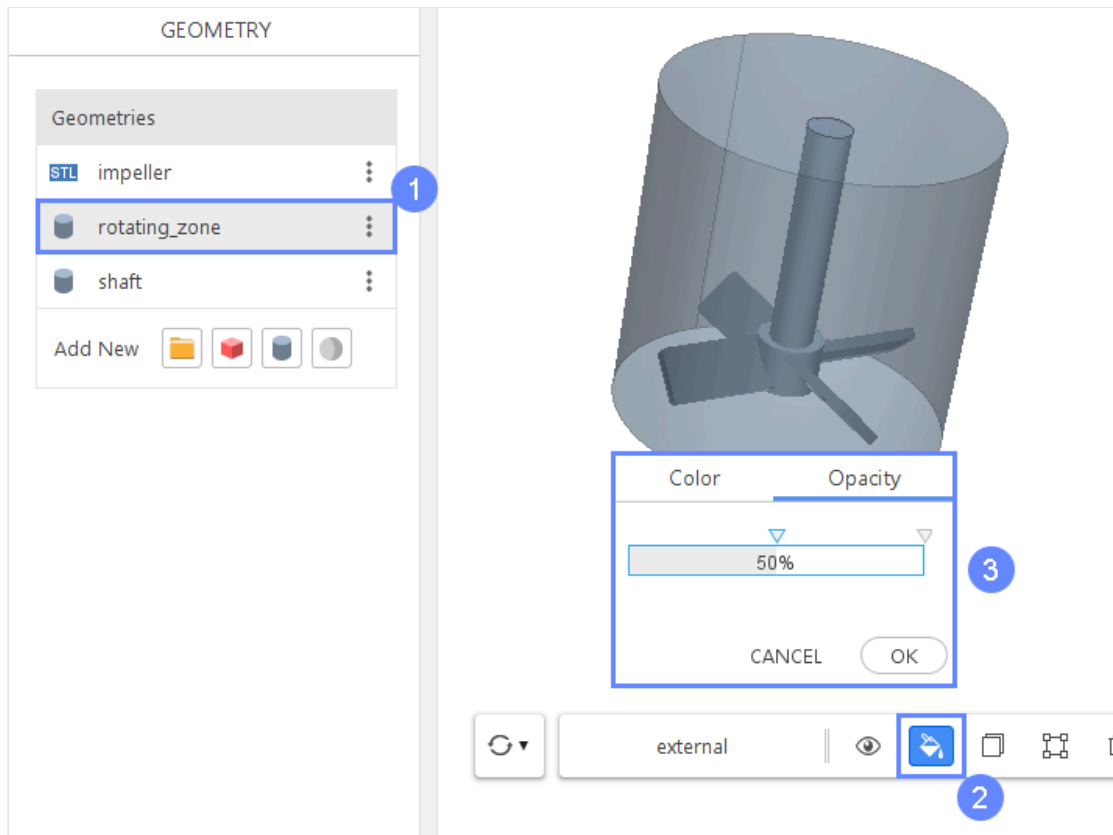
## 9. Show Impeller

All parts of the geometry are displayed in the graphics window now. Note that rotating\_zone encloses the impeller and shaft. In order to see all geometries, we will decrease the opacity of the rotating\_zone.

- 1 Press  to exit edit mode then select **rotating\_zone** if it is not selected
- 2 Click **Display Properties**
- 3 Adjust **Opacity** to **50%**





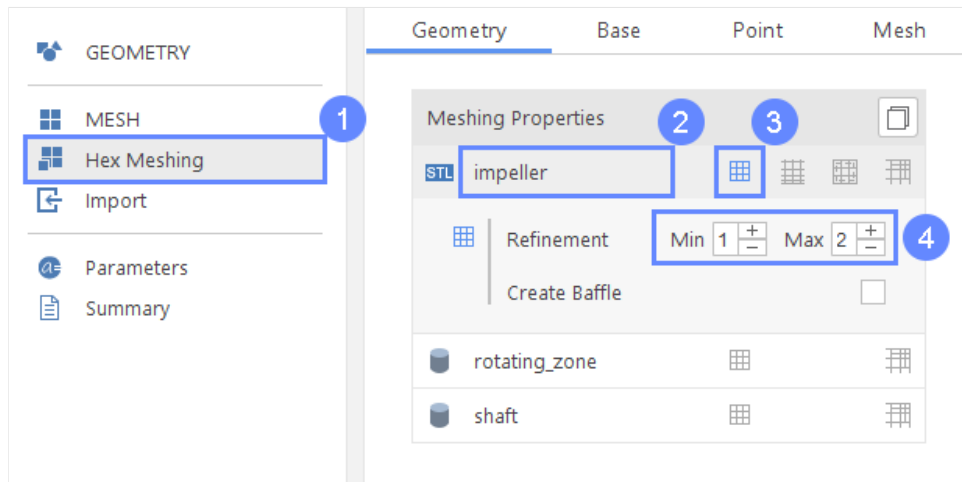


## 10. Meshing Properties - Impeller

We will enable meshing for impeller geometry.

- 1 Go to **Hex Meshing** panel
- 2 Select **impeller**
- 3 Check **Mesh Geometry**
- 4 Set **Refinement** to **Min 1 Max 2**



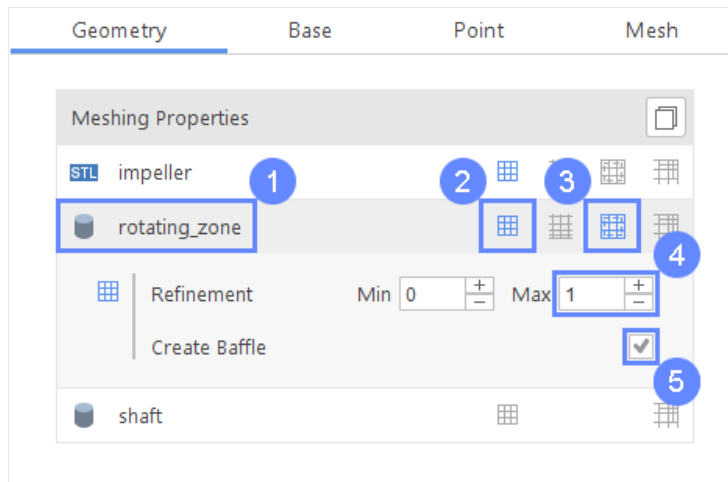


## 11. Meshing Properties - Rotating Zone

Since our domain will consist of a stationary and a rotating part we need to create a mesh interface between the parts. To do this we will create a baffle using the rotating\_zone geometry.

- 1 Select **rotating\_zone**
- 2 Check **Mesh Geometry**
- 3 Check **Create Cell Zone**
- 4 Change **Refinement Max** level to **1**
- 5 Check **Create Baffle**

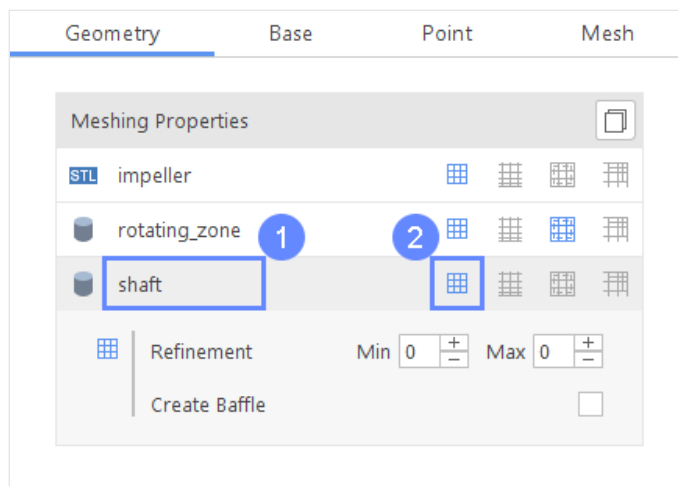




## 12. Meshing Properties - Shaft

We need to also mesh the shaft geometry.

- 1 Select **shaft**
- 2 Check **Mesh Geometry**



## 13. Base Mesh - Geometry

Now we will define the outer bounds of the computational domain.

- 1 Go to **Base** tab
- 2 Select **Cylinder** as the base mesh type
- 3 Define cylinder axis, origin, length, and radius

**Axis**    **Z**

**Origin [m]**    **-0.18**

**Length [m]**    **0.9**

**Radius [m]**    **0.6**

The screenshot shows the 'Base Mesh Type' dialog box. At the top, there are four tabs: 'Geometry', 'Base', 'Point', and 'Mesh'. The 'Base' tab is selected and highlighted with a blue box and a blue circle containing the number '1'. Below the tabs, there is a section titled 'Base Mesh Type' with four buttons: 'Box', 'Cylinder', 'Plate', and 'Wedge'. The 'Cylinder' button is selected and highlighted with a blue box and a blue circle containing the number '2'. Below this, there is a section titled 'Geometry' with several input fields. A blue box and a blue circle containing the number '3' highlight the 'Axis' dropdown menu (set to 'Z'), the 'Origin [m]' text box (containing '-0.18'), the 'Length [m]' text box (containing '0.9'), and the 'Radius [m]' text box (containing '0.6'). Other fields include 'Radial Fraction [-]' (containing '0.5') and an 'Autosize' button.

## 14. Base Mesh - Mesh

To have an appropriate mesh density we will use the following divisions and grading of the base mesh.

- 1
- 2 Define radial division, radial grading, axial division, central division

**Radial Division**    **15**

Radial Grading 1.02

Axial Division 30

Central Division 15

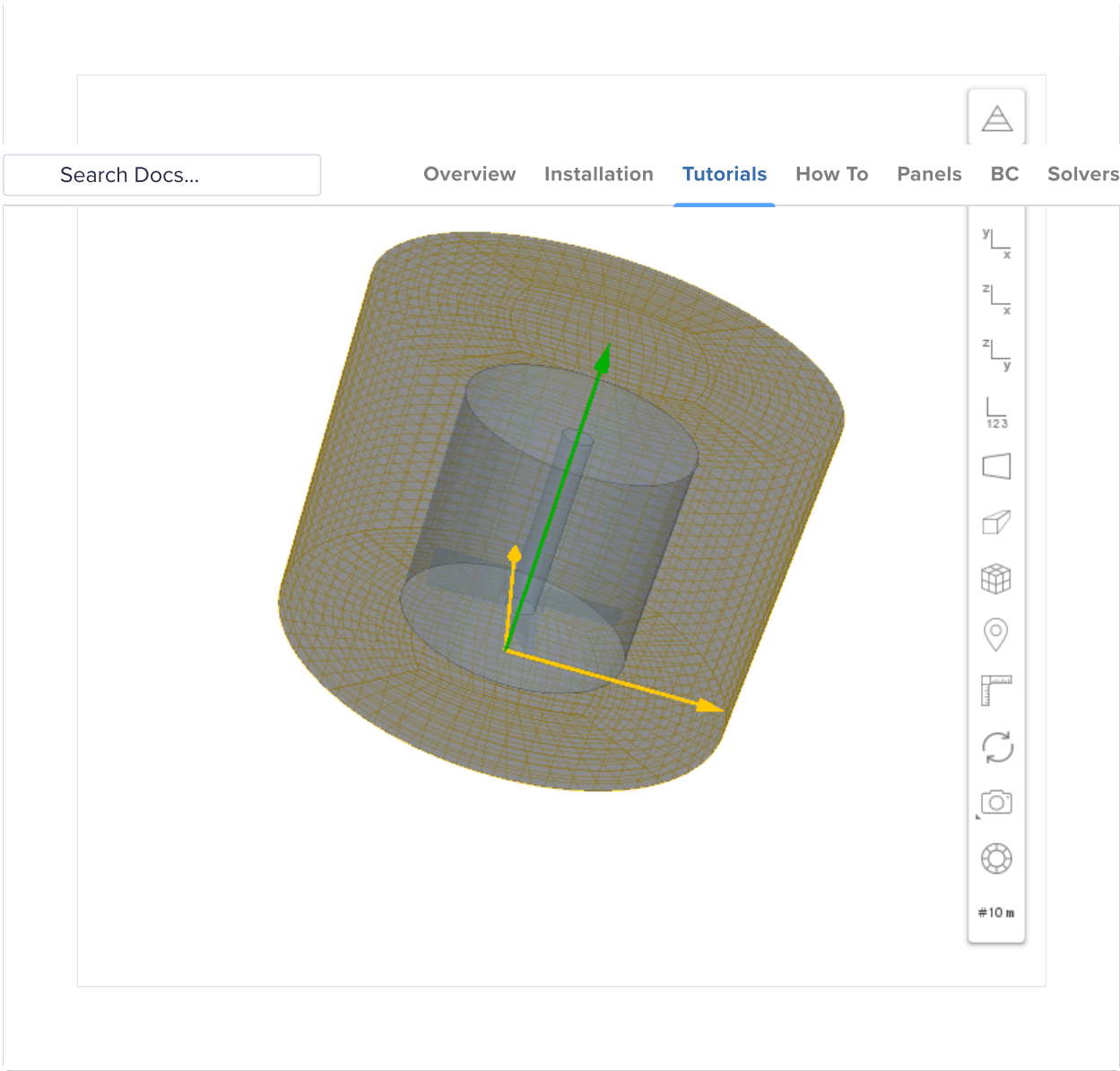
#### Mesh

Radial Division	<input type="text" value="15"/>	<input type="button" value="+"/> <input type="button" value="-"/>
Radial Grading	<input type="text" value="1.02"/>	
Axial Division	<input type="text" value="30"/>	<input type="button" value="+"/> <input type="button" value="-"/>
Axial Grading	<input type="text" value="1"/>	
Central Division	<input type="text" value="15"/>	<input type="button" value="+"/> <input type="button" value="-"/>

## 15. Base Mesh - View

We can preview the Base Mesh in the graphics window.



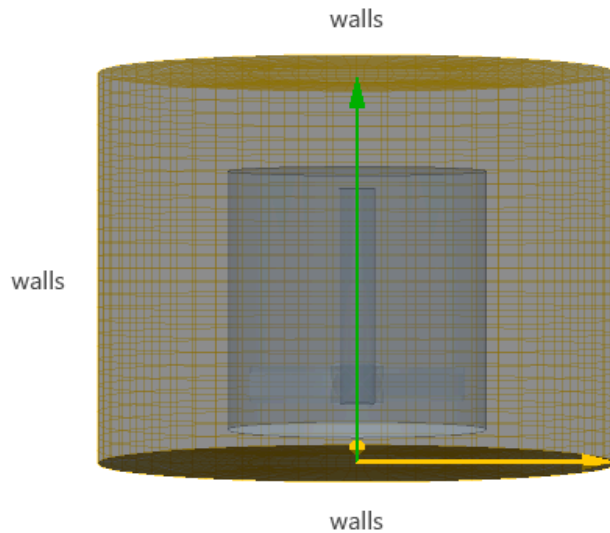


## 16. Base Mesh - Boundaries

Now we need to assign individual names to each side of the base mesh. Since we are modeling a closed tank all our boundaries will be defined as walls.

- 1 Type boundary names accordingly  
First Disk walls  
Second Disk walls  
Cylinder walls
- 2 Change type of all boundaries to wall  
First Disk wall  
Second Disk wall  
Cylinder wall

Boundaries			
First Disk	walls		
Second Disk	walls		
Cylinder	walls		

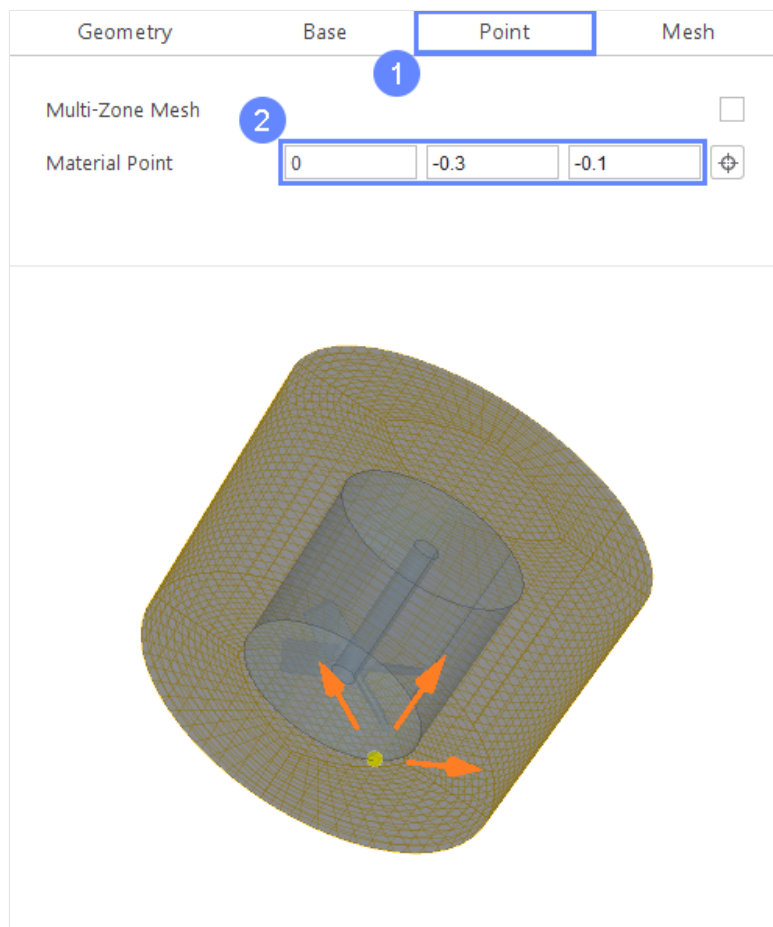


## 17. Material Point

Material Point tells the meshing algorithm on which side of the geometry the mesh is to be retained. Since we are modelling flow inside the tank we need to place the material point between the tank walls and the propeller.

- 1 Go to **Point** tab
- 2 Specify location inside the mesh  
**Material Point**    0    -0.3    -0.1





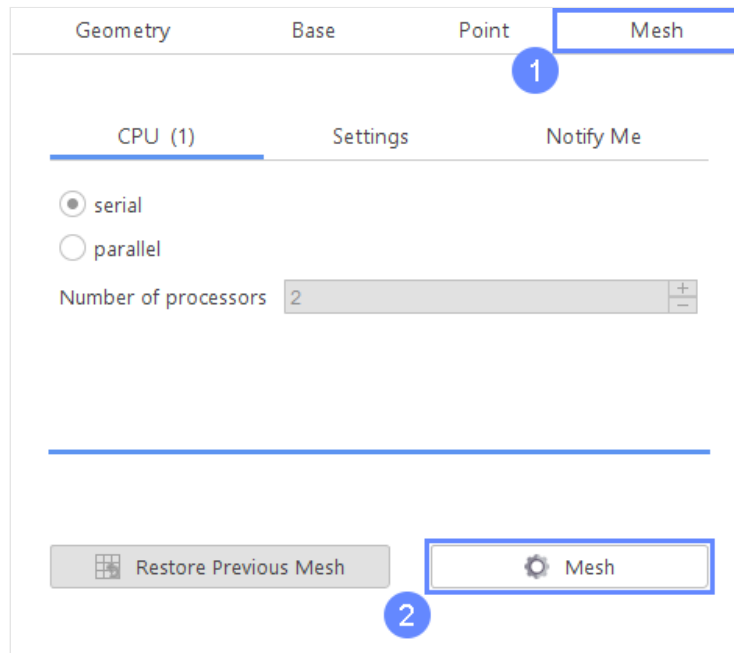
## 18. Meshing

Everything is now set up for meshing.

- 1 Go to **Mesh** tab
- 2 Click **Mesh** button to start meshing process





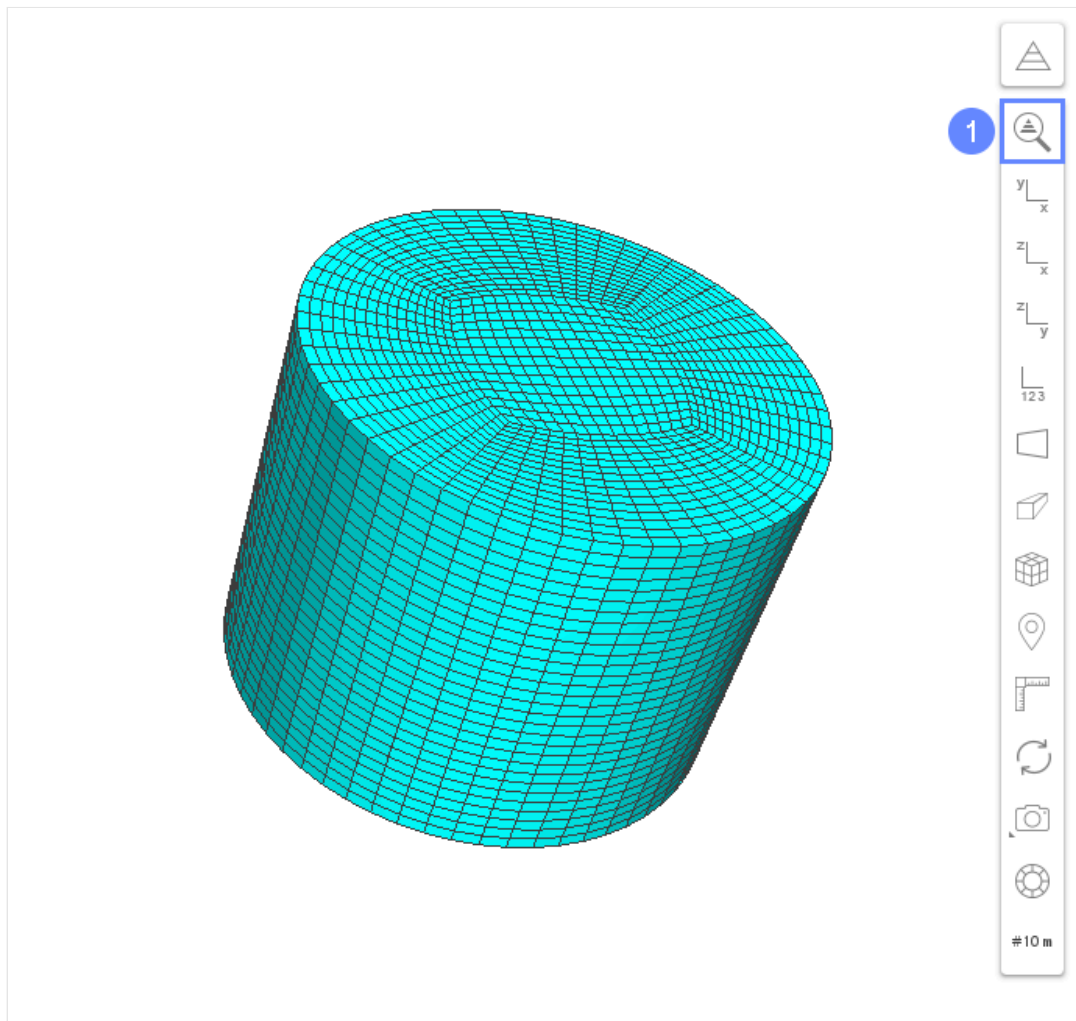


## 19. Mesh

After the meshing process is finished the mesh will appear in the 3D graphics window. The mesh used in this tutorial is created for demonstration purposes only, and its resolution should be considerably finer in real case scenario.

- 1 Click **Fit View** to zoom the geometry



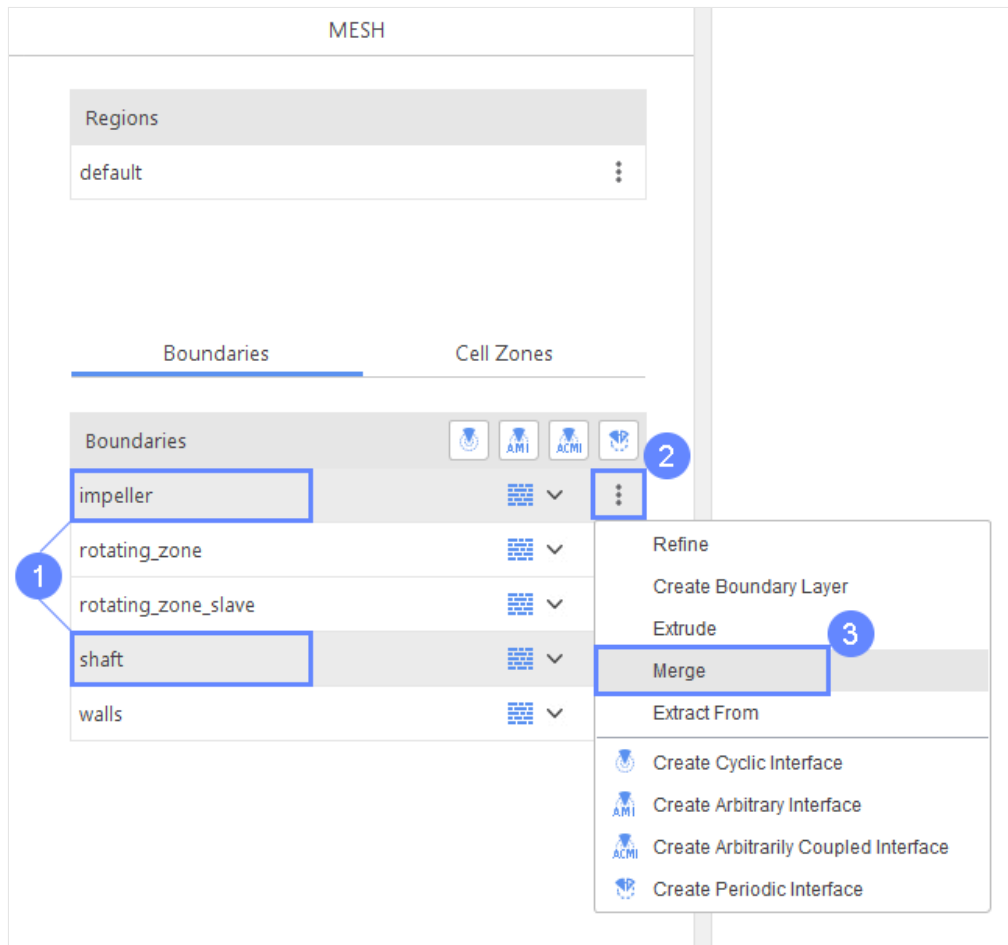


## 20. Merge Boundaries - Impeller and Shaft (I)

In order to simplify the definition of boundary conditions, we will merge the shaft and propeller boundaries into a single boundary.

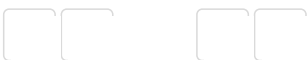
- 1 Hold **CTRL** key and select **impeller** and **shaft** boundaries
- 2 On the **impeller** boundary select **Options**
- 3 Choose **Merge** operation from the dropdown menu

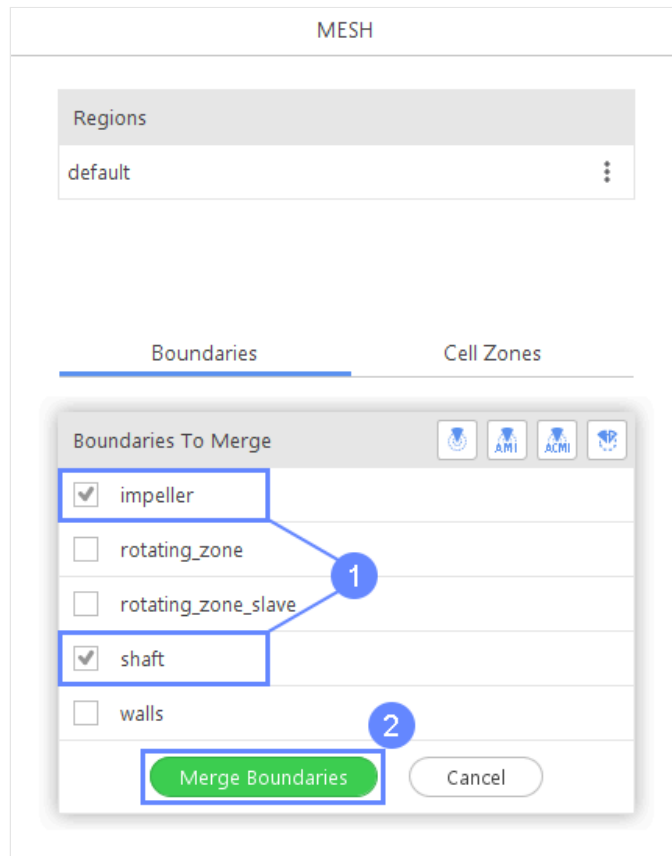




## 21. Merge Boundaries - Impeller and Shaft (II)

- 1 Make sure you have selected **impeller** and **shaft** boundaries
- 2 Click **Merge Boundaries** to confirm the operation

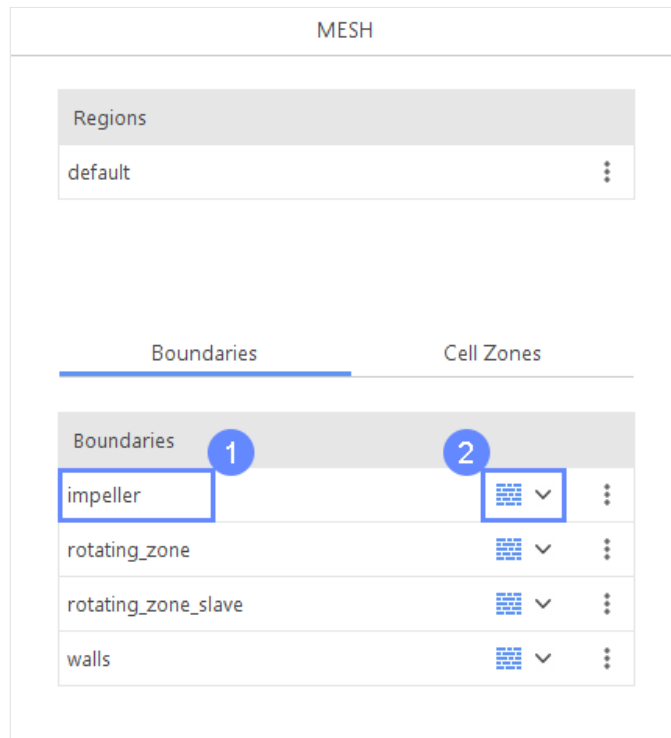




## 22. Merge Boundaries - Impeller and Shaft (III)

- 1 Rename newly created boundary from **impeller\_merged** to **impeller**
- 2 Change impeller's boundary type to **wall**



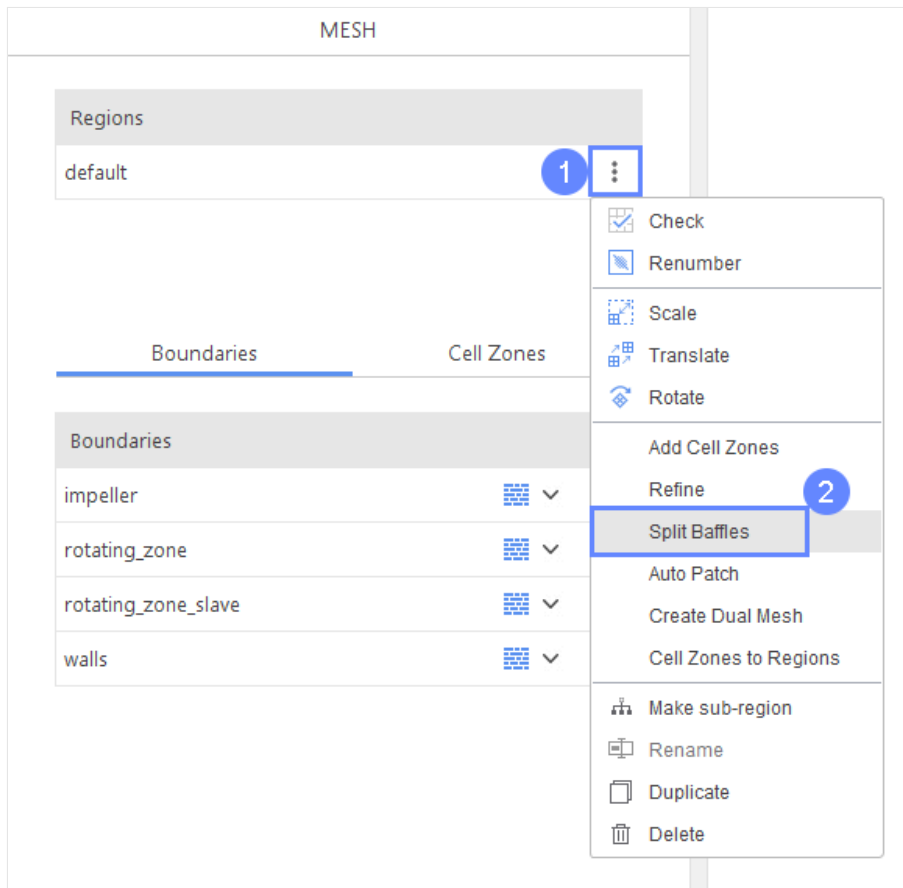


## 23. Split Baffles

Even though we requested the creation of baffles for the rotating\_zone, two sides of the baffle can still share some common nodes. In order to make sure that each side has its own nodes, we need to perform the Split Baffles operation on the mesh. This is a very important step when using dynamic mesh model.

- 1 Expand **Options** of the default region
- 2 Select **Split Baffles** from the drop-down list



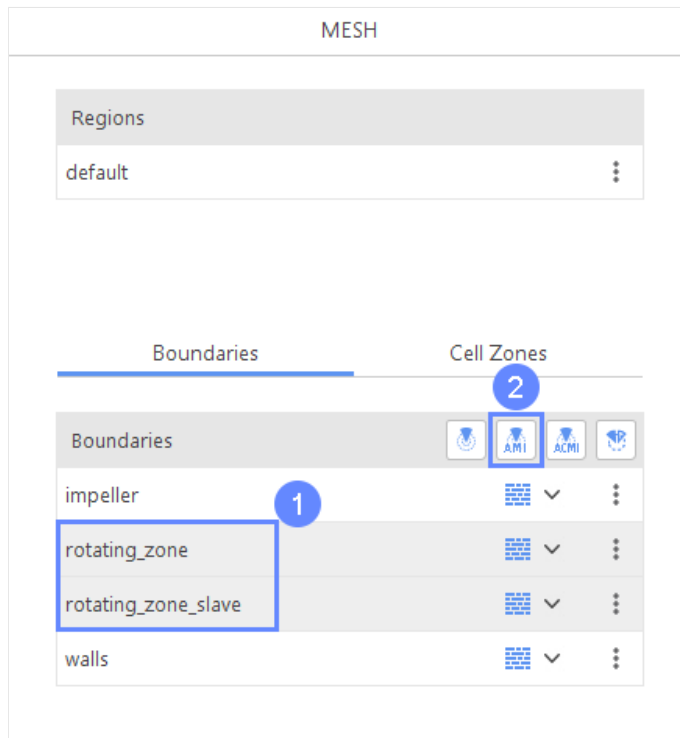


## 24. Create Mesh Interface

In the mesh setup, we requested creating baffles for `rotating_zone`, so that it can be a slip surface for the dynamic mesh. Now we will create the interface between the baffles.

- 1 Hold `CTRL` key and select `rotating_zone` and `rotating_zone_slave`
- 2 Click `Create Arbitrary Interface`

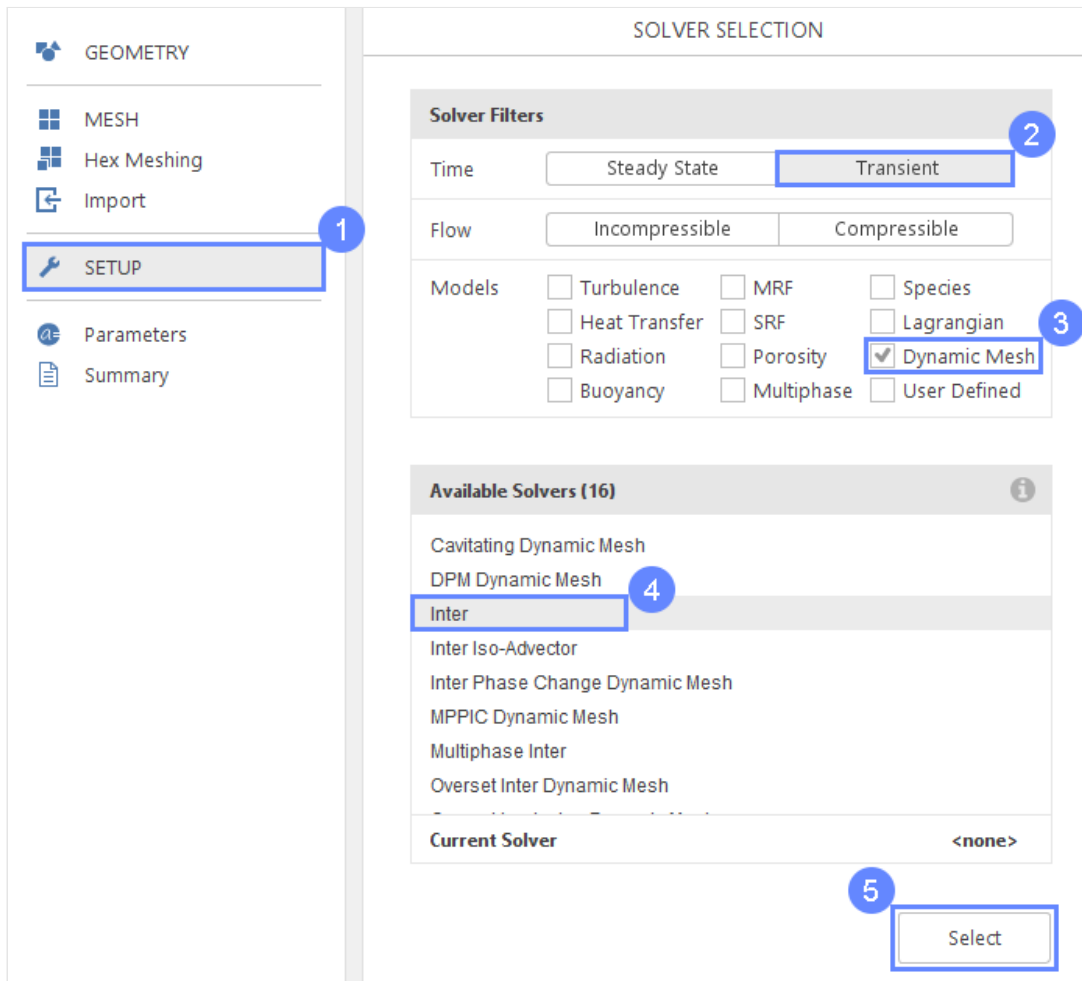




## 25. Setup Solver - Inter

We will use the Inter ([interFoam](#)) solver. This solver is able to model two-phase flow with free surface and supports dynamic mesh capabilities.

- 1 Go to **Setup** panel
- 2 Select **Transient** filter
- 3 Select **Dynamic Mesh** model filter
- 4 Pick **Inter** ([interFoam](#)) solver
- 5 **Select** solver



## 26. Dynamic Mesh

We will apply a rotational speed of 2 revolutions per second to the zone with the impeller. Due to the orientation of our geometry, our rotation axis will be the -Z-axis of the global coordinate system.

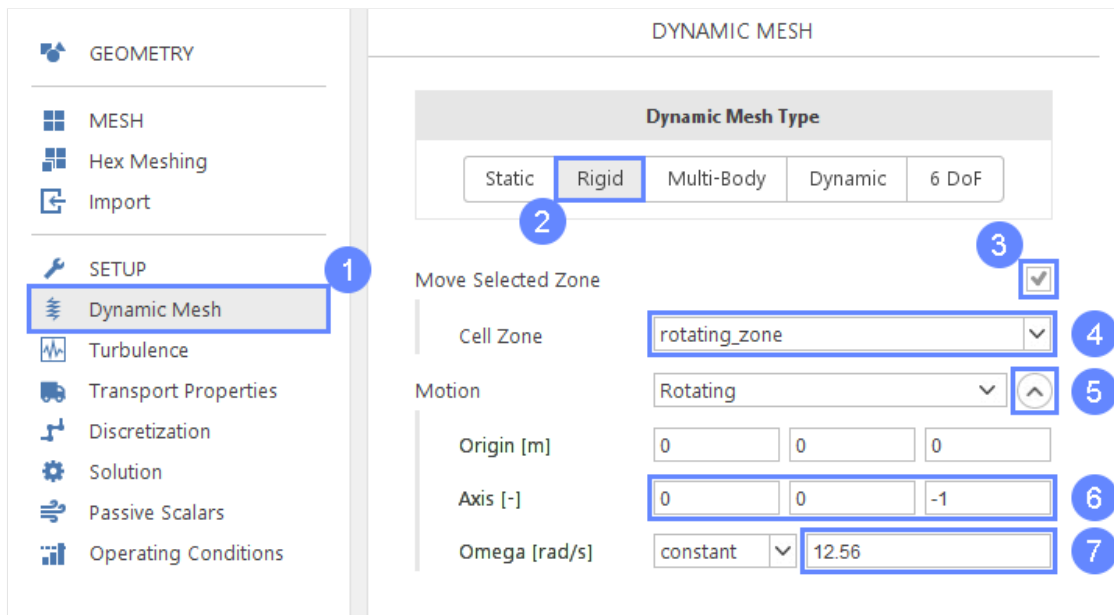
- 1 Go to **Dynamic mesh** panel
- 2 Select **Rigid** type
- 3 Check **Move selected zone**
- 4 Make sure **rotating\_zone** is selected as Cell Zone
- 5 Expand **Motion** options



Axis [-]    0    0    -1



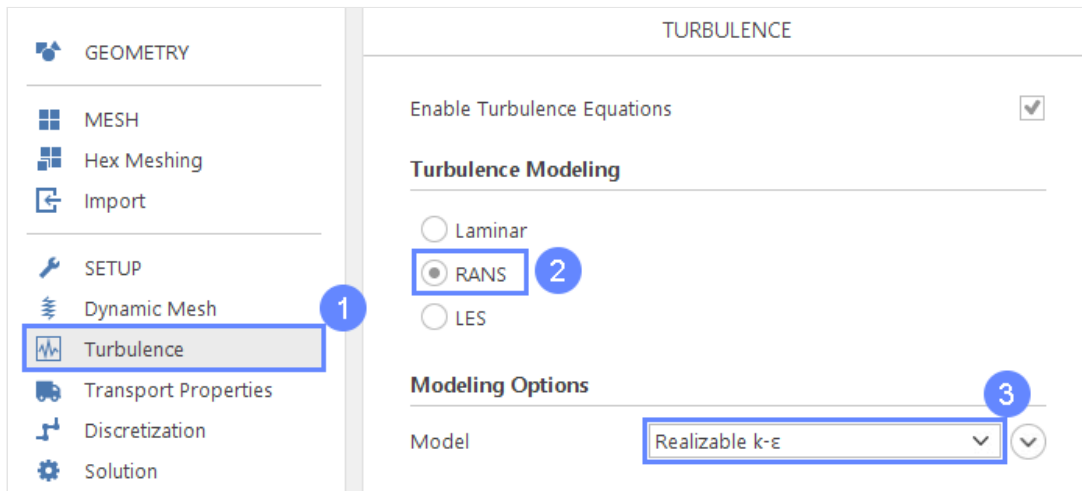
7 Set rotational speed  $\Omega$  [rad/s] 12.56



## 27. Turbulence

For turbulence modeling we will use *Realizable  $k-\epsilon$*  model.

- 1 Go to **Turbulence** panel
- 2 Select **RANS** turbulence formulation
- 3 Select *Realizable  $k-\epsilon$*  model



## 28. Operating Conditions

Incompressible solver like an *Inter*, operates on relative pressure (more [here](#)). In the incompressible Navier-Stokes equation, pressure is only related to the gradient and has no specific bounds. The pressure distribution can be determined by solving the N-S equation, but the exact value of pressure is irrelevant as long as the pressure difference between two points is the same.

To define precise value, it is necessary to define the pressure at a specific point. While the pressure at inlet or outlet boundaries is usually defined, if the domain is enclosed, a reference point must be established within the domain.

- 1 Go to **Operating Conditions** panel
- 2 Change the **Location** to **Point**
- 3 Set **Point** **0.4 0 0.5**



GEOMETRY

MESH

Hex Meshing

Import

Airfoil

SETUP

Dynamic Mesh

Turbulence

Transport Properties

Discretization

Solution

Passive Scalars

Operating Conditions

Cell Zones

Boundary Conditions

Initial Conditions

Monitors

OPERATING CONDITIONS

Gravitational Acceleration

$g$  [m/s<sup>2</sup>]

0

0

-9.81

$h_{ref}$  [m]

0

Reference Pressure

Location

Cell

Point

Point

0.4

0

0.5

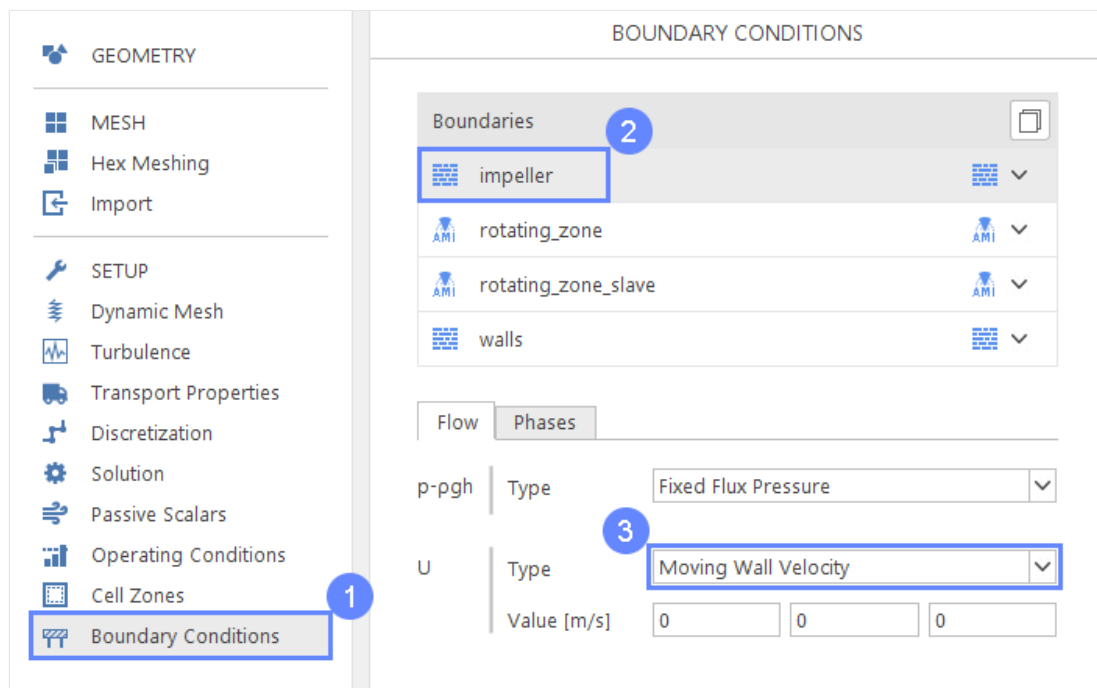
Value

0

## 29. Boundary Conditions - Impeller (Flow)

We need to make sure that impeller velocity is taken from the properties of the rotating zone. For this purpose, we need to apply the Moving Wall Velocity boundary condition.

- 1 Go to **Boundary Conditions** setup
- 2 Select **impeller** boundary
- 3 Change velocity type to **Moving Wall Velocity**



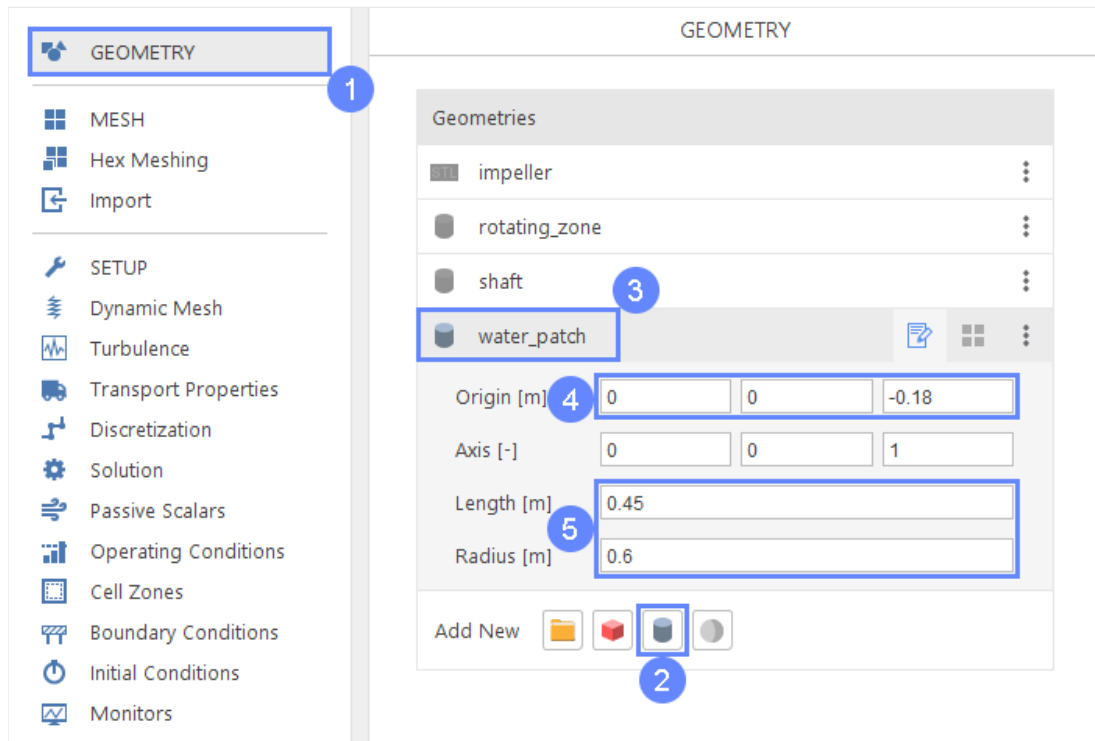
### 30. Geometry - Initial Conditions Patch

As an initial state, we want the mixing tank to be partly filled with water. For this purpose, we need to create geometry to indicate the initial water location.

- 1 Go to **Geometry** panel
- 2 Click **Create Cylinder**
- 3 Rename **cylinder\_1** to **water\_patch**
- 4 5 Define cylinder origin, length, and radius
 

**Origin [m]**    0    0    -0.18  
**Length [m]**    0.45  
**Radius [m]**    0.6



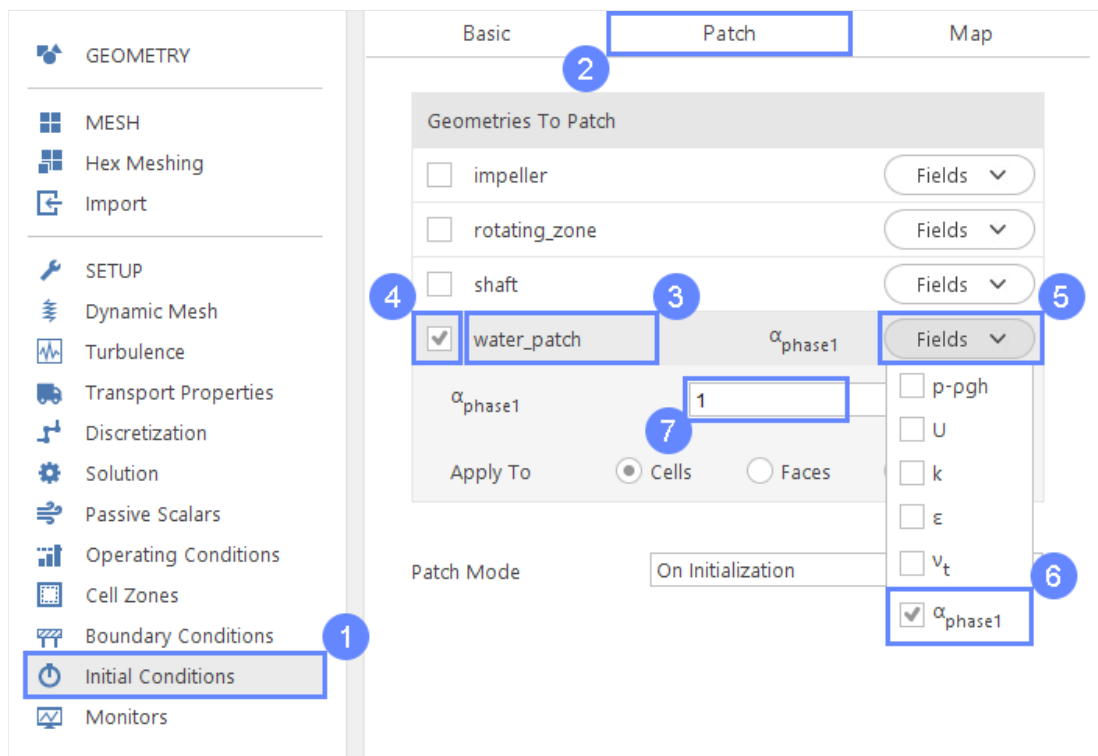


### 31. Initial Conditions - Patch

For initial conditions, we will leave all the default values, except inside the water patch, where the water phase fraction will be different.

- 1 Go to **Initial Conditions** panel
- 2 Switch to **Patch** tab
- 3 Select **water\_patch** geometry
- 4 Enable initialization on **water\_patch**
- 5 Expand **Fields** list
- 6 Select  $\alpha_{phase1}$  fraction for initialization
- 7 Set  $\alpha_{phase1}$  to **1**





## 32. Run - Time Control

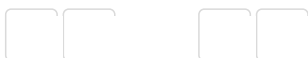
We will now adjust the time controls in order to capture the motion of the moving mesh in the output files. We will also enable automatic time step control for the simulation with reduced Courant number for better convergence.

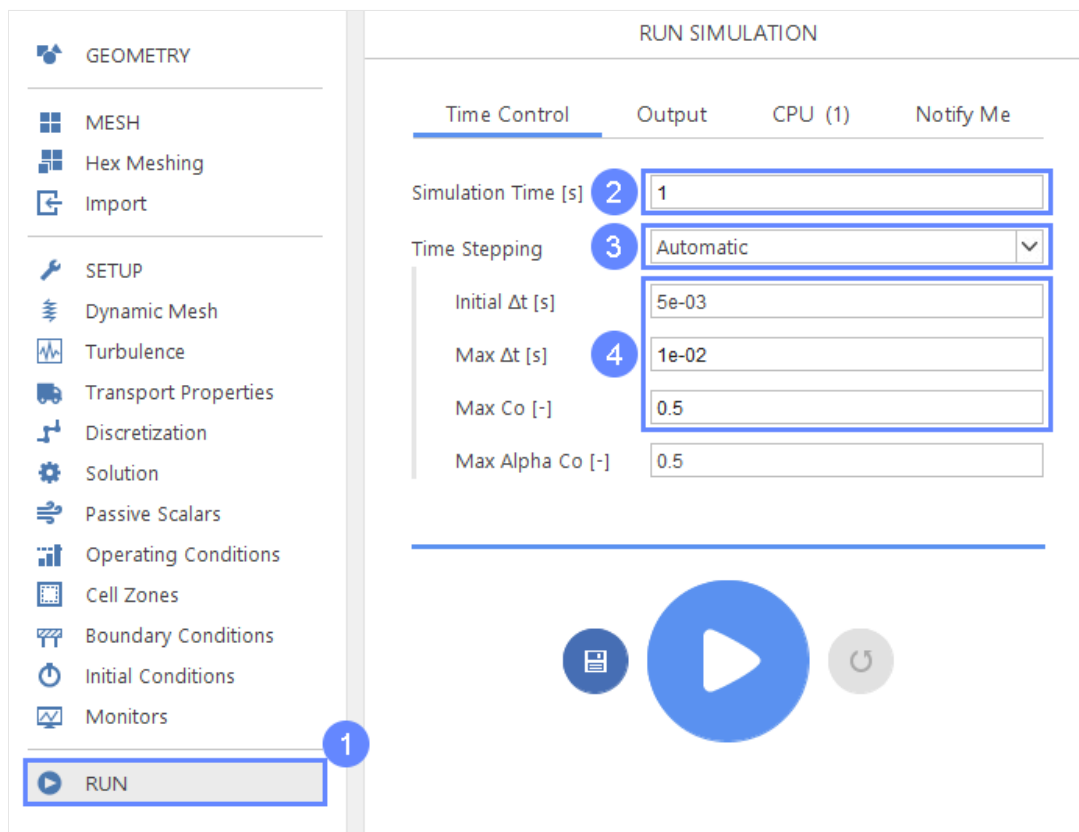
- 1 Go to **Run** panel
- 2 Set **Simulation Time [s]** to **1**
- 3 Change **Time Stepping** to **Automatic**
- 4 Set initial time step, time step limit and Courant number accordingly
 

**Initial  $\Delta t$  [s]**    **5e-03**

**Max  $\Delta t$  [s]**    **1e-02**

**Max Co [-]**    **0.5**

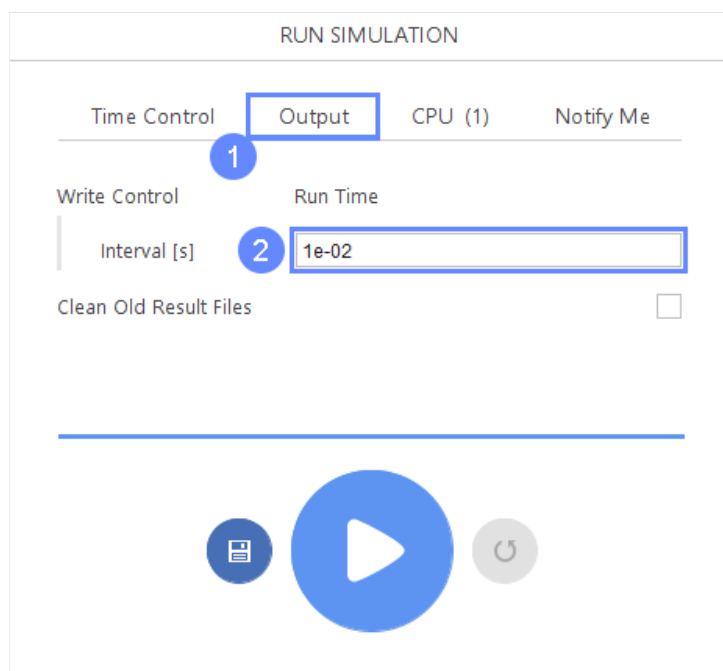




### 33. Run - Output

Now we just have to specify the frequency of writing results to the disk and start the simulation.

- 1 Switch to **Output** panel
- 2 Set Write Control **Interval [s]** to **1e-02**



## 34. Run - CPU

To speed up the calculation process, take advantage of parallel computing and increase the number of CPUs based on your PC's capability. The free version allows you to use only one processor (serial mode). To get the full version, you can use the contact form to

[Request 30-day Trial](#)

Estimated computation time for serial mode: 50 minutes

- 1 Switch to **CPU** tab
- 2 Click **Run Simulation** button





RUN SIMULATION

Time Control

Output

CPU (1)

Notify Me

☒ serial

☐ parallel

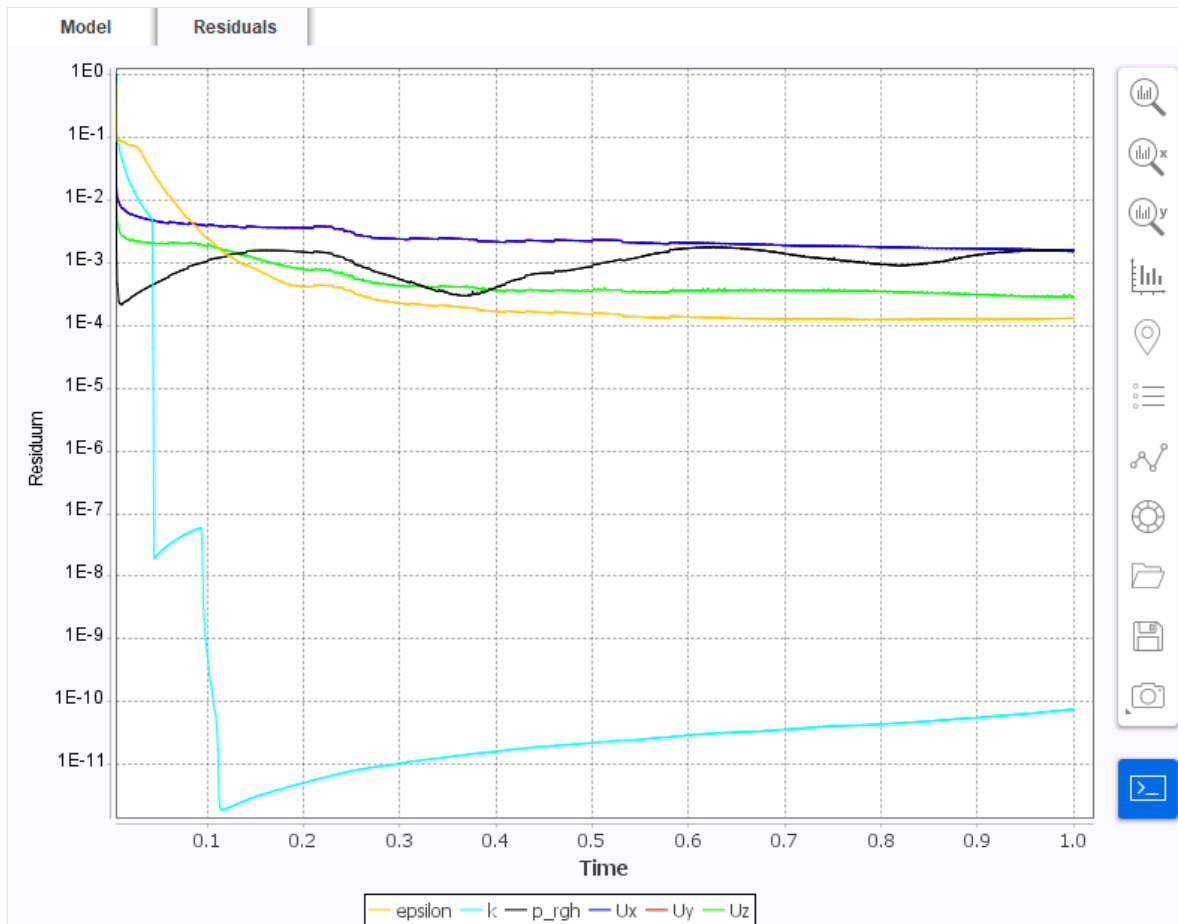
Number of processors 6

2

## 35. Residuals

When the simulation is finished we should see a similar residual plot.

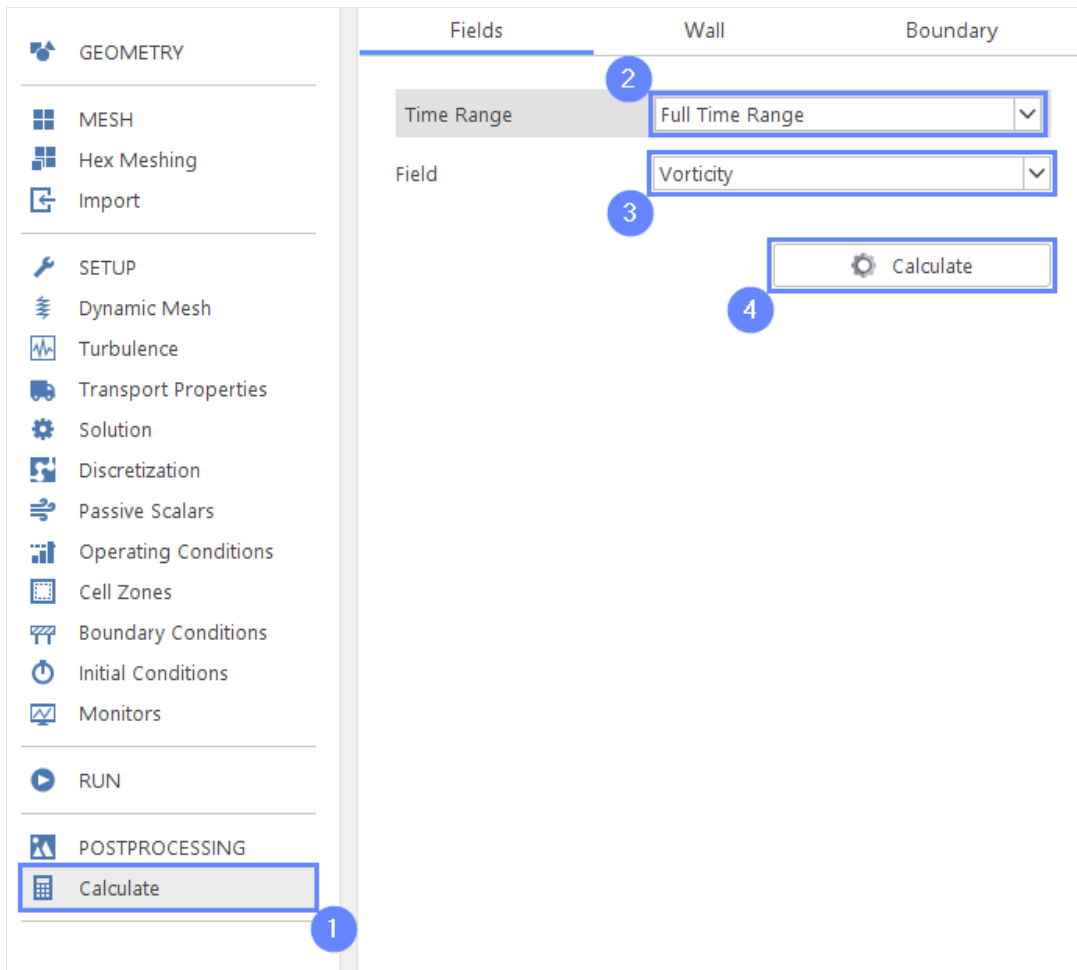




## 36. Calculate Additional Fields

When the simulation is finished we want to calculate additional flow variables to use later for postprocessing.

- 1 Go to **Calculate** panel
- 2 Select **Full Time Range**
- 3 Select **Vorticity** field
- 4 Click **Calculate**

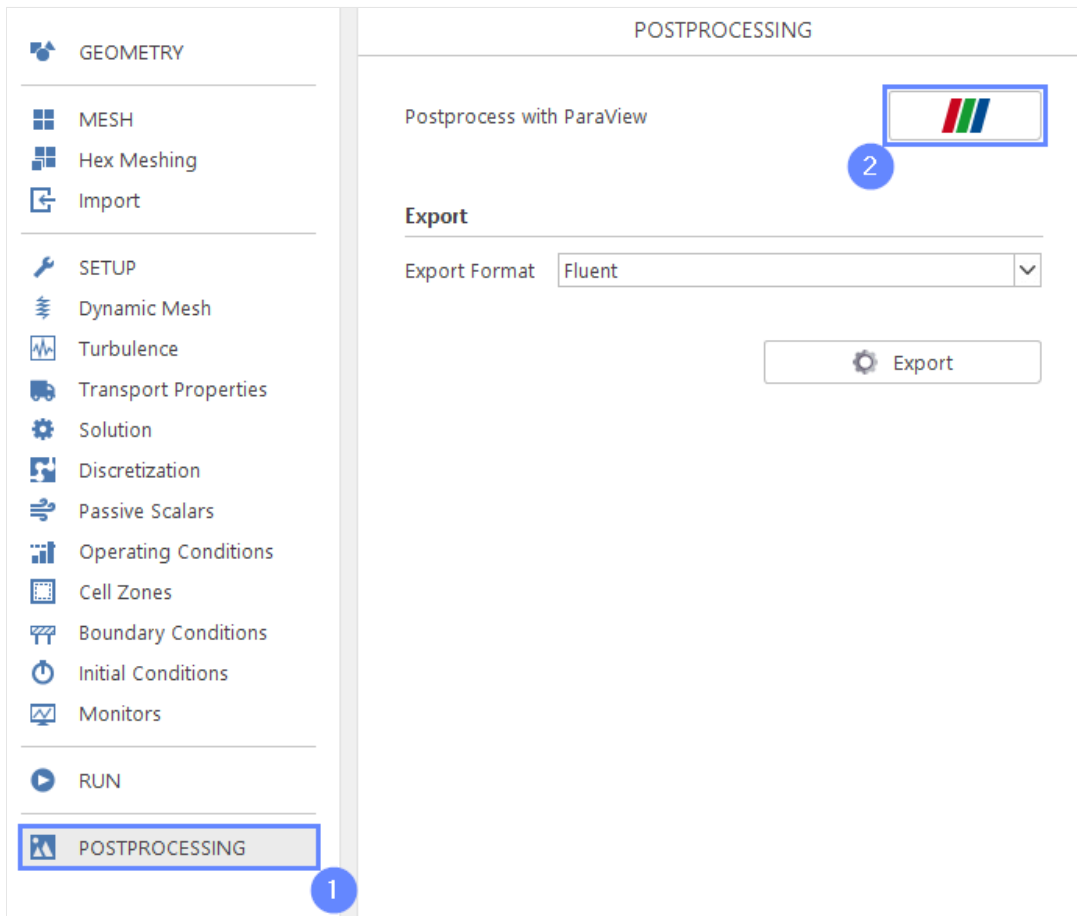


## 37. Start Postprocessing - ParaView

Start ParaView when the simulation is finished. Optionally you can start ParaView when the simulation is still in progress to observe the intermediate results.

- 1 Go to **Postprocessing** panel
- 2 Start **ParaView**

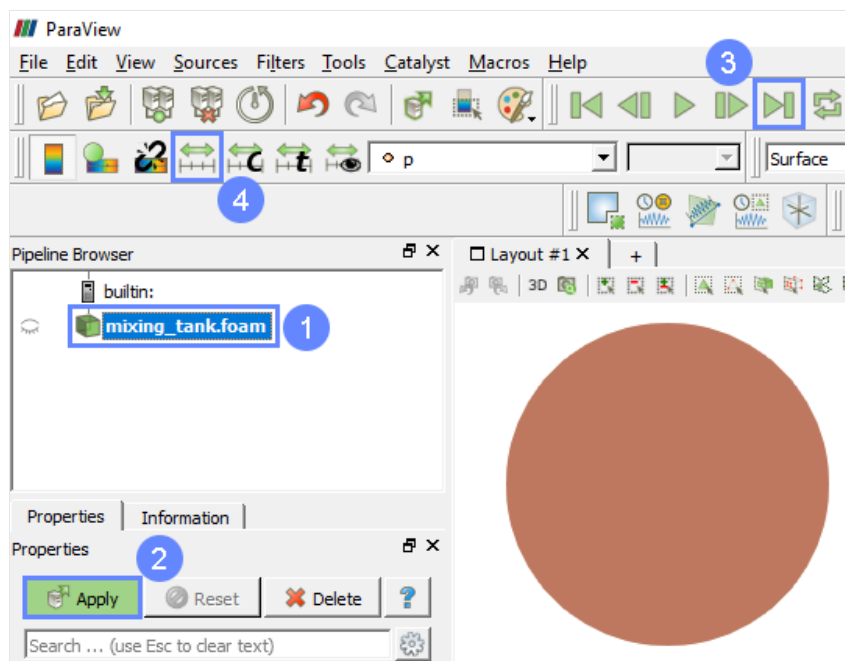




## 38. ParaView - Load Results

After opening ParaView, we have to load the results of the simulation.

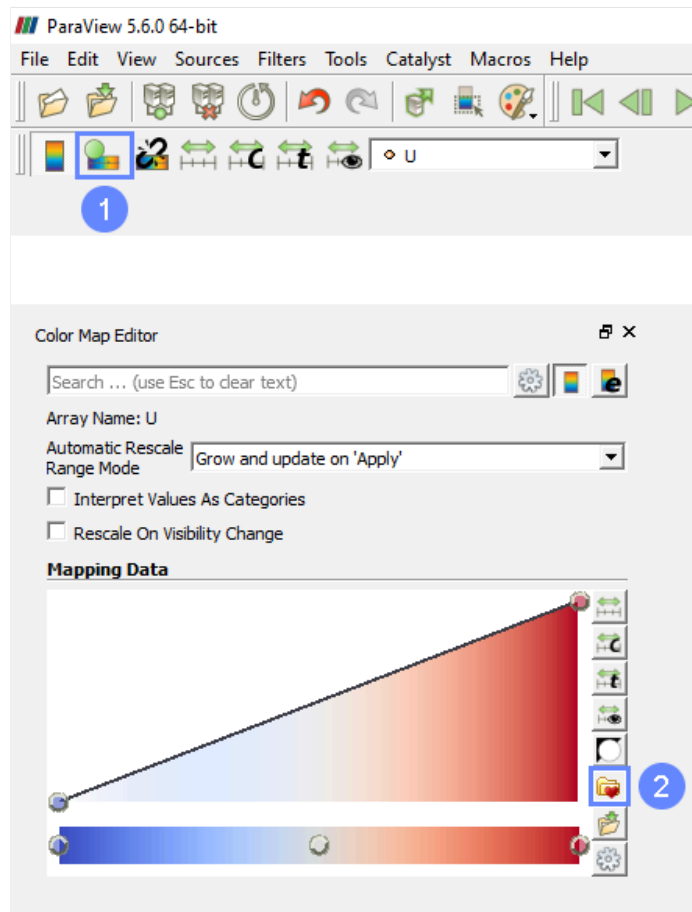
- 1 Select your case `mixing_tank.foam`
- 2 Click Apply
- 3 Click **Last Frame**
- 4 Click **Rescale to Data Range**
- 5 After rescale the contour will be shown in the 3D window.



## 39. ParaView - Coloring (I)

We can change the coloring scheme in ParaView to have nicer colors.

- 1 Click **Edit Color Map** from the menu placed on the left side, if the panel is not already shown.
- 2 Select **Choose Preset** from the Color Map Editor placed by default on the right side of the ParaView



## 40. ParaView - Coloring (II)

We can now select a new Color Preset.

- 1 Expand **Advanced Options**
- 2 Choose **Blue to Red Rainbow** preset
- 3 **Apply** changes
- 4 **Close** Choose Preset window



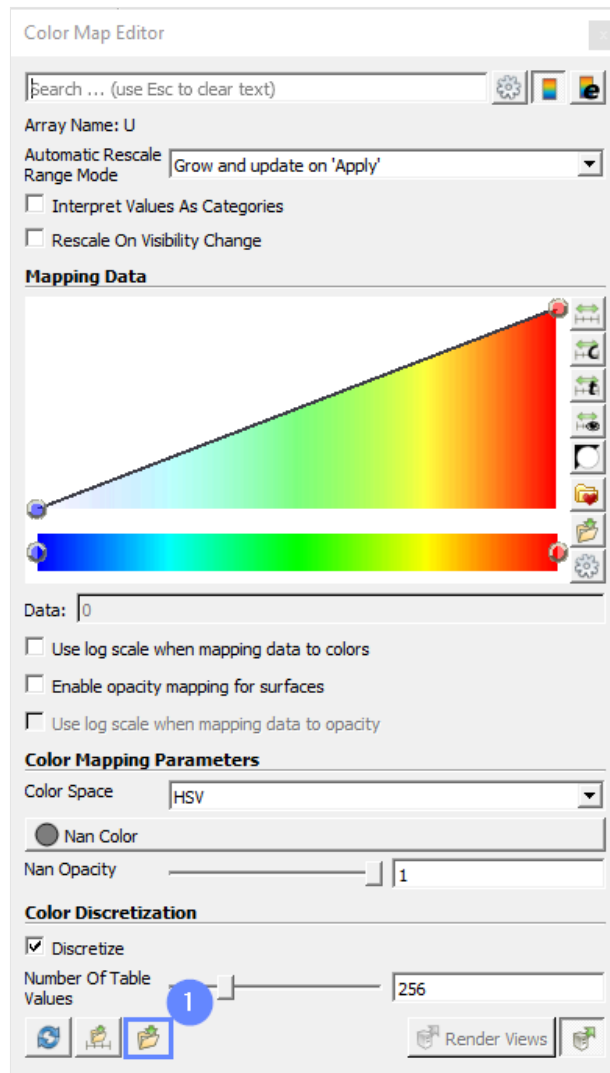


## 41. ParaView - Coloring (III)

To use the same color preset for all variables, we will save the current choice.

- 1 Click **Save current color map settings values as default for all arrays**





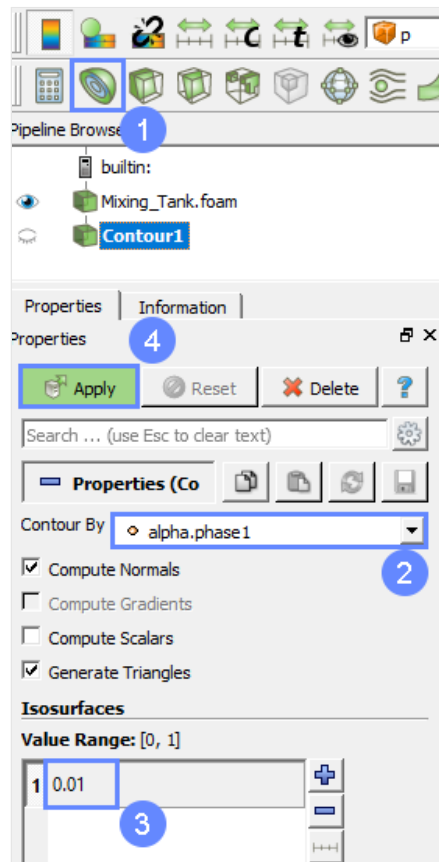
## 42. ParaView - Display Water Surface (I)

We will now plot the water surface colored by the water height.

- 1 Create **Contour**
- 2 Select phase fraction **alpha.phase1** field as a contour variable
- 3 Define water surface threshold at **0.01**
- 4 Click **Apply**





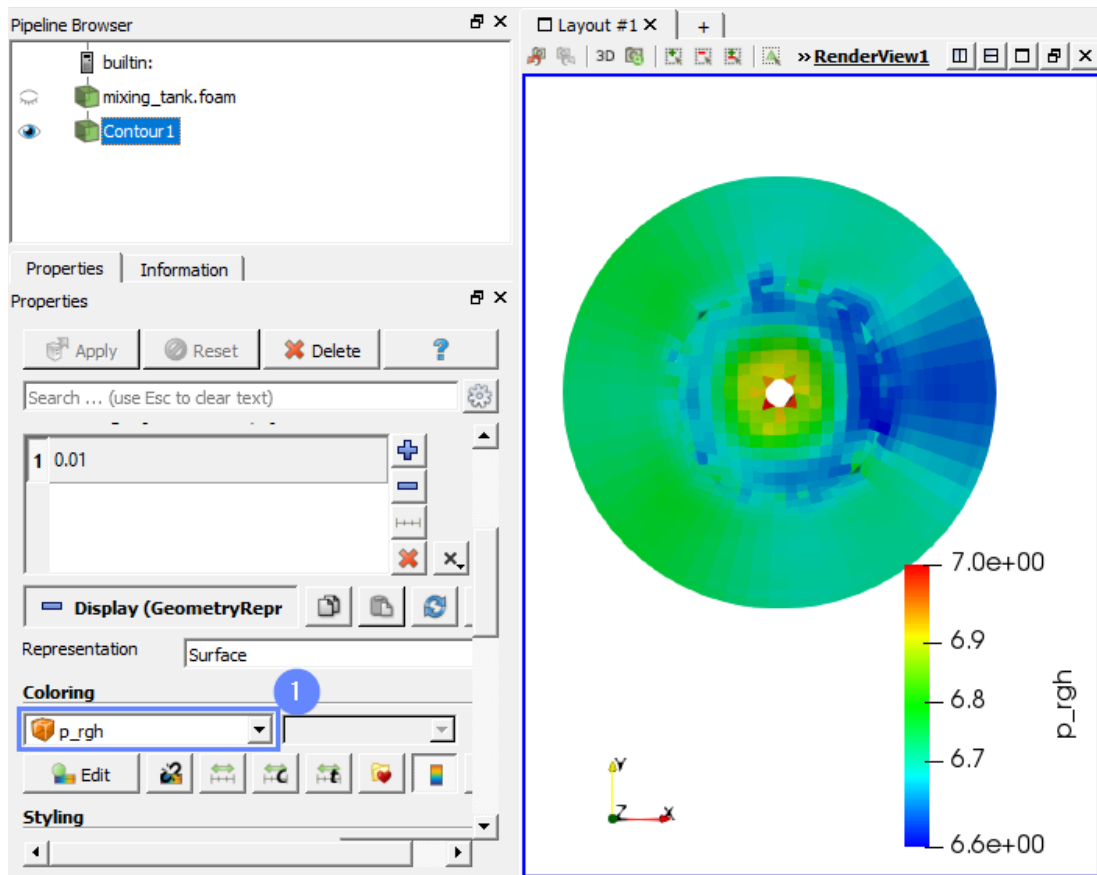


## 43. ParaView - Display Water Surface (II)

- 1 Select contour coloring variable `p_rgh`

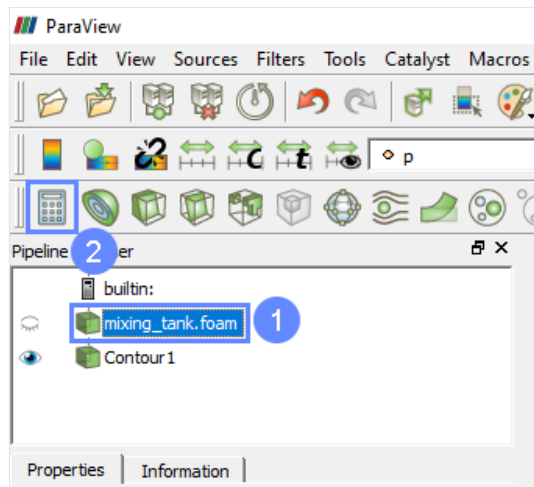
After change the coloring variable and opacity the contour will be shown in the 3D window. Note that the pressure on the water surface is constant.





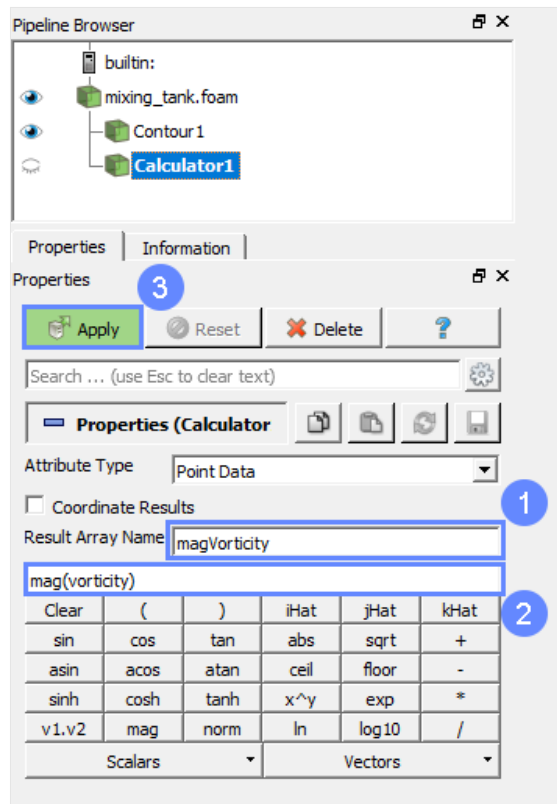
## 44. ParaView - Calculator (I)

- 1 Select **mixing\_tank.foam**
- 2 Clcik **Calculator**



## 45. ParaView - Calculator (II)

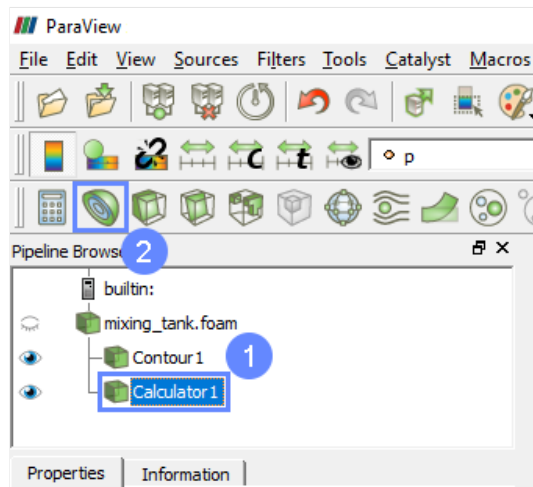
- 1 Change name to `magVorticity`
- 2 Write the formula `mag(vorticity)`
- 3 Click



## 46. ParaView - Create Contour (I)

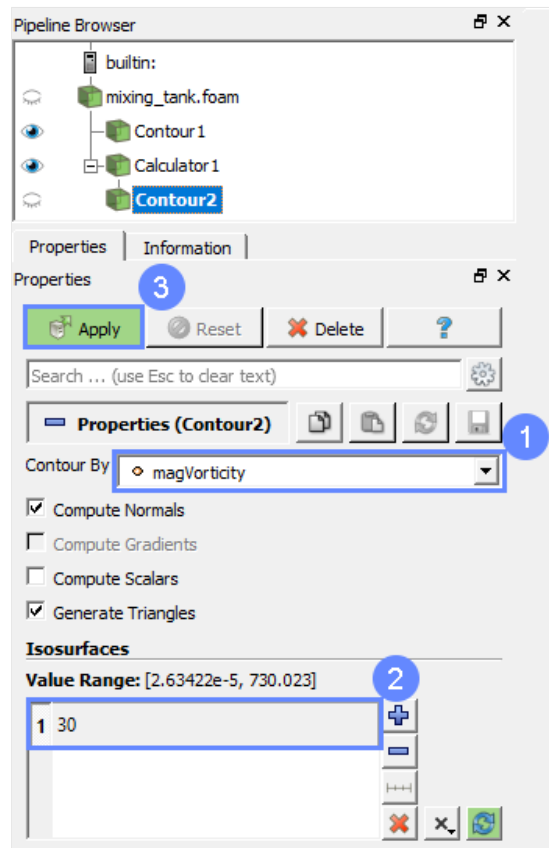
- 1 Select **Calculator 1**
- 2 Create **Contour**





## 47. ParaView - Create Contour (II)

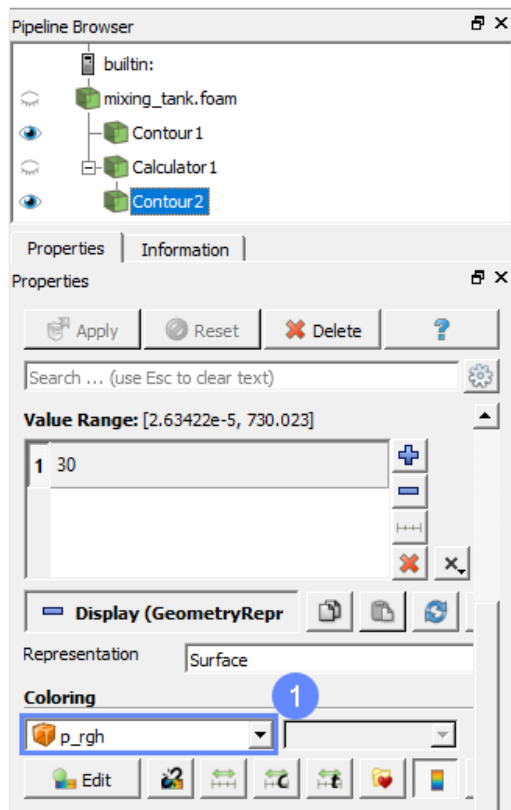
- 1 Select **magVorticity** as a contour variable
- 2 Set contour value to **30**
- 3 Click **Apply**



## 48. ParaView - Create Contour (III)

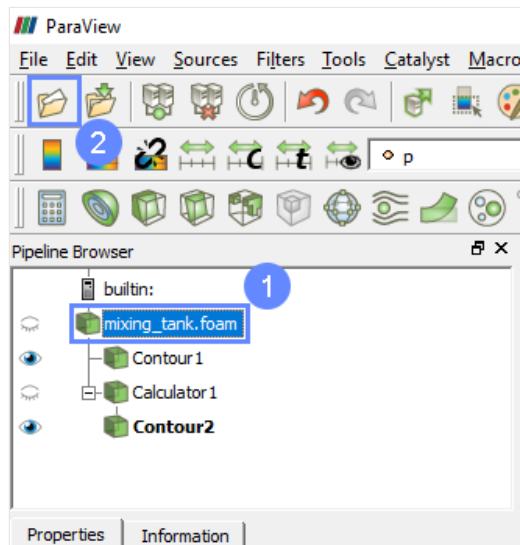
- 1 Select contour coloring variable `p_rgh`





## 49. ParaView - Load geometry (I)

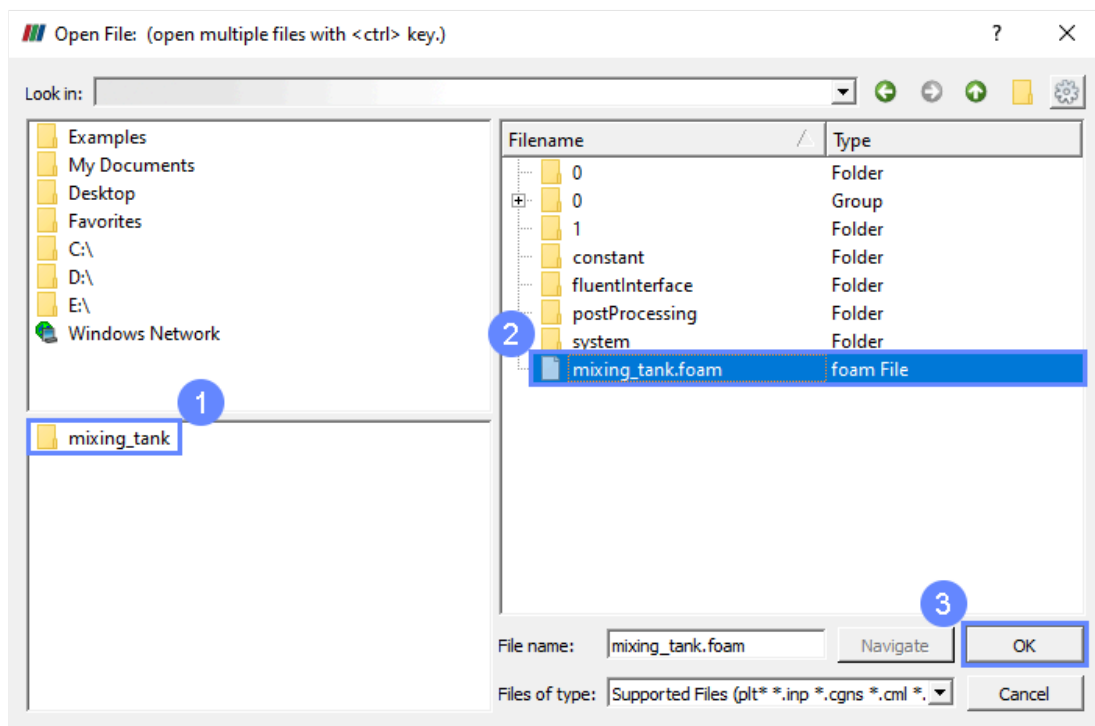
- 1 Select `mixing_tank.foam`
- 2 Click `Open`



## 50. ParaView - Load geometry (II)

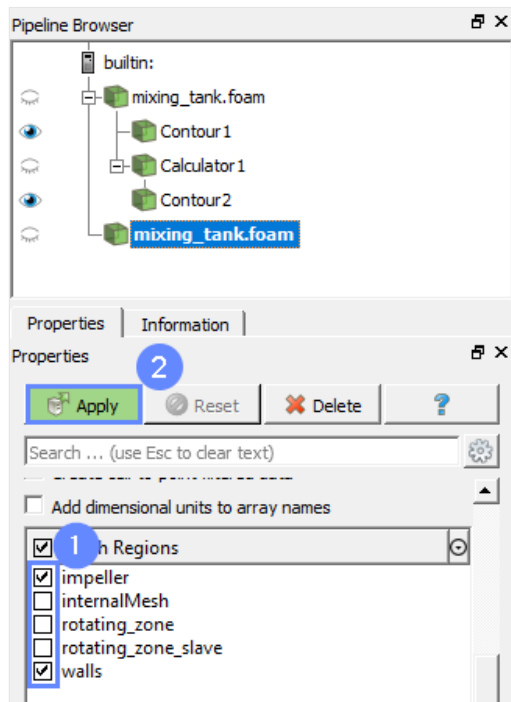
- 1 Click `mixing_tank`
- 2 Select `mixing_tank.foam` from file selection dialog
- 3 Click





## 51. ParaView - Load geometry (III)

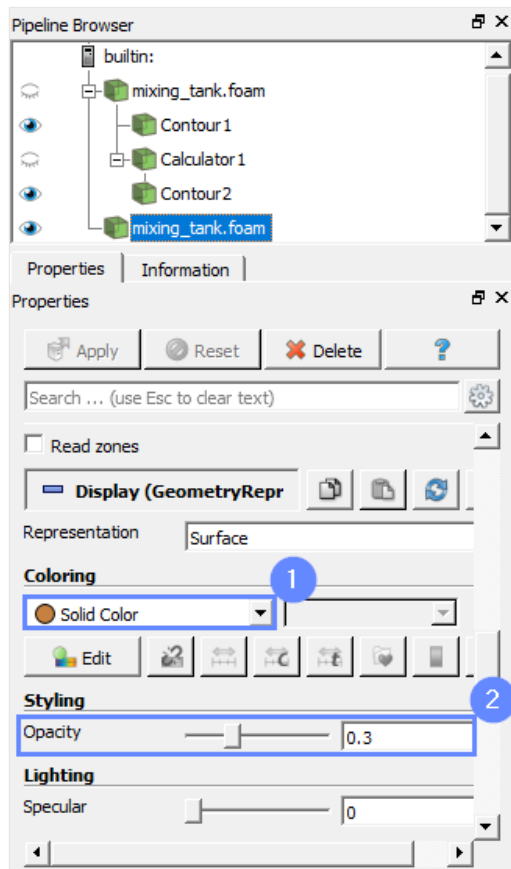
- 1 Select **impeller** and **walls** zones to plot
- 2 Click



## 52. ParaView - Load geometry (IV)

- 1 Set Coloring to **Solid Color**
- 2 Set the opacity to **0.3**

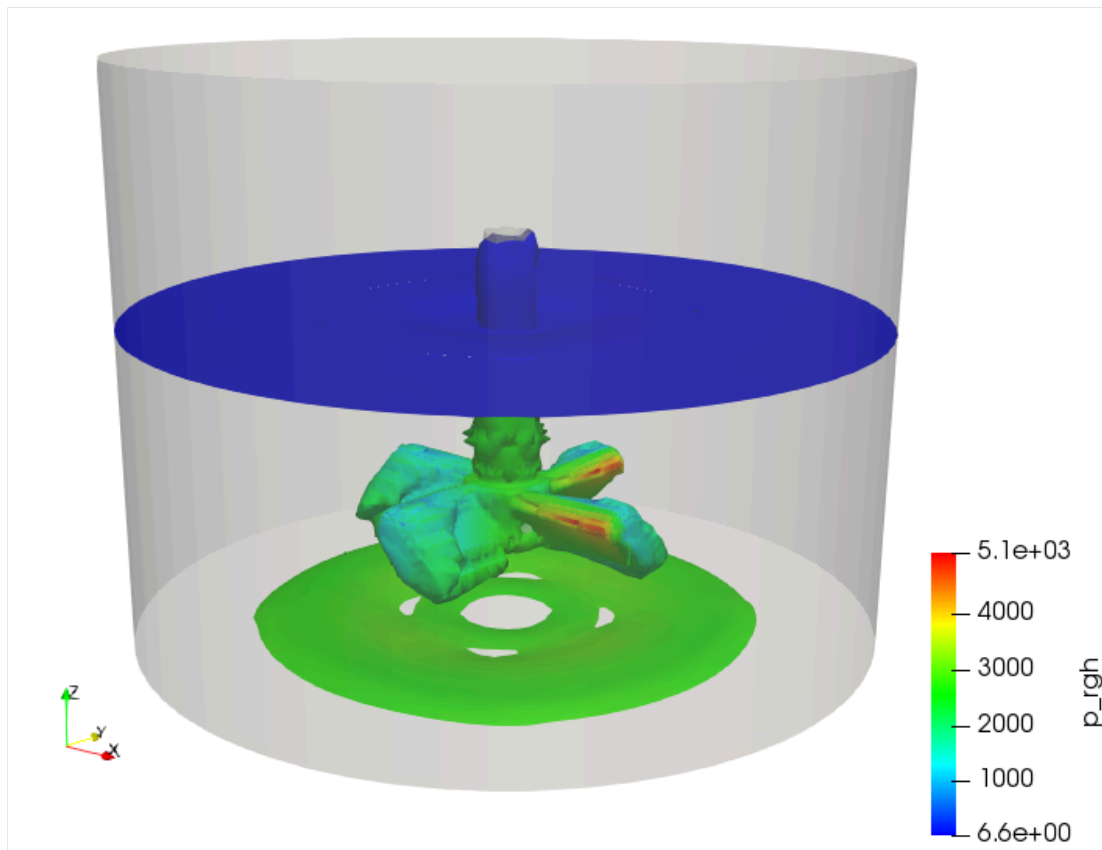




## 53. ParaView - Results

The results are displayed in the graphics window





## 54. Advanced Postprocessing with ParaView

This concludes the tutorial, covering all the aspects we intended to showcase. For a finely tuned presentation of the results, you may take advantage of the more advanced features.

In ParaView, you can display streamlines, contour plots, vector fields, line or time plots, calculating volume or surface integrals and create animations.

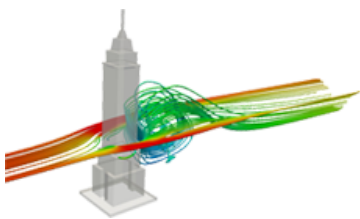
To familiarize yourself with the ParaView capabilities, it's worth checking out our video tutorial, *Paraview CFD Tutorial - Advanced Postprocessing in ParaView*, in which we demonstrate some of the most commonly used post-processing techniques.

[Paraview CFD Tutorial - Advanced Postprocessing in ParaView](#)



# SimFlow

CFD Software made EASY



## SimFlow CFD

[OpenFOAM GUI](#)  
[CFD Software](#)  
[Download](#)  
[Pricing](#)

## Resources

[Validation](#)  
[Tutorials](#)  
[What is CFD](#)  
[CFD Simulation](#)  
[CFD Analysis](#)  
[Faq](#)

## Company

[License Agreement](#)  
[Privacy Policy](#)  
[About](#)  
[Contact](#)

None of the OPENFOAM<sup>®</sup> related products and services offered by SIMFLOW Technologies are approved or endorsed by OpenCFD Limited, producer and distributor of the OpenFOAM software and owner of the

OPENFOAM<sup>®</sup> and OpenCFD<sup>®</sup> trade marks.

© 2012 - 2025 SimFlow Computational Fluid Dynamics (CFD) Software

