

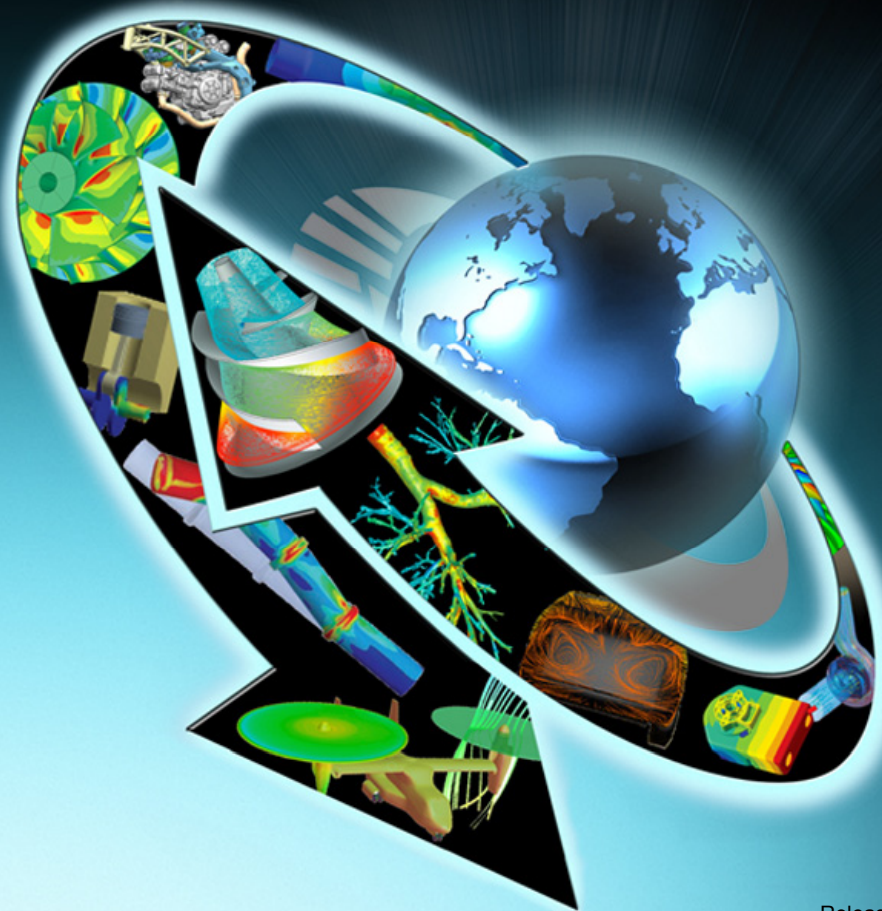


Customer Training Material

Workshop 7

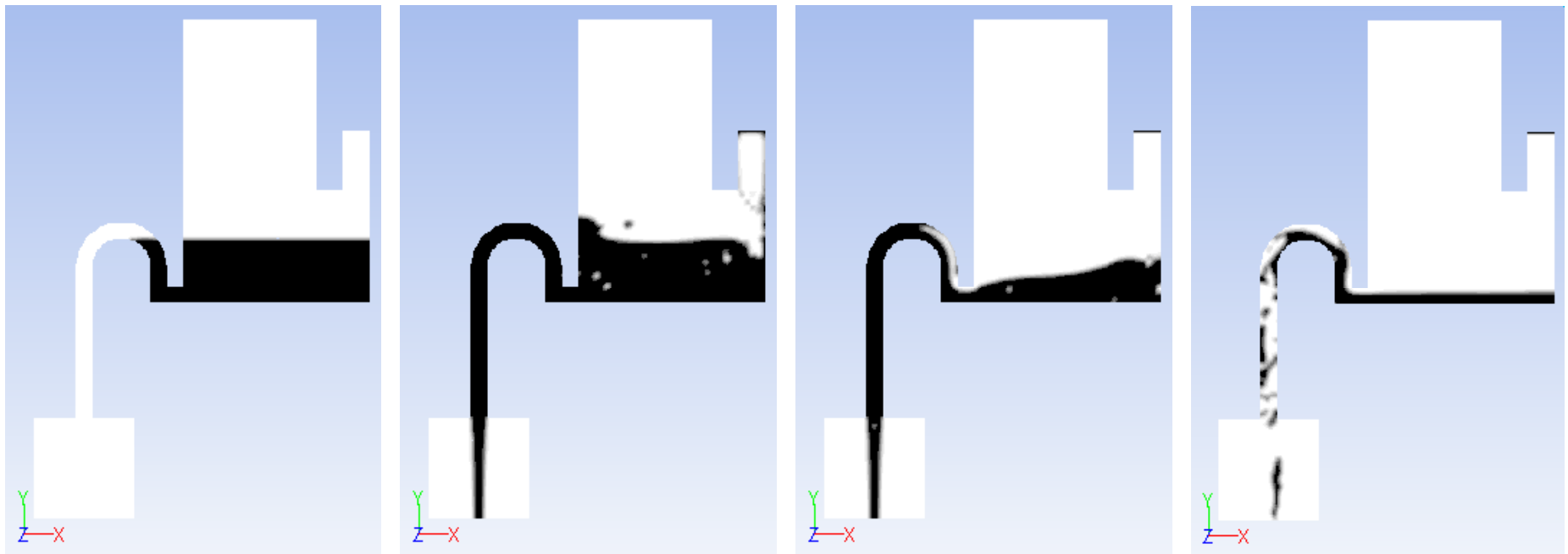
Tank Flushing Example

Introduction to ANSYS FLUENT



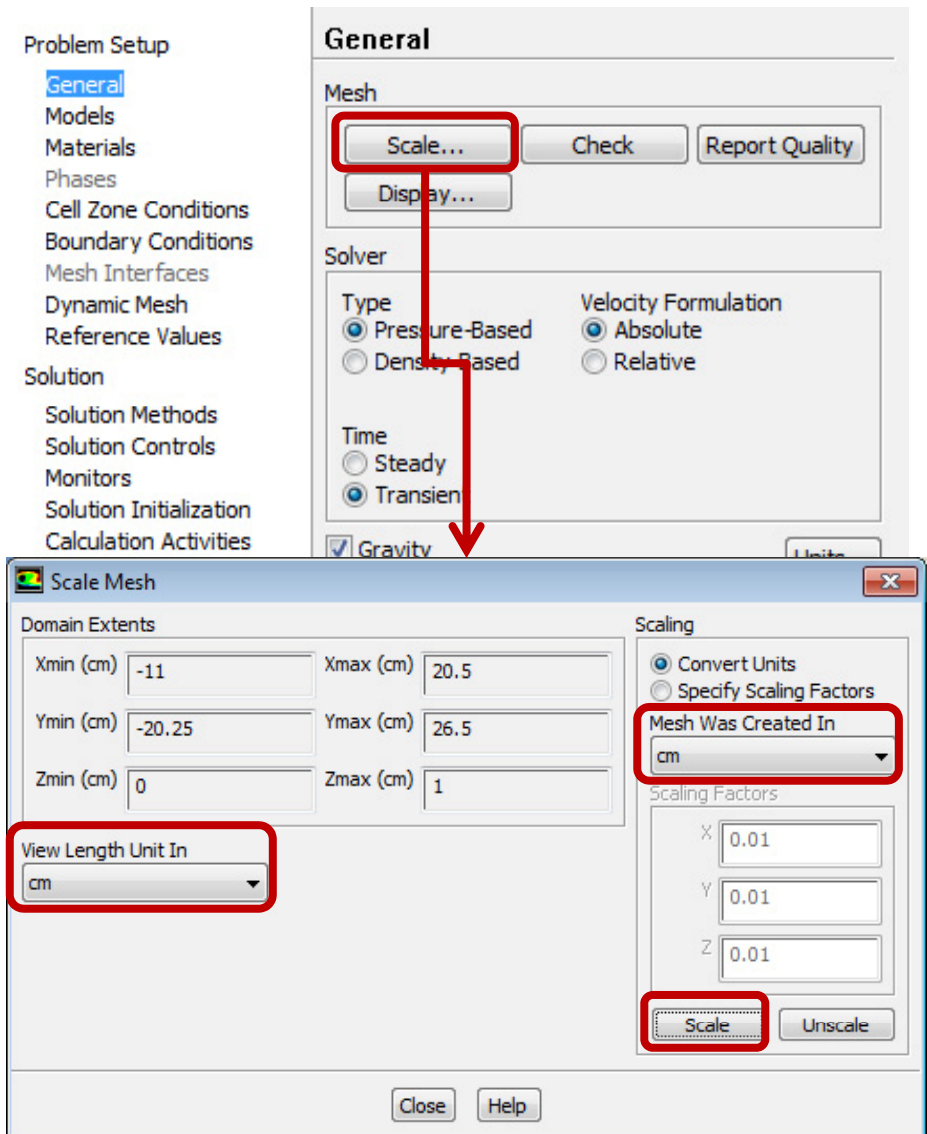
Introduction

- In this workshop, you will model the filling and emptying of a water tank. The transient problem is solved as a multiphase (air/water) simulation using the volume of fluid (VOF) multiphase model.
- An initial water level is set in the tank. The water supply is turned on for the first second of the simulation and then shut off for the rest of the simulation. The water level rises until water flows out the U-tube generating a siphoning effect which effectively empties the tank.



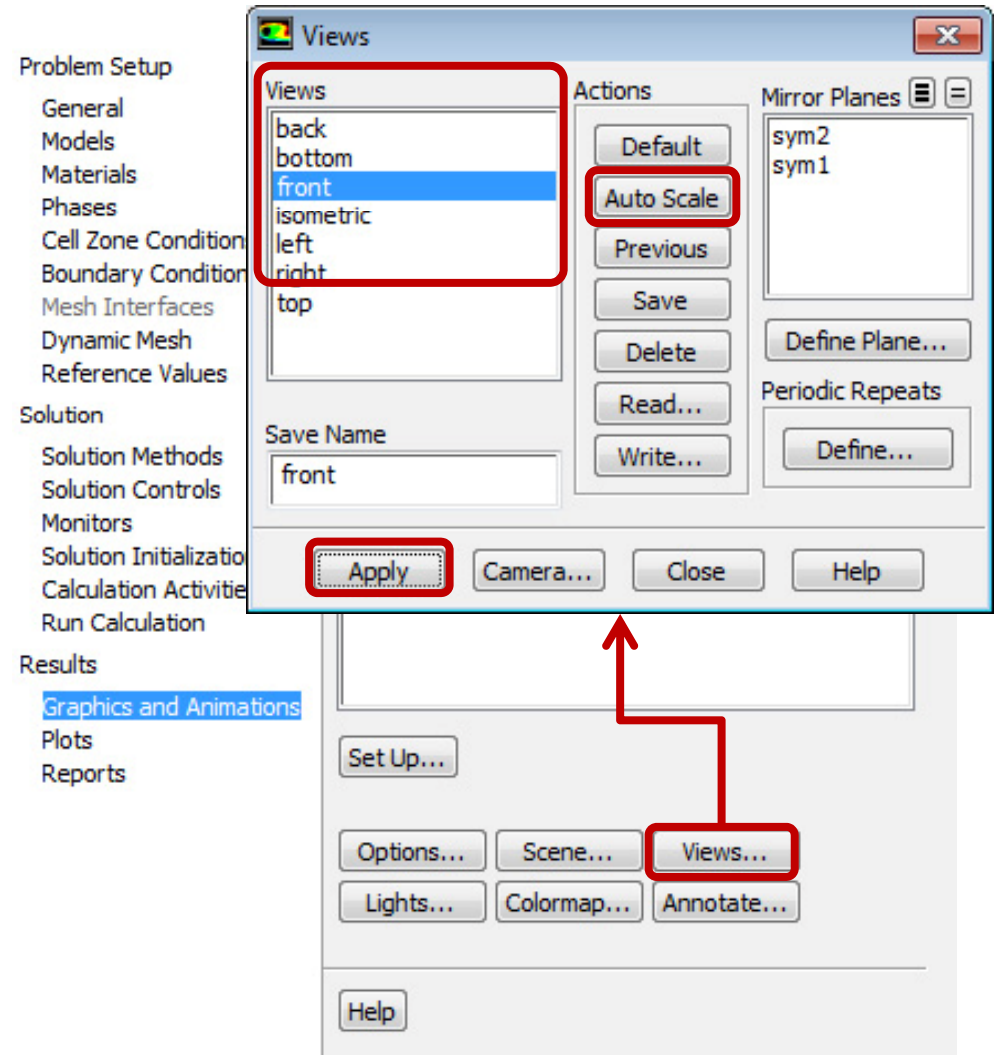
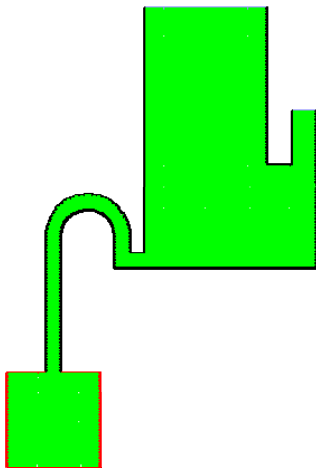
Mesh Import

- **Start a new 3D FLUENT session**
- Read or import the mesh file
[tankflush.msh.gz](#)
- **Click General in the outline tree.**
 - Scale the mesh to units of cm.
 - Set “View Length Unit In” to cm to have FLUENT display lengths in centimeters.
 - Verify the domain extents:
 $-11 < x < 20.5$ cm
 $-20.25 < y < 26.5$ cm
 $0 < z < 1$ cm
 - Check the mesh.
 - Redisplay the scaled mesh.



- **Orientate the view**

- Select 'Graphics and Animations' in the outline tree
- Click 'Views' button in the centre pane.
- In the panel that opens select 'front' under Views and click 'Apply' and 'Close'.



Define Simulation Type

- In the General panel, under Solver, set Time to **Transient**.
- Enable Gravity and set Gravitational Acceleration to **-9.81 m/s²** in the **y** direction.

Problem Setup

General

Models

Materials

Phases

Cell Zone Conditions

Boundary Conditions

Mesh Interfaces

Dynamic Mesh

Reference Values

Solution

Solution Methods

Solution Controls

Monitors

Solution Initialization

Calculation Activities

Run Calculation

Results

Graphics and Animations

Plots

Reports

General

Mesh

Scale...

Check

Report Quality

Display...

Solver

Type

☒ Pressure-Based

☐ Density-Based

Velocity Formulation

☒ Absolute

☐ Relative

Time

☐ Steady

☒ Transient

☒ Gravity

Units...

Gravitational Acceleration

X (m/s²)

0

P

Y (m/s²)

-9.81

P

Z (m/s²)

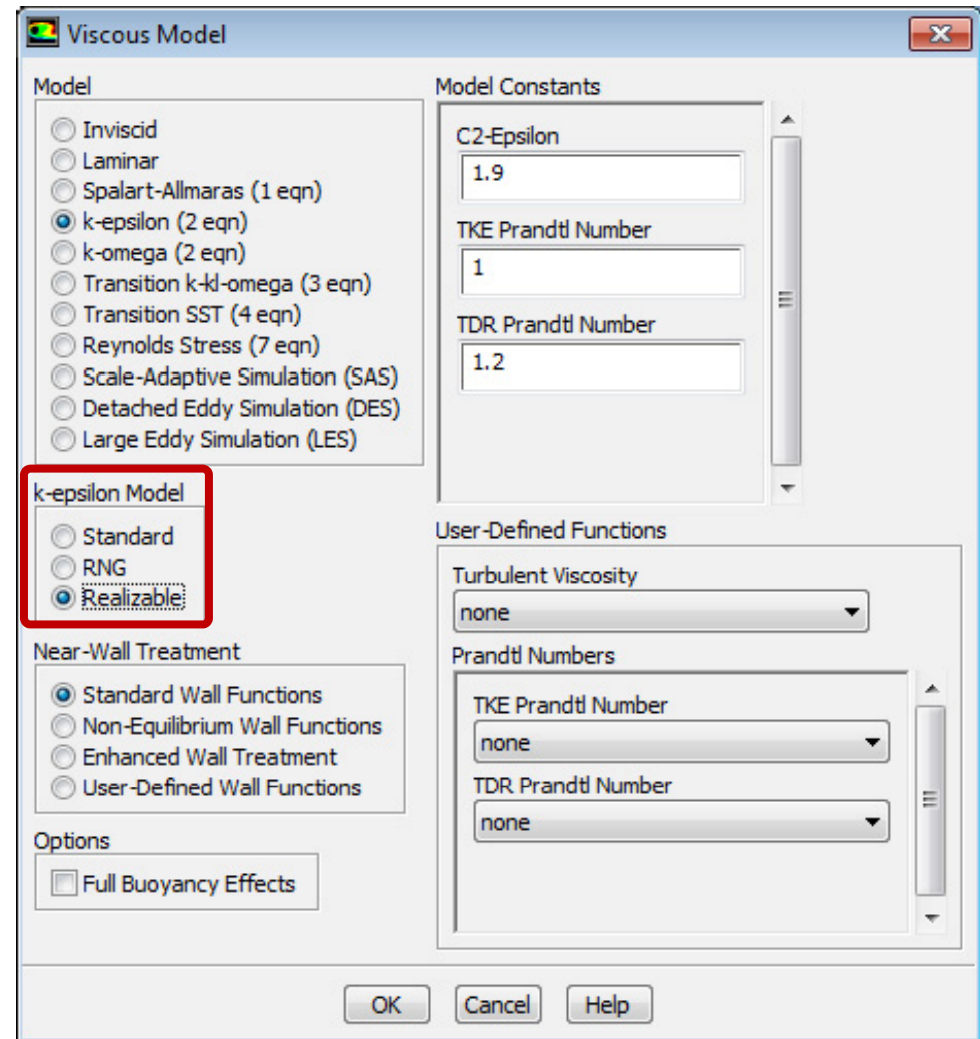
0

P

Help

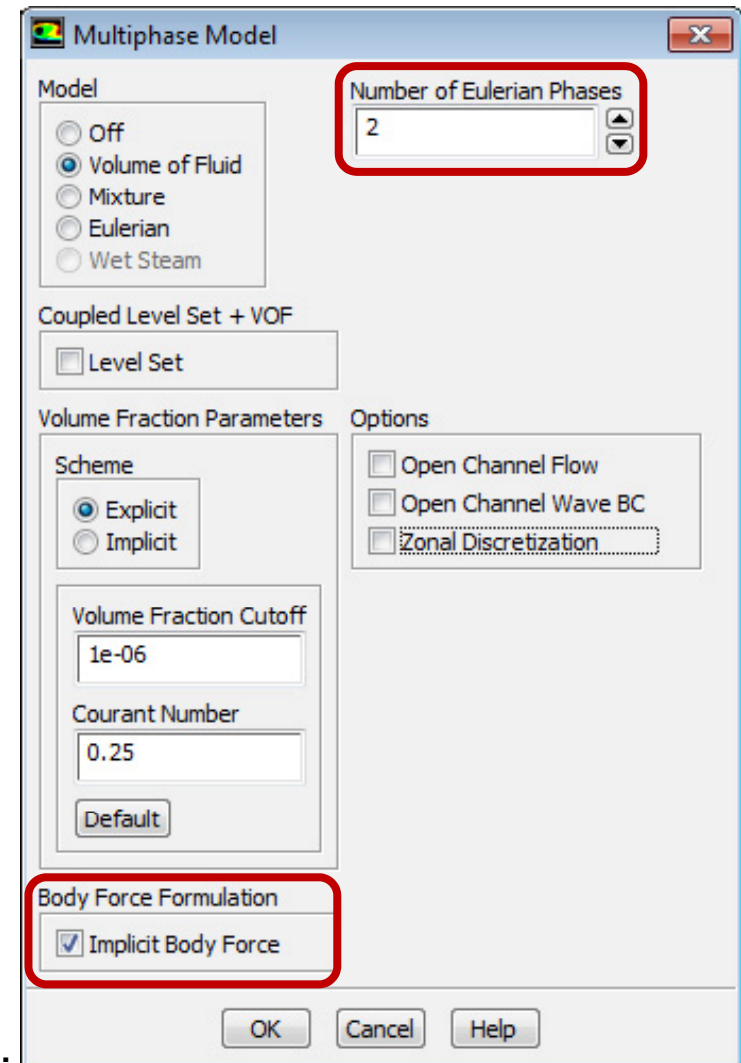
Enable Turbulence Model

- **Activate Models in the outline tree.**
 - Double-click Viscous-Laminar in the central pane under Models.
 - In the Viscous Model panel, select k-epsilon (2 eqn).
 - Under k-epsilon model, select Realizable.
 - Retain defaults for all other settings.
 - Click OK.

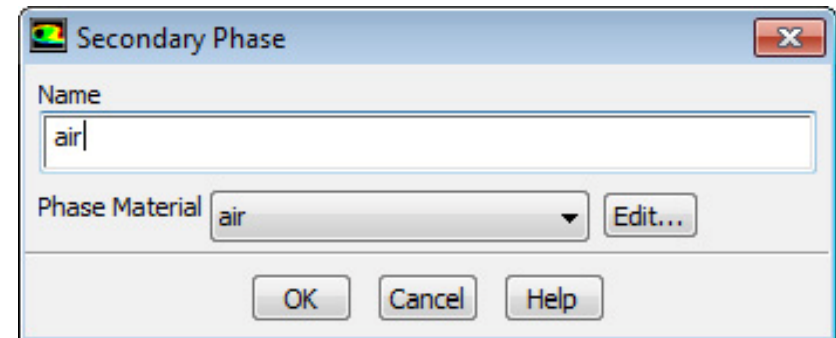
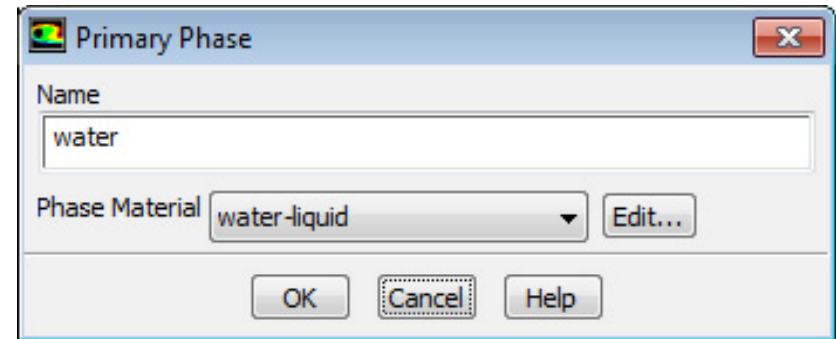


VOF Multiphase Model Setup

- **Enable the VOF multiphase model.**
 - Double-click on Multiphase.
 - In the panel that opens, enable “Volume of Fluid”.
 - Set “Number of Eulerian Phases” to 2.
 - Ensure that Scheme is set to Explicit.
 - Enable Implicit Body Force
 - Click OK.
- **Define the materials.**
 - Activate Materials in the outline tree.
 - Click Create/Edit...
 - In the Materials panel, click FLUENT Database...
 - Select water-liquid from the FLUENT Fluid Materials list, click Copy and then click Close.



- **Define the phases.**
 - Activate Phases in the outline tree.
 - Double-click **phase-1 - Primary Phase**.
 - In the Primary Phase panel, change the Name to **water**.
 - Ensure that **water-liquid** is selected under Phase Material.
 - Click OK.
- Double-click **phase-2 - Secondary Phase**.
- In the Secondary Phase panel, change the Name to **air**.
- Select **air** under Phase Material.
- Click OK.



Multiphase Model Setup

- **Define phase interactions**

- Click the Interaction button.
- In the Phase Interaction panel that opens, activate the Surface Tension tab.
- Select constant in the pull-down list and enter 0.072 N/m for the Surface Tension Coefficient. Click OK.

Problem Setup

General

Models

Materials

Phases

Cell Zone Conditions

Boundary Conditions

Mesh Interfaces

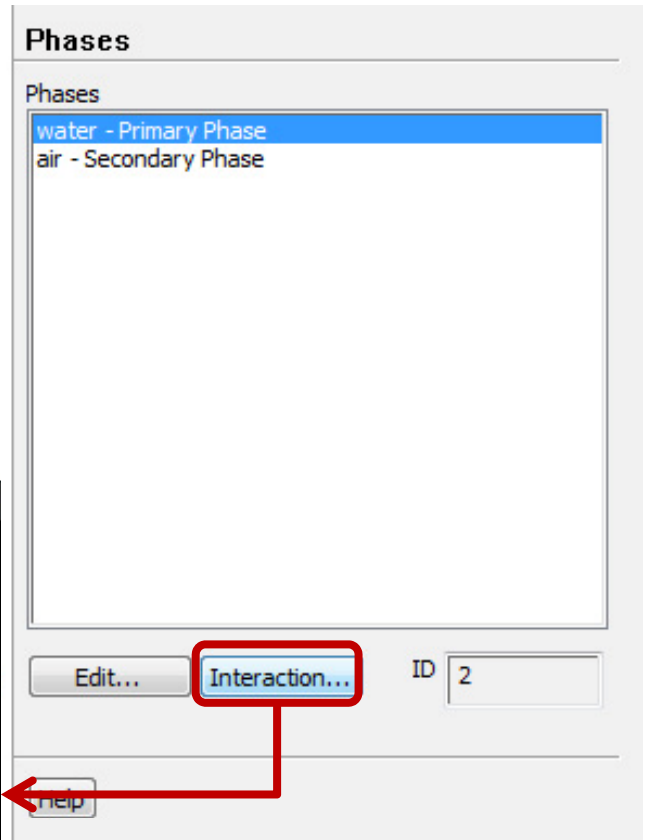
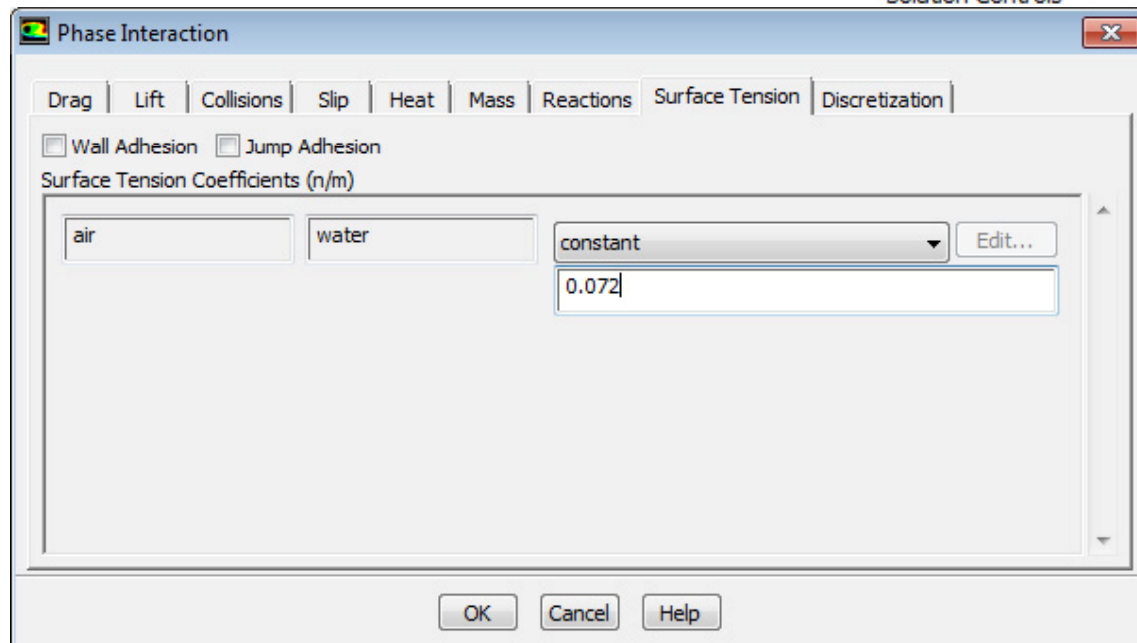
Dynamic Mesh

Reference Values

Solution

Solution Methods

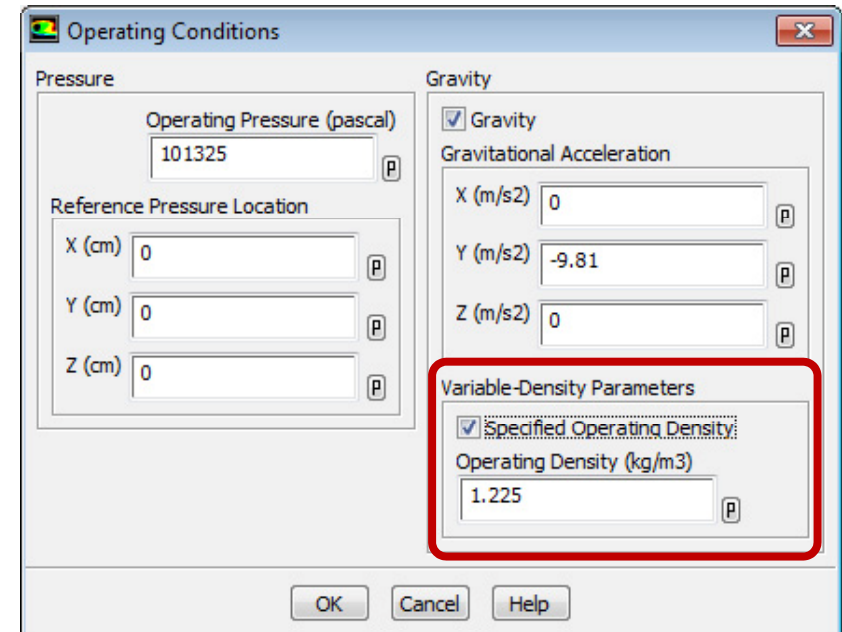
Solution Controls



Set Operating Conditions

- **Set the operating conditions.**
 - Activate Cell Zone Conditions in the outline tree.
 - Click Operating Conditions... in the centre pane.
 - Verify that Gravity is enabled and the Gravitational Acceleration is set correctly (-9.81 m/s^2 in the y direction).
 - Under Variable Density Parameters, activate Specified Operating Density.
 - Accept the default entry of 1.225 kg/m^3 for the Operating Density.

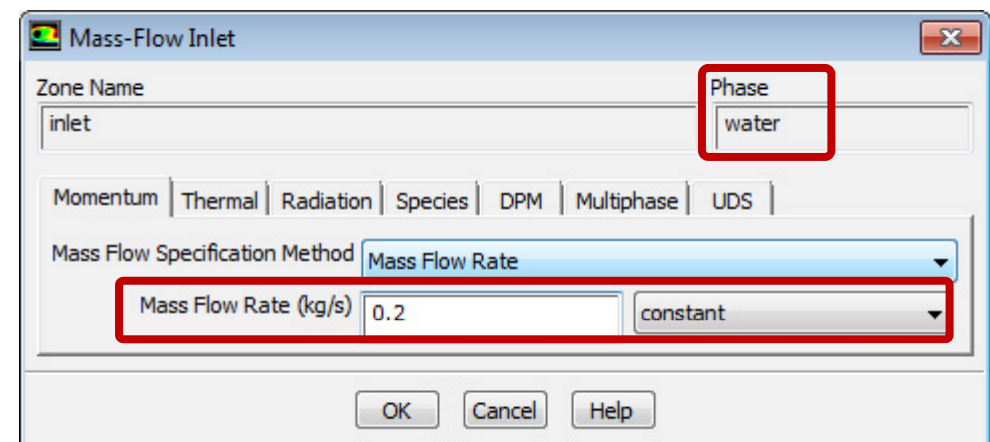
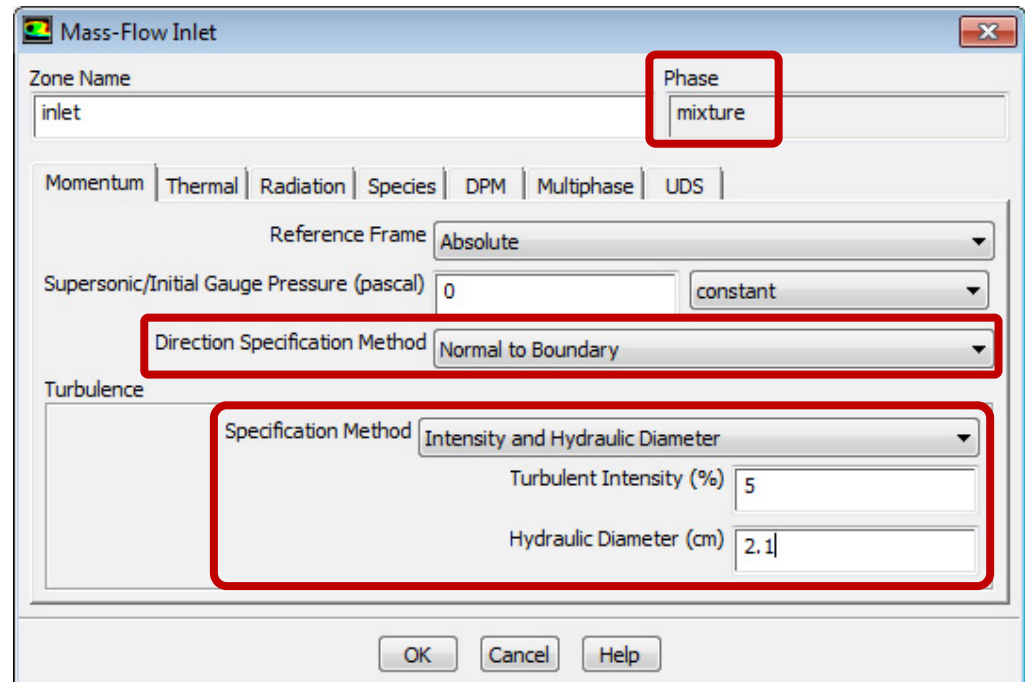
The operating density should be set to the density of the lightest fluid in the domain when using the VOF model; otherwise, an erroneous hydrostatic pressure distribution will occur.



Define Boundary Conditions [Inlet]

Problem Setup > Boundary Conditions

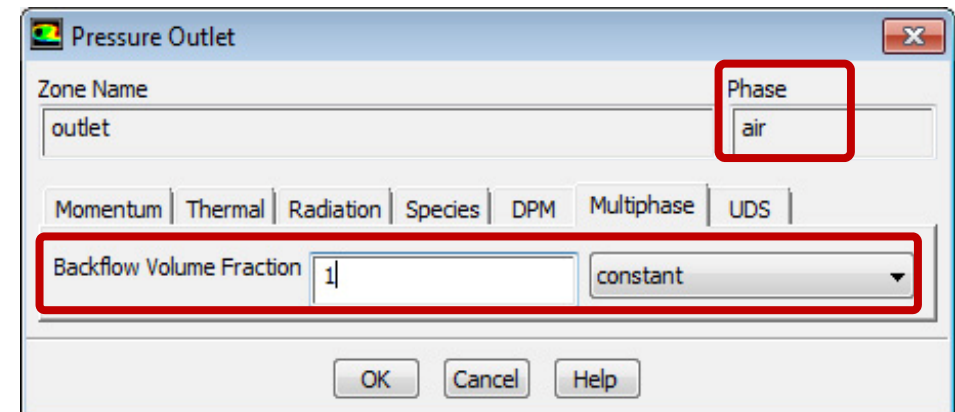
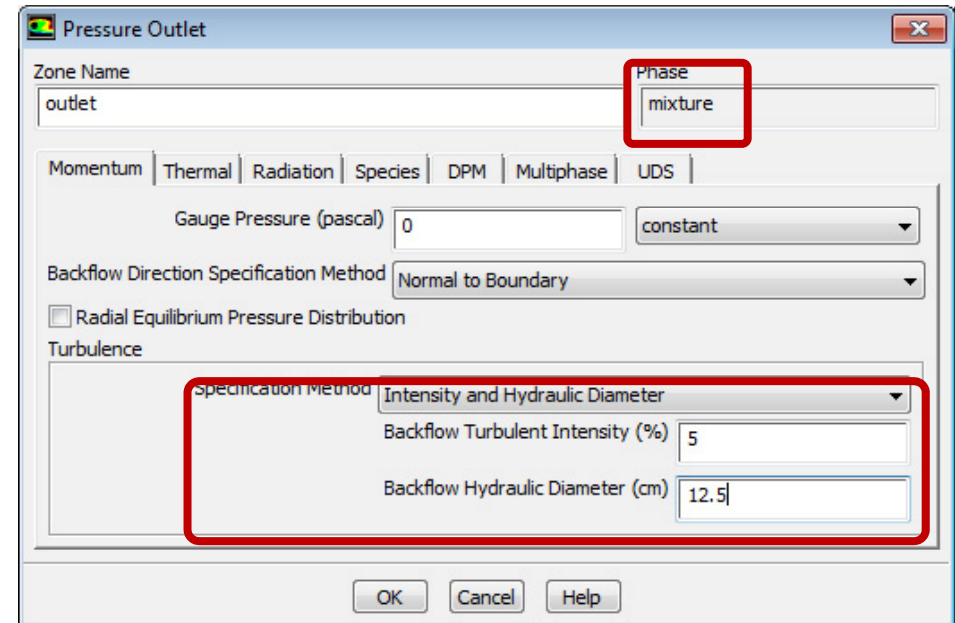
- Double click the **inlet** boundary.
- select **Normal to Boundary** for Direction Specification Method
- For the turbulent quantities, select **Intensity and Hydraulic Diameter**, with TI of 5% and HD of 2.1 cm.
- Click OK.
- In the centre pane, select **air** under Phase and double-click again on **inlet**.
- Set the Mass Flow Rate of air to 0.
- In the centre pane, select **water** under Phase and double click on **inlet**.
Set the mass flow rate to **0.2 kg/s**.
- Click OK.



Define Boundary Conditions [Lower 'Outlet']

Problem Setup > Boundary Conditions

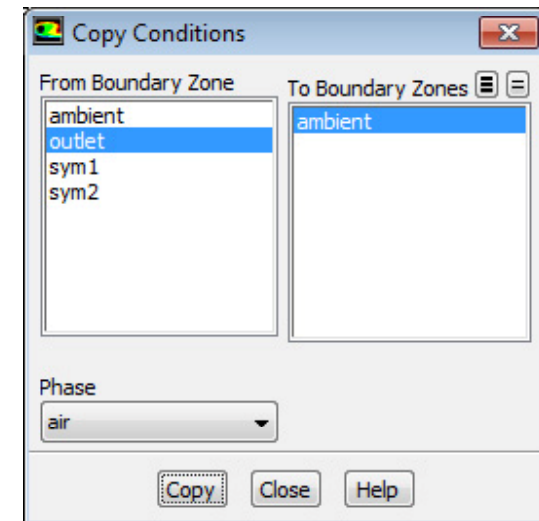
- Select **mixture** under Phase (in the centre pane).
- Select '**Outlet**' boundary
- For the turbulent quantities, select **Intensity and Hydraulic Diameter**, with TI of 5% and HD of 12.5 cm.
- Click OK.
- In the centre pane, select '**air**' under Phase and click 'Edit' again.
- Activate the Multiphase tab and enter **1** for **Backflow Volume Fraction**.
- Click OK.



Define Boundary Conditions [Top 'ambient']

- **Copy Boundary conditions from outlet to ambient.**
 - In the centre pane, click 'Copy'...
 - Under 'From Boundary Zone', select '**Outlet**'.
 - Under 'To Boundary Zone', select **Ambient**.
 - Select '**mixture**' under 'Phase' and click Copy.
 - Click OK when asked if you want to copy the boundary conditions.
 - This will copy settings specific to the mixture.
 - Select '**air**' under 'Phase' and again click Copy.
 - Click OK when asked if you want to copy the boundary conditions.
 - Close the Copy Conditions panel.

The Copy Conditions panel is a quick way of transferring common settings from one boundary to another. The 'To Boundary Zones' automatically displays boundaries of the same type as the 'From Boundary Zone' selection.



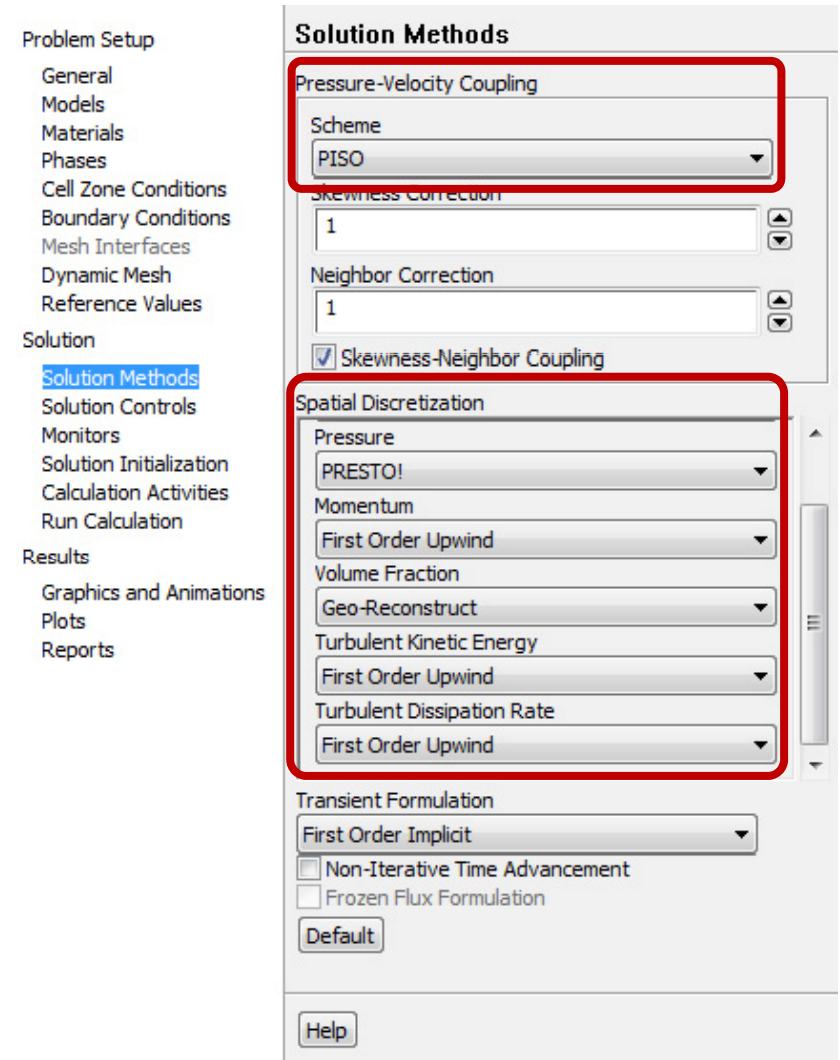
Define Solution Methods and Controls

- **Set Solution Methods:**

- Under Pressure-Velocity Coupling, set Scheme to PISO.
- Under Spatial Discretization, set the following discretization schemes:
 - Gradient – Least Squares Cell Based
 - Pressure – PRESTO!
 - Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate – First Order Upwind
 - Volume Fraction – Geo Reconstruct

- **Set Solution Controls.**

- Set the Under-Relaxation factor for momentum to 0.3.
- Set the under-relaxation factors for “Turbulent Kinetic Energy” and “Turbulent Dissipation Rate” to 0.5.



Initialize and Patch the Initial Solution

Initially, the tank is filled to a level of 6 cm with water. Here you will first initialize the flow solution, then create an adaption register and use the register to define the initial location of the liquid surface.

- **Initialize the flow field.**
 - Select Solution Initialization in the outline tree.
 - Select '**inlet**' from 'Compute from' dropdown list.
 - Set **air volume fraction** to 1.
 - Click Initialize.

This will instruct the solver to fill the tank with air. The next step is to partially fill the tank with water, resulting in the proper initial condition.

Problem Setup

General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values

Solution

Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation

Results

Graphics and Animations
Plots
Reports

Solution Initialization

Initialization Methods

☐ Hybrid Initialization
☒ Standard Initialization

Compute from
inlet

Reference Frame

☒ Relative to Cell Zone
☐ Absolute

Initial Values

X Velocity (m/s)
0

Y Velocity (m/s)
-0.8004598

Z Velocity (m/s)
0

Turbulent Kinetic Energy (m2/s2)
0.00240276

Turbulent Dissipation Rate (m2/s3)
0.01255714

air Volume Fraction
1

Initialize Reset Patch...

Reset DPM Sources Reset Statistics

Help

Define Adaption Register for Patching

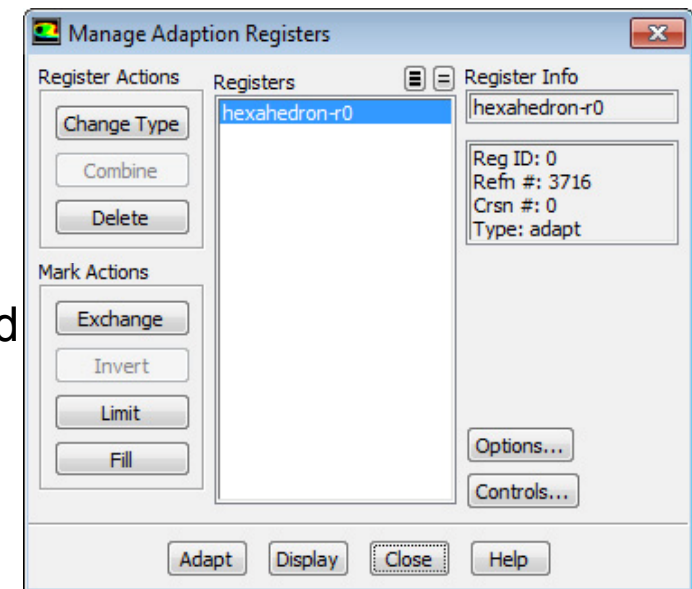
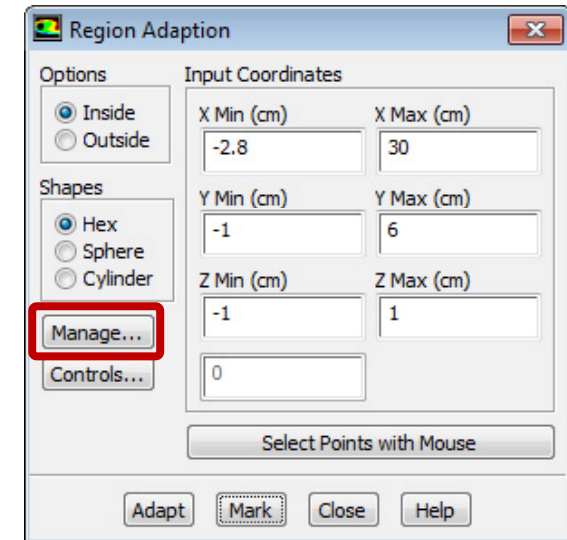
- Next, define the region of the domain to be filled with liquid.

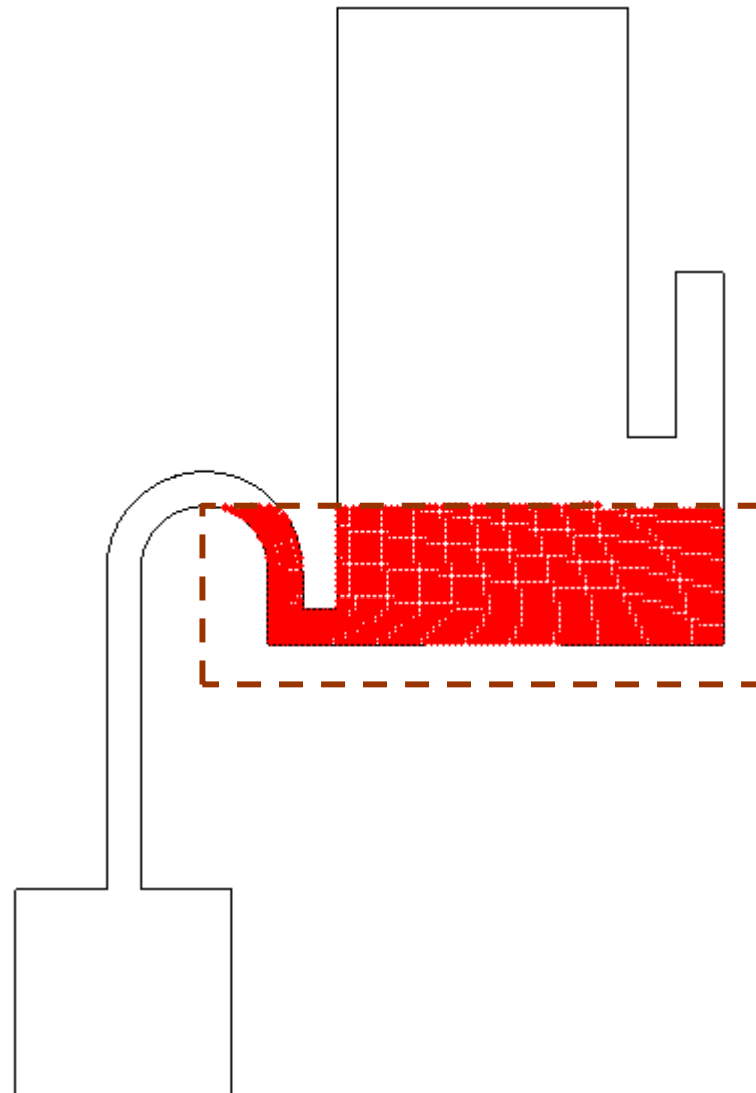
- In the top menu, select Adapt → Region.
- Enter the values shown in the panel to the right.
- Click **Mark**. **DO NOT CLICK ADAPT!**

A message appears in the FLUENT console informing you that 3716 cells have been marked.

- To view the marked cells, click Manage.
- Verify the register **hexahedron-r0** under Registers is selected and click Display
- You may need to zoom in (use the 'Fit to Window' icon) because the mesh was scaled since it was first displayed.
- Close the Manage Adaption Registers panel and the Region Adaption panel.

The marked cells will be displayed in the graphics window (see next page).

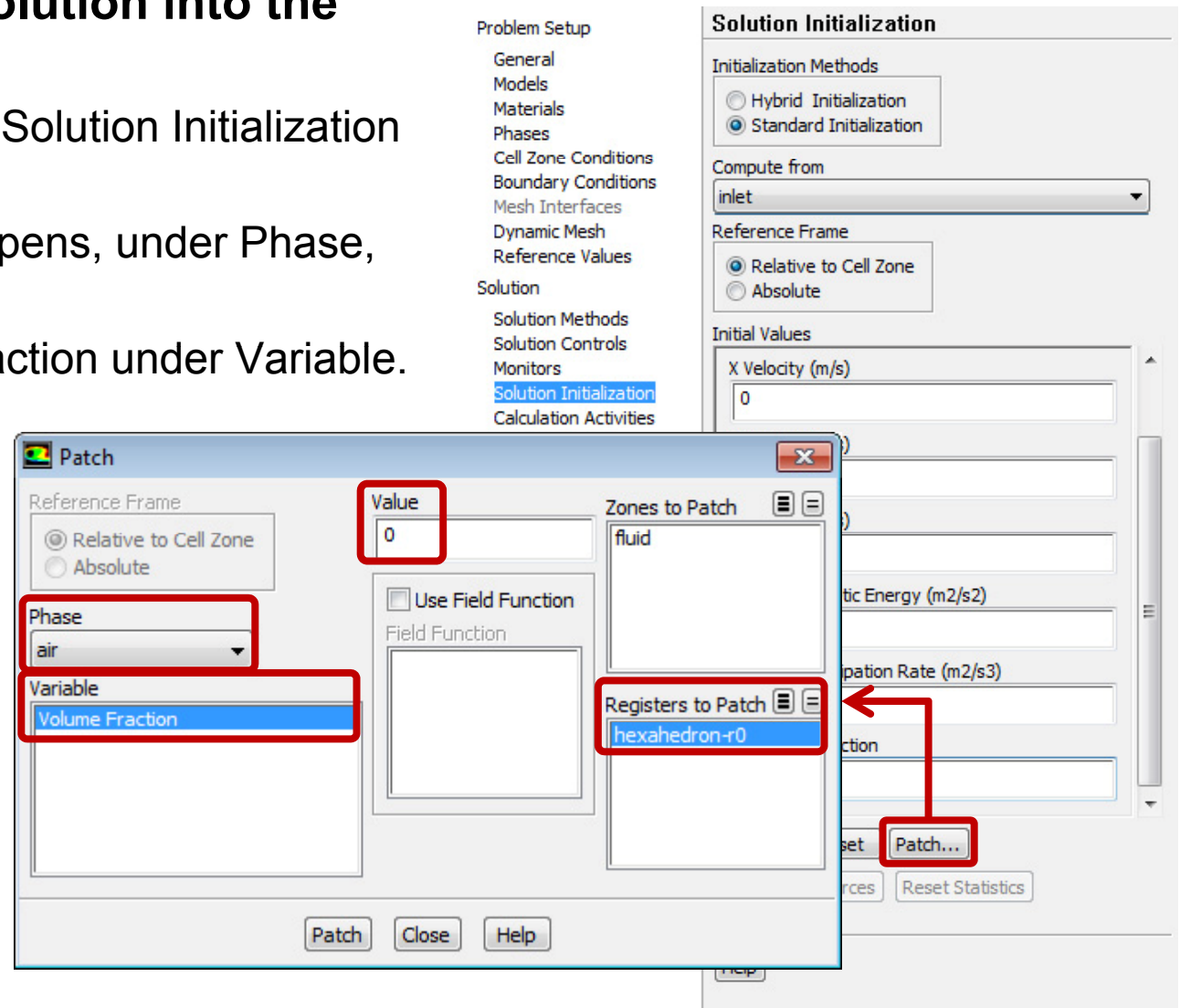




**Outline of region
adaption register**

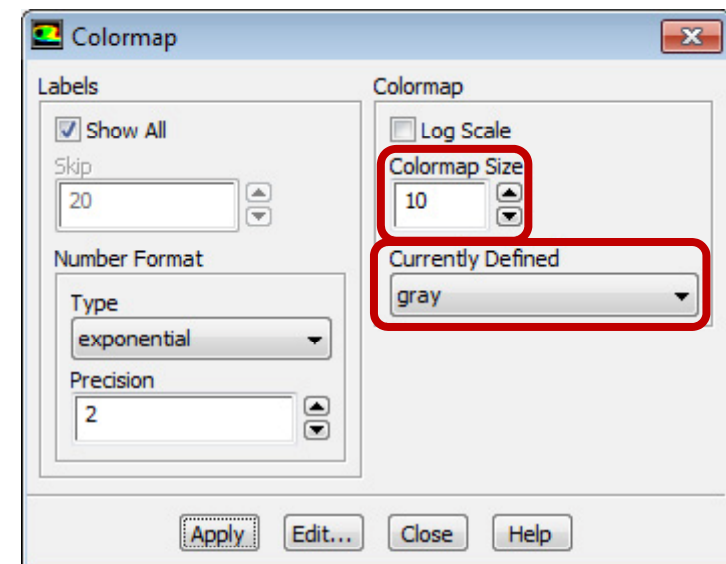
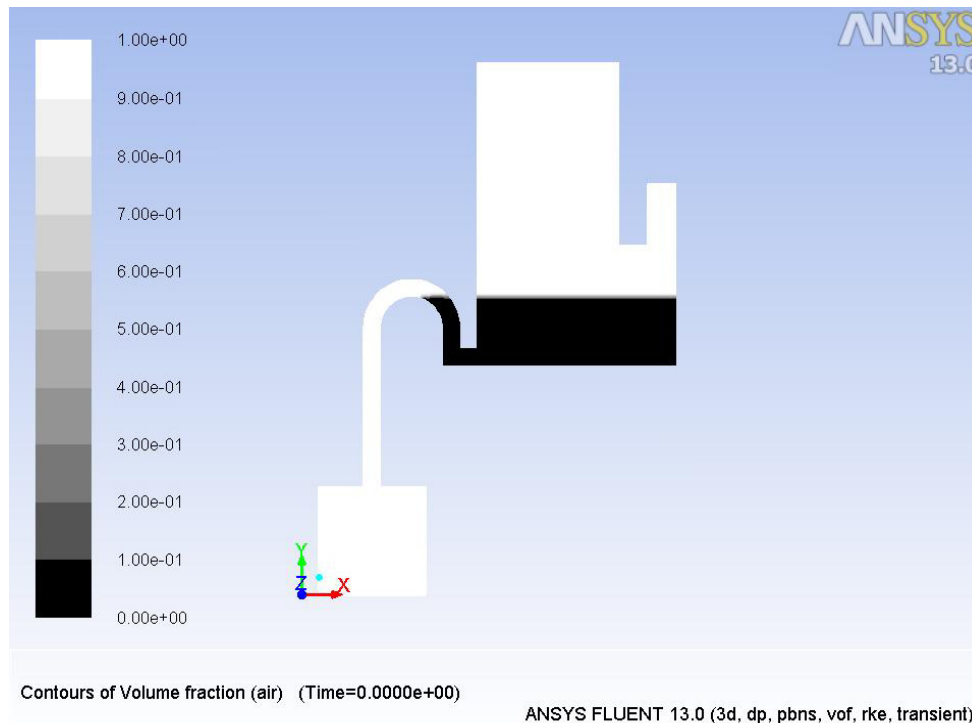
Initialize the Flow Solution

- **Patch the initial solution into the adaption register.**
 - Click Patch under Solution Initialization in the outline tree
 - In the panel that opens, under Phase, select air.
 - Select Volume Fraction under Variable.
 - Set Value to 0.
 - Under Registers to Patch, select the adaption register you created.
 - Click Patch.
 - Close the Patch panel.



Initialize Display Settings

- Create a contour plot of **air** volume fraction on plane **sym1** after patching. The variable to plot is under Phases...Volume Fraction...air
 - Note that the color scheme can optionally be changed from the default BGR to the 'gray' scheme. This can be done using the GUI menu: Display → Colormap. Also, for clarity, the Colormap Size has been reduced to 10 which will use 10 colors instead of the default 20.



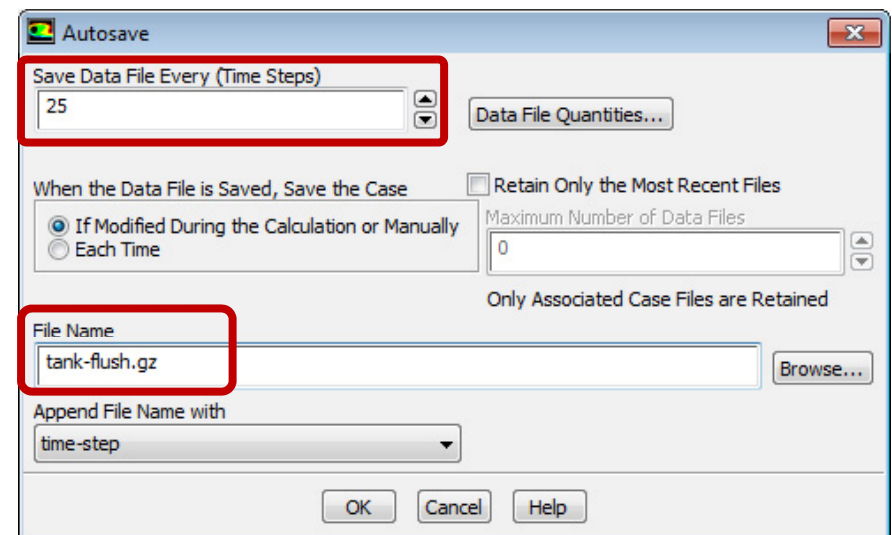
Define Calculation Activities

In this step you will define activities that FLUENT will perform during the calculation. These activities are as follows:

- To autosave case and data files.
- To turn off the supply of water after $t = 1$ second. (Mass flow rate boundary condition will be changed to zero).

- **Set autosave options.**

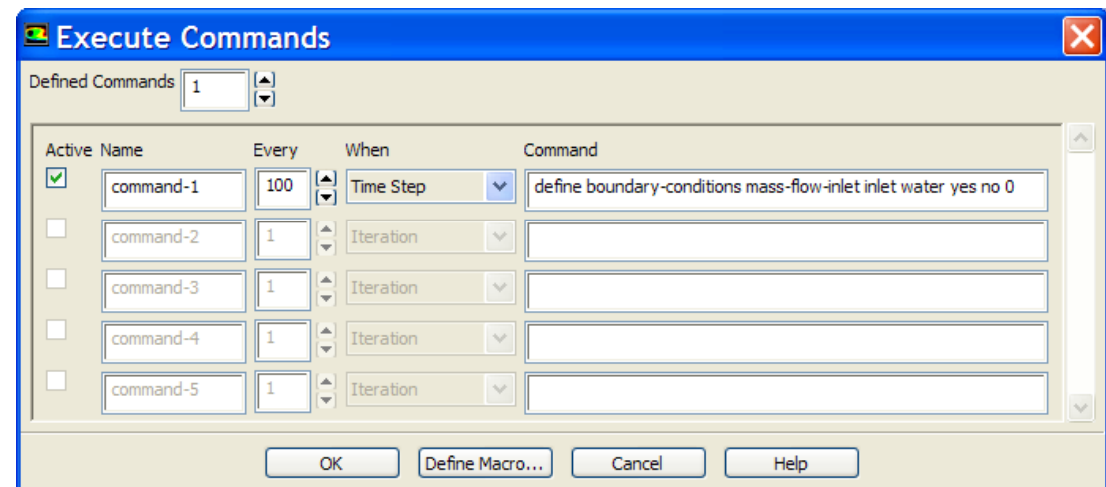
- Select Calculation Activities in the outline tree.
- Click Edit... next to Autosave.
- Set Autosave Every (Time Steps) to 25.



- [If running FLUENT standalone, rather than under workbench] In the panel that opens, enter the file name tank-flush.gz.
- [If running under workbench] No action needed.
- Retain the defaults for all other settings and click OK.

Define Calculation Activities

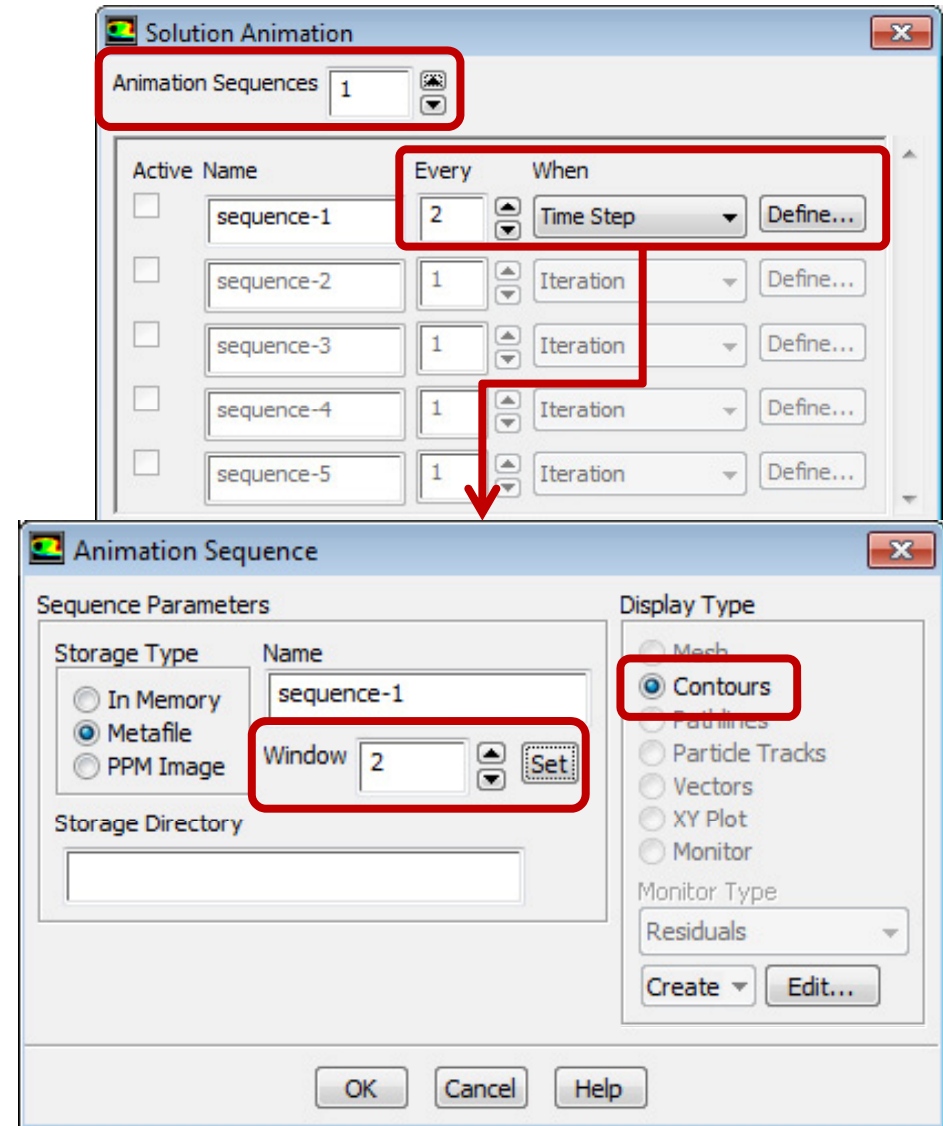
- **Define a command to modify the boundary condition after 1 second:**
 - In the centre pane, under Execute Commands, click Create/Edit.
 - In the panel that opens, set Defined Commands to 1.
 - Check Active next to the command line.
 - Enter the following command to be executed. Please make sure the spelling is **exactly as written as below**, take special care with the hyphens '-':
`define boundary-conditions mass-flow-inlet inlet water yes no 0`
 - Set 'Every' to 100.
 - Set 'When' to 'Time Step'.
 - Click OK.



Define Animation Solution

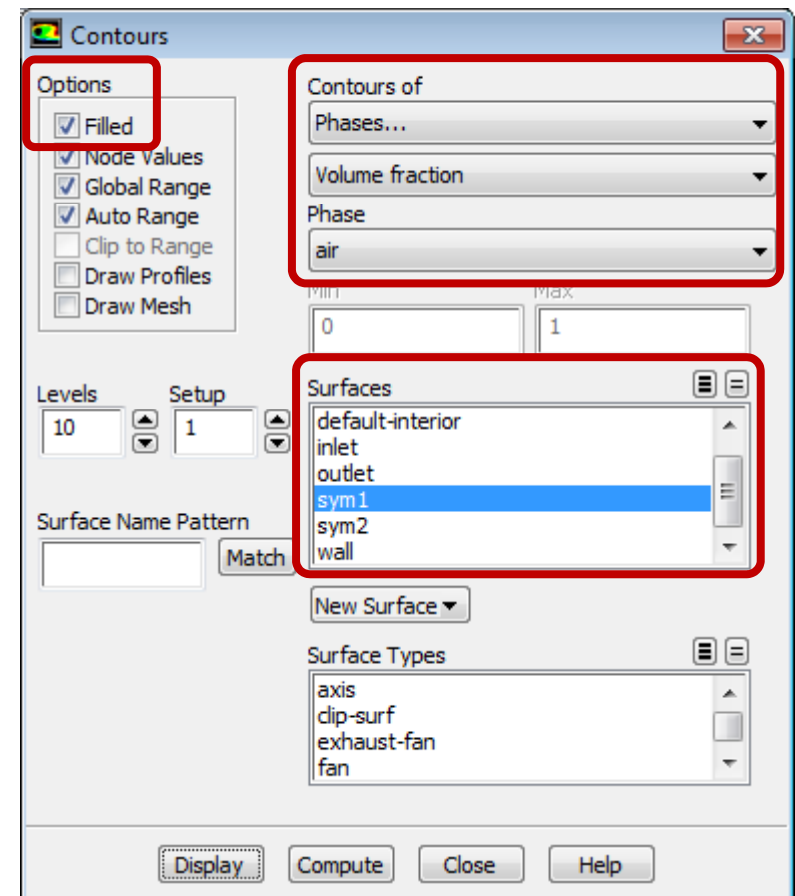
- **Set the animation sequence**

- Calculation Activities > Solution Animations > Create/Edit.
- In the panel that opens, set 'Animation Sequences' to 1, set 'Every' to 2, set 'When' to 'Time Step' and click 'Define'.
- In the 'Animation Sequence' panel that opens, set '**Window** number to 2 and click 'Set'. Under Display Type select 'Contours' to open the contours panel.



Define Animation Solution

- **Set the animation sequence cont ...**
 - In the Contours panel select 'Filled' under 'Options'.
 - Under 'Contours of' select 'Phases...' and choose 'air' for the 'Phase' to be displayed.
 - Under 'Surfaces' select 'sym1' zone.
 - Click 'Display' and close the panel.
 - Close remaining panels by clicking 'OK'.
- Enable display of both graphic windows (so both residuals and the contour plot are visible)



Run the Calculation

- Before running the calculation, you should save the case and data files. Use the Save toolbar button to write case and data files as [tank-flush-init.cas.gz](#).
 - If running FLUENT from within ANSYS Workbench, Select Save Project.
- Select Run Calculation from the outline tree.
 - Enter **0.01 s** for Time Step Size
 - Enter **350** under Number of Time Steps.
 - Click Calculate.

The solution will require approximately half an hour to compute. You can choose to run all of the calculations or stop the iterations and open a later data file to view the results.

Problem Setup

General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values

Solution

Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation

Results

Graphics and Animations
Plots
Reports

Run Calculation

Check Case... Preview Mesh Motion...

Time Stepping Method: Fixed

Time Step Size (s): 0.01

Number of Time Steps: 350

Options

☐ Extrapolate Variables

☐ Data Sampling for Time Statistics

Sampling Interval: 1

Max Iterations/Time Step: 20

Reporting Interval: 1

Profile Update Interval: 1

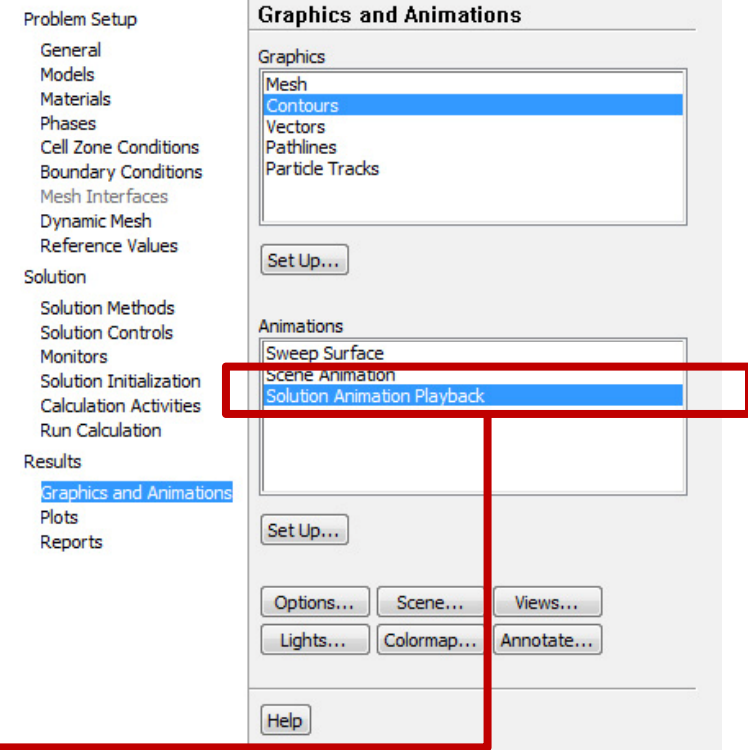
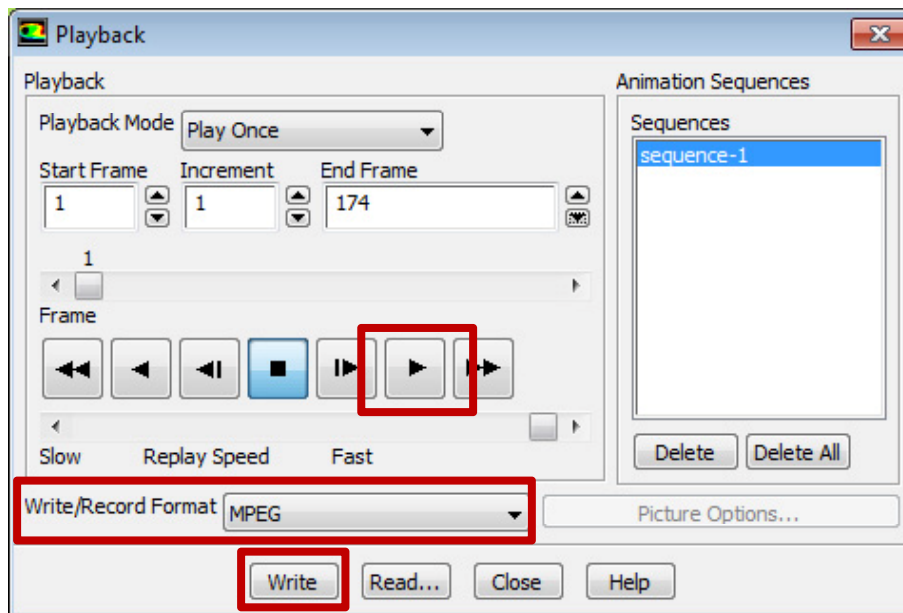
Data File Quantities... Acoustic Signals...

Calculate

Help

- **Generate Animation**

- Select 'Solution Animation Playback' from the 'Graphics and Animations' menu.
- Use the play button to view the animation on-screen
- Select 'MPEG' format and click to 'Write' to save the animation in your working directory.



Post-Process Results

- The animation can be played using most of the standard Multimedia Players like Windows Media Player.
- The 'Animation Playback' tool can also be used to generate a sequence of picture frames.

