Free Surface Wave Simulation Using the VOF Model

PURPOSE

The purpose of this tutorial is to provide guidelines for the basic setup and solution procedure for 3D, turbulent, free surface flow around a surface-piercing foil geometry.

In this tutorial you will learn how to:

- Model water-air interaction using the VOF model.
- Set up, run, and postprocess using FLUENT.

PREREQUISITES

This tutorial assumes that you are familiar with the user interface, basic setup and solution procedures in FLUENT. This tutorial does not cover the mechanics of using the VOF model, but focuses on setting up the problem for 3D, turbulent, free surface flow around a surface-piercing foil geometry and solving it.

If you have not used FLUENT before, it would be helpful to first review FLUENT 6.3 User's Guide and FLUENT 6.3 Tutorial Guide.

PROBLEM DESCRIPTION

The model is NACA 0024 vertical surface-piercing hydrofoil, with a chord length of 1.2m. The geometry is derived from the NACA four digit wing section series and features a symmetric profile, and a thickness of 24% of the chord length. When foil moves through calm water or when water flows over the stationary foil, it is important to know the free surface pattern and wave induced near the free surface with the chord length (1.2 m).

The foil was towed at 1.27 m/s in the Iowa Institute of Hydraulic Research towing tank. The computational conditions are set to the experimental conditions available at:

http://www.iihr.uiowa.edu/~shiphydro/efd_vdata_naca24.htm

The data includes far and near field wave elevations, surface pressures, and wave profiles on the hull at three Froude numbers (Fr = 0.19, 0.37, 0.55). In this tutorial, Fr=0.37 case is selected and simulated.

Step 1: Grid

1. Start 3D version of FLUENT.
2. Read the mesh file, "sp_naca0024_hf.msh".

   File > Read > Case…

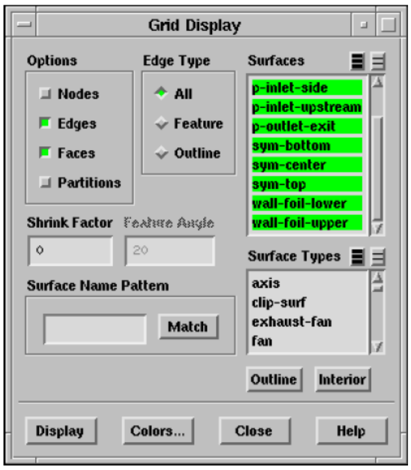
3. Check the grid.

   Grid > Check

FLUENT will perform various checks on the mesh and will report the progress in the console. Make sure the reported minimum volume is a positive number.

4. Display the grid outline.

   Display > Grid…



(a) Deselect all the surfaces and click "Outline".

(b) Under "Options", select "Faces".

(c) Click "Display" to visualize the grid, (Figure 1).

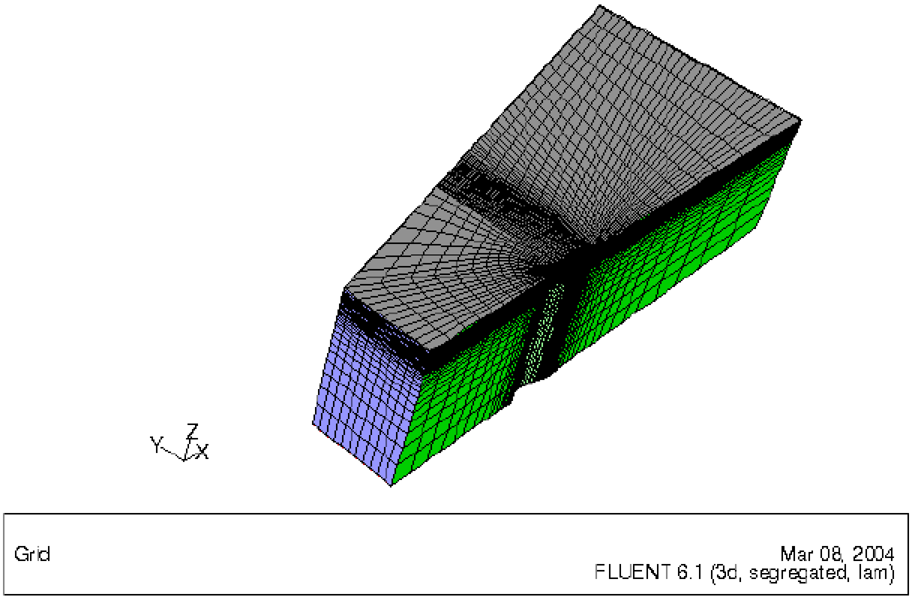


Figure 1: Grid Display

Step 2: Models

1. Keep the default solver settings.

   Define > Models > Solver…

2. Enable multiphase model.

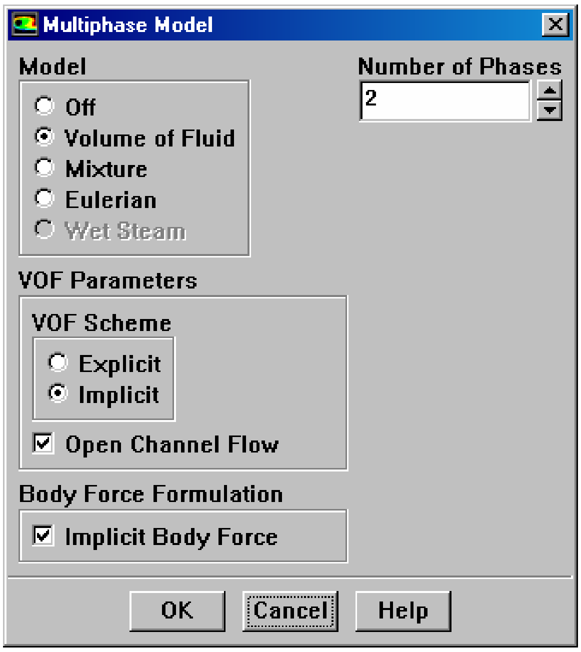
   Define > Models > Multiphase…

   (a) Under "Model", select "Volume of Fluid". The panel will expand showing the VOF Parameters.

   (b) Under "VOF Scheme", turn on "Implicit".

   (c) Under "Body Force Formulation", turn on "Implicit Body Force". Note: The "Implicit Body Force" option is turned on because gravity is taken into account.

   (d) Check the option "Open Channel Flow".



3. Enable standard k-epsilon turbulence model.

   Define > Models > Viscous…

   (a) Under Model, turn on "k-epsilon (2-eqn) ".

   (b) Keep the other default values.

Step 3: Materials

Define > Materials…

1. Click "Fluent Database..." in the Materials panel to open Database Materials panel.

   (a) Under Fluid Materials, select "water-liquid (h2o<l>)".

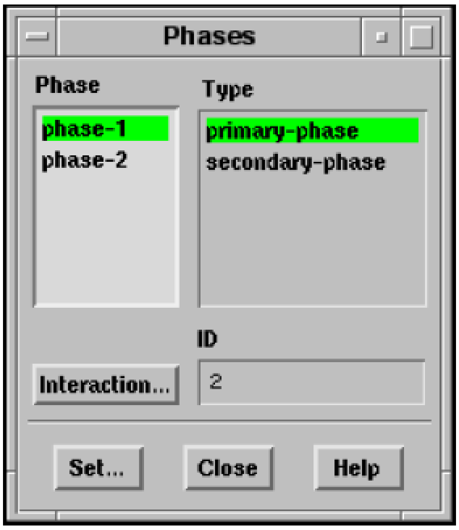
   (b) Click Copy and close the Database Materials panel.

2. Click Change/Create and close the Materials panel.

Step 4: Phases

1. Define the Primary Phase

   Define > Phases…



   (a) Select "phase-1" and click "Set…".

      i. Select "air" in the "Phase Material" drop-down list.

      ii. Enter the "Name" as "air".

2. Define Secondary phase.

   (a) Define "phase-2" by selecting "Phase Material" as "water-liquid" and rename it "water".

Step 5: Operating Conditions

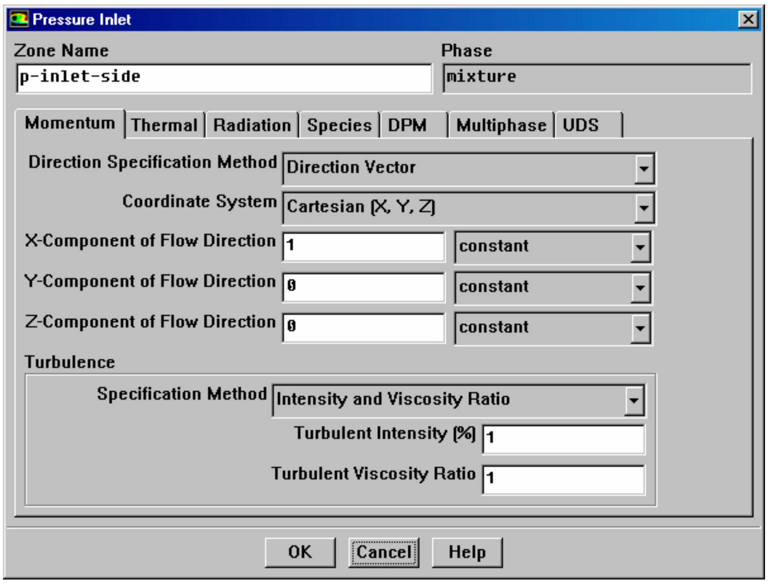
Define > Operating Conditions…

1. Enter "Reference Pressure Location" coordinates as X = -3.6, Y = 0, Z = 0.1. It is a usual practice in multiphase flow problems to set the Reference Pressure Location at a point, where only one phase exists.
2. Turn on "Gravity" and specify a value of -9.81 for "Gravitational Acceleration" in "Z direction".
3. Turn on "Specified Operating Density".

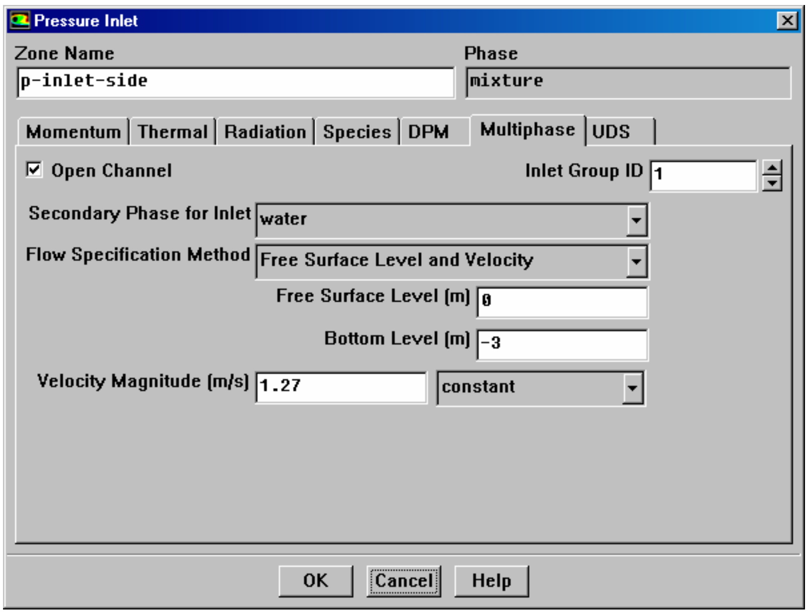
Step 6: Boundary Conditions

Define > Boundary Conditions…

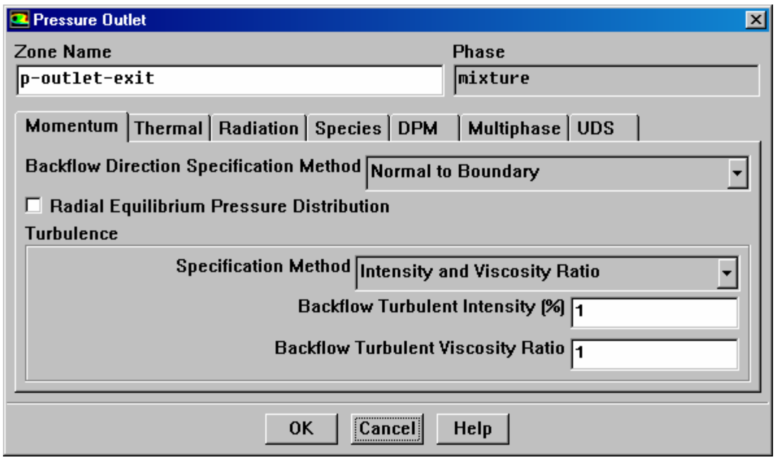
1. Set the boundary conditions for "p-inlet-side".
   (a) Select "mixture" as "Phase" in the "Boundary Conditions" panel and click "Set…".
      i. Select "Direction Vector" as "Direction Specification Method".
      ii. Select "Intensity" and "Viscosity Ratio" as "Turbulence Specification Method".
      iii. Specify a value of 1 for both "Turbulence Intensity" and "Turbulent Viscosity Ratio".



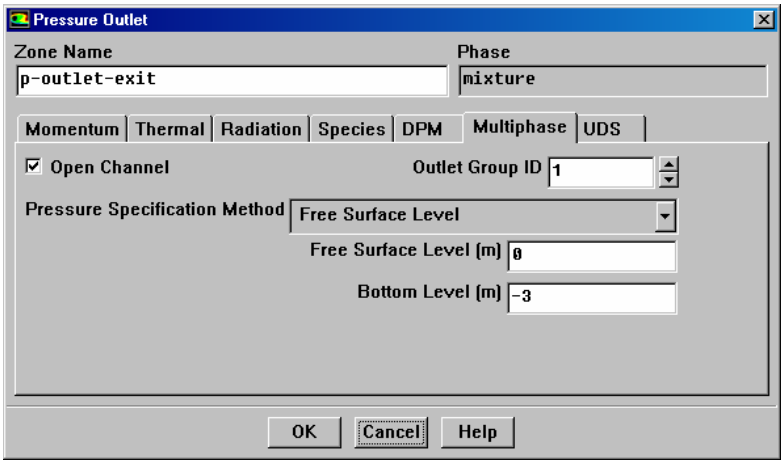
   (b) Go to "Multiphase" sub-panel and:
      i. Check "Open Channel". Then the panel will automatically expand.
      ii. Set -3 for "Bottom Level [m]" and 1.27 for "Velocity Magnitude [m/s]".



2. Specify the same boundary conditions for "p-inlet-upstream".
3. Set the boundary conditions for outlet ("p-outlet-exit").
   (a) Select "mixture" as "Phase" in the "Boundary Conditions" panel and click "Set…".
      i. Select "Normal to Boundary" as "Backflow Direction Specification Method".
      ii. Select "Intensity and Viscosity Ratio" as "Turbulence Specification Method".
      iii. Specify a value of 1 for both "Turbulence Intensity" and "Turbulent Viscosity Ratio".



   (b) Go to "Multiphase" sub-panel and:
      i. Check "Open Channel". Then the panel will automatically expand.
      ii. Set -3 for "Bottom Level [m]".



4. Keep the default boundary conditions for all other boundaries.

Step 7: Solution

1. Set the solution controls.

   Solve > Controls > Solution…

   (a) Set the "Under-Relaxation Factors" as shown in the following table:

Parameter	Value
Pressure	0.5
Momentum	0.1
Turbulence Kinetic Energy	0.5
Turbulence Dissipation Rate	0.5

(b) Under "Discretization", select "PRESTO!" in the Pressure and "SIMPLE" in the "Pressure-Velocity Coupling" drop-down lists respectively.

(c) Select "First Order scheme" for "Momentum", "Turbulence Kinetic Energy", and "Turbulence Dissipation Rate".

(d) Select "QUICK" scheme for "Volume Fraction".

(e) Click OK to close the panel.

2. Initialize the solution.

Solve > Initialize > Initialize…

(a) Click "Init".

(b) Select any pressure inlet boundary from "Compute From" drop-down list.

(c) Click "Init" again, and click "OK" for confirmation.

3. Define an adaption region for water.

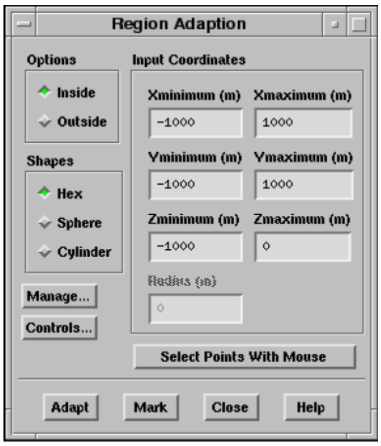
Define > Adaption > Region…

(a) Specify "Input Coordinates" as follows:

i. Specify a value of -1000 and 1000 for "Xminimum (m) " and "Xmaximum (m) ", respectively. This encompasses the entire X direction extent of the domain.

ii. Specify a value of -1000 and 1000 for "Yminimum (m) " and "Ymaximum (m) ", respectively. This encompasses the entire Y direction extent of the domain.

iii. Specify a value of -1000 and 0 for "Zminimum (m)" and "Zmaximum (m), respectively. This encompasses the water region below the calm water surface.



4. Patch the water region and the velocity.

   Solve > Initialize > Patch...

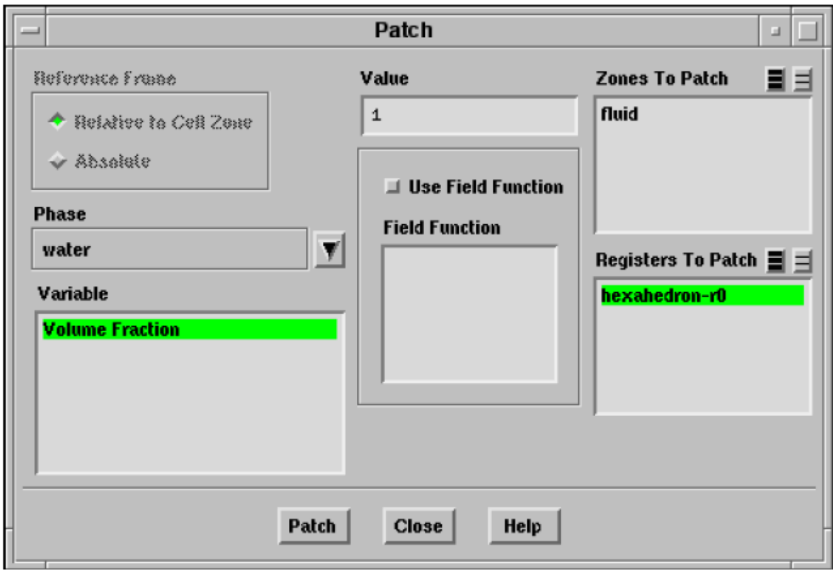
   (a) Patch the water region.

      i. Under "Phase", select "water".

      ii. Under "Registers To Patch", select "hexahedron-r0".

      iii. Under "Variable", select "Volume Fraction".

      iv. Enter Value as 1 and click "Patch".



   (b) Patch the velocity.

      i. Under "Phase", select "mixture".

      ii. Deselect "hexahedron-r0" under "Registers To Patch".

      iii. Under "Zones To Patch", select "fluid".

      iv. Select Variable as "X Velocity".

      v. Enter Value as 1.27, and click "Patch".

5. Enable the plotting of residuals.

   Solve > Monitors > Residuals...

   (a) Under Options, activate "Plot".

   (b) Set the convergence criteria for continuity to 1e-07.

6. Setup force monitors on foil walls.

   Solve > Monitors > Force...

   (a) Select "Drag" in the "Coeficient" drop-down list.

   (b) Under "Options", activate "Print" and "Plot".

   (c) Select all Wall Zones and click "Apply".

7. Request 50 iterations.

   Solve > Iterate...

Repeat 4(b) and request 50 more iterations. Save the case & data files as "stage1_first-order-schemes.cas".

8. Increase the accuracy of the numerical schemes from first-order to higher-order.

Solve > Controls > Solution…

(a) Select "Second Order Upwind" scheme for "Momentum".

(b) Select "Power Law " scheme for "Turbulence Kinetic Energy", and "Turbulence Dissipation Rate".

(c) Lower the Under-Relaxation Factor for "Turbulent Viscosity" from "1.0" to "0.7".

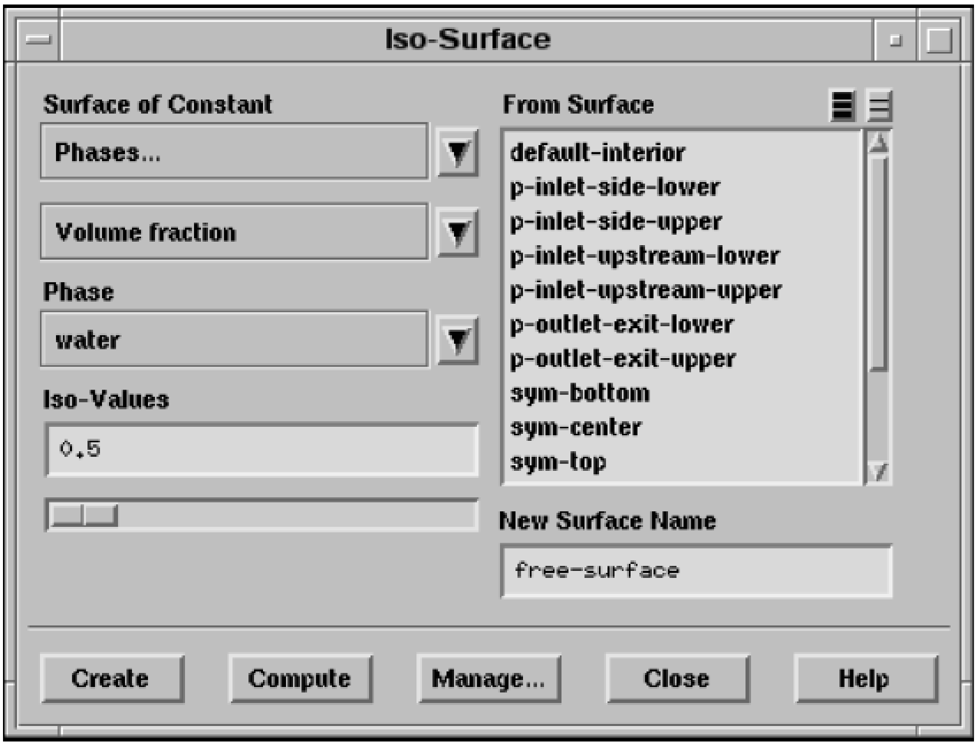
(d) Continue the iteration until the drag converges.

(e) Save the case and the data as "stage2_higher-order-schemes.cas".

Step 8: Postprocessing

1. Create the free surface, i.e. an iso-surface with water volume fraction of 0.5.

Surface > Iso-surface…



(a) Select "Phases..." and "Volume fraction" in the "Surface of Constant" drop-down lists.

(b) Under "Phase", select "water".

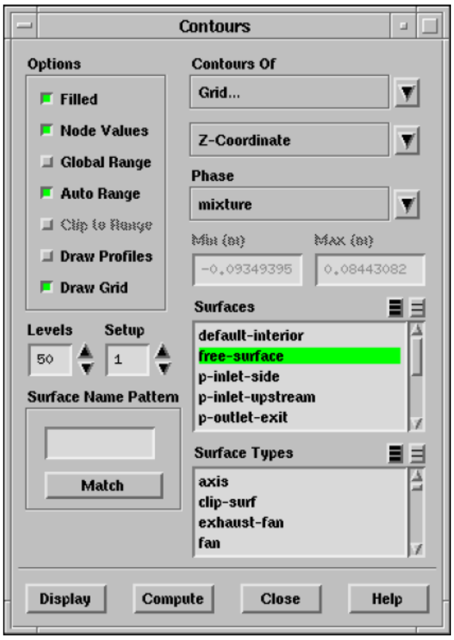
(c) Specify a value of 0.5 for "Iso-Values".

(d) Enter "free-surface" as "New Surface Name".

(e) Click "Create" and close the panel.

2. Display the free-surface elevation contours (Fig. 2).

Display > Contous…



(a) Select "Grid... " and "Z-Coordinate" in the "Contours Of" drop-down lists.

(b) Under "Options", turn on "Filled" and turn on "Global Range".

(c) Under "Surfaces", select "free-surface".

(d) Under "Options", activate "Draw Grid". Grid Display panel will come up.

(e) Close the "Grid Display" panel.

(f) Click "Display" and close the panel.

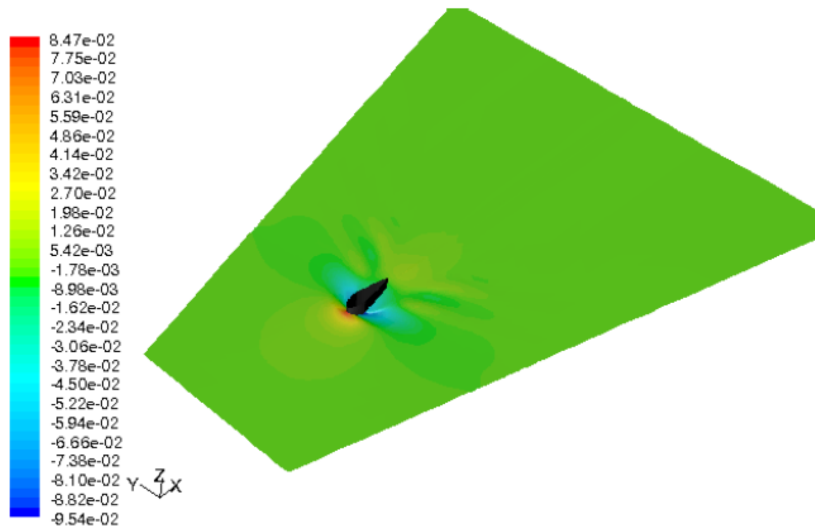
(g) Turn on symmetry.

Display > Views…

i. Under "Mirror Planes", select "sym-center".

ii. Click "Apply" and close the panel.

Rotate the contour display using mouse for better view.



Contours of Z-Coordinate (mixture) (m)
Mar 08, 2004
FLUENT 6.2 (3d, segregated, vof, ske)