
Tutorial 9. Modeling Turbulent Flow in a Mixing Tank

Introduction

The purpose of this tutorial is to illustrate the setup and solution of a 3D turbulent fluid flow for periodic section of a mixing tank.

Mixing is a very crucial unit operation in process industry. The efficiency of mixing depends on the type of agitator that will provide the required level of mixing in as short time as possible. Mixing time is usually the critical parameter in determining the efficiency of an agitated system. A CFD analysis yields values for species concentration, fluid velocity and temperature throughout the solution domain. This allows engineers to evaluate alternative designs and choose the optimum configuration.

This tutorial demonstrates how to do the following:

- Read an existing mesh file in FLUENT.
- Check the grid for dimensions and quality.
- Change the material properties and units.
- Set up boundary conditions for a moving fluid and wall zone.
- Set up boundary conditions for a periodic zone.
- Specify the solver settings and perform iterations.
- Create iso-surfaces and judge convergence by monitoring integrated quantities.
- Display the results over entire domain.

Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

Problem Description

Consider a cylindrical vessel of diameter (T) 1 m, filled with water up to $H = T$. The fluid is stirred by a standard six-blade Rushton turbine (Figure 9.1) rotating at a speed of 50 rpm. The turbine diameter (D) = $T/3$, blade height = $D/5$, and blade width = $D/4$.

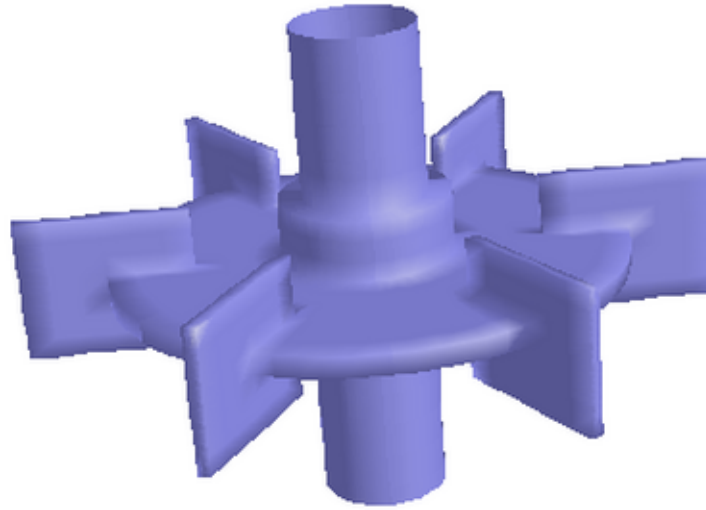


Figure 9.1: Schematic of Impeller

Preparation

1. Copy the mesh file, `tank.msh` to your working folder.
2. Start the 3D (3d) solver of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the grid file, `tank.msh`.

→ → Case...

FLUENT will read the mesh file and report the progress in the console.

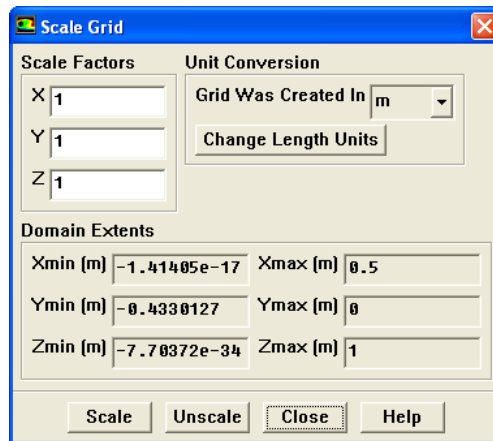
2. Check the grid.

→ Check

This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

→ Scale...

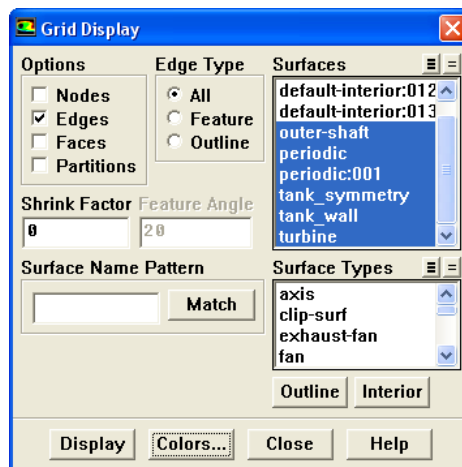


Check the domain extents to see if they correspond to the actual physical dimensions. If not, the grid has to be scaled with proper units.

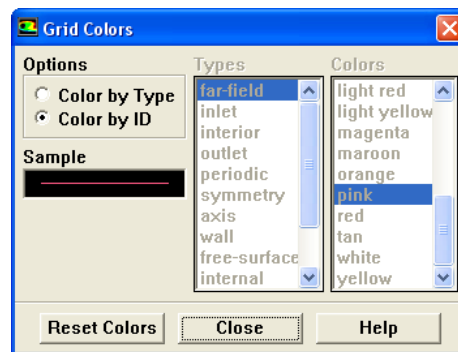
(a) Close the Scale Grid panel.

4. Display the grid.

Display → Grid...



(a) Click the Colors... button to open the Grid Colors panel.



- i. Select Color by ID in the Options group box.
 - ii. Close the Grid Colors panel.
- (b) Click Display in the Grid Display panel (Figure 9.2).

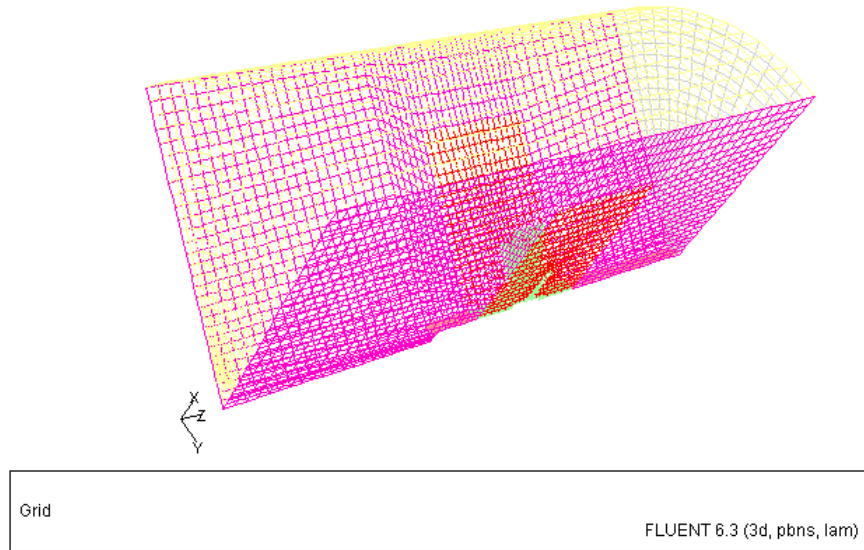
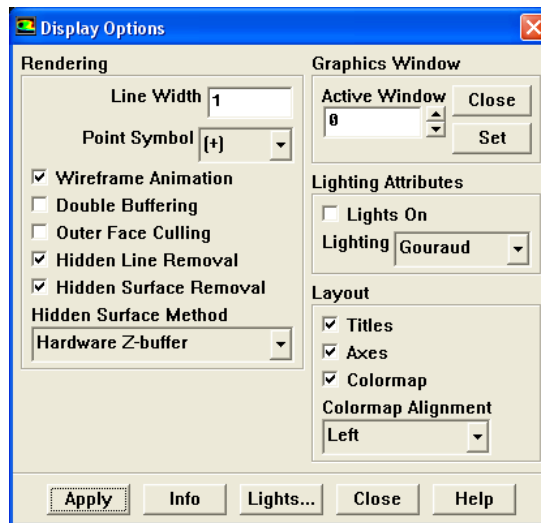


Figure 9.2: Grid Display

As the grid lines for all the zones are visible, the display looks cluttered. For clear visibility, enable the Hidden Line Removal option.

5. Display the grid without hidden lines.

Display → Options...



- (a) Enable Hidden Line Removal in the Rendering group box.
- (b) Click Apply and close the Display Options panel.
- (c) Deselect periodic:001 from the Surfaces selection list and click Display, in the Grid Display panel.

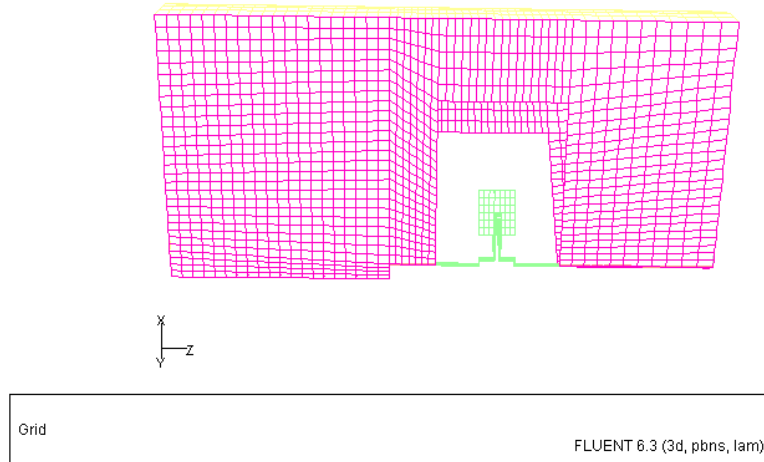


Figure 9.3: Grid Display Without Hidden Lines

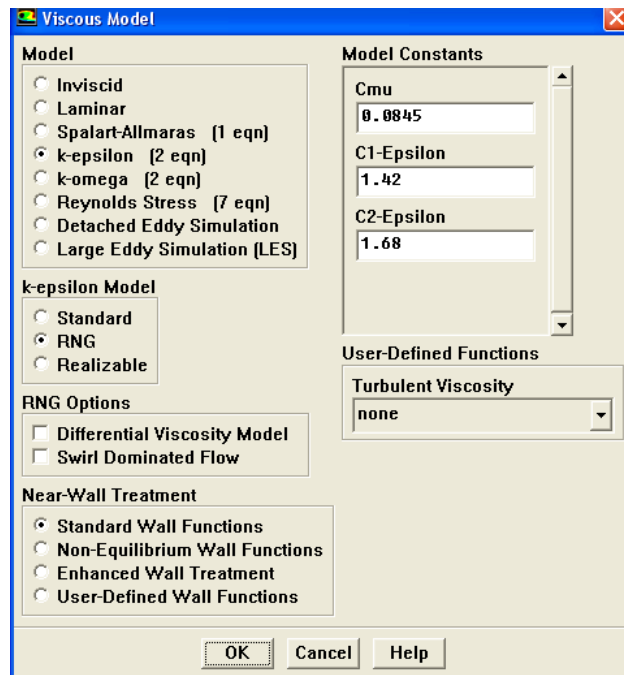
- (d) Close the Grid Display panel.

Step 2: Models

1. Enable the RNG k-epsilon model.

As the flow is turbulent, use a suitable turbulence model. For mixing tanks, it is recommended that you use the RNG k-epsilon model to resolve the correct flow features.

Define → Models → Viscous...

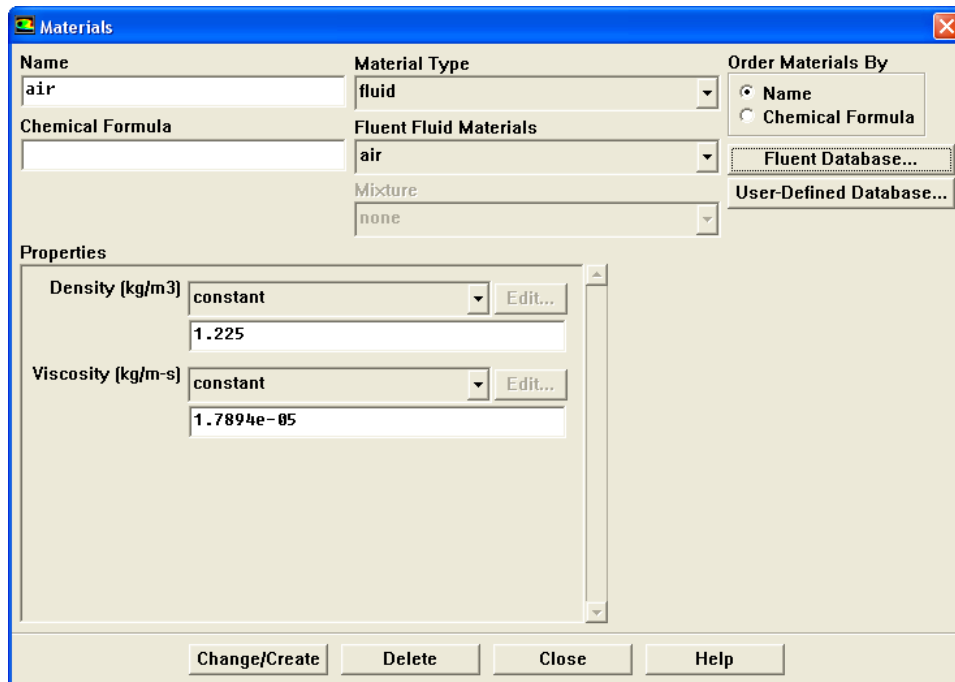


- (a) Select k-epsilon (2 eqn) from the Model list.
- (b) Select RNG from the k-epsilon Model list.
- (c) Click OK to close the Viscous Model panel.

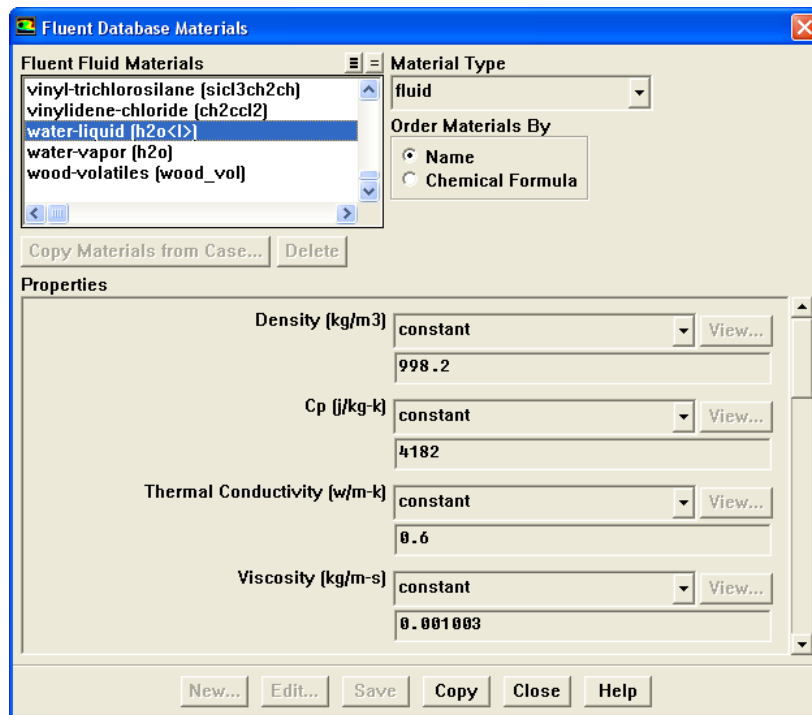
Step 3: Materials

1. Add liquid water to the list of fluid materials by copying it from the materials database.

Define → Materials...



- (a) Click the Fluent Database... button to open the Fluent Database Materials panel.
- i. Select water-liquid (h2o<l>) from the Fluent Fluid Materials selection list.
This will display the default settings for water-liquid.
 - ii. Click Copy and close the Fluent Database Materials panel.



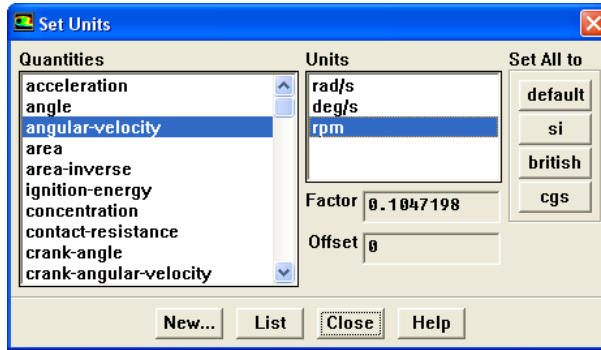
(b) Click Change/Create and close the Materials panel.

Step 4: Units

You can change the units of variables if required. The problem specifies angular velocity in rpm whereas the default unit is rad/s.

1. Change the unit of angular velocity.

Define → Units...



(a) Select angular-velocity from the Quantities selection list.

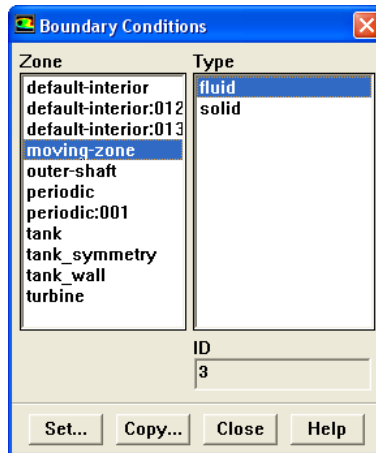
(b) Select rpm from the Units selection list and close the Set Units panel.

Step 5: Boundary Conditions

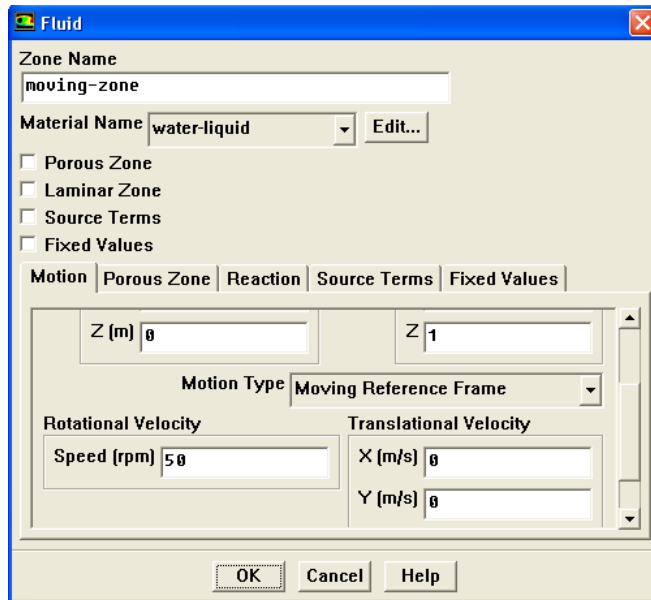
The problem is solved using rotating reference frame for the fluid. The turbine wall will then be defined to rotate with the moving frame.

Define → Boundary Conditions...

1. Set the boundary conditions for moving-zone.



- (a) Select moving-zone from the Zone selection list.
The Type will be reported as fluid.
- (b) Click the Set... button to open the Fluid panel.



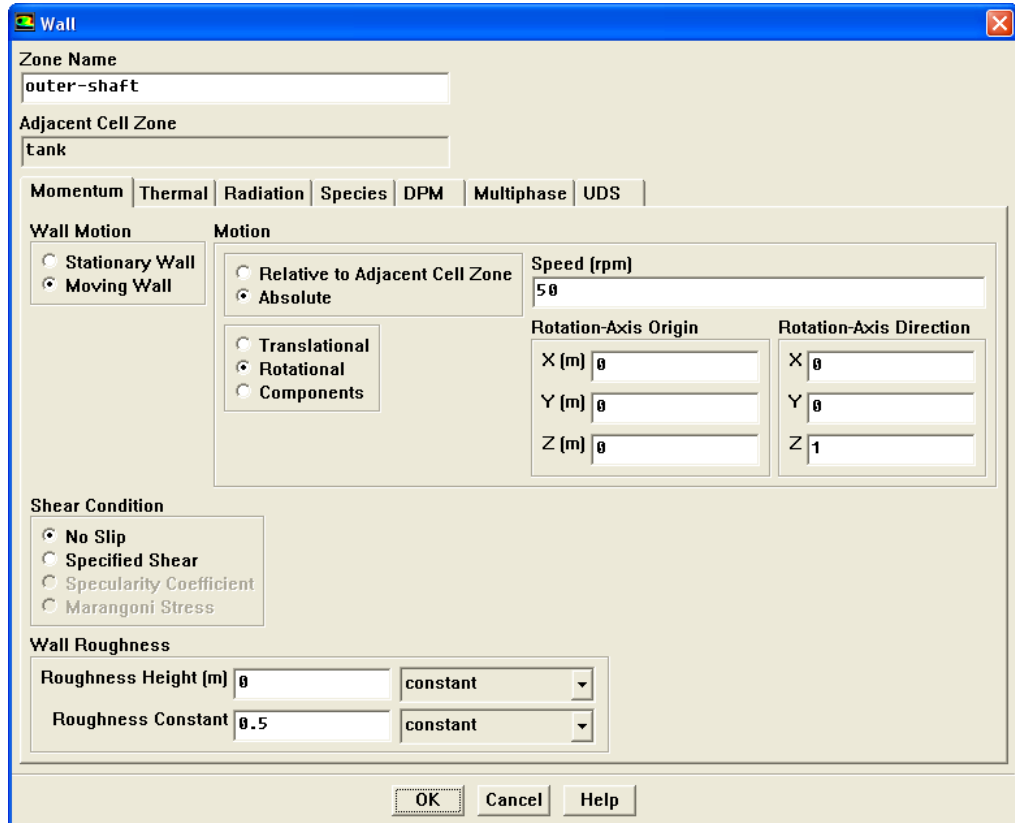
- i. Select water-liquid from the Material Name drop-down list.
 - ii. Select Moving Reference Frame from the Motion Type drop-down list.
 - iii. Enter 50 rpm for Speed and click OK to close the Fluid panel.
2. Set the boundary conditions for tank.
 - (a) Select tank from the Zone selection list.
The Type will be reported as fluid.
 - (b) Click the Set... button to open the Fluid panel.
 - i. Select water-liquid from the Material Name drop-down list.
 - ii. Click OK to close the Fluid panel.
3. Set the boundary conditions for wall zones and retain the default settings for turbine wall.

For a rotating reference frame, FLUENT assumes by default that all walls adjacent to the moving-zone rotate with the speed of moving reference frame. Hence all walls will rotate with respect to stationary (absolute) reference frame.

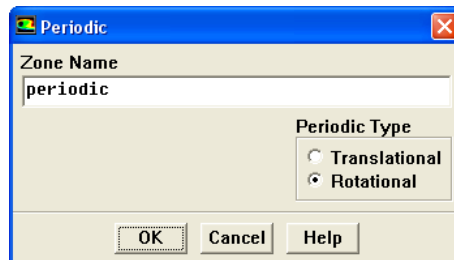
To specify a non-rotating wall, set a rotational speed of zero in the absolute frame. As the outer-shaft is a part of non-rotating fluid zone, explicitly set the rotation for this boundary.

- (a) Select outer-shaft from the Zone selection list.

- (b) Click the Set... button to open the Wall panel.
 - i. Select Moving Wall from the Wall Motion drop-down list.
 - ii. Select Absolute and Rotational in the Motion group box.
 - iii. Enter 50 rpm for Speed and click OK to close the Wall panel.



4. Set the boundary condition for periodic zones.
 - (a) Select periodic from the Zone selection list and click Set... button to open the Periodic panel.



- (b) Select Rotational in the Periodic Type group box.
- (c) Click OK to close the Periodic panel.
- (d) Similarly, set the boundary conditions for periodic:001.

5. Close the Boundary Conditions panel.

Step 6: Solution

1. Retain the default solver settings.

Solve → Controls → Solution...

2. Initialize the flow.

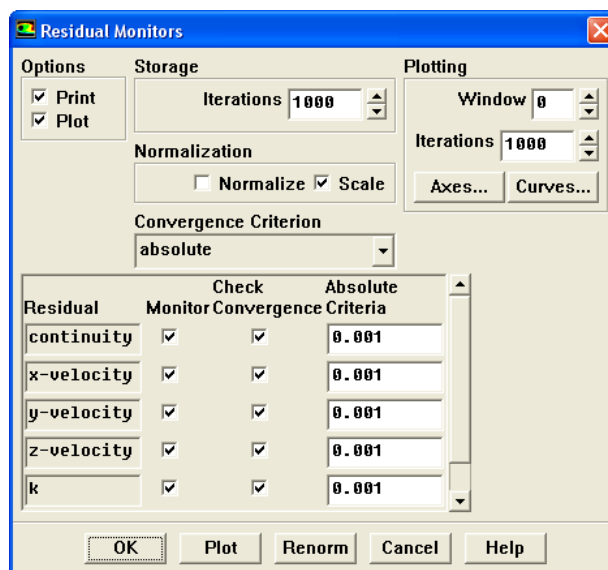
Solve → Initialize → Initialize...

- (a) Click Init and close the Solution Initialization panel.

The flow will get initialized with the default values of velocity and turbulence quantities.

3. Enable the plotting of residuals during the calculation.

Solve → Monitors → Residual...

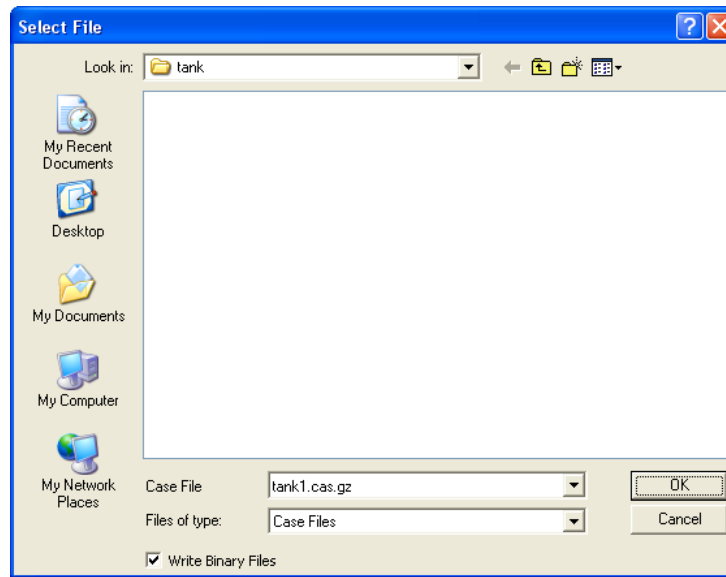


- (a) Enable Plot in the Options group box.
- (b) Click OK to close the Residual Monitors panel.

4. Save the case file (tank1.cas.gz).

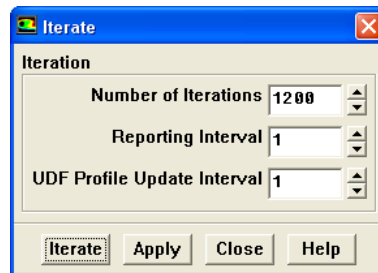
File → Write → Case...

Retain the default Write Binary Files option so that you can write a binary file. The .gz extension will save compressed files on both, Windows and Linux/UNIX platforms.



5. Start the calculation by requesting 1200 iterations.

→ Iterate...



- (a) Set Number of Iterations to 1200.
(b) Click Iterate.

The solution converges in approximately 1170 iterations with the default convergence criteria. The residuals plot is shown in Figure 9.4.

- (c) Close the Iterate panel.

The default convergence criteria are not sufficient to get the correct flow features in a mixing tank. To judge the convergence, some of the integrated quantities needs to be monitored along with velocity magnitude around the turbine.

In this problem, monitor the velocity magnitude on a surface just above and below the turbine. Also monitor the volume integral of kinetic energy in the tank.

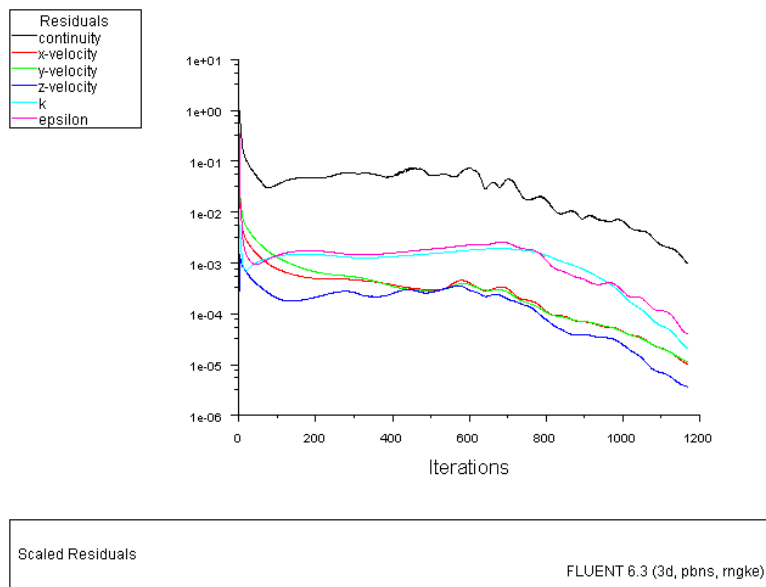
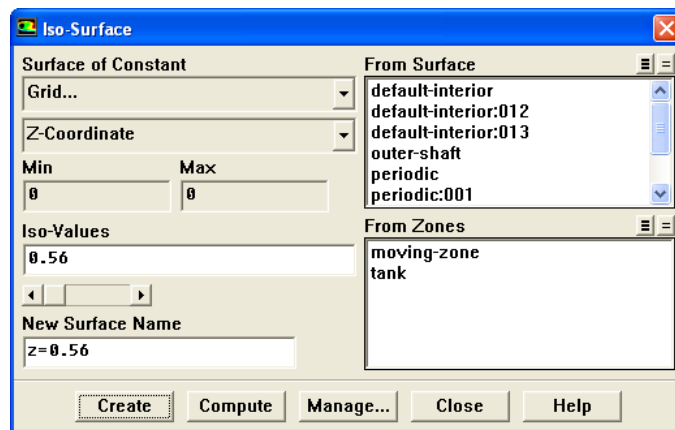


Figure 9.4: Scaled Residuals

6. Create isosurfaces.

Surface → Iso-Surface...

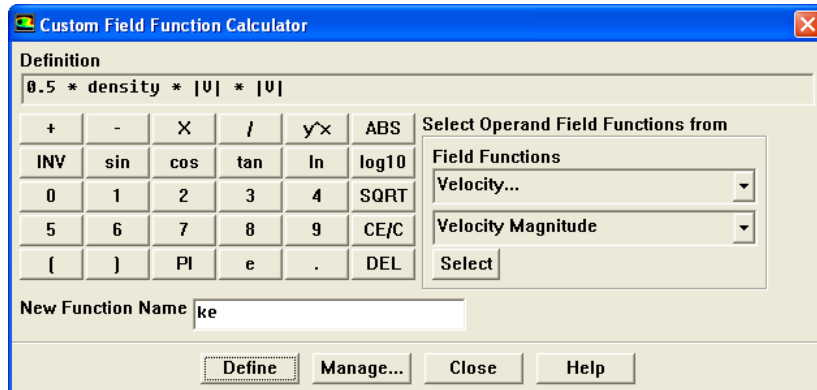


- (a) Select Grid... and Z-Coordinate from the Surface of Constant drop-down lists.
- (b) Enter 0.56 for Iso-Values.
- (c) Enter z=0.56 for New Surface Name.
- (d) Click Create.
- (e) Similarly, create another isosurface named z=0.64, with a value of 0.64.
- (f) Close the Iso-Surface panel.

7. Create a custom field function for kinetic energy,
($0.5 * \text{density} * \text{velocity-magnitude} * \text{velocity-magnitude}$).

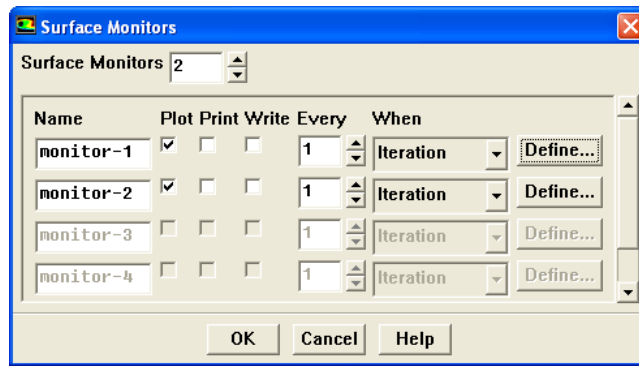
This custom field function can be used like any other standard variable reported by FLUENT. The value of the quantity will be evaluated at cell centers using the cell variables used in the definition.

Define → Custom Field Functions...

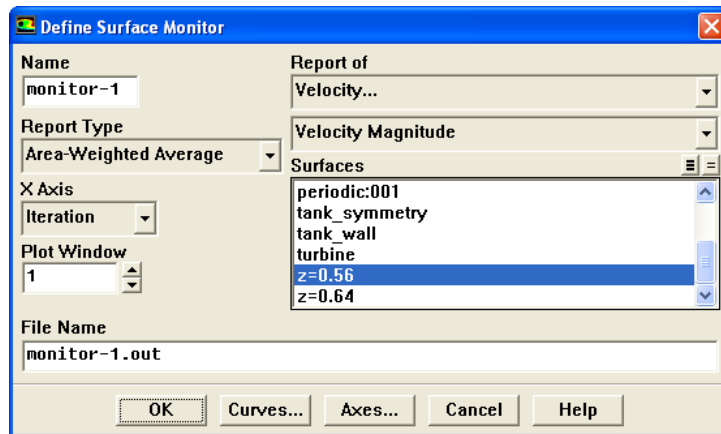


- (a) Enter 0.5 by clicking on the buttons available in the panel.
 - (b) Click the X button (multiplication operator).
 - (c) Select Density... and Density from the Field Functions drop-down lists.
 - (d) Click Select (to update the Definition text entry field).
 - (e) Click the X button.
 - (f) Select Velocity... and Velocity Magnitude from the Field Functions drop-down lists.
 - (g) Click Select and click the X button again.
 - (h) Select Velocity... and Velocity Magnitude from the Field Functions drop-down lists.
 - (i) Click Select.
 - (j) Enter ke for New Function Name and click Define.
 - (k) Close the Custom Field Function Calculator panel.
8. Set the surface monitors.

Solve → **Monitors** → Surface...



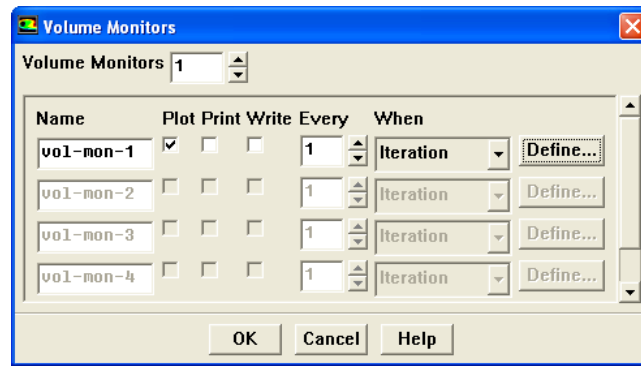
- (a) Increase the number of Surface Monitors to 2.
- (b) Enable Plot for both the monitors.
- (c) Click the Define... button for monitor-1 to open the Define Surface Monitor panel.



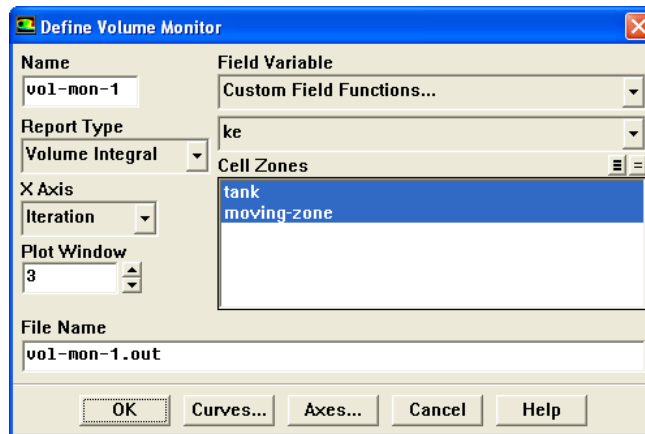
- i. Select Velocity... and Velocity Magnitude from the Report of drop-down lists.
 - ii. Select z=0.56 from the Surfaces selection list.
 - iii. Select Area-Weighted Average from the Report Type drop-down list.
 - iv. Click OK to close the Define Surface Monitor panel.
- (d) Similarly, define second surface monitor for surface, z=0.64.
- (e) Click OK to close the Surface Monitors panel.

9. Set the volume monitors.

Solve → Monitors → Volume...



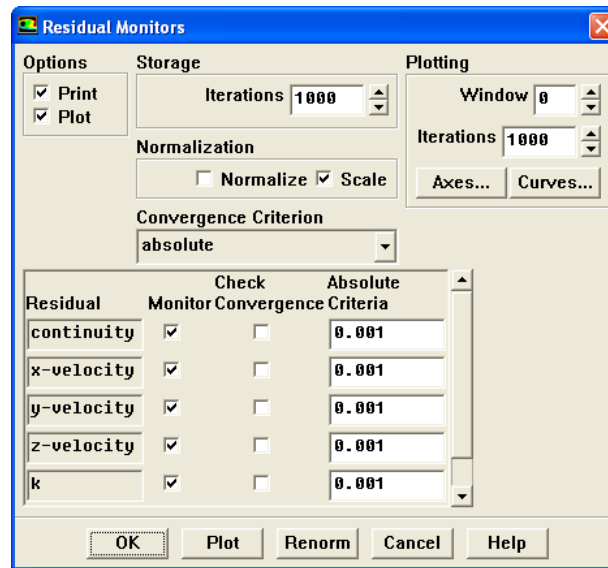
- (a) Increase the number of Volume Monitors to 1.
- (b) Enable Plot.
- (c) Click the Define... button for vol-mon-1 to open the Define Volume Monitor panel.



- i. Select Custom Field Functions... and ke from the Field Variable drop-down lists.
 - ii. Select both the zones from the Cell Zones selection list.
 - iii. Select Volume Integral from the Report Type drop-down list.
 - iv. Click OK to close the Define Volume Monitor panel.
- (d) Click OK to close the Volume Monitors panel.
10. Disable the convergence criteria for all the equations.

For a better convergence, perform the iterations till all the monitors flatten out. Therefore, disable the convergence criteria.

Solve → Monitors → Residual...



- (a) Disable Check Convergence for all the equations.
- (b) Click OK to close the Residual Monitors panel.

11. Iterate the solution till all the monitors reach a constant value, as shown in Figures 9.5, 9.6 and 9.7.

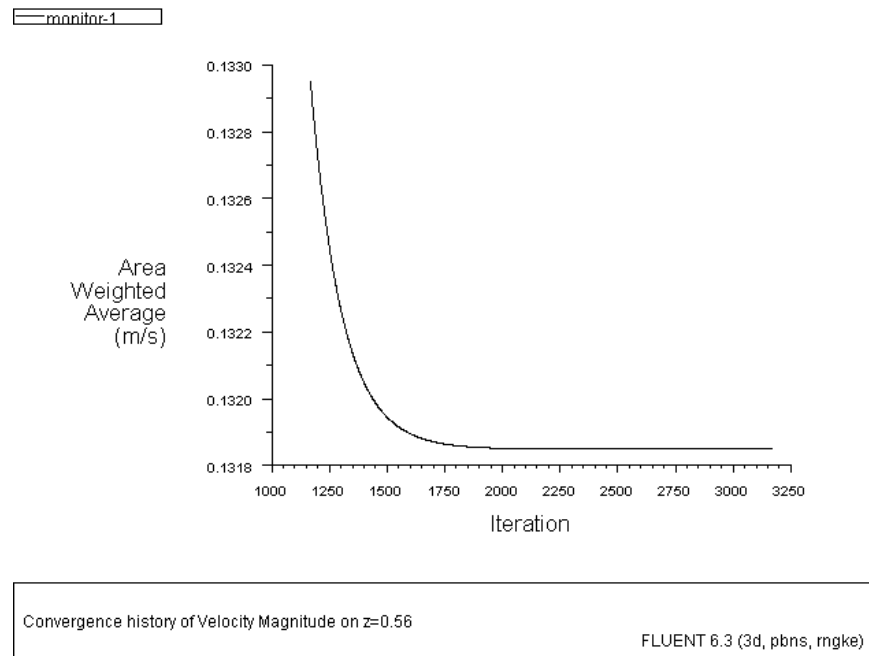
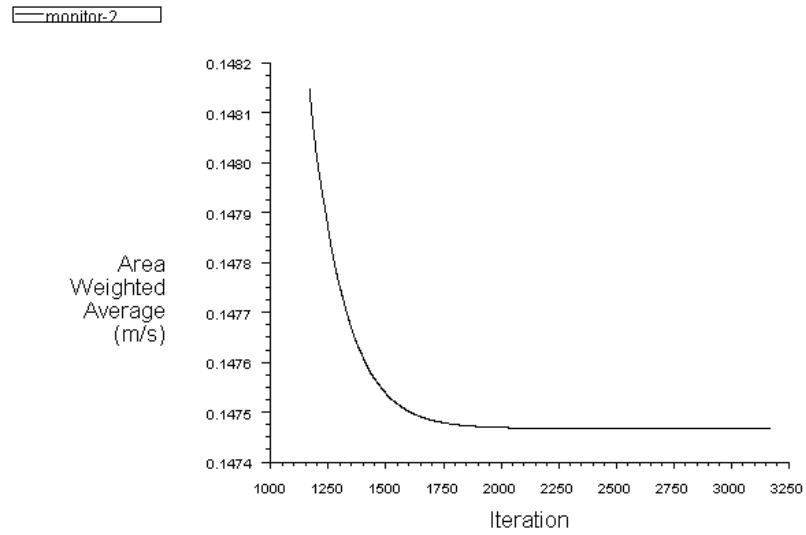
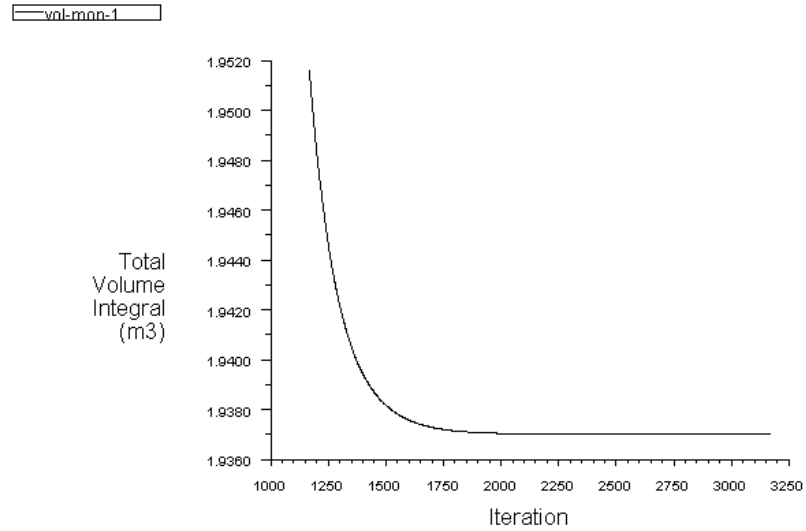


Figure 9.5: Surface Monitor for Velocity Magnitude on $z = 0.56$



Convergence history of Velocity Magnitude on z=0.64
FLUENT 6.3 (3d, pbns, rngke)

Figure 9.6: Surface Monitor for Velocity Magnitude on $z = 0.64$



Convergence history of ke on tank etc.
FLUENT 6.3 (3d, pbns, rngke)

Figure 9.7: Volume Monitor for ke

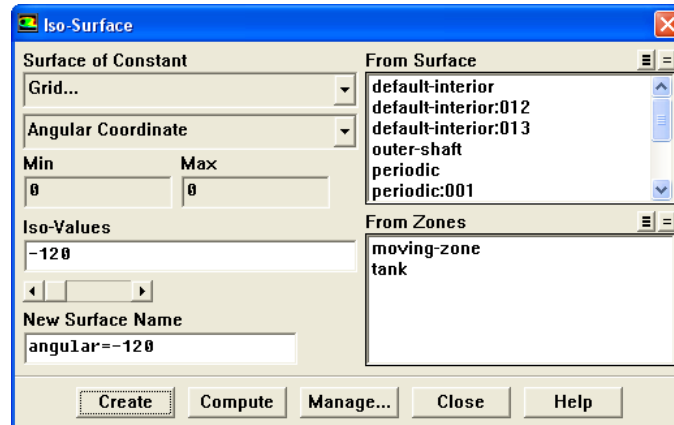
- Save the case and data files (tank2.cas.gz and tank2.dat.gz).

File → Write → Case & Data...

Step 7: Postprocessing

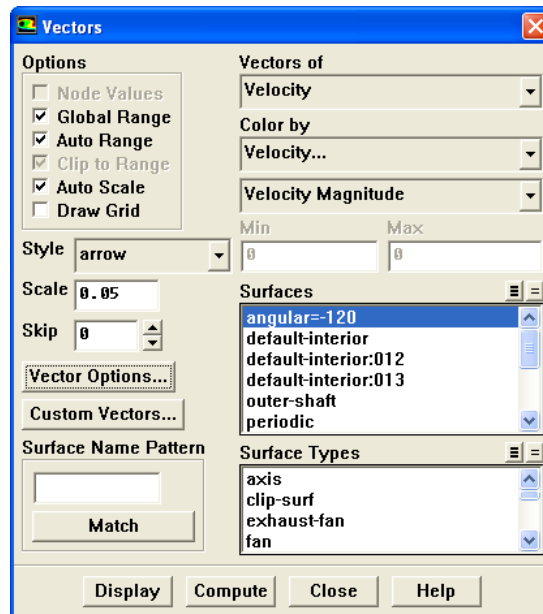
- Create an angular co-ordinate.

Surface → Iso-Surface...



- Select Grid... and Angular Coordinate from the Surface of Constant drop-down lists.
 - Enter -120 for Iso-Values.
 - Enter angular--120 for New Surface Name and click Create.
 - Close the Iso-Surface panel.
- Display velocity vectors on an iso-surface created using angular co-ordinate (Figure 9.8).

Display → Vectors...



- (a) Select Velocity from the Vectors of drop-down list.
- (b) Select Velocity... and Velocity Magnitude from the Color by drop-down lists.
- (c) Select angular=-120 from the Surfaces selection list.
- (d) Click the Vector Options... button to open the Vector Options panel.

- i. Enable Fixed Length.

This allows you to display all the vectors with the same length.

- ii. Click Apply and close the Vector Options panel.

- (e) Enter 0.05 for Scale.
- (f) Click Display and close the Vectors panel.

There are two circulation loops which enhance mixing, one at the top of the turbine and another below.

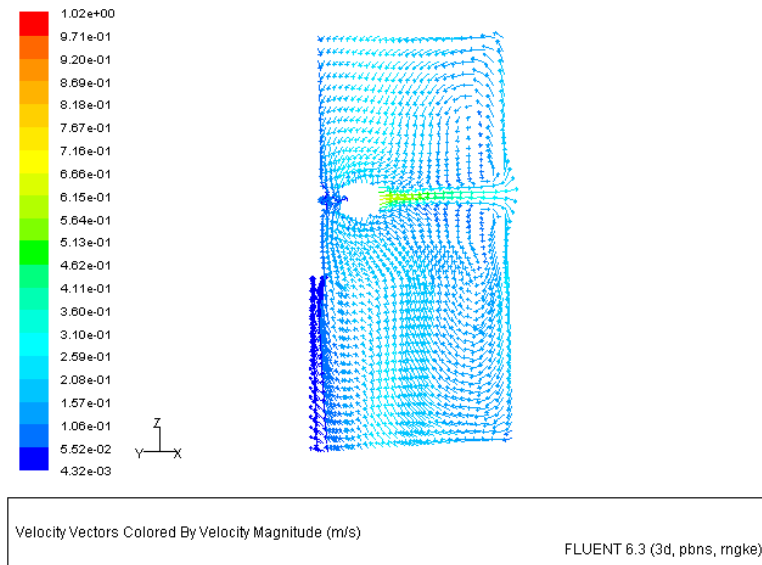
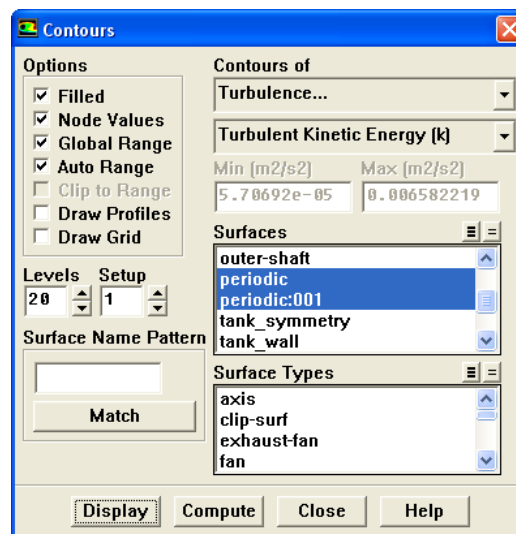


Figure 9.8: Velocity Vectors on angular = -120

3. Display turbulent kinetic energy on the periodic surfaces (Figure 9.9).

Display → Contours...



- (a) Select Turbulence... and Turbulent Kinetic Energy from the Contours of drop-down lists.
- (b) Select periodic and periodic:001 from the Surfaces selection list.
- (c) Enable Filled and click Display.
- (d) Close the Contours panel.

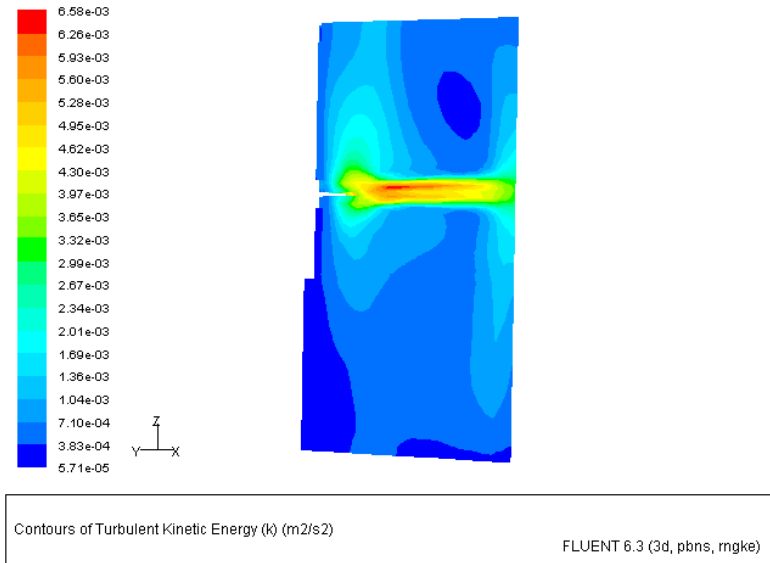
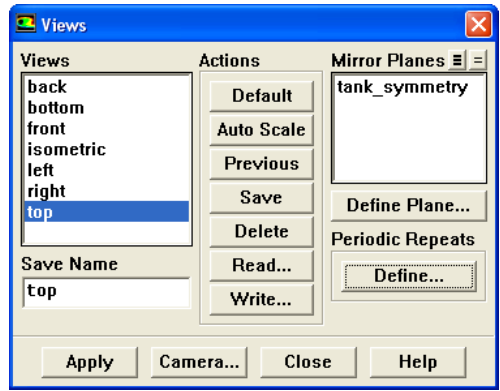


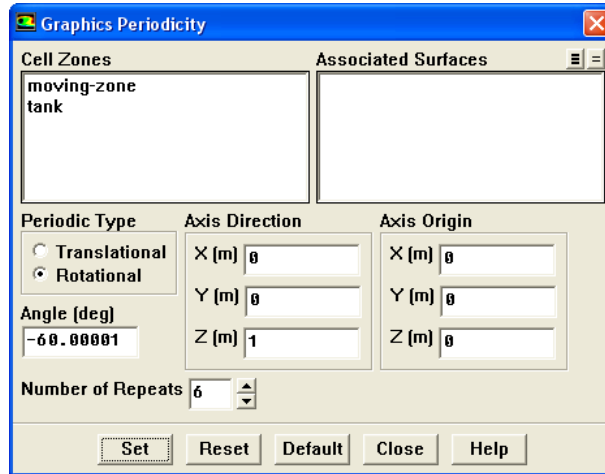
Figure 9.9: Contours of Turbulent Kinetic Energy on Periodic Surfaces

4. Change the view so that results can be viewed for complete domain.

Display → Views...



- (a) Click the Define... button in the Periodic Repeats group box to open the Graphics Periodicity panel.



i. Click the Set button.

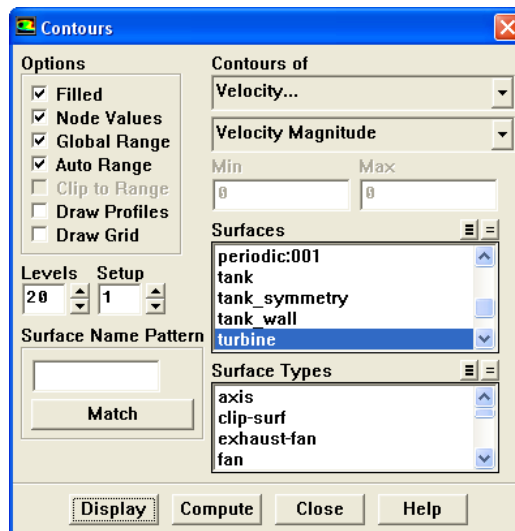
The display in graphics window will get updated and will repeat the surfaces six times.

ii. Close the Graphics Periodicity panel.

(b) Click Apply and close the Views panel.

5. Display the velocity contours on surface, turbine (Figure 9.10).

Display → Contours...



(a) Select Velocity... and Velocity Magnitude from the Contours of drop-down lists.

(b) Select turbine from the Surfaces selection list.

(c) Click Display and close the Contours panel.

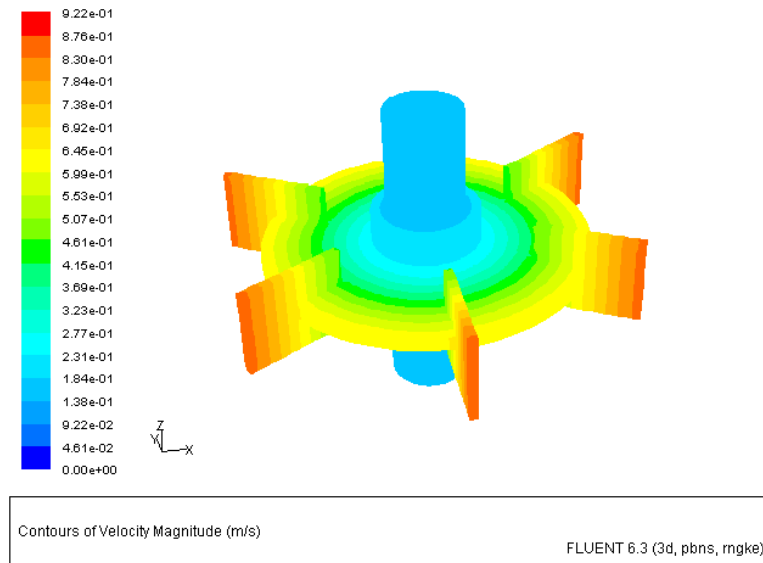


Figure 9.10: Contours of Velocity Magnitude on turbine

Summary

This example demonstrates the use of moving reference frame (MRF) to model the flow in mixing tanks. Monitors were used to judge convergence of crucial quantities. In actual CFD analysis, much finer mesh needs to be employed around the blade to resolve the velocity and pressure gradients correctly.

References

M. Campolo, F. Sbrizzai, A. Soldati, *Time-dependent flow structure and Lagrangian mixing in Ruston-impeller baffled-tank reactor*, Chemical Engineering Science 58 (2003) 1615-1629.

Exercises/Discussions

1. Can you estimate the power that would be needed to drive such system?
2. What will be the flow pattern if:
 - (a) The turbine is rotated in opposite direction
 - (b) The rotational speed of turbine is increased
3. Display pathlines by creating some points in the tank to visualize the flow pattern.
4. Check what can be the maximum speed for which we can get a converged solution with the existing mesh.

Links for Further Reading

- <http://www.bakker.org>
- http://www.postmixing.com/About/mixing_course.htm
- <http://www.fluent.com/solutions/brochures/mixsim.pdf>

