
Tutorial 9. Using Multiple Rotating Reference Frames

Introduction

Many engineering problems involve rotating flow domains. One example is the centrifugal blower unit that is typically used in automotive climate control systems. For problems where all the moving parts (fan blades, hub and shaft surfaces, etc.) are rotating at a prescribed angular velocity, and the stationary walls (e.g., shrouds, duct walls) are surfaces of revolution with respect to the axis of rotation, the entire domain can be referred to as a single rotating frame of reference. However, when each of several parts is rotating about a different axis of rotation, or about the same axis at different speeds, or when the stationary walls are not surfaces of revolution (such as the volute around a centrifugal blower wheel), a single rotating coordinate system is not sufficient to “immobilize” the computational domain so as to predict a steady-state flow field.

In FLUENT, the flow features associated with multiple rotating parts can be analyzed using the multiple reference frame (MRF) capability. This model is powerful in that multiple rotating reference frames can be included in a single domain. The resulting flow field is representative of a snapshot of the transient flow field in which the rotating parts are moving. However, in many cases the interface can be chosen in such a way that the flow field at this location is independent of the orientation of the moving parts. In other words, if an interface can be drawn on which there is little or no angular dependence, the model can be a reliable tool for simulating time-averaged flow fields. It is therefore very useful in complicated situations where one or more rotating parts are present.

This tutorial illustrates the procedure for setting up and solving a problem using the MRF capability. As an example, the flow field on a 2D section of a centrifugal blower will be calculated. The example will be limited to a single rotating reference frame.

This tutorial demonstrates how to do the following:

- Specify different frames of reference for different fluid zones.
- Set the relative velocity of each wall.
- Calculate a solution using the pressure-based solver.

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed [Tutorial 1](#). Some steps in the setup and solution procedure will not be shown explicitly.

In general, to solve problems using the MRF feature, you should be familiar with the concept of creating multiple fluid zones in your grid generator.

Problem Description

This problem considers a 2D section of a generic centrifugal blower. A schematic of the problem is shown in Figure 9.1. The blower consists of 32 blades, each with a chord length of 13.5 mm. The blades are located approximately 56.5 mm (measured from the leading edge) from the center of rotation. The radius of the outer wall varies logarithmically from 80 mm to 146.5 mm. The total pressure at the inlet is defined to be 200 Pa and the flow discharges to ambient conditions (static pressure = 0 Pa). The blades are rotating with an angular velocity of 261 rad/s. The flow is assumed to be turbulent.

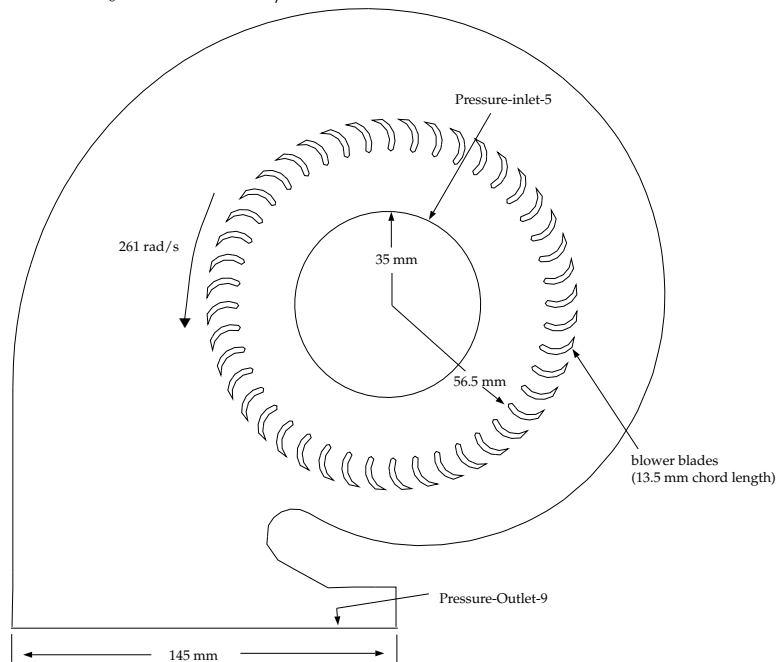


Figure 9.1: Schematic of the Problem

Setup and Solution

Preparation

1. Download `multiple_rotating.zip` from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working folder (as described in Tutorial 1).
2. Unzip `multiple_rotating.zip`.
The file, `blower.msh.gz` can be found in the `multiple_rotating` folder created after unzipping the file.
3. Start the 2D (2d) version of FLUENT.

Step 1: Grid

1. Read in the mesh file (`blower.msh.gz`) in the FLUENT serial solver.

`File` → `Read` → `Case...`

The mesh file is opened in the serial solver because the Smooth/Swap... operation is available only in serial FLUENT.

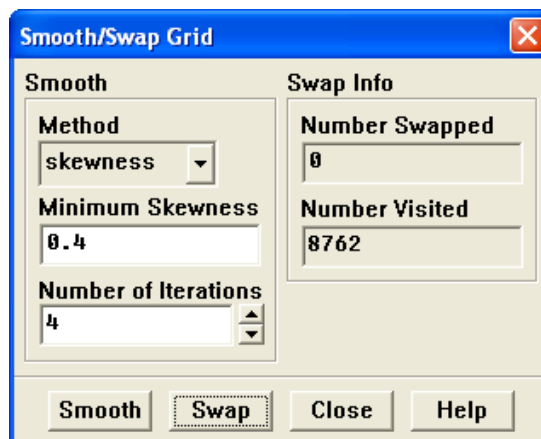
2. Check the grid.

`Grid` → `Check`

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

3. Smooth and swap the grid.

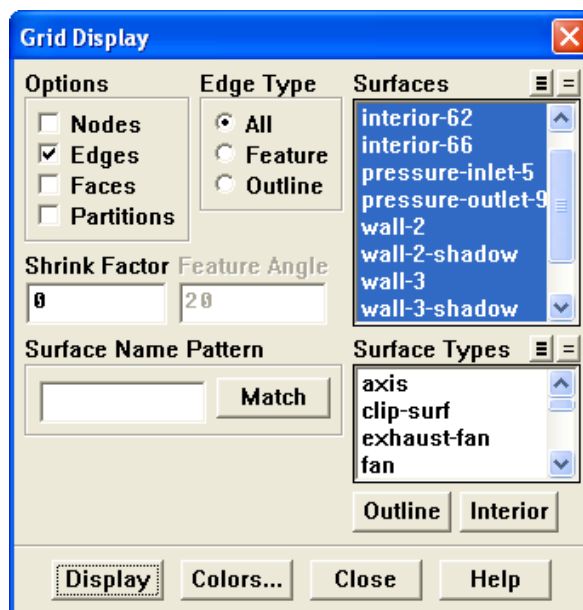
`Grid` → `Smooth/Swap...`



The smooth and swap function is available only in serial FLUENT. If you want to solve using FLUENT parallel, you can do so only after node smoothing and face swapping. Node smoothing and face swapping will improve the mesh quality. This step is recommended for triangular and tetrahedral meshes.

- (a) Retain the default smoothing parameters and click Smooth.
 - (b) Click Swap repeatedly until the Number Swapped under Swap Info is zero.
 - (c) Close the Smooth/Swap Grid panel.
4. Display the mesh (Figure 9.2).

Display → Grid...



- (a) Retain the default settings.
- (b) Click Display and close the Grid Display panel.

The mesh consists of three fluid zones, fluid-13, fluid-14, and fluid-18. These are reported in the console when the grid is read. In the Grid Display panel, the fluid zones are reported as interior zones interior-61, interior-62 and interior-66. In a later step, you will learn how to associate a fluid zone with an interior zone. The fluid zone containing the blades will be solved in a rotational reference frame.

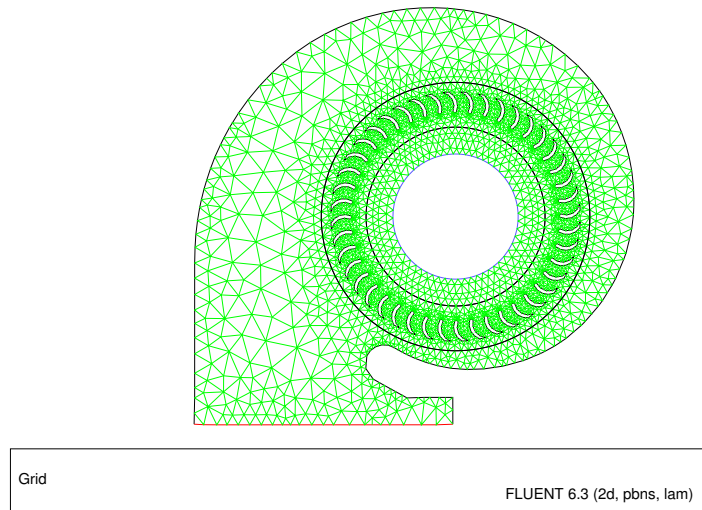


Figure 9.2: Mesh of the 2D Centrifugal Blower

The fluid zones are separated by wall boundaries. These boundaries were used in the grid generator to separate the fluid zones, and will be converted to interior zones when the boundary conditions are set later in this tutorial. Each of these wall zones also has an associated “shadow wall” which was created by FLUENT when it read the grid. Shadow walls are created whenever a wall has fluid zones on both sides.

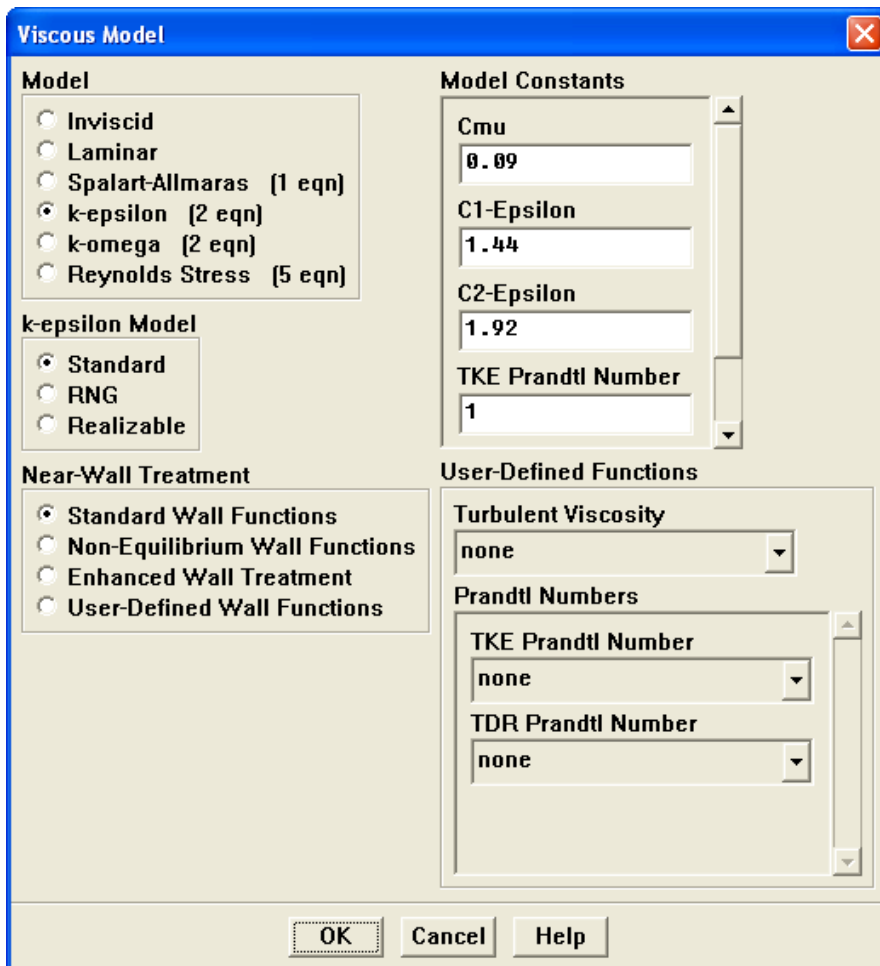
Step 2: Models

1. Retain the default solver settings.

Define → Models → Solver...

2. Enable the standard k - ϵ turbulence model.

Define → Models → Viscous...

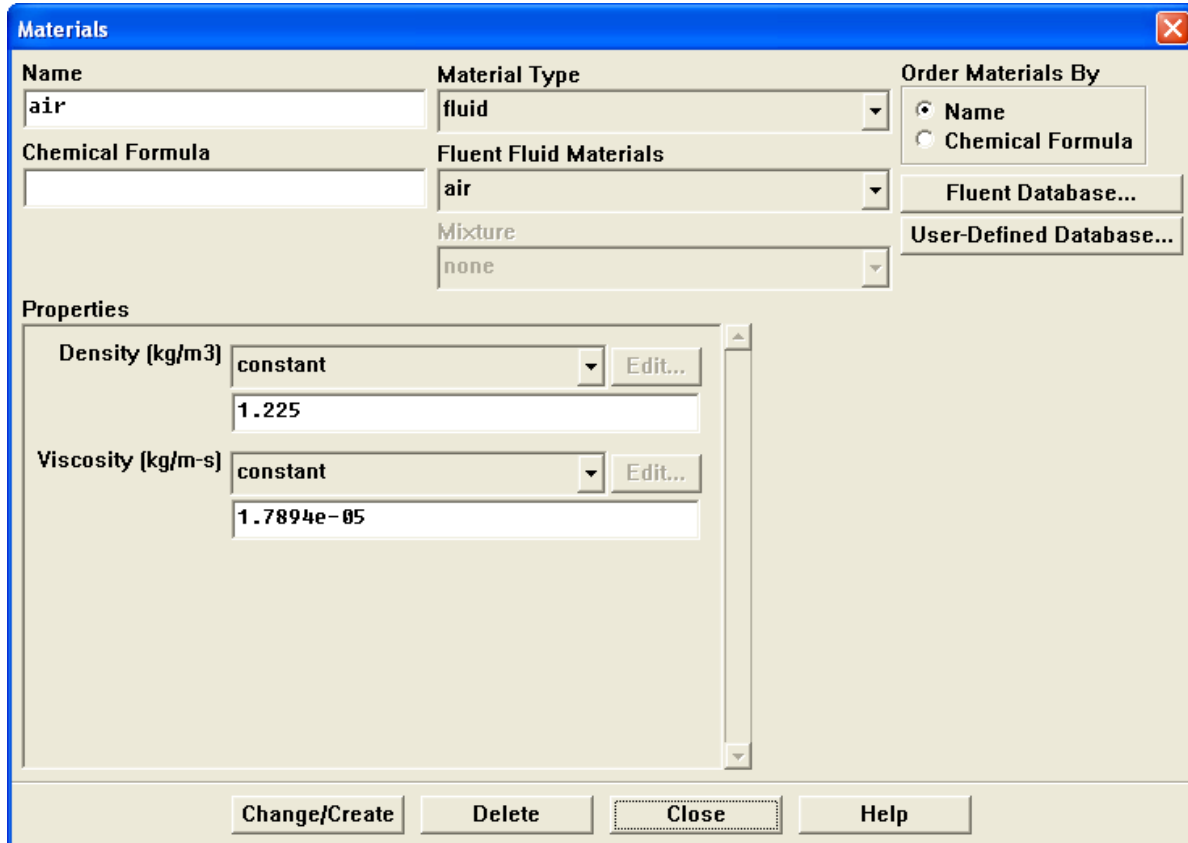


- (a) Select k - ϵ (2 eqn) from the Model list.
- (b) Click OK to close the Viscous Model panel.

Step 3: Materials

Retain the default material, air, with its predefined properties, for all fluid zones.

Define → Materials...

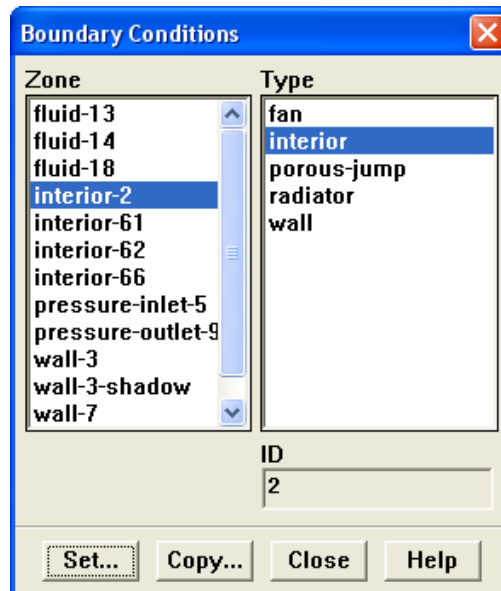


Extra: If needed, you could modify the fluid properties for air or copy another material from the database.

See Chapter 8 of the [User's Guide](#) for details.

Step 4: Boundary Conditions

Define → Boundary Conditions...

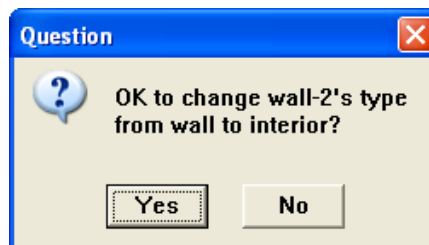


1. Change wall-2 and wall-3 type to interior.

The zones wall-2 and wall-3 are the interfaces between the three fluid zones. They need to be changed to type interior, as discussed earlier. The resulting interior faces are those that have fluid cells on both sides but do not require any boundary conditions to be set.

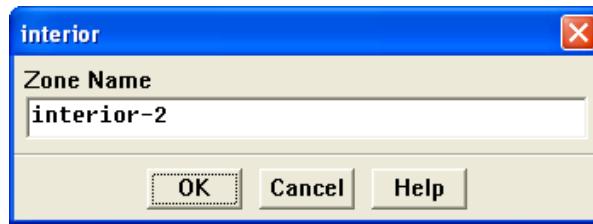
- (a) Select wall-2 in the Zone selection list and then select interior in the Type selection list.

A Question dialog box will open, asking if you want to change the type from wall to interior.



- i. Click Yes to open the interior panel.

FLUENT will fuse wall-2 and wall-2-shadow together to form interior-2.



- ii. Click OK to retain the default Zone Name.
- (b) Similarly, change wall-3 to an interior zone named interior-3.
2. Identify the rotating fluid zone (i.e., the zone containing the blades) by displaying the mesh for each zone.

→ Grid...

It is unclear when you read the grid which fluid zone corresponds to which interior zone. While the interior zones can be selected individually in the Grid Display panel, the fluid zones cannot. Commands in the text interface, however, can be used to make this association.

- (a) Deselect all surfaces by clicking on the unshaded icon to the right of Surfaces.
- (b) Click the Outline button at the bottom of the panel to select only the outline surfaces of the domain.
- (c) Click Display.

Only the domain boundaries and interior walls will be displayed.
- (d) In the console, type the commands as shown in the boxes.

Hint: You may need to press the <Enter> key to get the > prompt.

```
> 
/display> 
()
zone id/name(1) [()] 
zone id/name(2) [()] <Enter>
```

The resulting display (Figure 9.3) shows that zone fluid-13 corresponds to the rotating region.

3. Close the Grid Display panel.

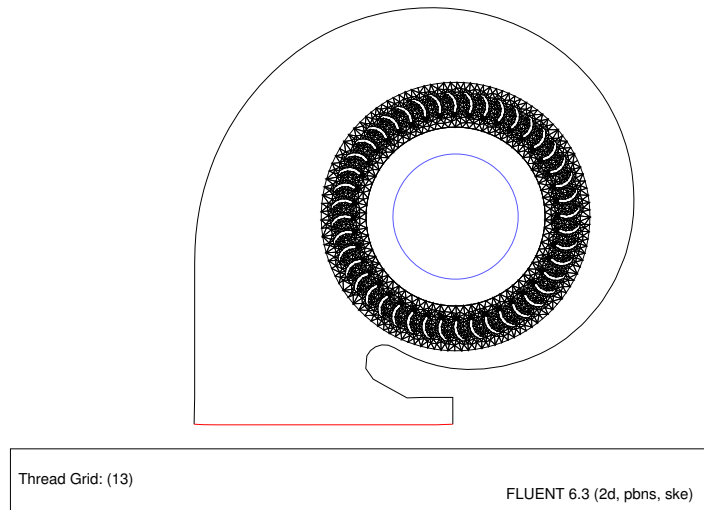
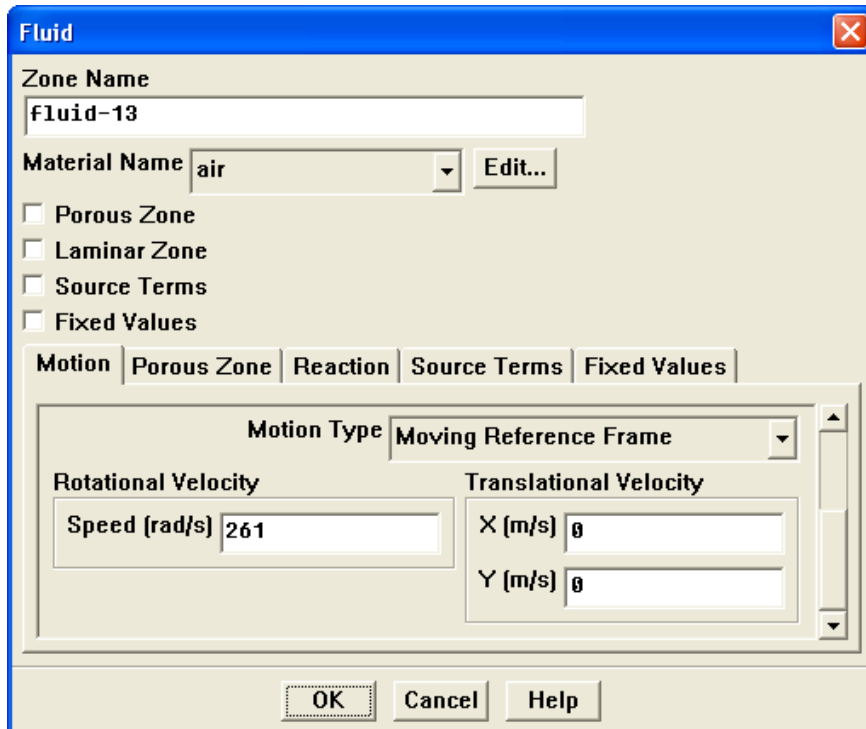


Figure 9.3: Mesh in fluid-13

4. Define the boundary conditions for the rotational reference frame (fluid-13).



- (a) Retain the Rotation-Axis Origin default setting of (0,0).

This is the center of curvature for the circular boundaries of the rotating zone.

- (b) Select Moving Reference Frame from the Motion Type drop-down list.
- (c) Enter 261 rad/s for Speed in the Rotational Velocity group box.
Scroll down to find the Speed number-entry box.
- (d) Click OK to close the Fluid panel.

Note: *Since the other fluid zones are stationary, you do not need to set any boundary conditions for them. If one of the remaining fluid zones was also rotating, you would need to set the appropriate rotational speed for it.*

5. Set the boundary conditions (see Figure 9.1) for the flow inlet (pressure-inlet-5).

- (a) Enter 200 Pa for the Gauge Total Pressure.
- (b) Select Intensity and Hydraulic Diameter from the Specification Method drop-down list.
- (c) Enter 5 % for Turbulent Intensity.
- (d) Enter 0.05 m for Hydraulic Diameter.
- (e) Click OK to close the Pressure Inlet panel.

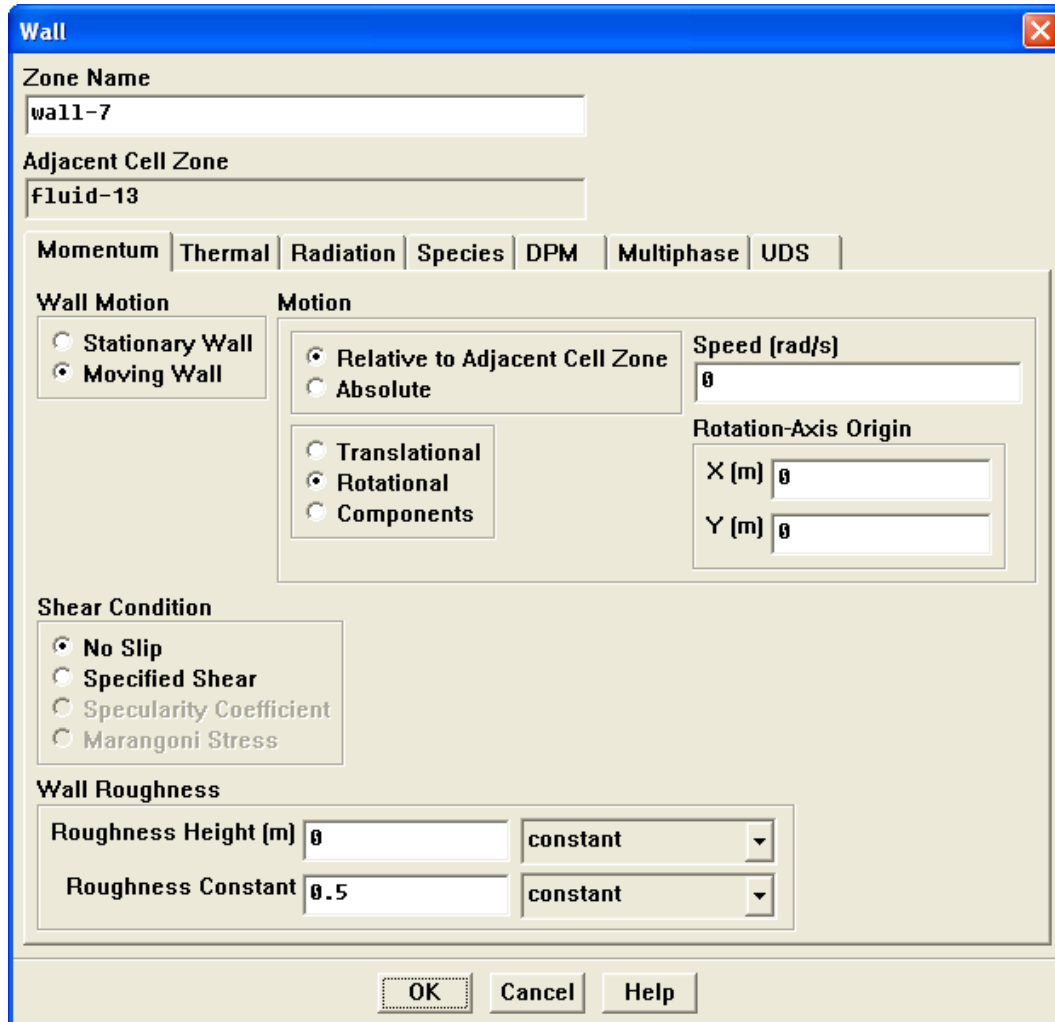
Note: *All pressures that you specify in FLUENT are gauge pressures, relative to the operating pressure specified in the Operating Conditions panel. By default, the operating pressure is 101325 Pa.*

See Section 8.14 of the [User's Guide](#) for details.

- Set the backflow turbulence parameters for the flow outlet (pressure-outlet-9) to the same values used for pressure-inlet-5.

Note: *The backflow values are used only if reversed flow occurs at the outlet, but it is a good idea to use reasonable values, even if you do not expect any backflow to occur.*

- Define the velocity of the wall zone representing the blades (wall-7) relative to the moving fluid zone.



With fluid-13 set to a rotating reference frame, wall-7 becomes a moving wall.

- Select Moving Wall from the Wall Motion list.
The Wall panel will expand to show the wall motion parameters.
- Select Relative to Adjacent Cell Zone and Rotational from the Motion lists.
- Set the (relative) Speed to 0 rad/s.

- (d) Click OK to close the Wall panel.

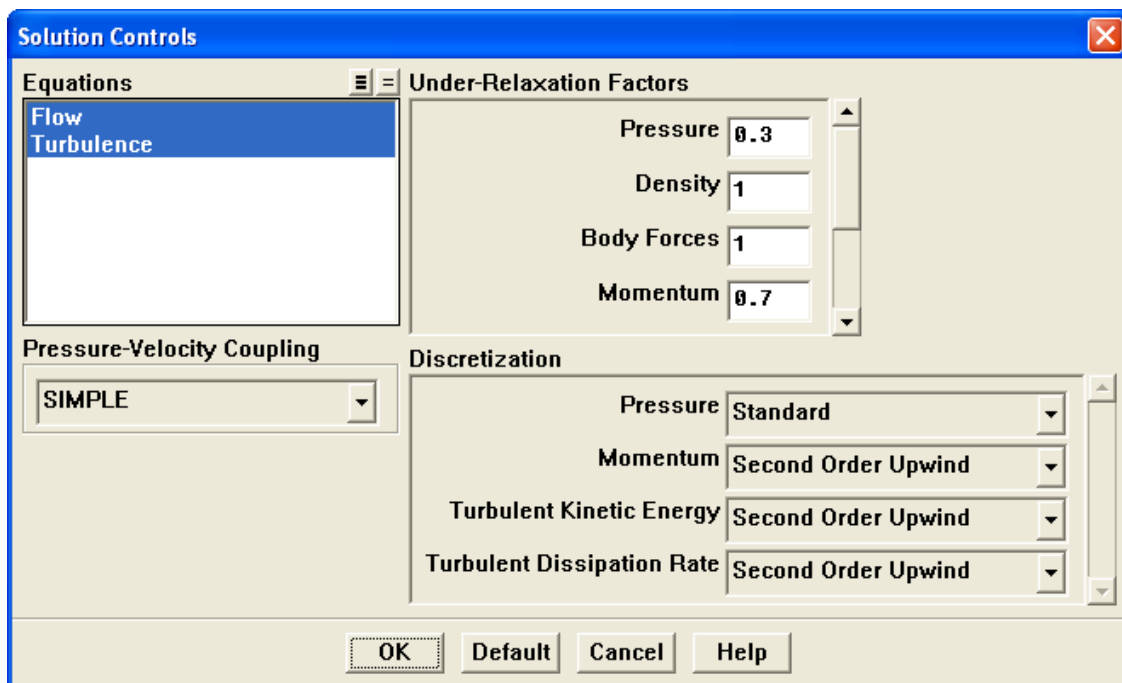
The Rotation-Axis Origin should be located at $x = 0$ m and $y = 0$ m. With these settings, the blades will move at the same speed as the surrounding fluid.

8. Close the Boundary Conditions panel.

Step 5: Solution

1. Set the parameters that control the solution.

Solve → Controls → Solution...



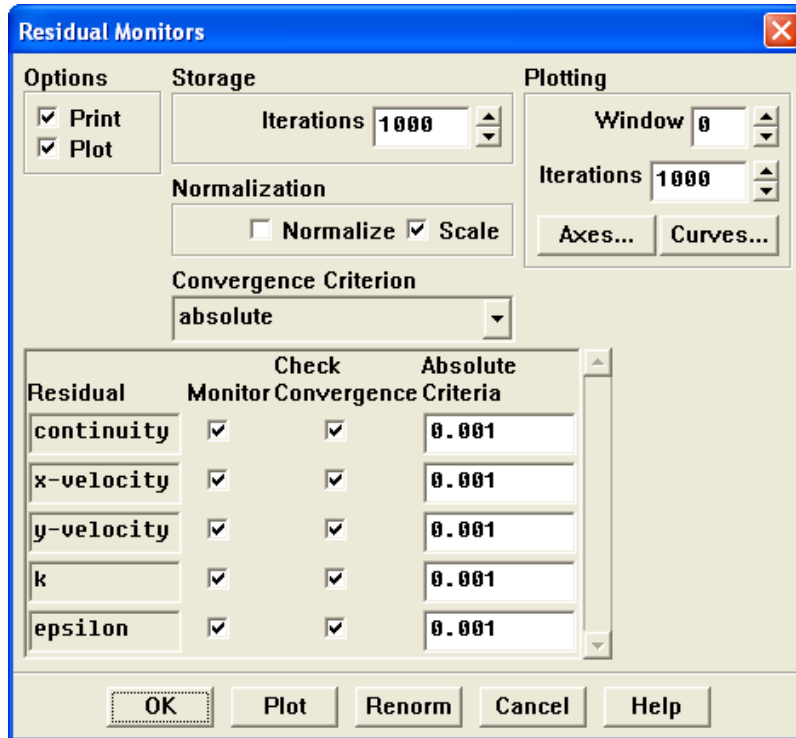
- (a) Select Second Order Upwind from the Momentum, Turbulent Kinetic Energy, and Turbulent Dissipation Rate drop-down lists in the Discretization group box.

The second-order scheme will provide a more accurate solution.

- (b) Retain the default parameters for all other solution controls and click OK to close the Solution Controls panel.

2. Enable the plotting of residuals during the calculation.

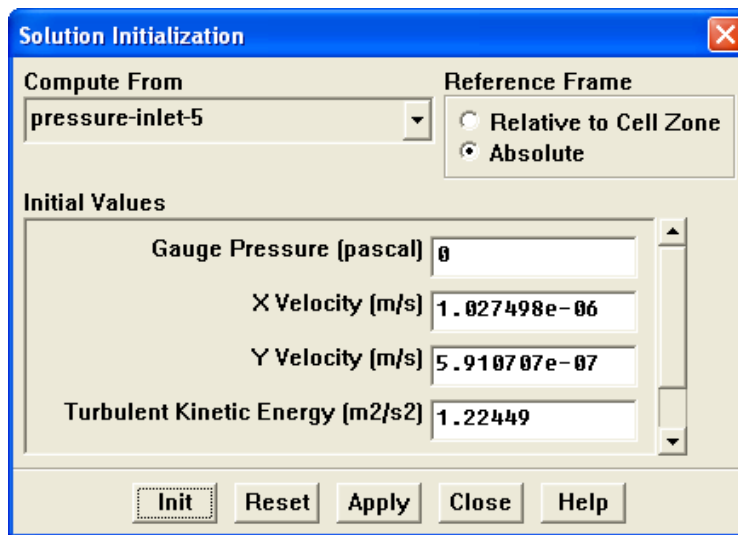
Solve → Monitors → Residual...



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

3. Initialize the solution using the boundary conditions set at pressure-inlet-5.

Solve → Initialize → Initialize...



- (a) Select `pressure-inlet-5` from the Compute From drop-down list.
- (b) Select `Absolute` from the Reference Frame list.
- (c) Click `Init` to initialize the solution.
- (d) Close the Solution Initialization panel.

Note: *In this tutorial, you chose an Absolute reference frame for initializing the solution. In certain cases, Relative to Cell Zone may help the solution converge faster.*

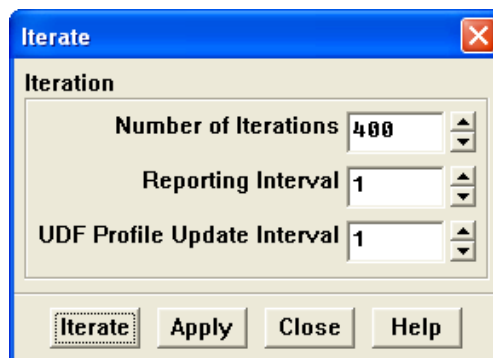
See Section 25.14 of the [User's Guide](#) for guidelines.

4. Save the case file (`blower.cas.gz`).

`File` → `Write` → `Case...`

5. Start the calculation by requesting 400 iterations.

`Solve` → `Iterate...`



- (a) Enter 400 for the Number of Iterations.
- (b) Click `Iterate`.

During the calculation, FLUENT will report that there is reversed flow occurring at the exit. This is due to the sudden expansion, which results in a recirculating flow near the exit.

The solution will converge in around 160 iterations (when all residuals have dropped below 0.001).

- (c) Close the `Iterate` panel.

6. Save the case and data files (`blower2.cas.gz` and `blower2.dat.gz`).

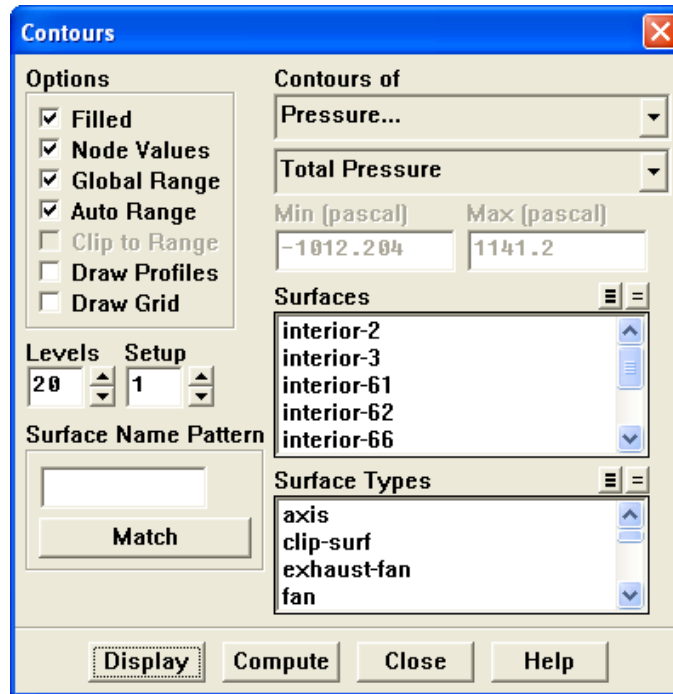
`File` → `Write` → `Case & Data...`

Note: *It is good practice to save the case file whenever you are saving the data. This will ensure that the relevant parameters corresponding to the current solution data are saved accordingly.*

Step 6: Postprocessing

1. Display filled contours of total pressure (Figure 9.4).

Display → Contours...



- (a) Enable Filled in the Options group box.
- (b) Select Pressure... and Total Pressure from the Contours of drop-down lists.
- (c) Click Display and close the Contours panel.

Total pressure contours show the expected pressure jump across the blower blades.

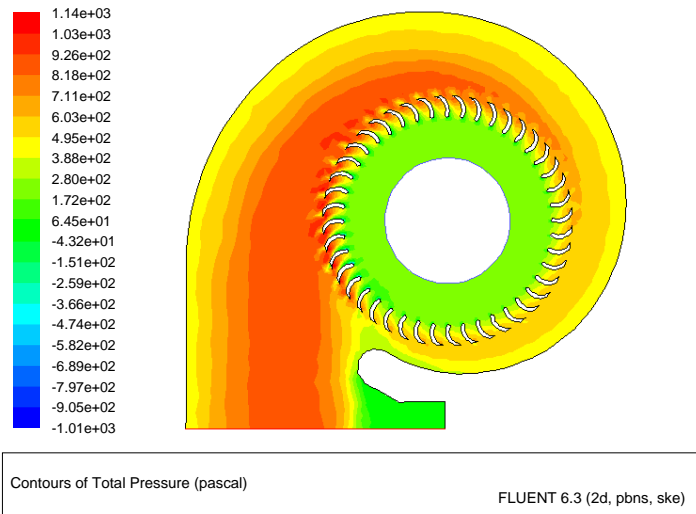
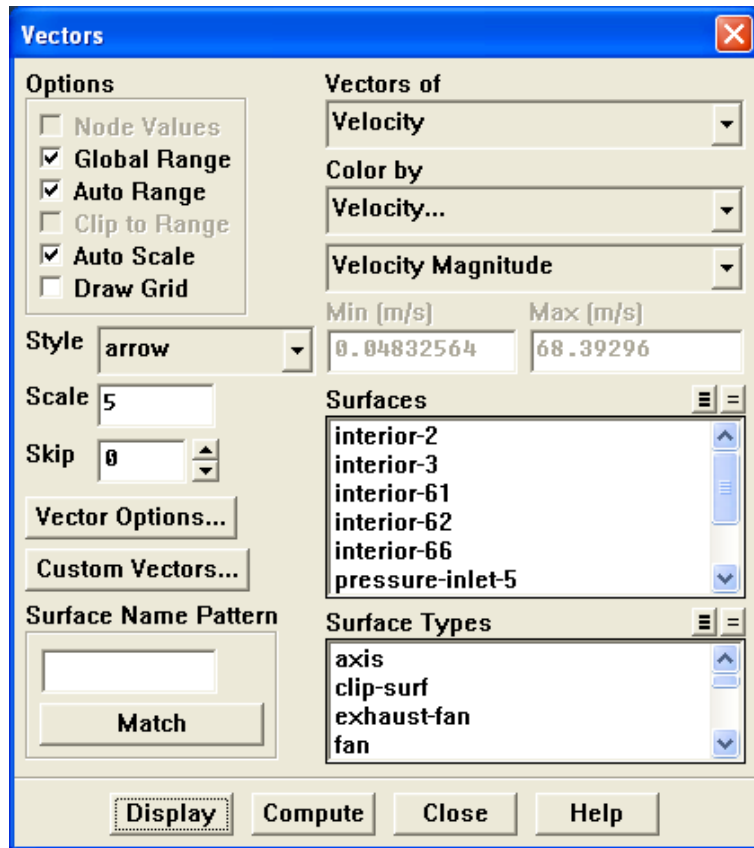


Figure 9.4: Contours of Total Pressure

2. Display velocity vectors (Figure 9.5).

Display → Vectors...



- (a) Enter 5 for Scale.
- (b) Click Display and close the Vectors panel.

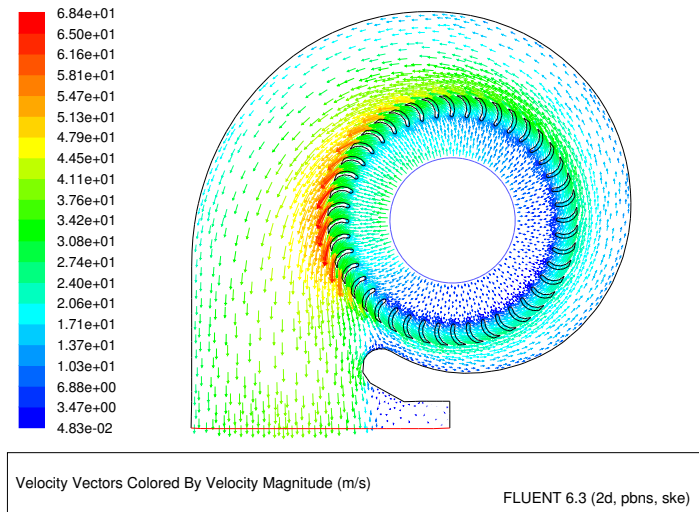
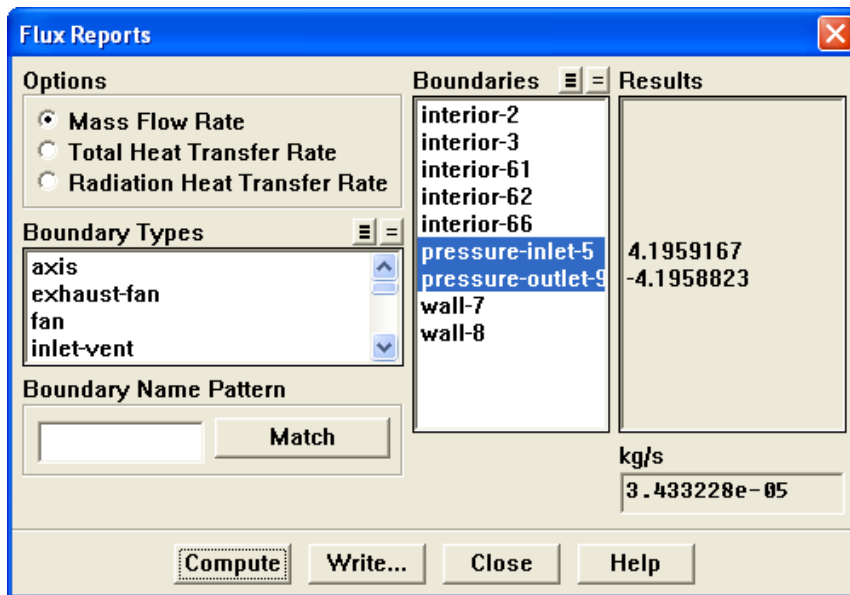


Figure 9.5: Velocity Vectors

By default, Auto Scale is chosen. This will automatically scale the length of velocity vectors relative to the size of the smallest cell in the mesh. To increase the length of the “scaled” vectors, set the Scale factor to a value greater than 1.

3. Report the mass flux at pressure-inlet-5 and pressure-outlet-9.

Report → Fluxes...



- Retain the selection of Mass Flow Rate in the Options group box.
- Select pressure-inlet-5 and pressure-outlet-9 from the Boundaries selection list.
- Click Compute.

The net mass imbalance should be no more than a small fraction (say, 0.5%) of the total flux through the system. If a significant imbalance occurs, you should decrease your residual tolerances by at least an order of magnitude and continue iterating.

The flux report will compute fluxes only for boundary zones. To report fluxes on surfaces or planes, use the Surface Integrals... option in the Report menu.

- Close the Flux Reports panel.

Summary

This tutorial illustrates the procedure for setting up and solving problems with multiple reference frames using FLUENT. Although this tutorial considers only one rotating fluid zone, extension to multiple rotating fluid zones is straightforward as long as you delineate each fluid zone.

Note that this tutorial was solved using the default absolute velocity formulation. For some problems involving rotating reference frames, you may wish to use the relative velocity formulation. See the User's Guide for details.

Further Improvements

This tutorial guides you through the steps to reach an initial solution. You may be able to obtain a more accurate solution by using an appropriate higher-order discretization scheme and by adapting the grid. Grid adaption can also ensure that the solution is independent of the grid. These steps are demonstrated in [Tutorial 1](#).