Tutorial 8. 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.

The following FLUENT features will be demonstrated in this tutorial:

- Specifying different frames of reference for different fluid zones.
- Setting the relative velocity of each wall.
- Calculating a solution using the segregated solver

Prerequisites: This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have solved or read Tutorial 1. Some steps 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 8.1. 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.

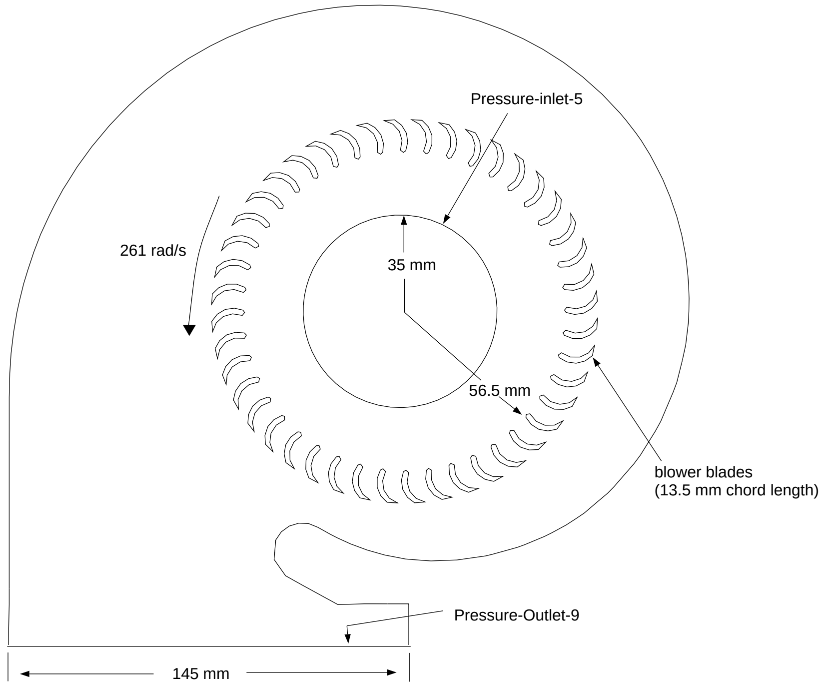


Figure 8.1: Schematic of the Problem

Preparation

1. Copy the file `blower/blower.msh` from the FLUENT documentation CD to your working directory (as described in Tutorial 1).
2. Start the 2D version of FLUENT.

Step 1: Grid

1. Read in the mesh file (`blower.msh`).

   File ⟶ Read ⟶ Case...

2. Check the grid.

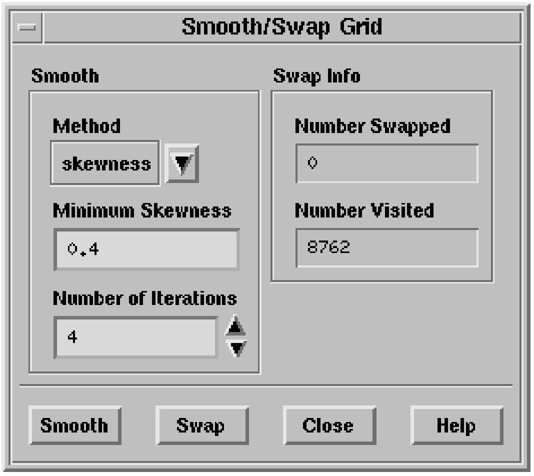
   Grid ⟶ Check

   Note: FLUENT *will perform various checks on the mesh and will report the progress in the console window. Pay particular attention to the reported minimum volume. Make sure this is a positive number.*

3. Smooth and swap the grid.

   Grid ⟶ Smooth/Swap...

   Node smoothing and face swapping will improve the mesh quality. This step is recommended for triangular and tetrahedral meshes.



Method: skewness

Minimum Skewness: 0.4

Number of Iterations: 4

Number Swapped: 0

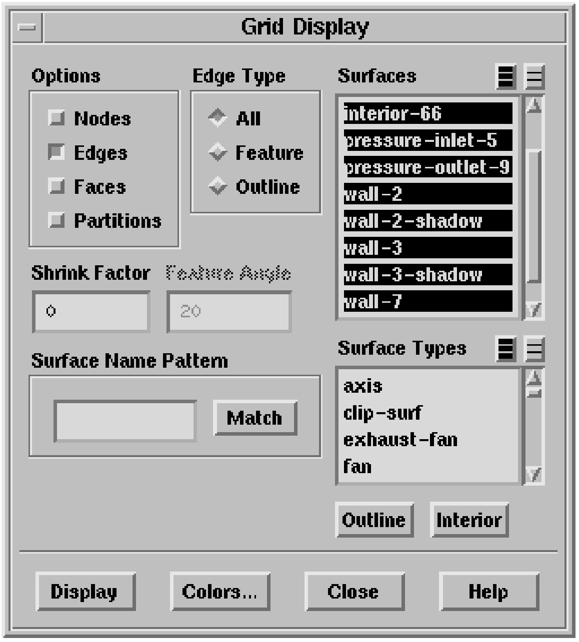
Number Visited: 8762

   (a) Retain the default smoothing parameters and click on Smooth.

   (b) Click on Swap repeatedly until the Number Swapped under Swap Info is zero.

4. Display the mesh (Figure 8.2).

   Display ⟶Grid...



Options: ☐ Nodes ☑ Edges ☐ Faces ☐ Partitions

Edge Type: ● All ○ Feature ○ Outline

Shrink Factor: 0

Feature Angle: 20

Surfaces (selected): interior-66, pressure-inlet-5, pressure-outlet-9, wall-2, wall-2-shadow, wall-3, wall-3-shadow, wall-7

The mesh consists of three fluid zones, fluid-13, fluid-14, *and* fluid-18. *These are reported in the console window 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.*

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.*

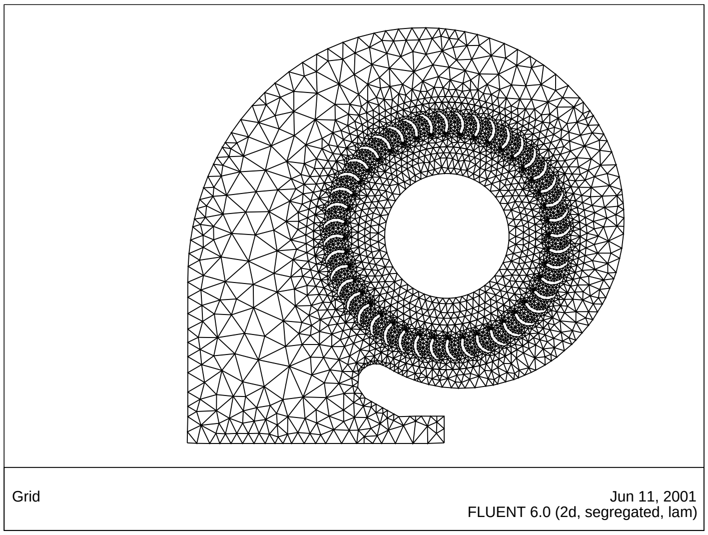
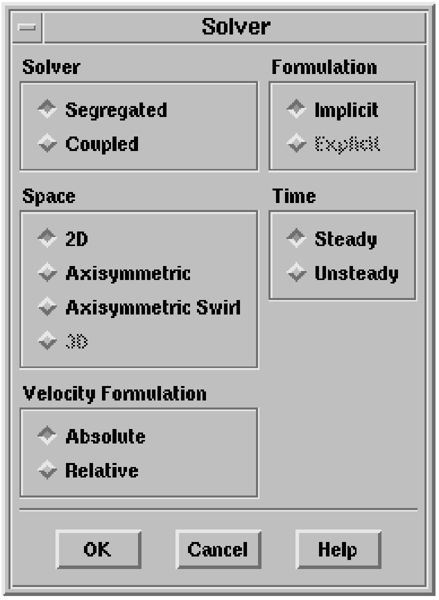


Figure 8.2: Mesh of the 2D Centrifugal Blower

Step 2: Models

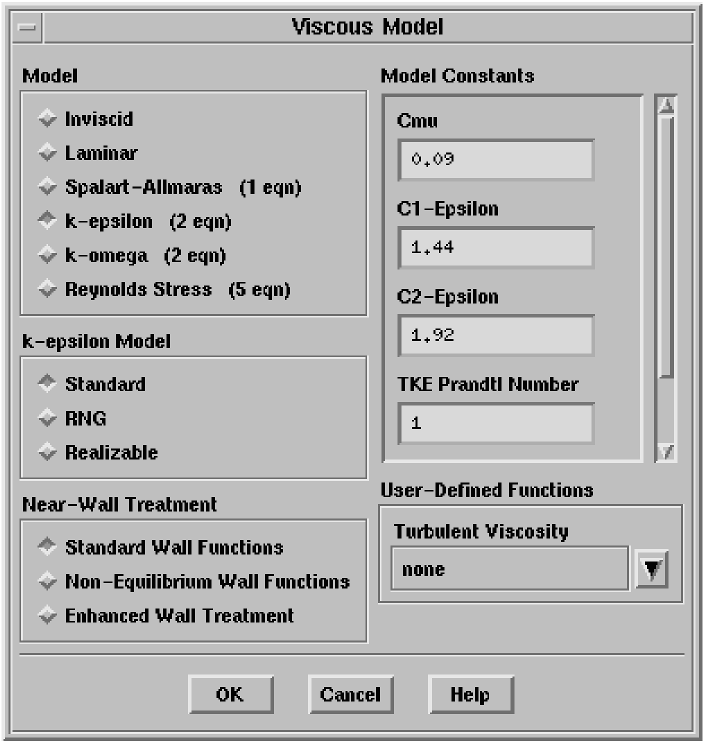
1. Keep the default solver settings.

   Define ⟶ Models ⟶ Solver...



2. Turn on the standard k-ϵ turbulence model.

   Define ⟶ Models ⟶ Viscous...



Model

- ○ Inviscid
- ○ Laminar
- ○ Spalart-Allmaras (1 eqn)
- ● k-epsilon (2 eqn)
- ○ k-omega (2 eqn)
- ○ Reynolds Stress (5 eqn)

k-epsilon Model

- ● Standard
- ○ RNG
- ○ Realizable

Near-Wall Treatment

- ● Standard Wall Functions
- ○ Non-Equilibrium Wall Functions
- ○ Enhanced Wall Treatment

Model Constants

Cmu: 0.09

C1-Epsilon: 1.44

C2-Epsilon: 1.92

TKE Prandtl Number: 1

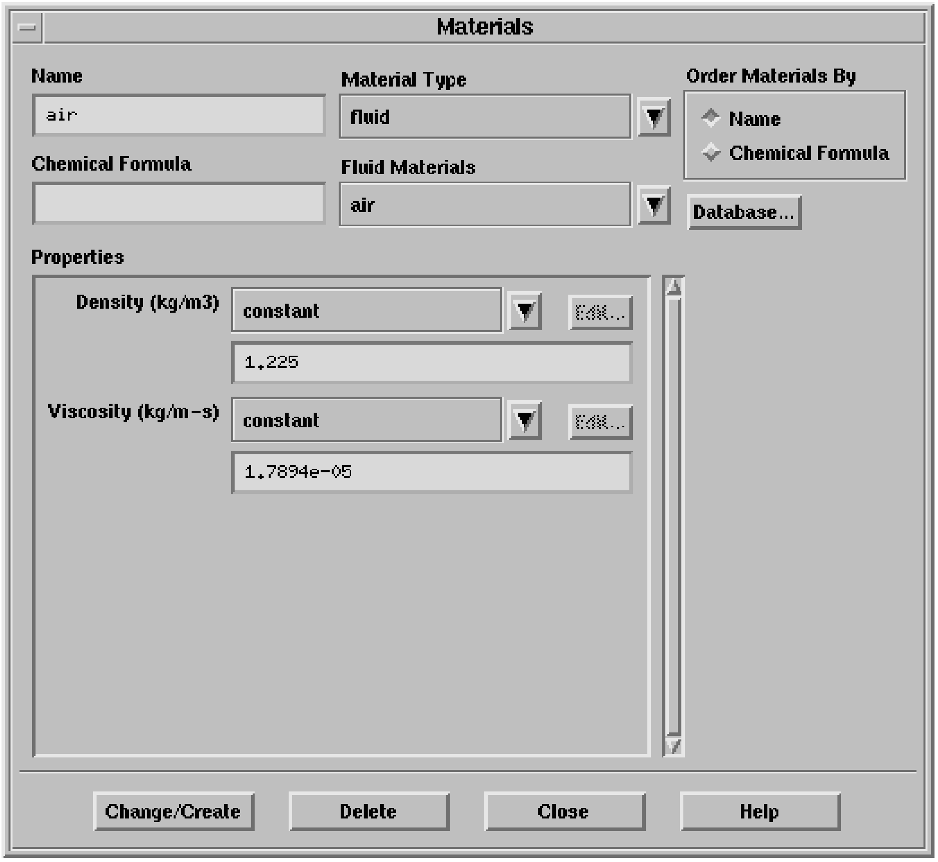
User-Defined Functions

Turbulent Viscosity: none

Step 3: Materials

You will use the default material, air, with its predefined properties, for all fluid zones. No action is required in the panel.

Define ⟶Materials...



Name: air

Chemical Formula:

Material Type: fluid

Fluid Materials: air

Order Materials By: ◆ Name ◇ Chemical Formula

Properties

Density (kg/m3): constant

1.225

Viscosity (kg/m-s): constant

1.7894e-05

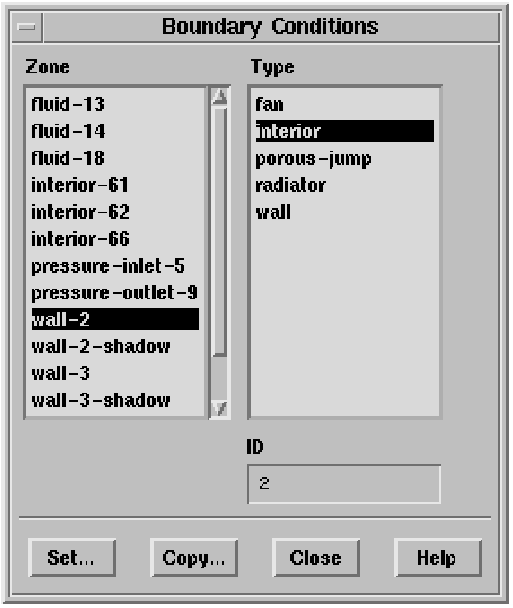
Extra: *If needed, you could modify the fluid properties for air or copy another material from the database. See the "Physical Properties" chapter of the User's Guide for details.*

Step 4: Boundary Conditions

Define ⟶Boundary Conditions...

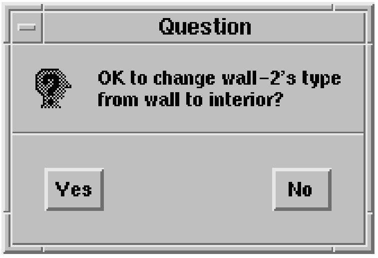
1. Change wall-2 and wall-3 to type interior.

   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.*



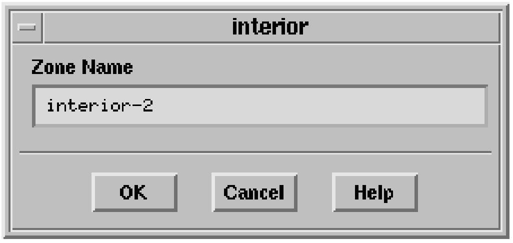
   (a) Select wall-2 in the Zone list and then select interior in the Type list.

   FLUENT *will prompt for confirmation before changing the zone type.*



OK to change wall-2's type from wall to interior?

(b) Click on Yes and FLUENT will fuse wall-2 and wall-shadow-2 together to form interior-2.



Zone Name: interior-2

(c) Click OK to keep the default Zone Name.

(d) Repeat the previous steps to 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.

   Display ⟶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 on the Outline button at the bottom of the panel to select only the outline surfaces of the domain.

   (c) Click on Display.

Only the domain boundaries and interior walls will be displayed.

(d) In the console window, type the commands shown in boxes in the dialog below.

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

```
> display
/display> zone-grid
()
zone id/name(1) [()] 13
zone id/name(2) [()] <CR>
```

The resulting display (Figure 8.3) shows that zone fluid-13 *corresponds to the rotating region.*

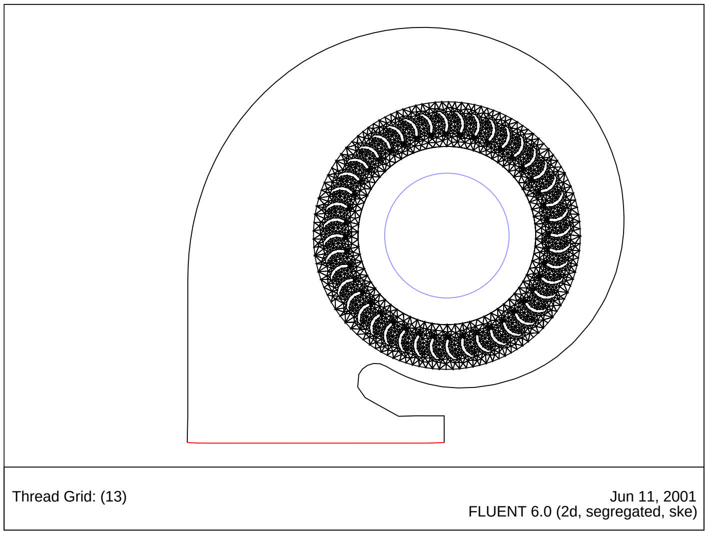


Figure 8.3: Mesh in fluid-13

3. Define a rotational reference frame for fluid-13.

   Define ⟶Boundary Conditions...

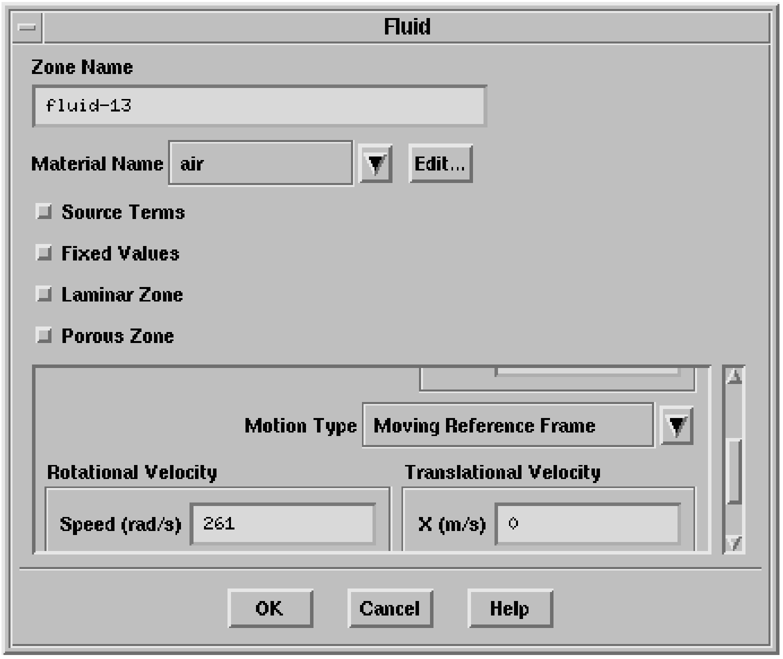
   (a) Keep the Rotation-Axis Origin default setting of `(0,0)`.

   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.

   Hint: *Use the scroll bar to access the* Motion Type *list.*

   (c) Scroll down further, and set the Speed under Rotational Velocity to 261 rad/s.



Fluid

Zone Name: fluid-13

Material Name: air

☐ Source Terms

☐ Fixed Values

☐ Laminar Zone

☐ Porous Zone

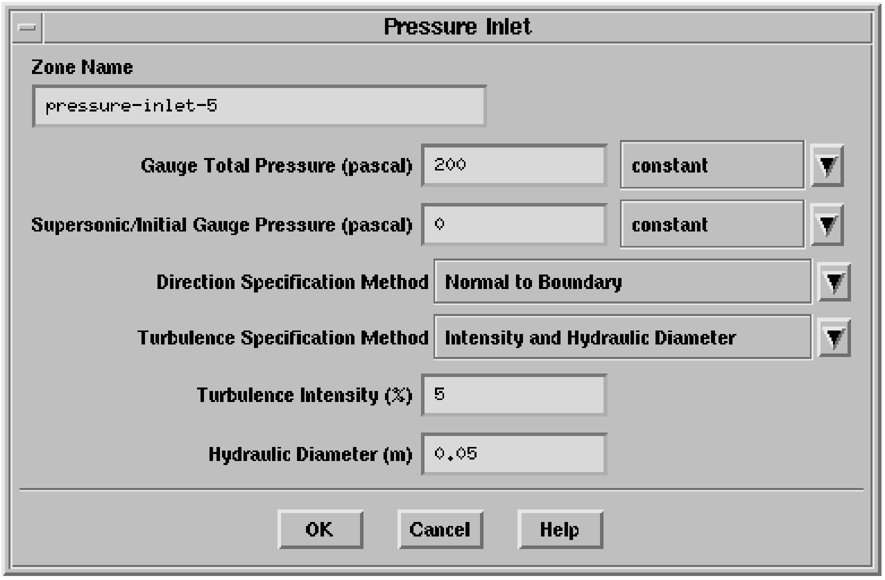
Motion Type: Moving Reference Frame

Rotational Velocity — Speed (rad/s): 261

Translational Velocity — X (m/s): 0

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.*

4. Set the following conditions (see Figure 8.1) for the flow inlet (pressure-inlet-5).



Zone Name: pressure-inlet-5

Gauge Total Pressure (pascal): 200 constant

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Direction Specification Method: Normal to Boundary

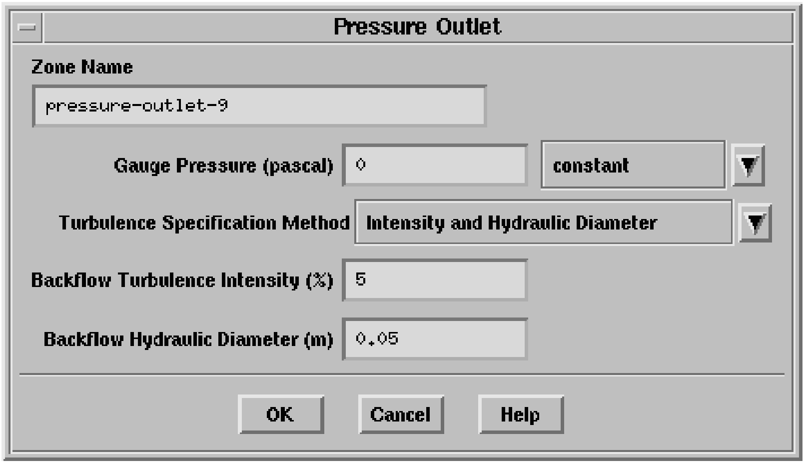
Turbulence Specification Method: Intensity and Hydraulic Diameter

Turbulence Intensity (%): 5

Hydraulic Diameter (m): 0.05

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 the User's Guide for details.*

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



Zone Name: pressure-outlet-9

Gauge Pressure (pascal): 0 constant

Turbulence Specification Method: Intensity and Hydraulic Diameter

Backflow Turbulence Intensity (%): 5

Backflow Hydraulic Diameter (m): 0.05

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.*

6. 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.*

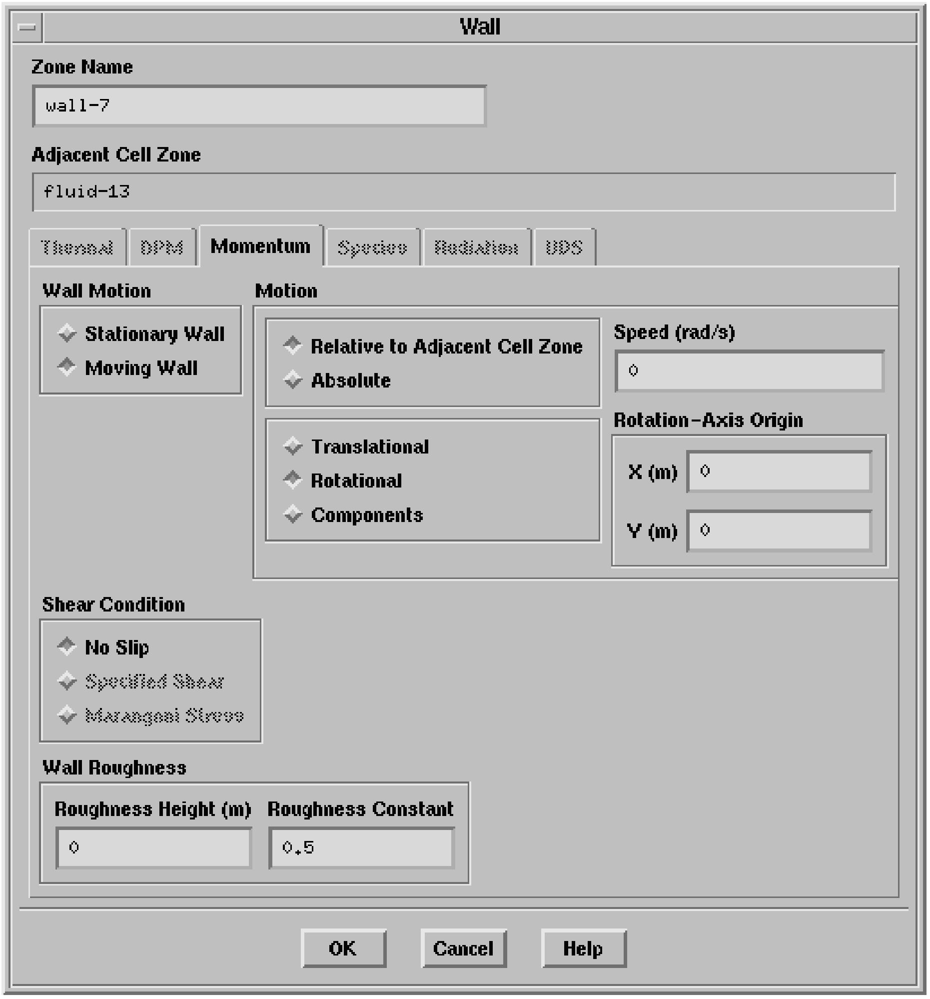
   (a) In the Momentum section of the Wall panel, enable the Moving Wall option.

   The panel will expand to show the wall motion parameters.

   (b) Under Motion, select Relative to Adjacent Cell Zone and Rotational.

   (c) Set the (relative) Speed to 0 rad/s.

   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.*



Zone Name: wall-7

Adjacent Cell Zone: fluid-13

Wall Motion
○ Stationary Wall
● Moving Wall

Motion
● Relative to Adjacent Cell Zone
○ Absolute

○ Translational
● Rotational
○ Components

Speed (rad/s): 0

Rotation-Axis Origin
X (m): 0
Y (m): 0

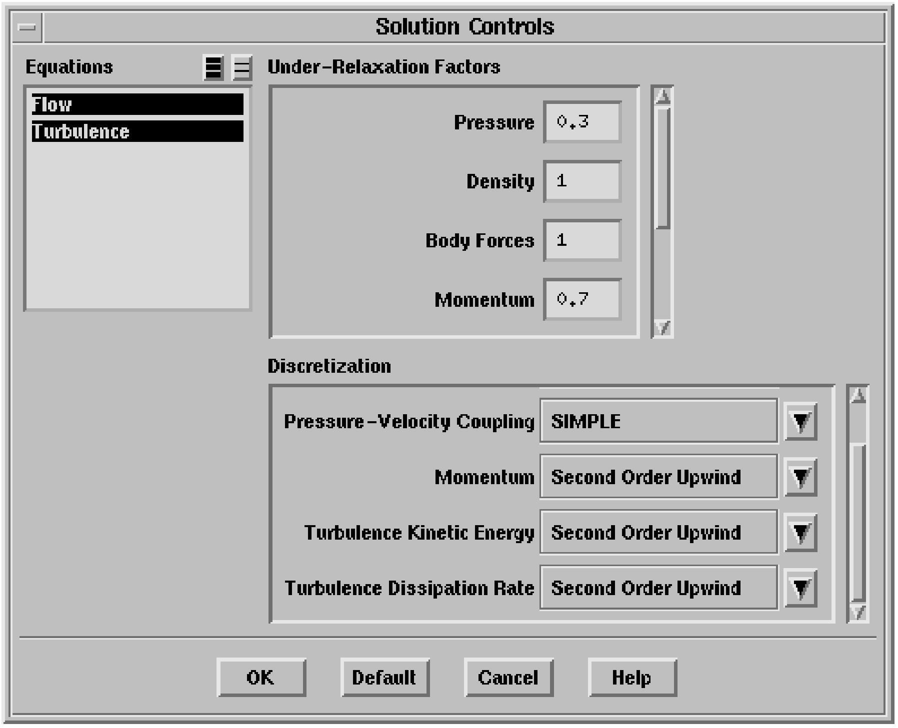
Shear Condition
● No Slip
○ Specified Shear
○ Marangoni Stress

Wall Roughness
Roughness Height (m): 0
Roughness Constant: 0.5

Step 5: Solution

1. Choose the second-order discretization scheme for the governing equations.

   Solve ⟶ Controls ⟶ Solution...



Solution Controls

Equations

Flow

Turbulence

Under-Relaxation Factors

Pressure 0.3

Density 1

Body Forces 1

Momentum 0.7

Discretization

Pressure-Velocity Coupling SIMPLE

Momentum Second Order Upwind

Turbulence Kinetic Energy Second Order Upwind

Turbulence Dissipation Rate Second Order Upwind

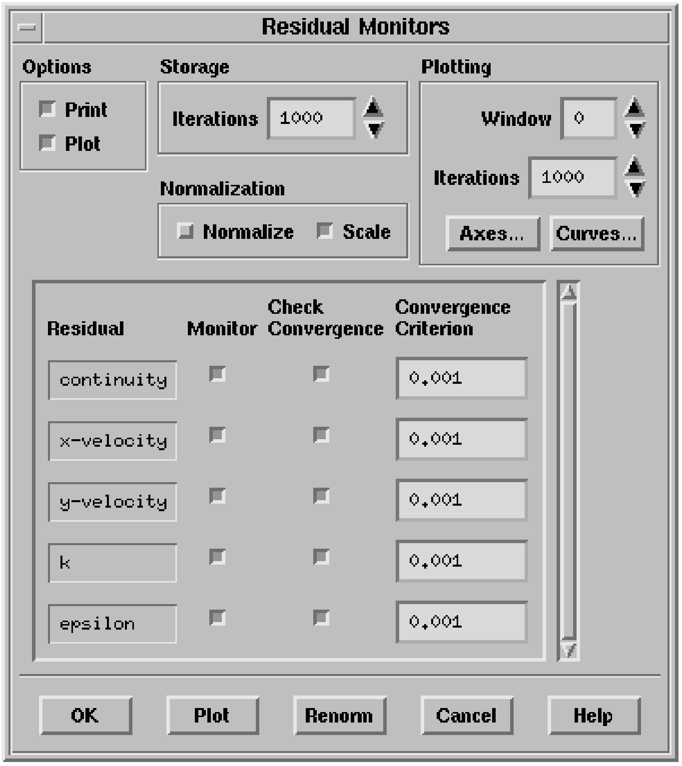
   (a) In the drop-down lists next to Momentum, Turbulence Kinetic Energy, and Turbulence Dissipation Rate, select Second Order Upwind.

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

   (b) Keep the default parameters for all other solution controls.

2. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residual...



Options: ☑ Print ☑ Plot

Storage: Iterations 1000

Normalization: ☐ Normalize ☑ Scale

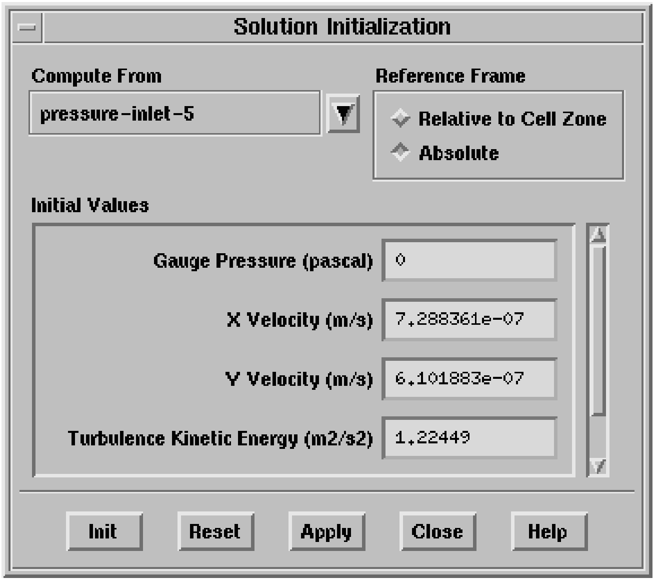
Plotting: Window 0, Iterations 1000

Residual	Monitor	Check Convergence	Convergence Criterion
continuity	☑	☑	0.001
x-velocity	☑	☑	0.001
y-velocity	☑	☑	0.001
k	☑	☑	0.001
epsilon	☑	☑	0.001

   (a) Select Plot under Options, and click on OK.

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

   Solve ⟶ Initialize ⟶ Initialize...



Solution Initialization

Compute From: pressure-inlet-5

Reference Frame: Relative to Cell Zone / Absolute

Initial Values

Gauge Pressure (pascal): 0

X Velocity (m/s): 7.288361e-07

Y Velocity (m/s): 6.101883e-07

Turbulence Kinetic Energy (m2/s2): 1.22449

   (a) Select pressure-inlet-5 in the Compute From drop-down list.
   (b) Select Absolute under Reference Frame.
   (c) Click Init to initialize the solution.

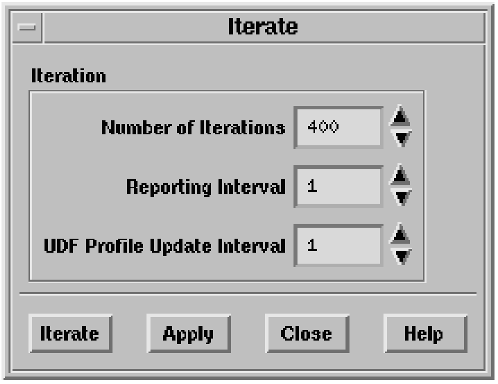
   Note: *In this tutorial, you chose an* Absolute *reference frame for initializing the solution. In certain cases,* Relative to Cell Zone *may help the solution converge faster. See the User's Guide for guidelines.*

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

   File ⟶ Write ⟶ Case...

5. Start the calculation by requesting 400 iterations.

   Solve ⟶Iterate...



Number of Iterations 400

Reporting Interval 1

UDF Profile Update Interval 1

During the calculation, FLUENT *will report that there is reversed flow occuring 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).

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

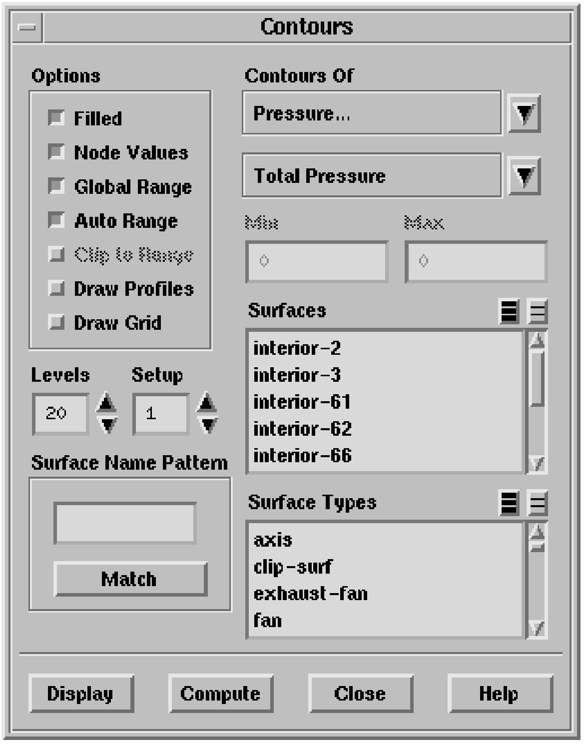
   File ⟶ Write ⟶Case & Data...

   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 8.4).

   Display ⟶Contours...



Contours

Options

☑ Filled

☑ Node Values

☑ Global Range

☑ Auto Range

☐ Clip to Range

☐ Draw Profiles

☐ Draw Grid

Contours Of: Pressure...

Total Pressure

Levels: 20

Setup: 1

   (a) Select Pressure... and Total Pressure in the Contours Of drop-down lists.

   (b) Select Filled under Options.

   (c) Click Display.

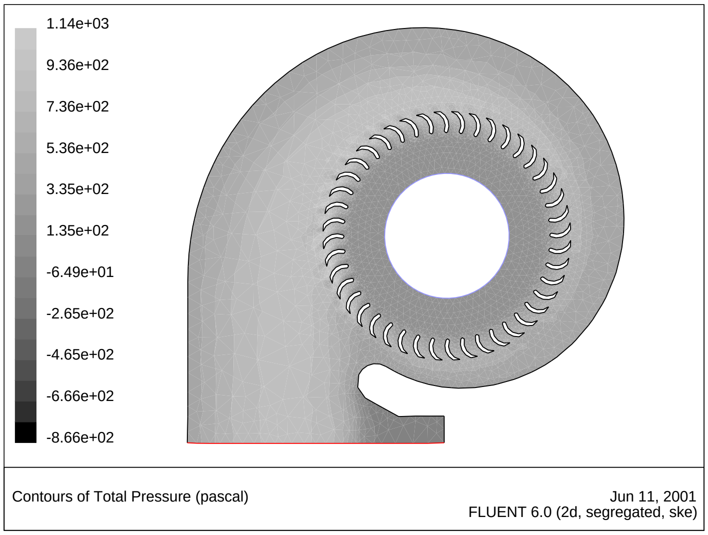
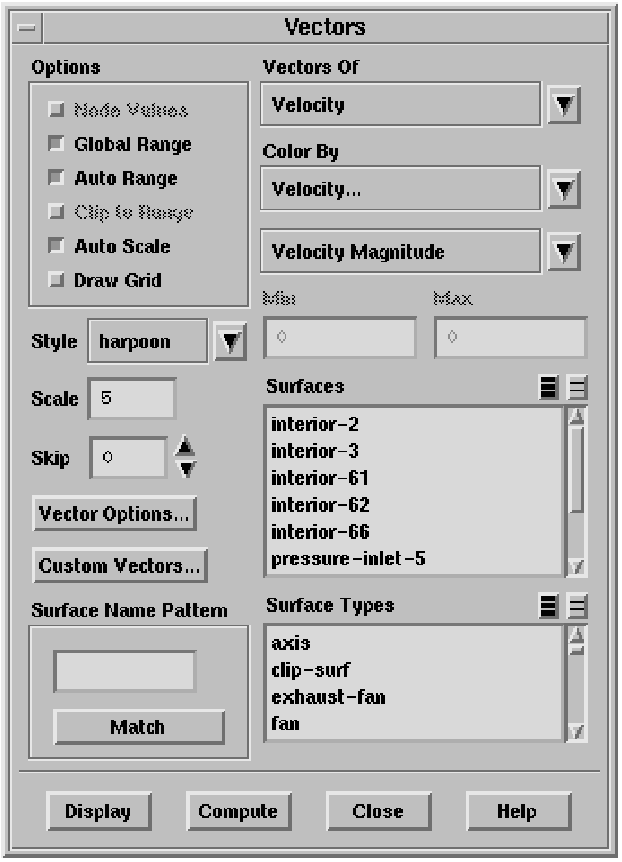


Figure 8.4: Contours of Total Pressure

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

2. Display velocity vectors (Figure 8.5).

   Display →Vectors...



Options

☐ Node Values
☑ Global Range
☑ Auto Range
☐ Clip to Range
☑ Auto Scale
☐ Draw Grid

Vectors Of: Velocity

Color By: Velocity...

Velocity Magnitude

Min: 0

Max: 0

Style: harpoon

Scale: 5

Skip: 0

Surfaces

interior-2
interior-3
interior-61
interior-62
interior-66
pressure-inlet-5

Surface Types

axis
clip-surf
exhaust-fan
fan

   (a) Set the Scale factor to 5.

   (b) Click on Display to view the vectors.

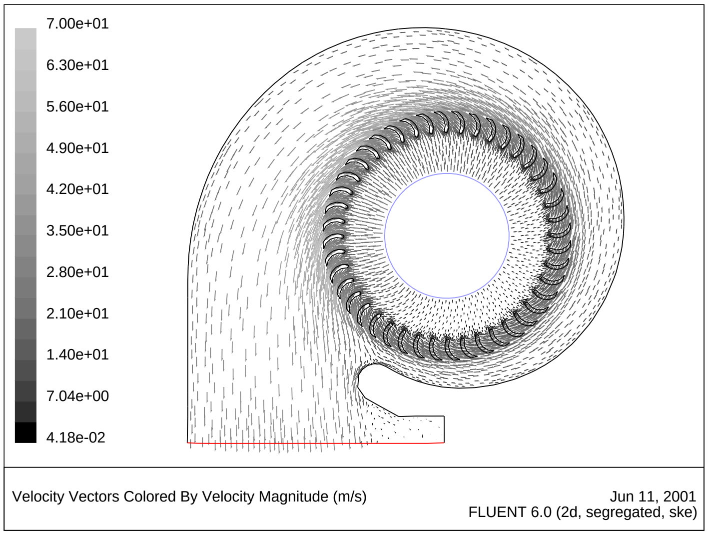
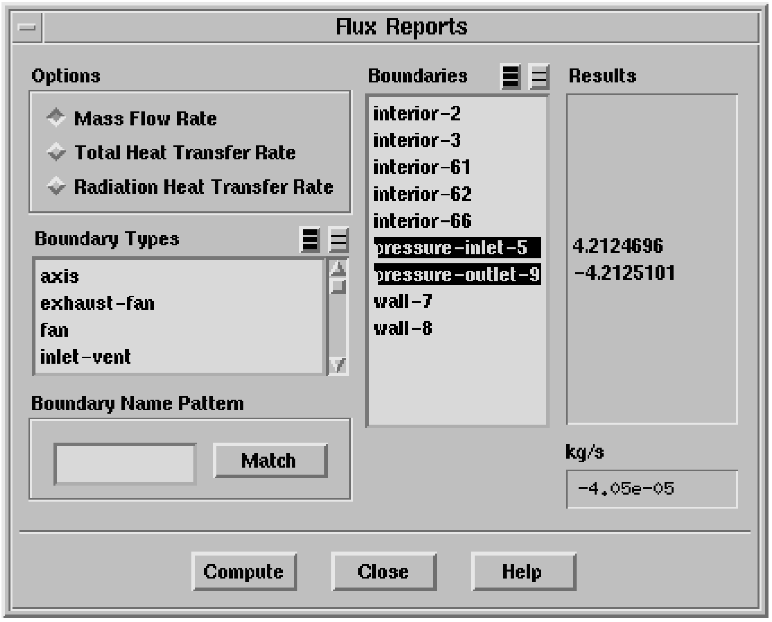


Figure 8.5: Velocity Vectors

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...



Flux Reports

Options: Mass Flow Rate

Boundaries: pressure-inlet-5, pressure-outlet-9

Results:

```
4.2124696
-4.2125101
```

kg/s: -4.05e-05

   (a) Keep the Mass Flow Rate setting under Options.

   (b) Select pressure-inlet-5 and pressure-outlet-9 in the Boundaries list.

   (c) Click on 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.*

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.