
Tutorial 10. Using Sliding Meshes

Introduction: In this tutorial, the sliding mesh capability of FLUENT is used to predict the time-dependent flow through a two-dimensional rotor-stator blade row. The time-varying rotor-stator interaction is modeled by allowing the mesh associated with the moving rotor to translate (slide) relative to the stationary mesh associated with the stator blade.

In this tutorial you will learn how to:

- Merge two meshes into a single mesh, using `tmerge`
- Define boundary conditions and create grid-interface planes for sliding mesh simulations
- Calculate a steady-state solution (using the coupled explicit solver) as an initial guess for a transient flow prediction
- Calculate a transient solution using the second-order implicit unsteady formulation and the coupled explicit solver
- Monitor solution history for time-dependent parameters
- Postprocess and store transient data sets, using the automatic procedures available in FLUENT

Prerequisites: This tutorial assumes that you are familiar with the basic menu structure and solution procedure used by FLUENT and that you have solved Tutorial 1.

Problem Description: The rotor-stator geometry considered in this tutorial is shown in Figure 10.1. The geometry consists of a planar slice through the rotor and stator blades, extracted by unrolling a plane of constant radius ($R = 0.686$ m) in an axial flow turbomachine. The speed of rotation, 410 RPM, yields a linear velocity of the rotor, $R\Omega$, equal to 29.4 m/s, as indicated in the figure. The

Using Sliding Meshes

fluid, assumed to be air, enters the stator row at the specified total pressure and temperature and exits the rotor at the specified exit static pressure. The inlet Mach number is 0.07 and the flow will be treated as compressible.

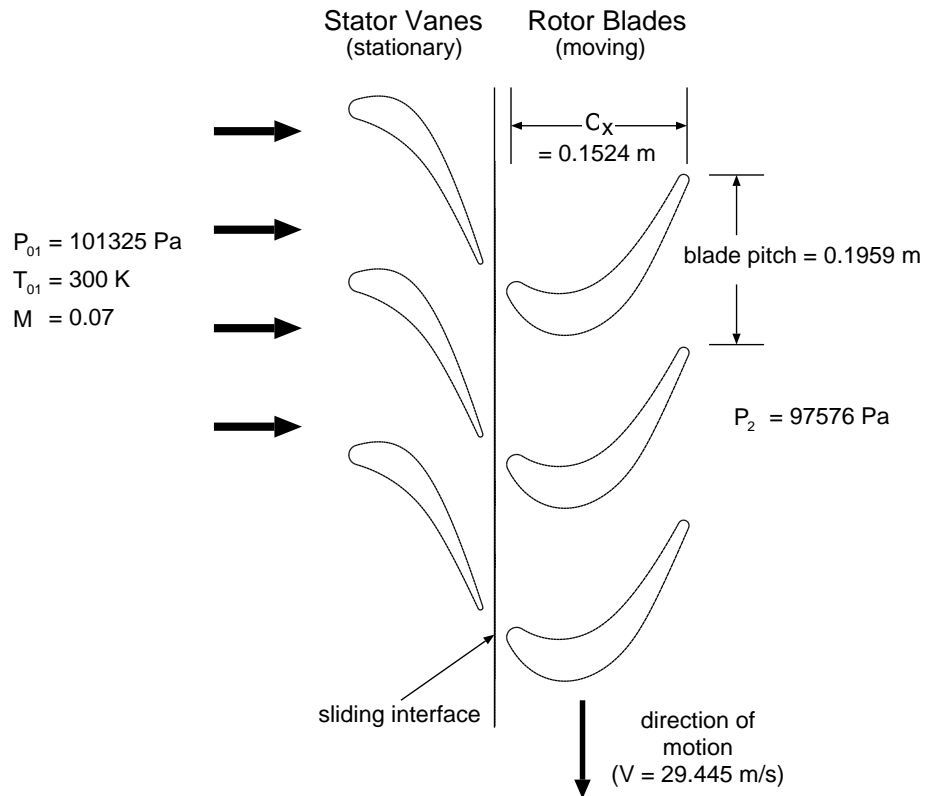


Figure 10.1: Rotor-Stator Problem Description

Preparation

1. Copy the files `slide/rotor.msh` and `slide/stator.msh` from the FLUENT documentation CD to your working directory (as described in Tutorial 1).

Note: *The geometries of the rotor and stator flow domains have been meshed separately. This is the usual procedure when the sliding mesh capability is used: separate mesh files are created for the sliding and stationary mesh regions. This ensures that the sliding interface between the two regions is defined by two separate boundary zones which share no common nodes. The two separate mesh files must be merged prior to reading them into FLUENT, as detailed in Step 1, below.*

Step 1: Merging the Mesh Files

1. Start `tmerge` by typing utility `tmerge -2d` at the system prompt.
2. Provide the mesh file names, `rotor.msh` and `stator.msh`, as prompted. Provide scaling factors of 1 and translations and rotations of zero for each mesh file. Save the new merged mesh file as `slide.msh`.

```
Append 2D grid files.
tmerge2D Fluent Inc, Version 2.1.8

Enter name of grid file (ENTER to continue) : rotor.msh
x,y scaling factor, eg. 1 1                : 1 1
x,y translation, eg. 0 1                   : 0 0
rotation angle (deg), eg. 45               : 0

Enter name of grid file (ENTER to continue) : stator.msh
x,y scaling factor, eg. 1 1                : 1 1
x,y translation, eg. 0 1                   : 0 0
rotation angle (deg), eg. 45               : 0

Enter name of grid file (ENTER to continue) : <ENTER>

Enter name of output file                   : slide.msh
```

! *The mesh files must be read into `tmerge` in this order for the tutorial to run as written. Otherwise, zone names and numbers will be assigned differently when the files are merged together. In general, however, you can specify files to be read into `tmerge` in any order.*

Step 2: Grid

1. Start the 2D version of FLUENT.
2. Read in the mesh file `slide.msh`.

File → **Read** → Case...

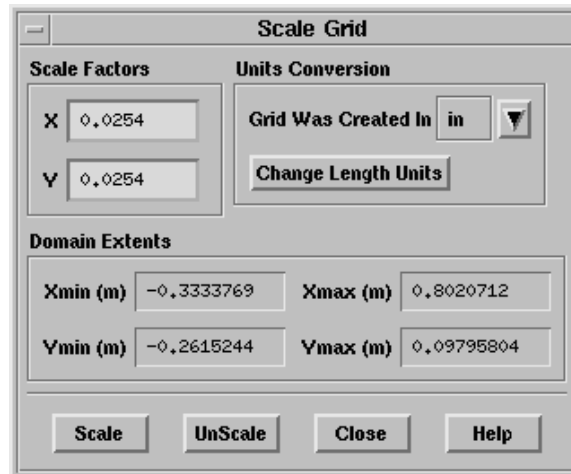
3. Check the grid.

Grid → Check

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

4. Scale the grid.

Grid → Scale...

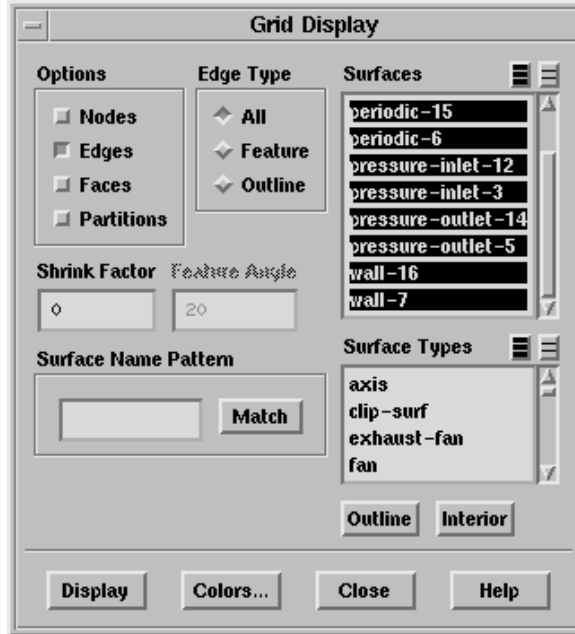


- (a) Select in in the Units Conversion drop-down list to complete the phrase Grid Was Created In in (inches).
- (b) Click on Scale to scale the grid.

The final domain extents should appear as in the panel above.

5. Display the grid (Figure 10.2).

Display → Grid...



Note: You can use the mouse probe button (right button, by default) to find out the boundary zone labels. As annotated in Figure 10.3, the upstream boundary is a pressure inlet, the downstream boundary is a pressure outlet, and the lateral top and bottom boundaries are periodic. The stator blade and stator-side fluid are identified as wall-7 and fluid-9. The rotor blade and rotor-side fluid are wall-16 and fluid-18. (Your mouse probe will report the fluid regions as interior-8 and interior-17 zones, for the stator and rotor sides, respectively. These are the face zones associated with the fluid regions.)

To determine which fluid zone is the stator-side fluid, you can create the fluid-9 and fluid-18 display surfaces using the Zone Surface panel. Then, display the zones (one at a time) using the Grid Display panel.

If you wish to annotate your own graphics display, you can use the Annotate panel.

Display → Annotate...

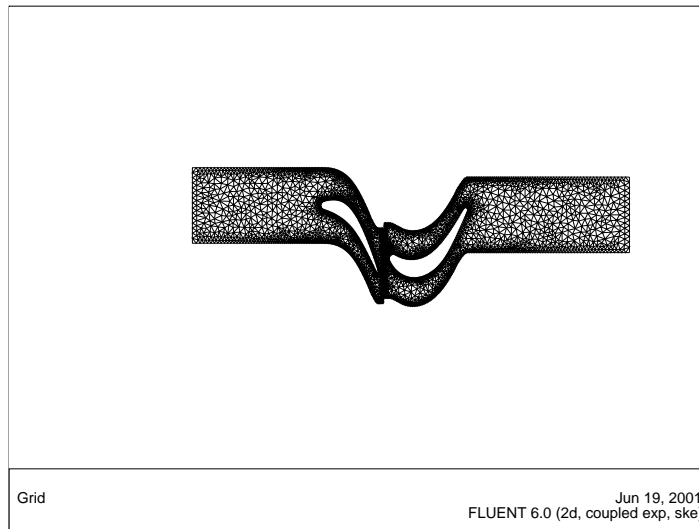


Figure 10.2: Rotor-Stator Mesh Display

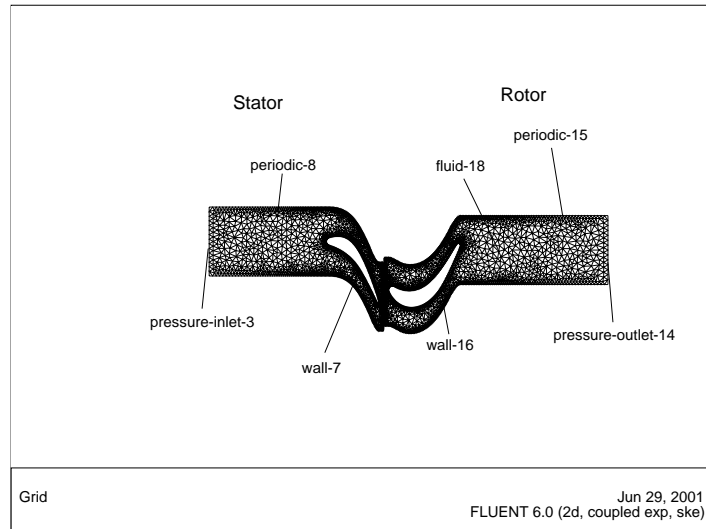
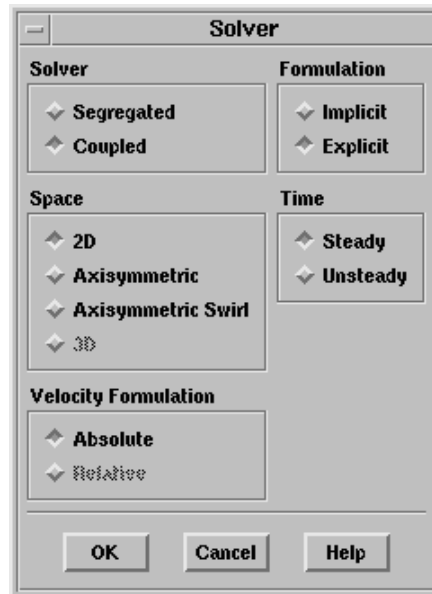


Figure 10.3: Annotated Mesh

Step 3: Models

1. Select the coupled explicit solver.

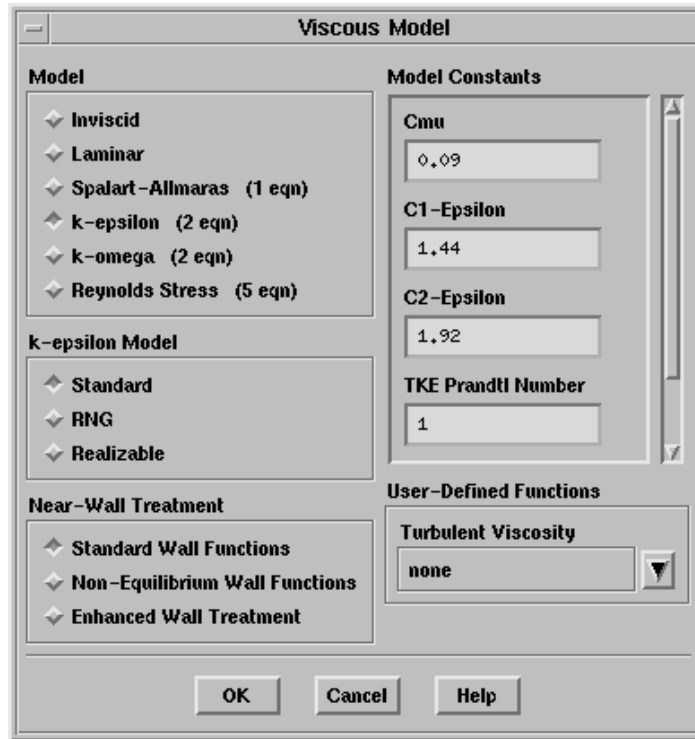
Define → Models → Solver...



Note: *Initially, you will solve for the steady flow through the blade passage. Later, after obtaining the steady flow as the starting point for the transient calculation, you will revisit this panel to turn on time-dependent flow.*

2. Enable the standard k - ϵ turbulence model.

Define → Models → Viscous...



Note: The Reynolds number of the flow is about 10^5 , and the flow will be treated as fully turbulent.

Step 4: Materials

1. Select air (the default material) as the fluid material, and use the ideal-gas law to compute density. Retain the default values for all other properties.

Define → Materials...

Materials

Name: air

Material Type: fluid

Order Materials By: Name, Chemical Formula

Chemical Formula:

Fluid Materials: air

Database...

Properties

Density (kg/m³): ideal-gas [Edit...]

Cp (j/kg-k): constant [Edit...]
1006.43

Thermal Conductivity (w/m-k): constant [Edit...]
0.0242

Viscosity (kg/m-s): constant [Edit...]
1.7894e-05

Molecular Weight (kg/kgmol): constant [Edit...]

Change/Create Delete Close Help

! Don't forget to click the Change/Create button after selecting ideal-gas in the drop-down list for Density.

Step 5: Operating Conditions

1. Set the operating pressure to 0 Pa.

Define → Operating Conditions...



Here, the operating pressure is set to zero and boundary condition inputs for pressure will be defined in terms of absolute pressures. Boundary condition inputs should always be relative to the value used for operating pressure.

Step 6: Boundary Conditions

Define → Boundary Conditions...

1. Set the conditions for the upstream boundary (pressure-inlet-3).

Pressure Inlet

Zone Name
pressure-inlet

Gauge Total Pressure (pascal) 101325 constant ▼

Supersonic/Initial Gauge Pressure (pascal) 100978,2 constant ▼

Total Temperature (k) 300 constant ▼

Direction Specification Method Direction Vector ▼

X-Component of Flow Direction 1 constant ▼

Y-Component of Flow Direction 0 constant ▼

Turbulence Specification Method Intensity and Hydraulic Diameter ▼

Turbulence Intensity (%) 5

Hydraulic Diameter (m) 0,1959

OK Cancel Help

- (a) Change the Zone Name from pressure-inlet-3 to pressure-inlet.
- (b) Set the Gauge Total Pressure to 101325 Pa.

Using Sliding Meshes

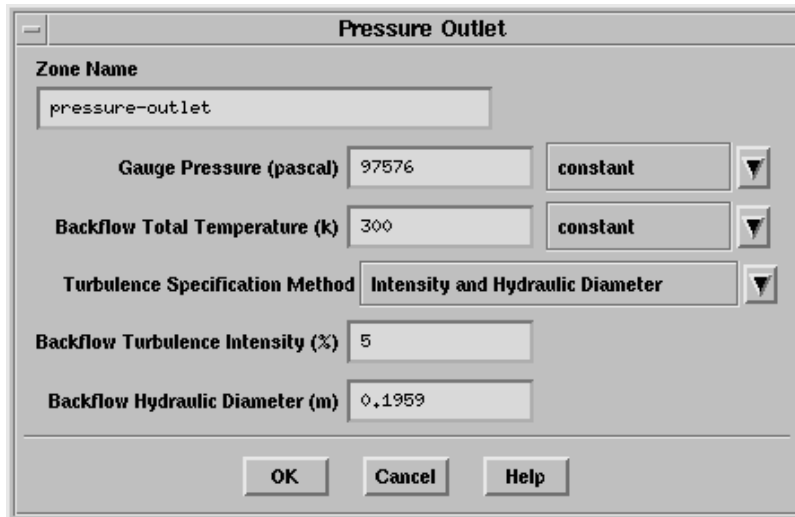
- (c) Set the Supersonic/Initial Gauge Pressure to 100978.2 Pa.

The inlet static pressure estimate is computed from the assumed inlet total pressure and Mach number (see Figure 10.1).

- (d) Set the Total Temperature to 300 K.
- (e) In the Direction Specification Method drop-down list, select Direction Vector.
- (f) In the Turbulence Specification Method drop-down list, select Intensity and Hydraulic Diameter.
- (g) Set the Turbulence Intensity to 5%, and the Hydraulic Diameter to 0.1959 m.

The inlet turbulence length scale will be computed using the blade pitch as an equivalent “hydraulic diameter”.

2. Set the conditions for the exit plane boundary (pressure-outlet-14).



- (a) Change the Zone Name from pressure-outlet-14 to pressure-outlet.
- (b) Set the Gauge Pressure to 97576 Pa.
- (c) Set the Backflow Total Temperature to 300 K.
- (d) In the Turbulence Specification Method drop-down list, select Intensity and Hydraulic Diameter.
- (e) Set the Turbulence Intensity to 5%, and the Hydraulic Diameter to 0.1959 m.

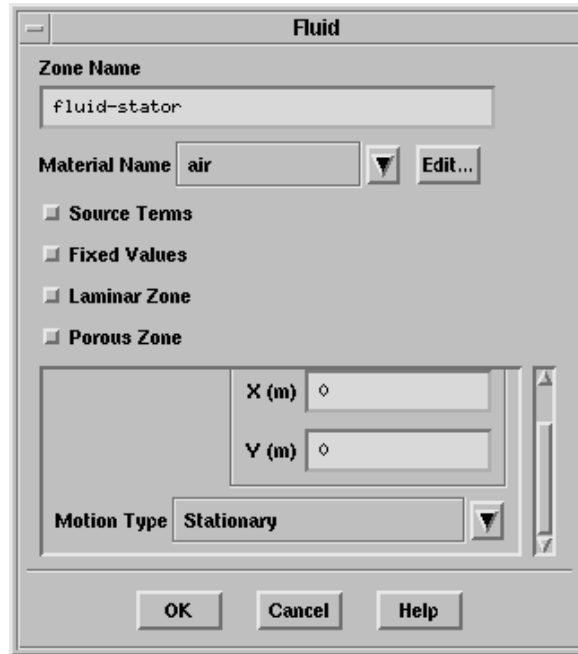
Note: *The temperature and turbulence conditions you input at the pressure outlet will be used only if flow enters the domain through this boundary. You can set them equal to the inlet values, as no flow reversal is expected at the domain exit in this problem. In general, however, it is important to set reasonable values for these downstream scalar values, in case flow reversal occurs at some point during the calculation.*

3. Keep the default Momentum boundary conditions for the stator blades (wall-7) and the rotor blades (wall-16).



The velocity of a “non-moving” wall is assumed to match that of the adjacent mesh region, yielding a no-slip condition in the reference frame of the mesh. Thus, FLUENT will assume that the stator blade is stationary in the non-moving reference frame of the stator mesh. Similarly, FLUENT will assume that the rotor blade is moving at the grid speed in the sliding rotor region. Therefore, you will not modify the wall velocity of the rotor (wall-16), even though the rotor is moving. The default setting of a non-moving wall is correct and implies zero velocity in the moving reference frame of the sliding region. (The motion of the mesh region will be defined as a boundary condition for the fluid zone, below.)

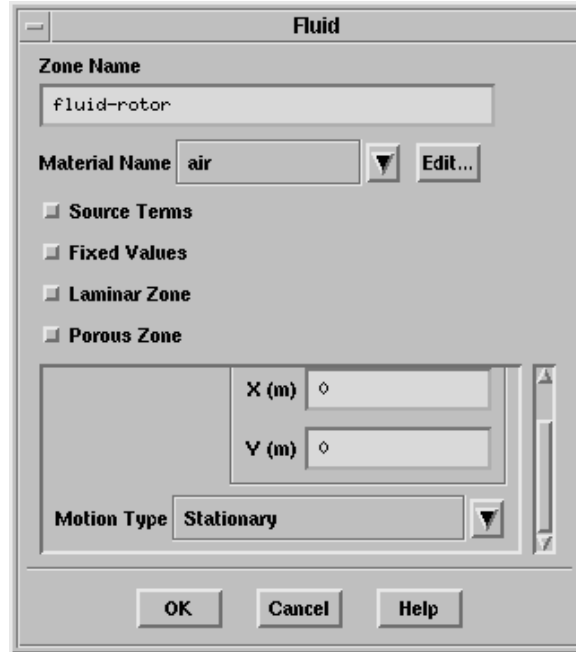
4. Set the conditions for the stator-side fluid (fluid-9).



- (a) Change the Zone Name from fluid-9 to fluid-stator.
- (b) Keep the default selection of air as the Material Name, and the Motion Type as Stationary.

Hint: Use the scroll bar to access the Motion Type field.

5. Set the conditions for the rotor-side fluid (fluid-18).



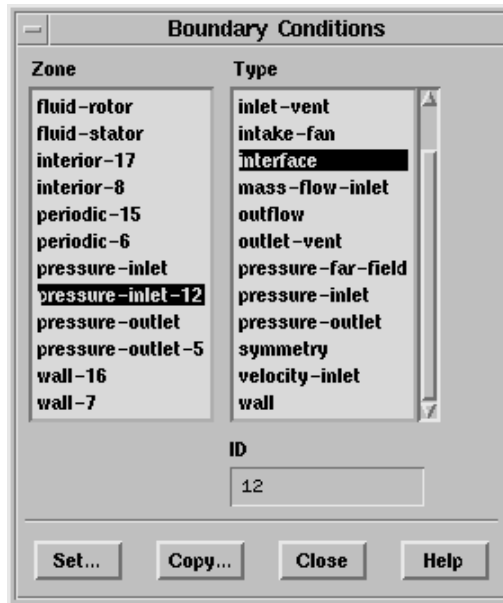
- (a) Change the Zone Name from fluid-18 to fluid-rotor.
- (b) Keep the default selection of air as the Material Name, and the Motion Type as Stationary.

Later, after solving the steady flow through the non-moving rotor passage, you will return to this panel to specify that the rotor zone is sliding, as illustrated in Figure 10.1.

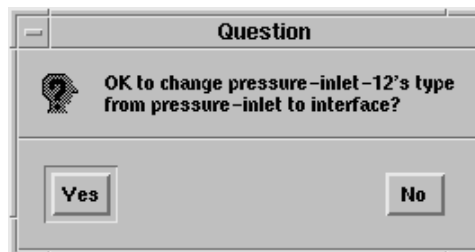
6. Define the zones on the sliding boundary as interface zones.

The sliding grid interface contains two boundary zones: pressure-inlet-12 and pressure-outlet-5. pressure-inlet-12 is the upstream boundary of the rotor-side fluid region, and pressure-outlet-5 is the downstream boundary of the stator-side fluid region. These boundaries were defined as the upstream and downstream boundaries of the original mesh files, rotor.msh and stator.msh, and must be re-defined as interface boundary types for the merged mesh.

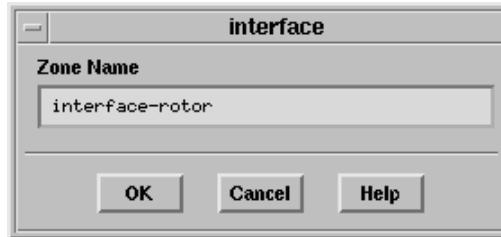
- (a) Select pressure-inlet-12 in the Zone list and choose interface as the new Type.



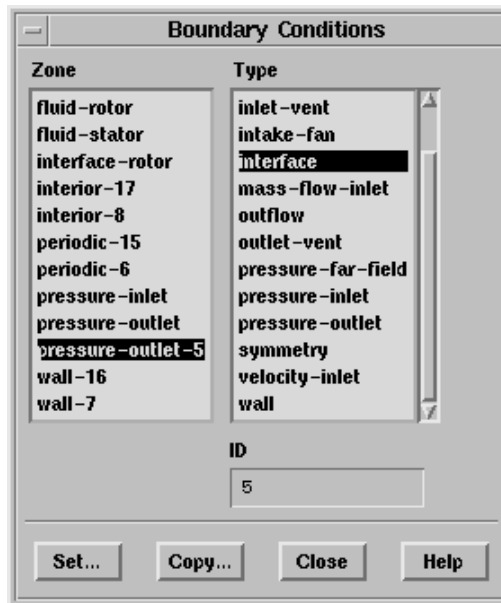
- (b) Confirm that it is OK to change the boundary type.



(c) Change the Zone Name to interface-rotor.



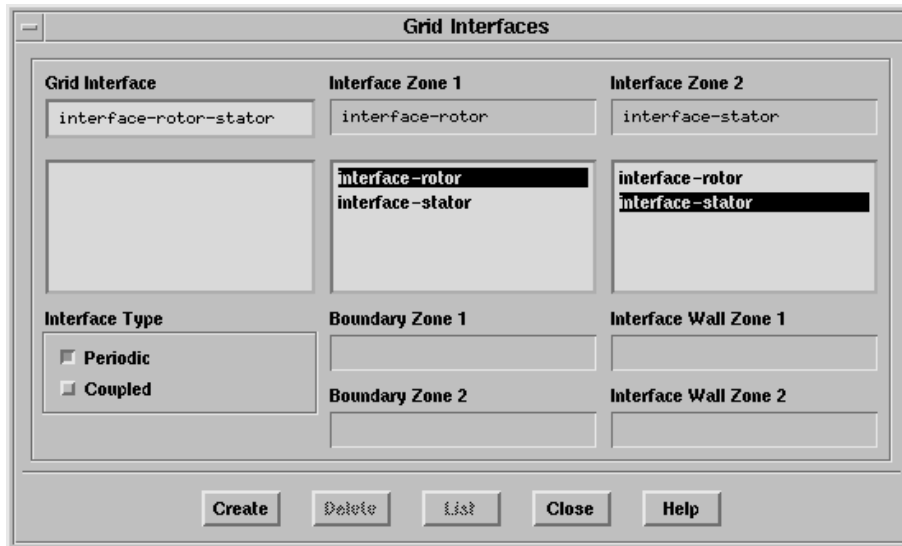
(d) Repeat this procedure to convert pressure-outlet-5 to an interface boundary named interface-stator.



Step 7: Grid Interfaces

In this step, you will create a periodic grid interface between the rotor and stator mesh regions.

Define → Grid Interfaces...



1. Select interface-rotor in the Interface Zone 1 list.

Note: *In general, when one interface zone is smaller than the other, it is recommended that you choose the smaller zone as Interface Zone 1. In this case, however, since both zones are the same size, the order is not significant.*

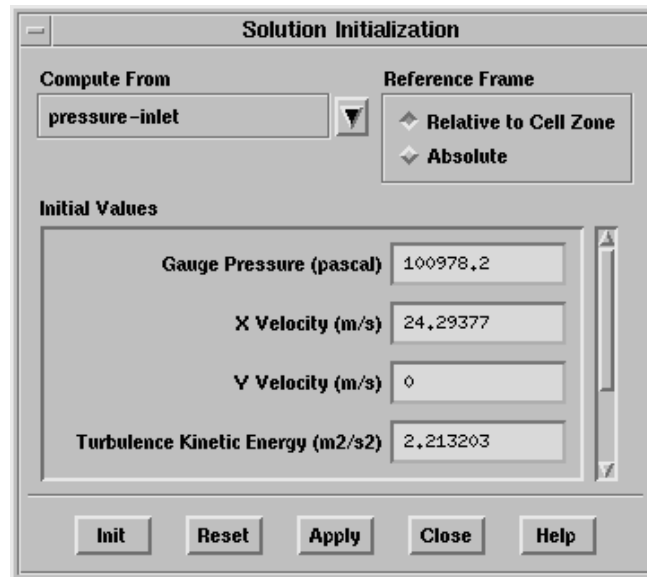
2. Select interface-stator in the Interface Zone 2 list.
3. Enter the name interface-rotor-stator under Grid Interface.
4. Select Periodic under Interface Type, and click Create.

The interface between the sliding and non-sliding zones will be treated as periodic where the two zones are non-overlapping.

Step 8: Solution: Steady Flow with Non-Moving Rotor

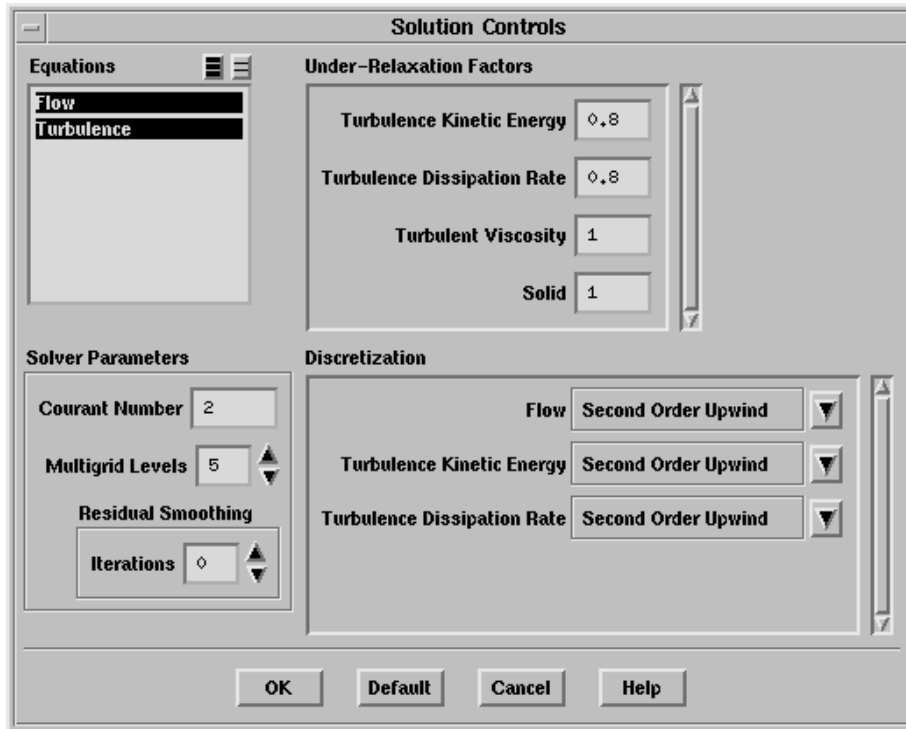
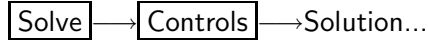
1. Initialize the solution for steady flow.

Solve → Initialize → Initialize...



- (a) Select pressure-inlet in the Compute From drop-down list.
- (b) Click Init, and Close the panel.

2. Set the solution parameters.



(a) Set the Courant Number to 2.

Setting the Courant number to 2.0 promotes rapid convergence for the steady flow simulation. (For more information on the Courant number, see Step 9: Enable Time Dependence and Sliding Rotor Motion).

(b) Change Multigrid Levels to 5.

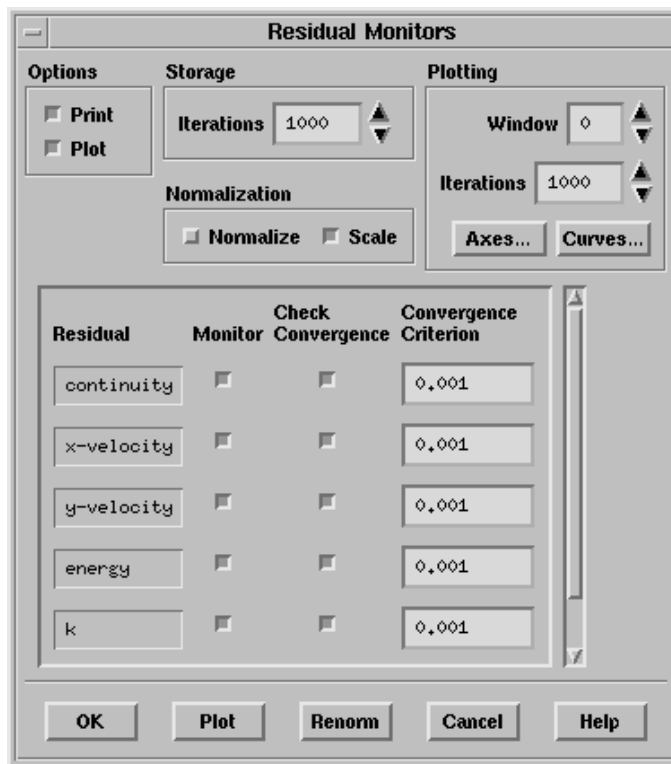
Five levels of multigrid enable boundary condition information to propagate rapidly across the solution domain.

Using Sliding Meshes

- (c) Under Discretization, select Second Order Upwind for Turbulence Kinetic Energy and Turbulence Dissipation Rate.
Second-order discretization provides optimum accuracy.

3. Enable the plotting of residuals.

Solve → Monitors → Residual...

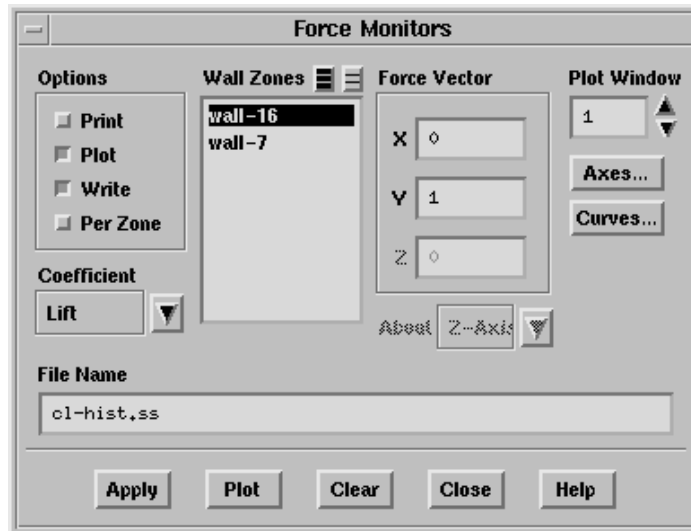


- (a) Select Plot under Options, and click OK.

4. Enable the monitoring of the lift force on the rotor blade (wall-16).

Note: *Monitoring forces on the rotor blade provides a good measure of convergence during the initial steady-state flow prediction. Here, you will request dynamic plotting of lift as the solution proceeds. In addition, you will write the lift information to a file, `cl-hist.ss`. You could choose to monitor any other variable (e.g., mass flow rate), including a custom field function.*

Solve → Monitors → Force...

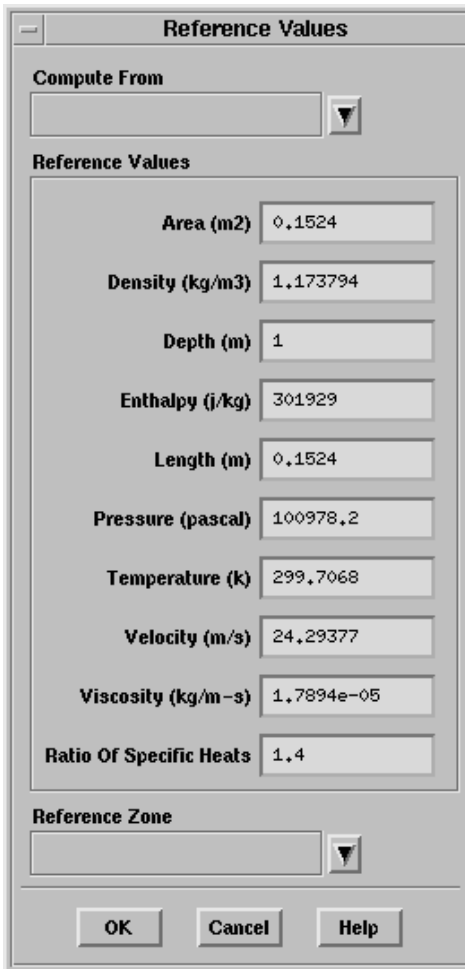


- (a) Select Lift in the Coefficient drop-down list.
- (b) In the Wall Zones list, select wall-16 (the rotor).
- (c) Enable the Plot and Write options.
- (d) In the File Name field, enter `cl-hist.ss`.
- (e) Keep the Plot Window set to 1.
- (f) Click Apply, and Close the panel.

Using Sliding Meshes

5. Set the reference values to be used in the lift coefficient calculation.

Report → Reference Values...



The image shows a 'Reference Values' dialog box with the following fields and values:

Parameter	Value
Area (m ²)	0.1524
Density (kg/m ³)	1.173794
Depth (m)	1
Enthalpy (J/kg)	301929
Length (m)	0.1524
Pressure (pascal)	100978.2
Temperature (K)	299.7068
Velocity (m/s)	24.29377
Viscosity (kg/m-s)	1.7894e-05
Ratio Of Specific Heats	1.4

At the bottom of the dialog box are three buttons: **OK**, **Cancel**, and **Help**.

- (a) Select pressure-inlet in the Compute From drop-down list.
- (b) Change the Area to 0.1524 m².
- (c) Change the Length to 0.1524 m (the axial chord length).

6. Save the steady flow case file (`slide_ss.cas`).

File → **Write** → Case...

7. Start the calculation by requesting 500 iterations.

Solve → Iterate...

The residual history and lift force history will be displayed as the calculation proceeds. The lift history should be similar to Figure 10.4.

Note: *After 500 iterations, the steady flow calculation may not be fully converged. Here, this is not of concern, as the steady-state prediction will be used only as a starting solution for the transient sliding-mesh calculation.*

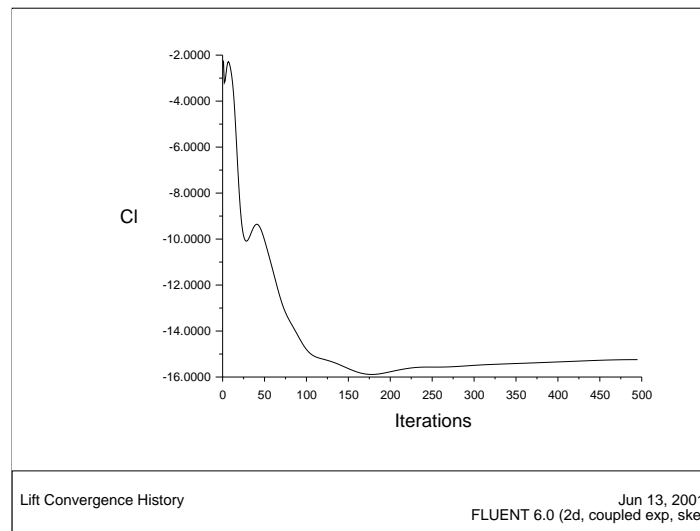


Figure 10.4: Lift Coefficient History: Steady Flow, Non-Moving Rotor

Using Sliding Meshes

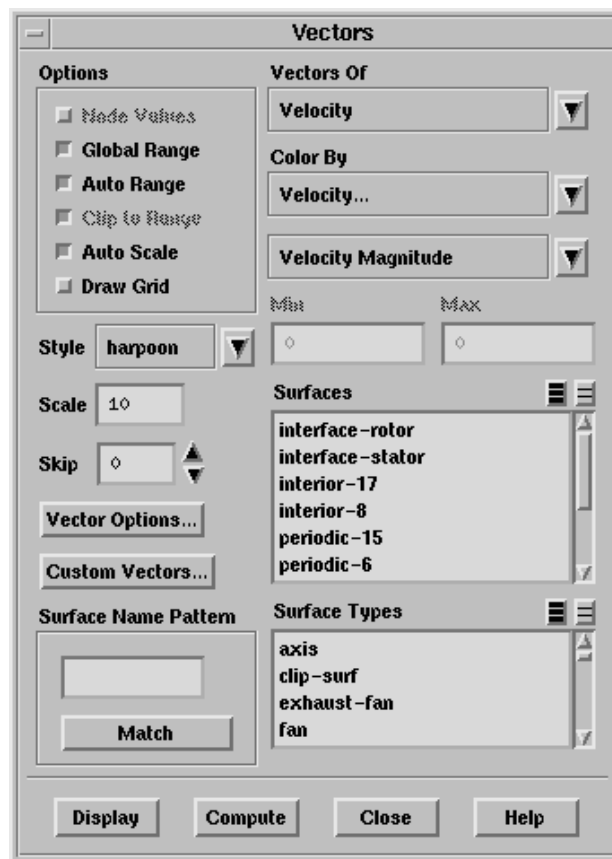
8. Save the case and data files (slide_ss.cas and slide_ss.dat).

File → **Write** → Case & Data...

Note: If you choose a file name that already exists in the current directory, FLUENT will prompt you for confirmation to overwrite the file.

9. Display the steady flow velocity vectors (Figure 10.5).

Display → Vectors...

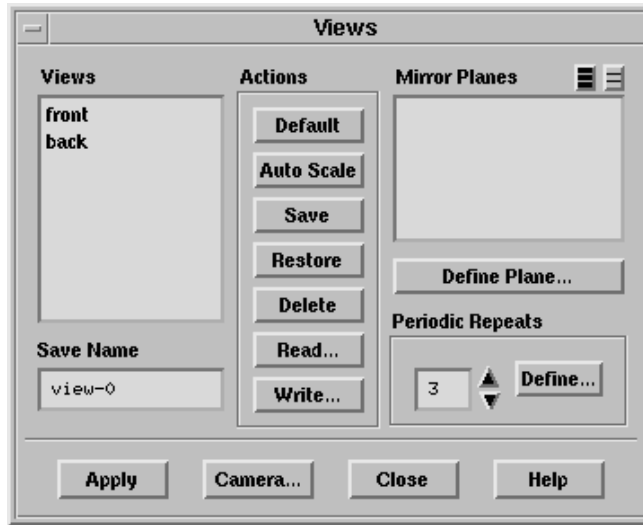


- (a) Change the Scale to 10.
(b) Click Display.

The steady flow prediction shows the expected form, with peak velocity of about 140 m/s through the passage.

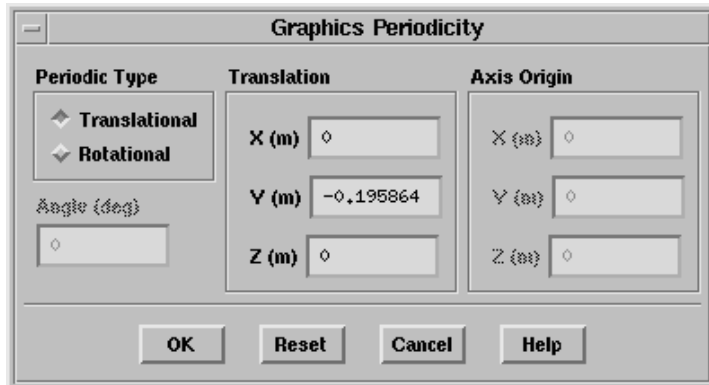
10. Enable the display of three periodic repeats of the solution domain (Figure 10.5).

Display → Views...



- (a) Set the number of Periodic Repeats to 3.
- (b) Under Periodic Repeats, click the Define... button.

This will open the Graphics Periodicity panel.



Using Sliding Meshes

(c) Click Reset.

This will reset Y Translation to the value shown above.

(d) Click OK in the Graphics Periodicity panel.

(e) In the Views panel, click Apply.

The grid display will be updated to show three periodic repeats. You may need to translate the display using your mouse to get the view shown in Figure 10.5.

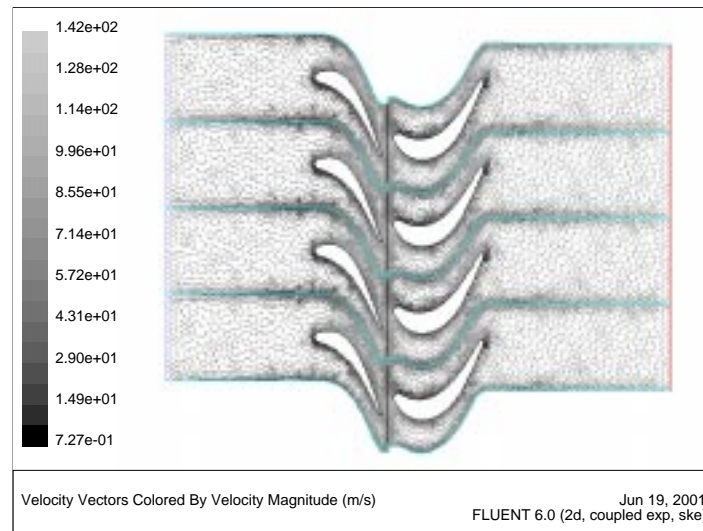
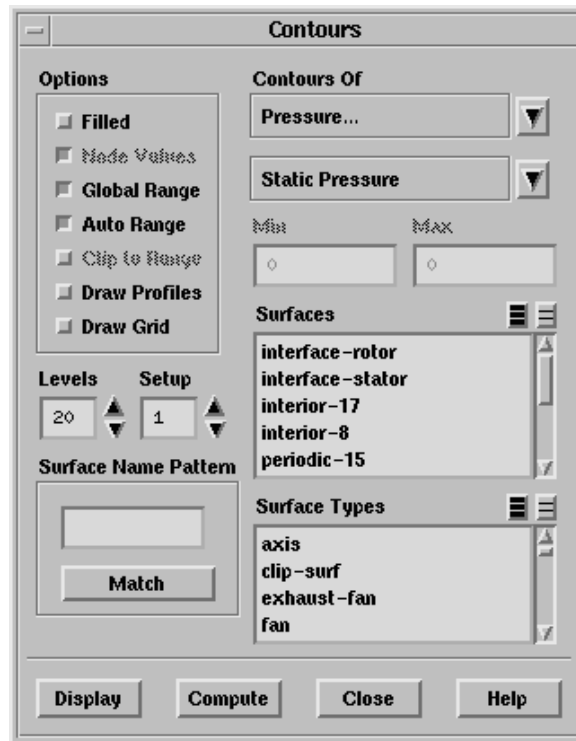


Figure 10.5: Velocity Vectors: Steady Flow, Non-Moving Rotor

11. Display the steady flow contours of static pressure (Figure 10.6).

Display → Contours...



The steady flow prediction shows the expected pressure distribution through the passage, with low pressure on the suction surfaces and stagnation around the impingement point on the rotor.

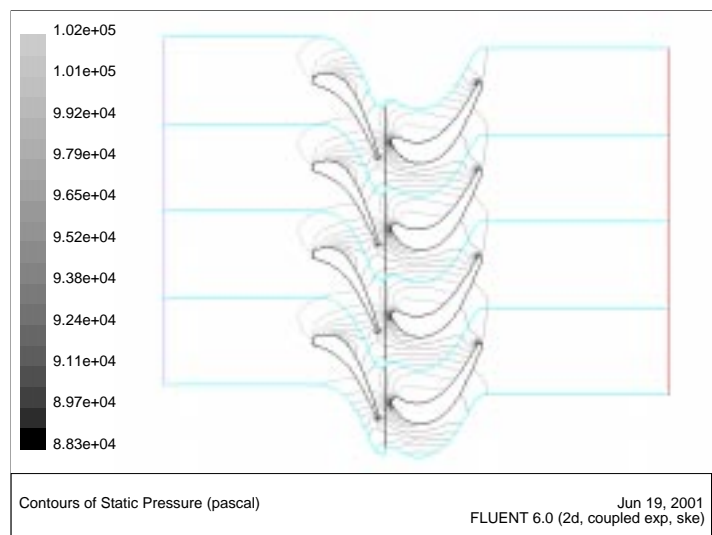


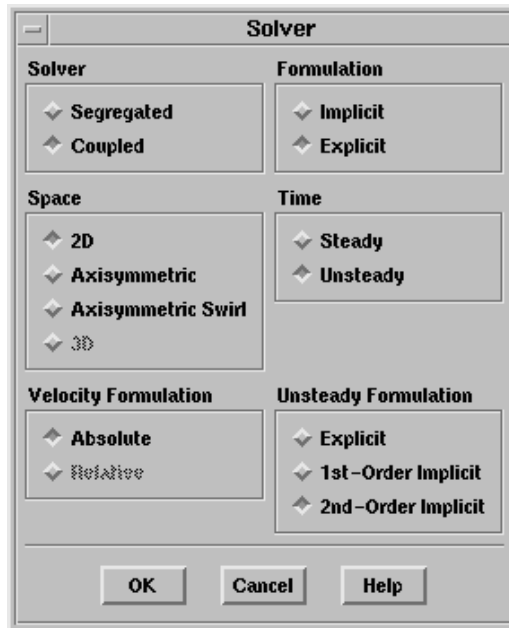
Figure 10.6: Contours of Static Pressure: Steady Flow, Non-Moving Rotor

Step 9: Enable Time Dependence and Sliding Rotor Motion

In this step you will enable the rotor motion by turning on time dependence and setting the sliding velocity of the rotor fluid zone.

1. Enable a time-dependent flow calculation.

Define → Models → Solver...



- (a) Under Time, select Unsteady.
- (b) Under Unsteady Formulation, select 2nd-Order Implicit.

Implicit (dual) time-stepping allows you to set the physical time step used for the transient flow prediction (while FLUENT continues to determine the time step used for inner iterations based on a Courant condition). Here, second-order implicit time-stepping is enabled: this provides higher accuracy in time than the first-order option.

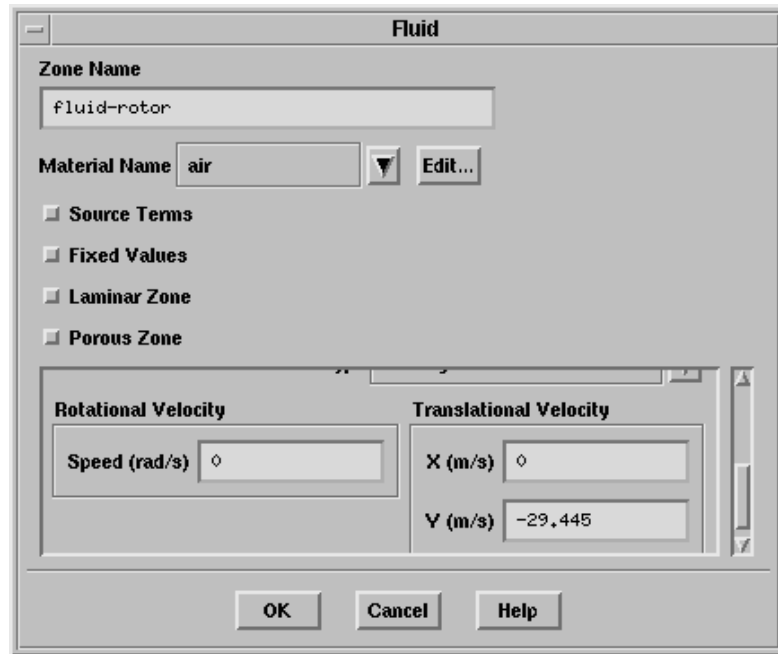
In explicit (global) time-stepping, FLUENT uses a single time step for the transient calculation. This time step is based on the Courant condition:

$$\Delta t = \frac{CFL \Delta x}{\lambda_{\max}}$$

where CFL is the Courant number. Explicit time-stepping might be the optimum solution strategy if you are modeling a traveling shock wave, where the Courant condition is ideal for determination of the time step value.

2. Define the sliding motion of the rotor-side fluid zone (fluid-rotor).

Define → Boundary Conditions...

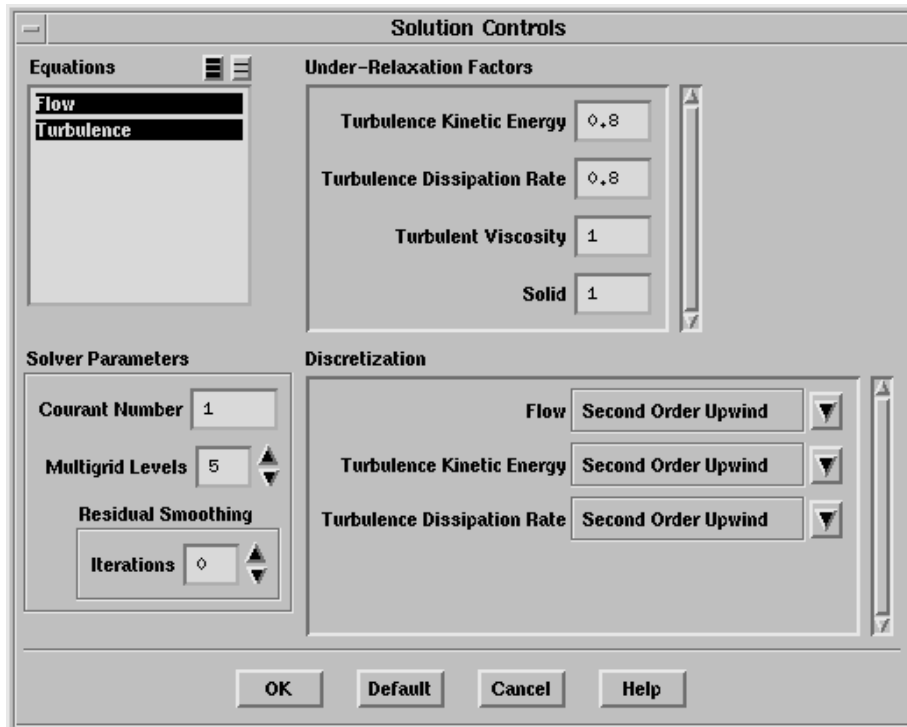


- (a) Scroll down and select Moving Mesh in the Motion Type drop-down list.
- (b) Change the Translational Velocity in the Y direction to -29.445.

Step 10: Solution: Unsteady Flow with Moving Rotor

1. Change the Courant number to 1 for the transient calculation.

Solve → Controls → Solution...



The Courant number controls the time step used by FLUENT during the inner iterations performed during each time step. A Courant number of 1 is a conservative setting that should ensure the stability of the inner iterations.

2. Reset the lift force monitor.

Solve → Monitors → Force...



- (a) In the File Name field, enter `cl-hist.td`.
- (b) Click Apply.
- (c) Click Clear.

This will remove the lift coefficient data for the steady-state calculation (i.e., the contents of the file `cl-hist.ss`) from memory.

- (d) Click Yes when asked to confirm the data discard.

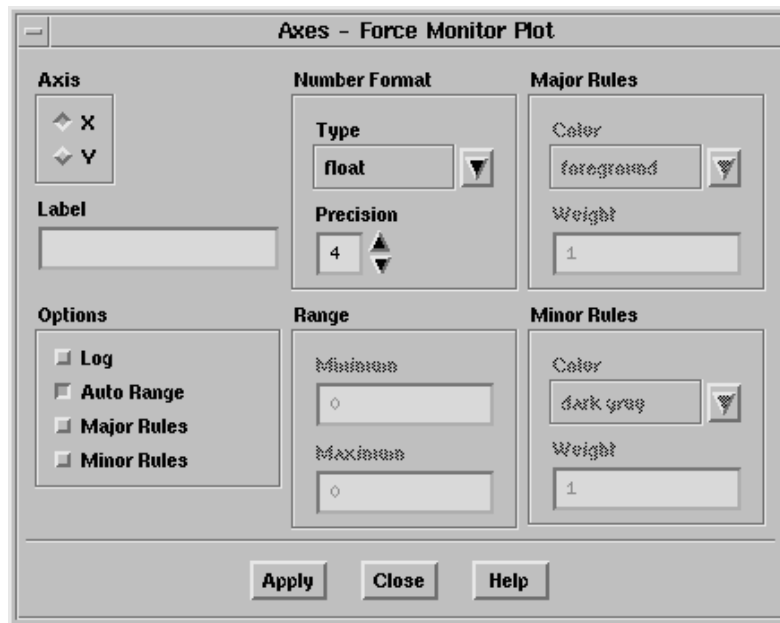


Using Sliding Meshes

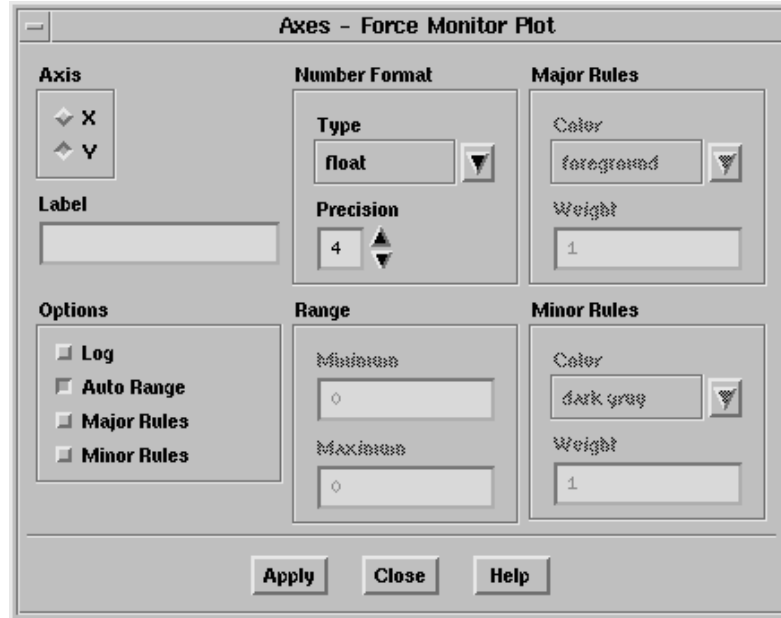
! If you do not clear the old force-monitoring data, FLUENT will plot them with the new data, corrupting the new lift coefficient plot.

- (e) Click on the Axes... button.

This will open the Axes - Force Monitor Plot panel.



- (f) Under Axis, select X (the default).
(g) Under Number Format, check that the Type is float.
(h) Set the Precision to 4.
(i) Click Apply.



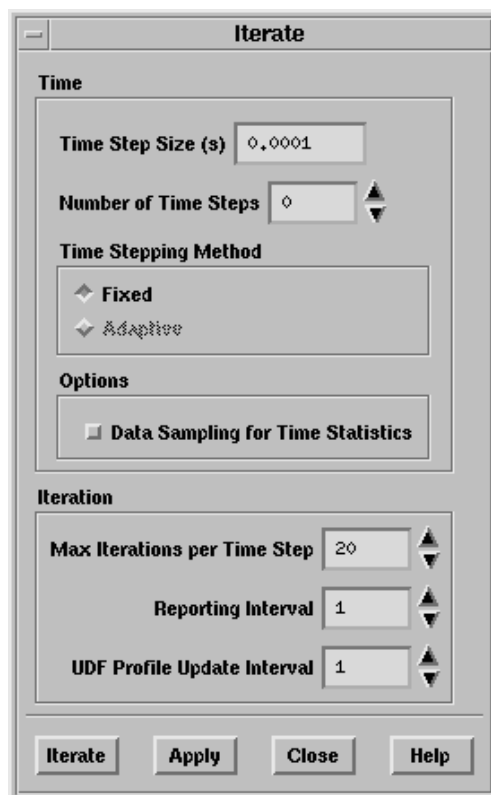
- (j) Under Axis, select Y.
- (k) Under Number Format, check that the Type is float and the Precision is 4.
- (l) Close the Axes - Force Monitor Plot and Force Monitors panels.

Note: *Monitoring the lift force is an ideal way to determine when the transient flow prediction becomes time-periodic (independent of the initial condition). In the time-periodic solution, the lift force variation will repeat identically from one passing period to the next.*

Using Sliding Meshes

3. Set the time step parameters.

Solve → Iterate...



- (a) Set the Time Step Size to 0.0001 second.
- (b) Check that the Max Iterations per Time Step is set to 20.
- (c) Click Apply.

Note: *The selection of the time step is critical for accurate time-dependent flow predictions. Here, the time step is chosen to be about 1/70 of the passing period, T . The passing period is the time it takes for the rotor blade to pass from one stator blade row to the next:*

$$T = (0.1959 \text{ m}) / (29.4 \text{ m/s}) = 6.7 \times 10^{-3} \text{ sec}$$

Using a time step of 0.0001 second, 67 time steps will be performed as the rotor performs one pass.

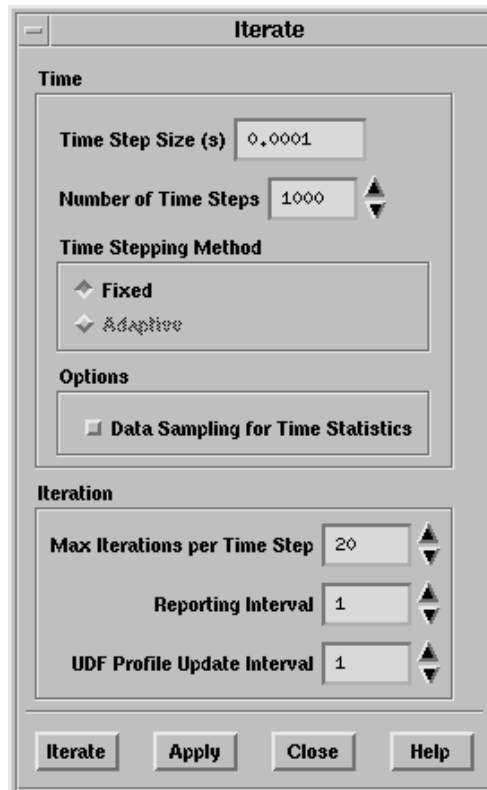
! *The maximum number of iterations per time step should be set large enough so that the inner iterations converge before the solution moves to the next time value. The value of 20, selected here, is quite small: it is likely that the initial time steps will not fully converge within 20 inner iterations. While this would be unsuitable for prediction of most time-dependent flows, the current simulation does not require high accuracy during the initial time steps. The rotor-stator flow prediction will be continued in time until a time-periodic flow is obtained. Low accuracy during the initial passing periods is acceptable as long as convergence is achieved during each time step of the final passing periods.*

Using Sliding Meshes

4. Save the transient sliding mesh case file (slide_td.cas).

File → **Write** → Case...

5. Start the transient calculation by requesting 1000 time steps.



- ! *Calculation of 1000 time steps will require significant CPU resources. Instead of calculating, you can read the case and data files saved after 0.1 seconds:*

`slide01.cas` and `slide01.dat`

After reading the files, skip to Step 11: Postprocessing at $t=0.1$ Second. (The case and data files are available in the same directory where you found the mesh files.)

By requesting 1000 time steps, you are asking FLUENT to compute until time is equal to 0.1 second. This will include roughly 15 passing periods ($15 \times 6.7e-3 \text{ sec} = 0.10 \text{ sec}$). Experience shows that the flow becomes time-periodic after about 12 passing periods. The lift history display allows you to confirm this. Your lift force history should be similar to that shown in Figure 10.7. Notice the periodicity of the lift coefficient after approximately 0.05 seconds.

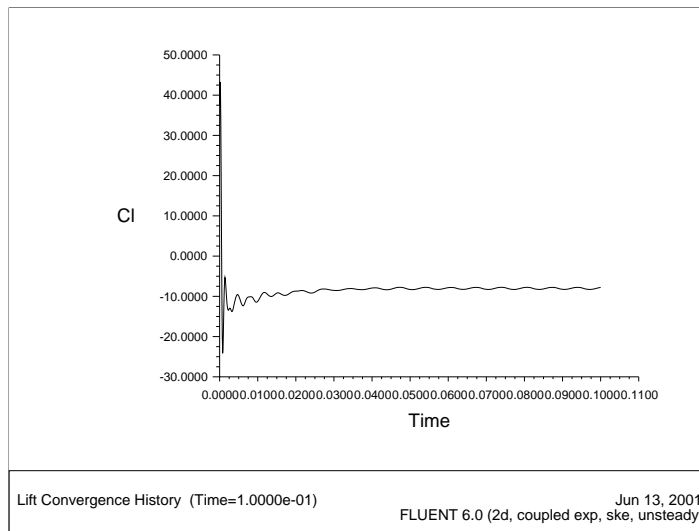


Figure 10.7: Lift Coefficient History: Unsteady Flow, Moving Rotor

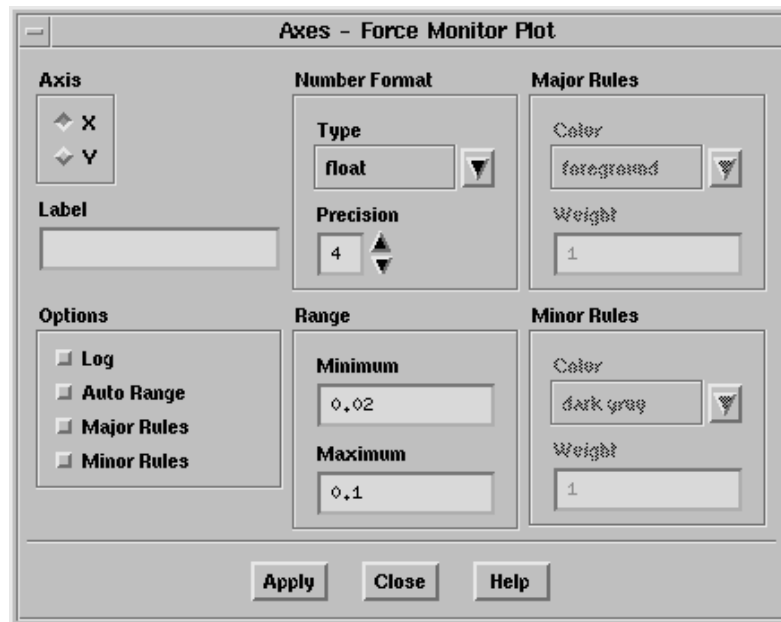
Using Sliding Meshes

6. Reset the lift coefficient history range to focus on the periodicity.

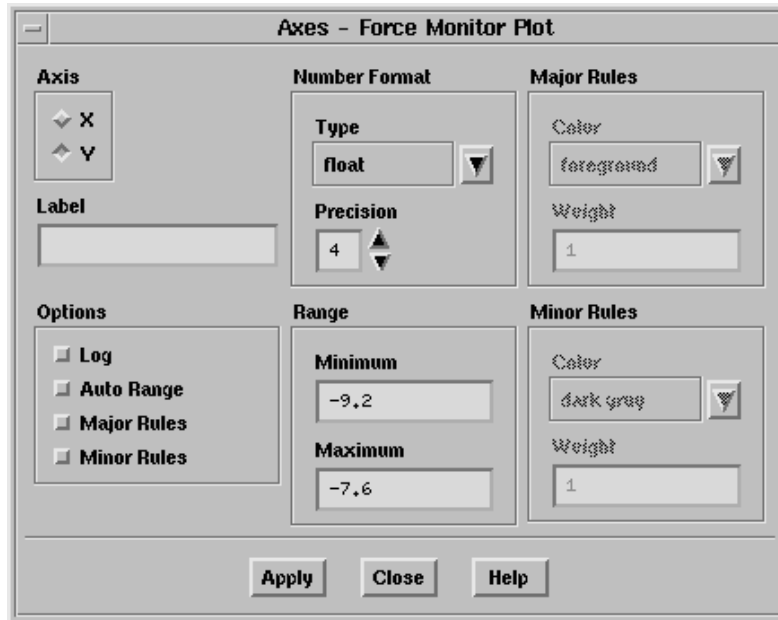
Solve → Monitors → Force...

- (a) Click the Axes... button.

This will open the Axes - Force Monitor Plot panel.



- i. Under Axis, select X.
- ii. Under Options, deselect Auto Range.
- iii. Under Range, set the Minimum to 0.02, and the Maximum to 0.1.
- iv. Click Apply.



- v. Under Axis, select Y.
 - vi. Under Options, deselect Auto Range.
 - vii. Under Range, set the Minimum to -9.2, and the Maximum to -7.6.
 - viii. Click Apply and Close the panel.
- (b) In the Force Monitors panel, click Plot.

In Figure 10.8, you can see the periodicity more clearly.

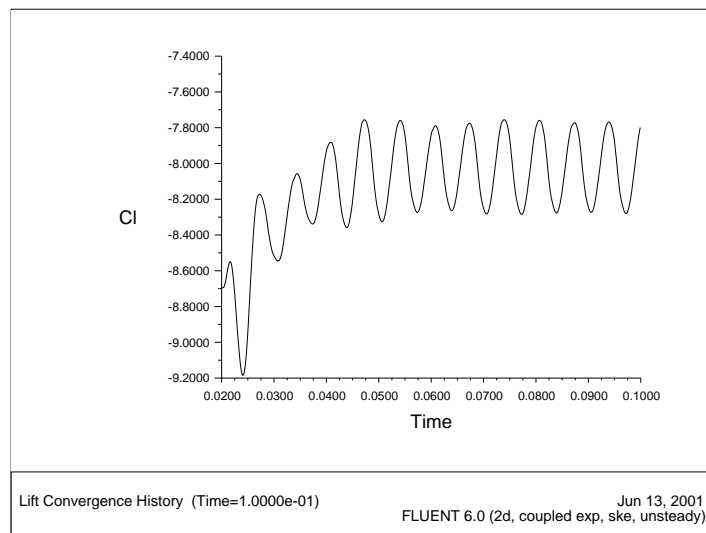


Figure 10.8: Lift Coefficient History: Narrowed Range

7. Save the case and data files at $t = 0.1$ second (slide01.cas and slide01.dat).

File → **Write** → Case & Data...

! *When the sliding mesh model is used, you must save a case file whenever a data file is saved. This is because the case file contains the grid information, which is changing with time.*

Step 11: Postprocessing at $t = 0.1$ Second

The solution data saved at $t = 0.1$ second can be reviewed using any of FLUENT's postprocessing features. Time-dependent data are analyzed just like steady-state results: you read a case and data file for each time value of interest. Note that this means you must save case and data files (or graphics files) at all intermediate time values for which results are of interest. Automatic saving of results during a transient calculation is used in Step 12, *Saving and Postprocessing Time-Dependent Data Sets*.

! For sliding mesh cases, it is important that you read the associated case file whenever you read a data file, because the case file contains the grid information, which is changing with time.

1. Display the velocity vectors at $t = 0.1$ second (Figure 10.9).

Display → Vectors...

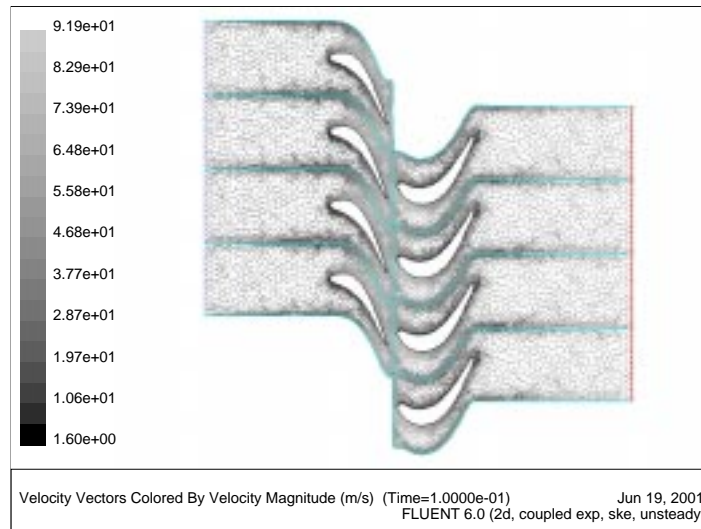


Figure 10.9: Velocity Vectors at $T = 0.1$ Second: Unsteady Flow

Note: The velocity vectors in Figure 10.9 are displayed with respect to the absolute reference frame (default). If you want to

display the vectors with respect to the moving reference frame, you can select Relative Velocity in the Vectors Of drop-down list and select the rotor fluid zone as the Reference Zone in the Reference Values panel.

2. Display contours of static pressure at $t = 0.1$ second (Figure 10.10).

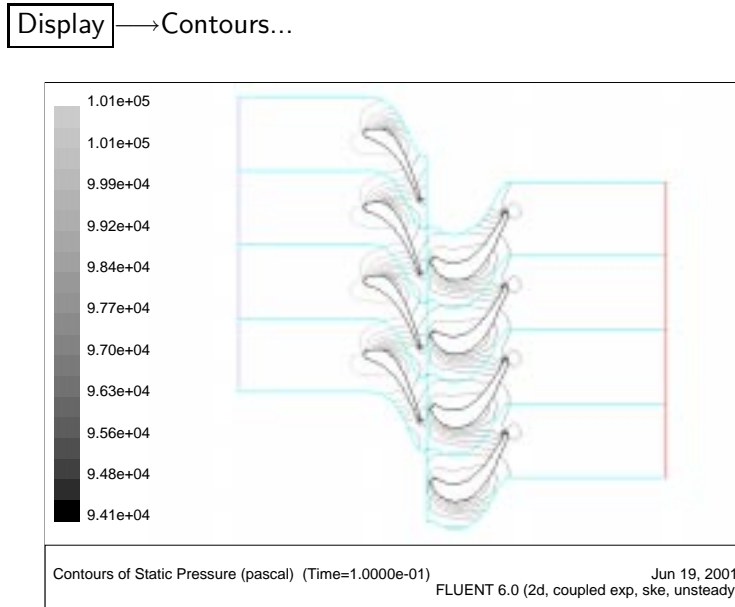
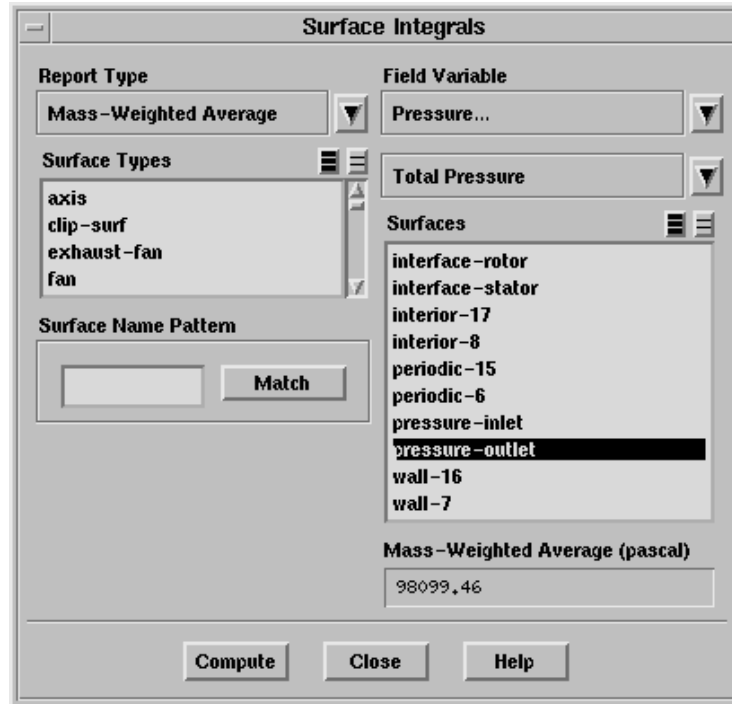


Figure 10.10: Contours of Static Pressure at Time = 0.1 Second: Unsteady Flow

Note: *Slight discontinuities in the pressure contours along the sliding interface are expected. This is because the contour plotting uses one-sided interpolation on either side of the sliding plane. This is purely a display issue.*

- Determine the instantaneous total pressure loss through the system.

Report → Surface Integrals...

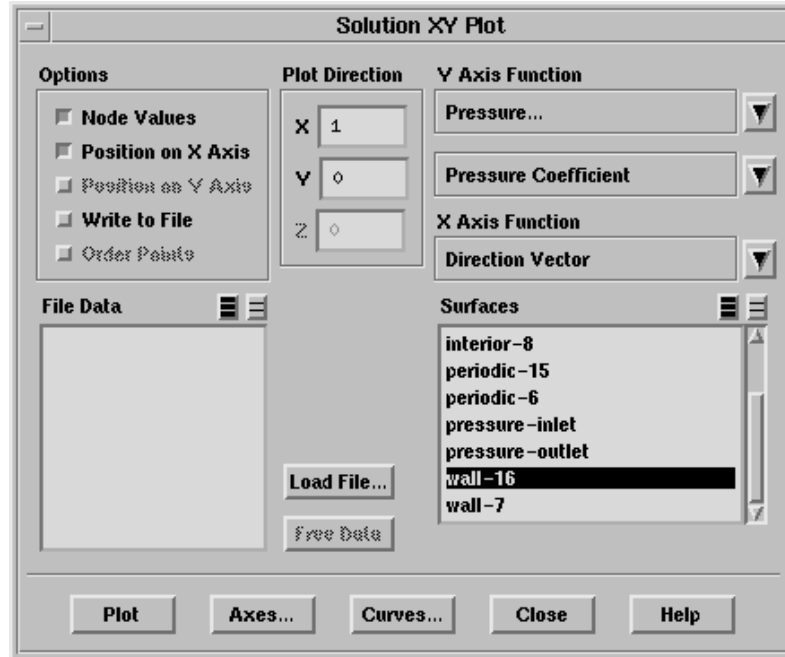


- In the Report Type drop-down list, select Mass-Weighted Average.
- Select Pressure... and Total Pressure in the Field Variable drop-down lists.
- In the Surfaces list, select pressure-outlet.
- Click Compute.

The mass-averaged (instantaneous) total pressure at the exit is about 98100 Pa, implying a loss of about 3200 Pa from the inlet total pressure of 101325 Pa.

- Plot the instantaneous pressure coefficient on the rotor blade at $t = 0.1$ second (Figure 10.11).

Plot → XY Plot...



- Select Pressure... and Pressure Coefficient in the Y Axis Function drop-down lists.
- In the Surfaces list, select wall-16.
- Click Plot.

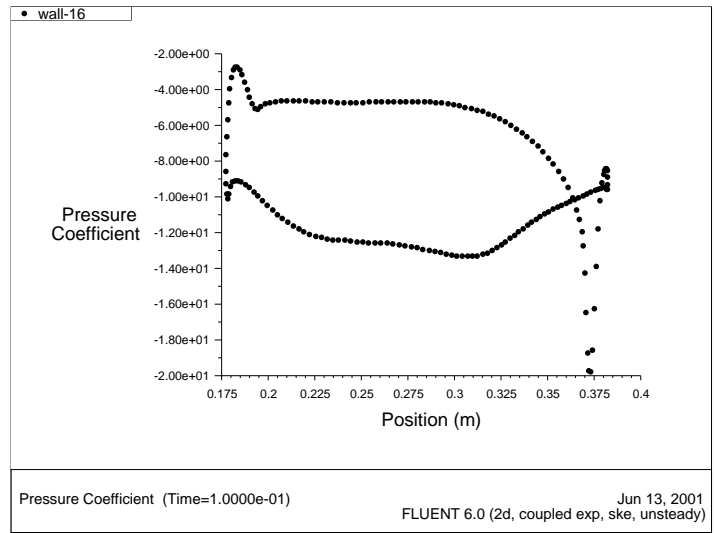


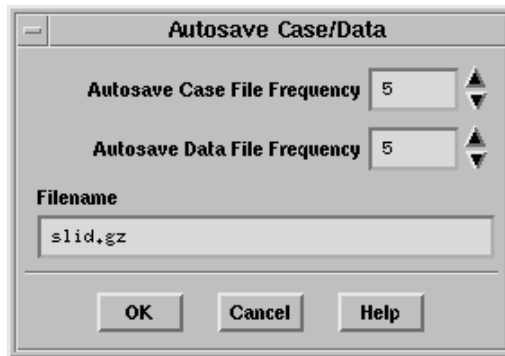
Figure 10.11: Pressure Coefficient on the Moving Rotor at Time = 0.1 Second

Step 12: Saving and Postprocessing Time-Dependent Data Sets

After 0.1 second, the sliding mesh flow prediction has achieved a time-periodic state. In order to study how the flow changes within a single passing period, you will now continue the time-marching through the next period and save results every 5 time steps.

1. Request saving of case and data files every 5 time steps.

File → Write → Autosave...



- (a) Set the Autosave Case File Frequency and Autosave Data File Frequency to 5.
- (b) In the Filename field, enter `slid.gz`.
- (c) Click OK.

FLUENT will append the time step value to the file name prefix (`slid`). The standard extensions (`.cas` and `.dat`) will also be appended. This will yield file names of the form `slid1005.cas` and `slid1005.dat`, where 1005 is the time step number.

Using Sliding Meshes

The extension “.gz”, supplied in the panel above, instructs FLUENT to save the case and data files in compressed format, yielding file names of the form `slid1005.cas.gz` and `slid1005.dat.gz`. If you leave off the “.gz” extension, FLUENT will save the files as `slid1005.cas` and `slid1005.dat`, etc.

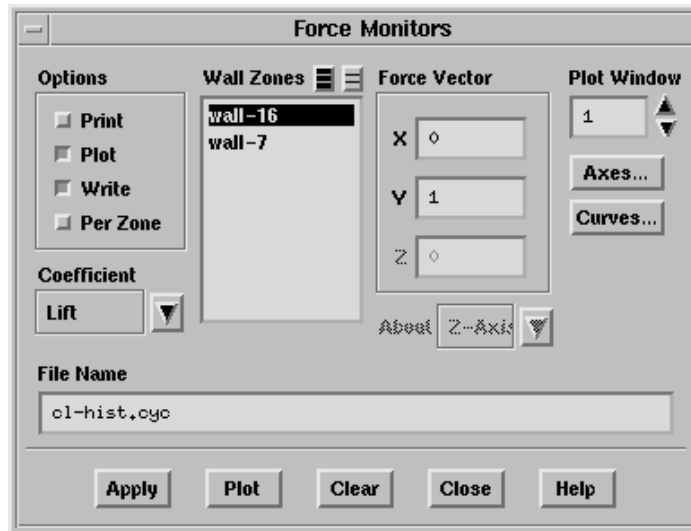
! When the sliding mesh model is used, you must save a case file whenever a data file is saved. This is because the case file contains the grid information, which is changing with time.

Extra: If you want to generate a solution animation by plotting, for example, pressure contours every 5 time steps, you can use the Solution Animation panel to set up the animation before you begin the calculation. Tutorial 4 demonstrates how to do this.

2. Reset the lift force monitor.

Supplying a new file name eliminates overwriting of the file stored during the previous time steps.

Solve → Monitors → Force...



- (a) In the File Name field, enter `cl-hist.cyc`.
- (b) Click Apply.
- (c) Click Clear.

This will remove the lift coefficient data for the previous time steps from memory (i.e., `cl-hist.td`).

- (d) Click Yes when asked to confirm the data discard.
- (e) Click on the Axes... button to reset the domain and range.

This will open the Axes - Force Monitor Plot panel.

- i. Under Axis, select X.
- ii. Under Options, select Auto Range.

This will deactivate the Minimum and Maximum range fields.

Using Sliding Meshes

- iii. Click Apply.
 - iv. Under Axis, select Y.
 - v. Under Options, select Auto Range.
This will deactivate the Minimum and Maximum range fields.
 - vi. Click Apply, and Close the Axes - Force Monitor Plot panel.
- (f) Close the Force Monitors panel.

3. Continue the calculation by requesting 70 time steps.

This performs the time-marching iterations for the next passing period, starting from the current solution data at time = 0.1 second.

Solve → Iterate...

Note: Requesting 70 time steps will march the solution through 0.007 seconds, or roughly one passing period. With the autosave and command monitor features active (as defined above), the case, data, and pressure contour plot files will be saved every 0.0005 seconds. The lift history should appear as in Figure 10.12.

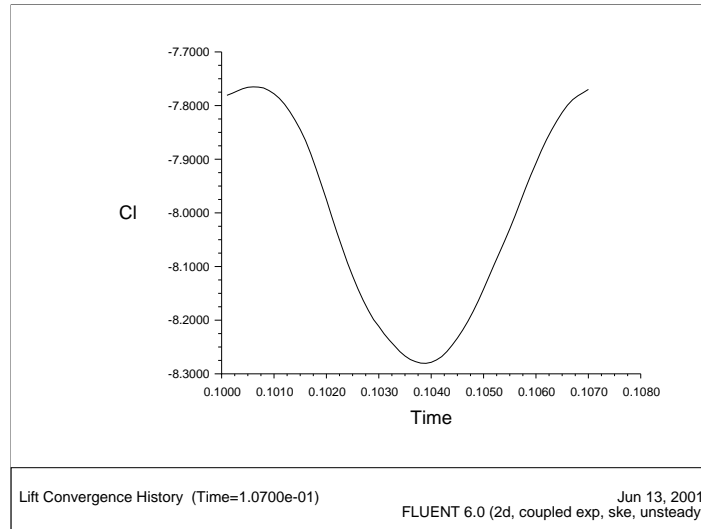


Figure 10.12: Lift History During the Final Passing Period

4. Examine the results at different time steps within a single passing period.

- (a) Read in the case and data files of interest.

File → **Read** → Case & Data...

- (b) Display contours of static pressure.

The display of pressure contours every five time steps will show the time-varying pressure distribution and the motion of the rotor. Examples of two pressure contour plots at $t = 0.1030$ and $t = 1.070$ seconds are shown in Figures 10.13 and 10.14, respectively.

Display → Contours...

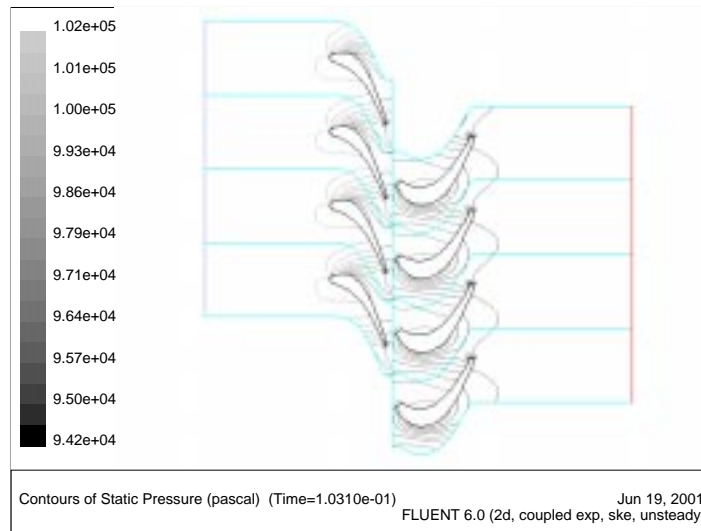


Figure 10.13: Pressure Contours at Time = 0.103 Seconds

Extra: *If you generated a solution animation during the calculation as mentioned earlier in this tutorial, you could play it back inside FLUENT to see the pressure contour animation over time. See Tutorial 4 for details.*

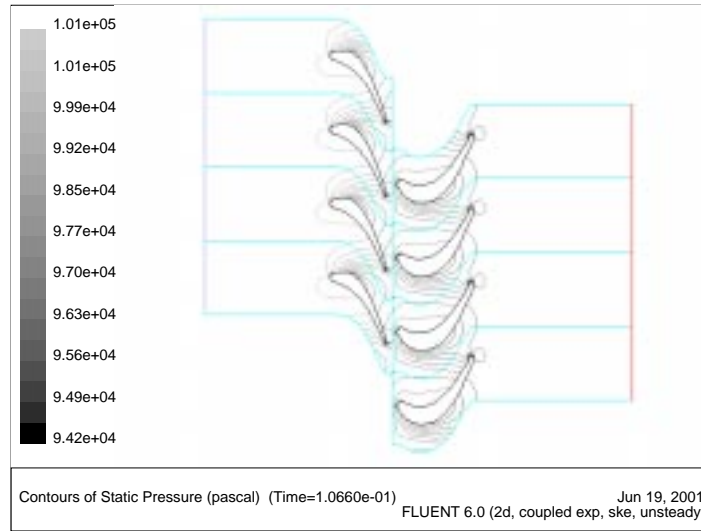


Figure 10.14: Pressure Contours at Time = 0.107 Seconds

Summary: In this tutorial, you have modeled the time-periodic flow involving rotor-stator interaction. You have learned how to merge the separate rotor and stator meshes using `tmerge`, and to create the grid-interface zones along the sliding interface. Similar procedures can be used to tie together meshes for non-sliding mesh analyses.

You have used FLUENT's time-dependent flow prediction capability, and you have learned how to set solution parameters for implicit time-stepping. These time-dependent flow prediction procedures can also be applied to other, non-sliding mesh, analyses. The procedures in this tutorial, however, are applicable to time-periodic calculations, in which the initial condition and initial phase of the transient calculation are treated without concern for time accuracy.

You have also learned how to manage the file saving and graphical postprocessing for time-dependent flows, using file autosaving and command monitors to automatically save solution information as the transient calculation proceeds.