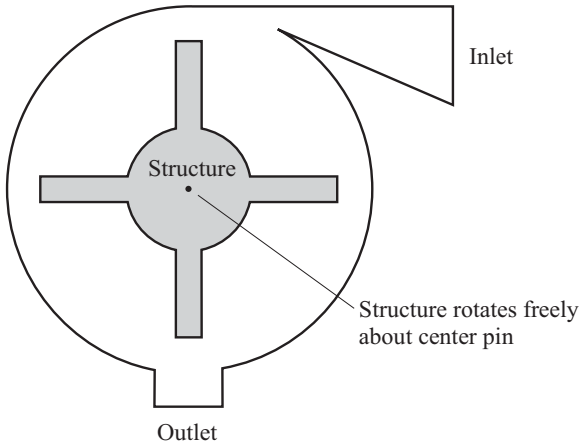
Problem description

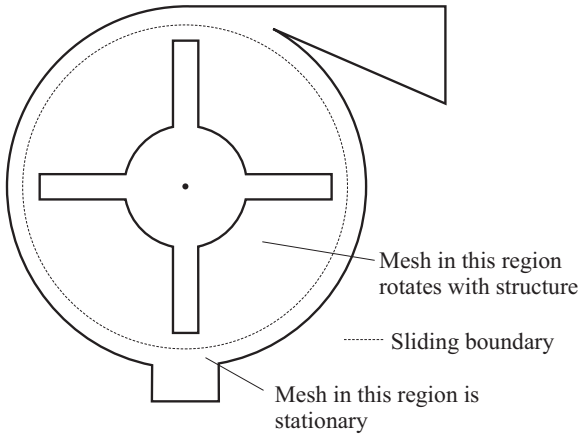
A simplified turbine is immersed in a fluid as shown:



At the beginning of the analysis, the turbine is at rest. A normal traction is suddenly imposed at the turbine inlet. The fluid flows through the turbine housing, causing the turbine to rotate.

The model is planar and two-dimensional.

Because the turbine can rotate an arbitrary amount, it is convenient to model fluid surrounding the turbine with elements that rotate with the turbine. These elements slide past the elements that are near the turbine housing, as shown:



Fluid is allowed to flow through the sliding mesh boundary.

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Defining a boundary condition of type sliding-mesh

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem cannot be solved with the 900 nodes version of the ADINA System because the 900 nodes version of the ADINA system does not contain ADINA-FSI.

Note that you must have an ADINA-M license to do this problem. In addition you should be able to allocate as much memory as possible to the AUI, at least 400 MB. This memory is needed to perform the particle tracing at the end of this problem description.

Much of the input for this problem is stored in files `prob37_1.in`, `prob37_2.in`, `prob37_3.in` and `prob37_1.plo`. You need to copy file `prob37_1.in`, `prob37_2.in`, `prob37_3.in`, `prob37_1.plo` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoke the AUI and set the Program Module drop-down list to ADINA CFD. Choose Edit→Memory Usage and make sure that the ADINA/AUI memory is at least 400 MB.

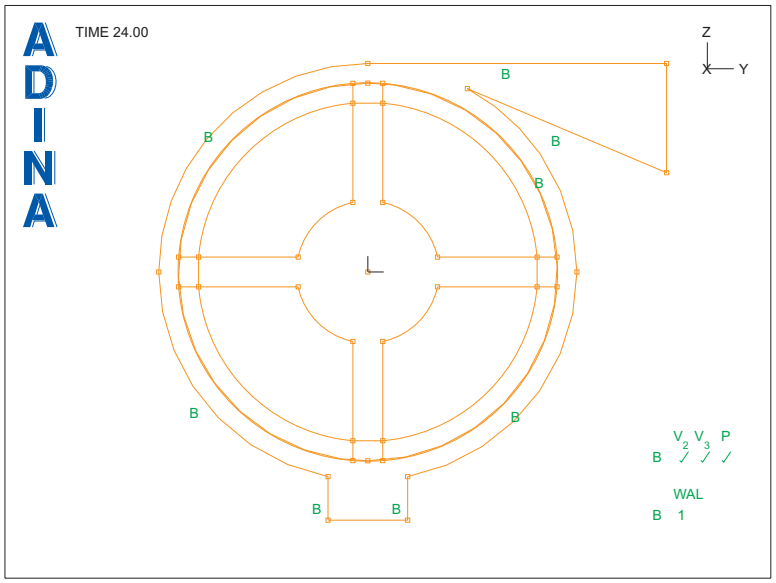
ADINA CFD model

Defining model control data, geometry, wall boundary condition

We have prepared a batch file (`prob37_1.in`) that performs the following operations:

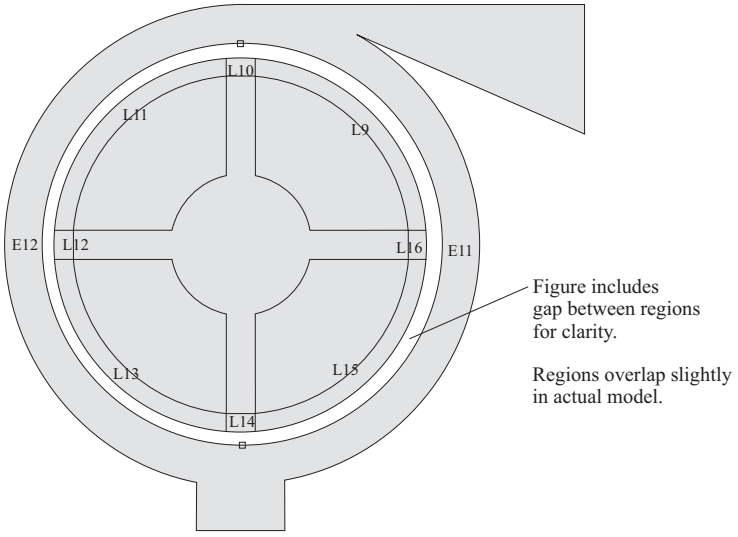
- ► Specifies a transient FSI analysis.
- ► Specifies the time stepping
- ► Defines points, lines and surfaces.
- ► Defines a sheet body
- ► Defines a wall boundary condition
- ► Plots the model

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob37_1.in` and click Open. The graphics window should look something like the figure on the next page.



Defining the sliding-mesh boundary conditions

The following figure shows the lines and edges on the sliding-mesh boundaries:

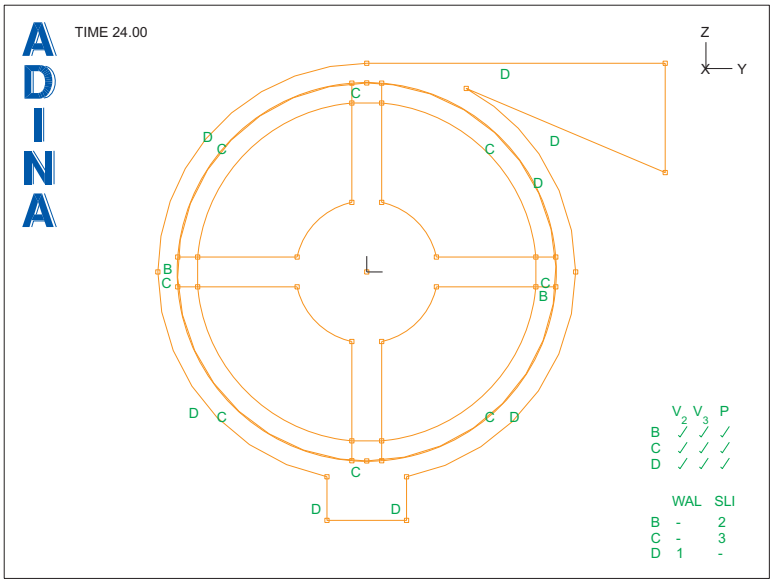


Although the figure shows a slight gap between the two meshes, actually the two meshes overlap slightly.

Click the Special Boundary Conditions icon SBC, add condition number 2 and set the Type to Sliding Mesh. Set the 'Apply to' field to Edges, then enter 12, 11 in the first two rows of the table and click Save. Add condition number 3 and make sure that the Type is Sliding Mesh. Set the 'Apply to' field to Lines, enter line numbers 9 to 16 in the first eight rows of the table, then click Save (do not close the dialog box).

We also need to create a boundary condition pair to link the two boundary conditions of type sliding-mesh. Click the Boundary Condition Pair button, and, in the first row of the table, set B.C. #1 to 2 and B.C. #2 to 3, then click OK twice to close both dialog boxes.

When you click the Redraw icon, the graphics window should look something like this:



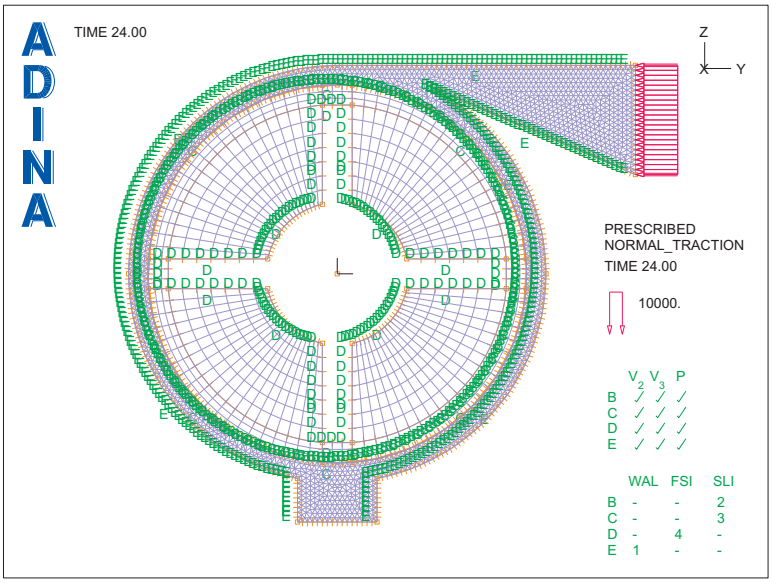
Finishing the ADINA CFD model definition

We have prepared a batch file (`prob37_2.in`) that performs the following operations:

- Defines remaining special boundary conditions
- Defines leader-follower relationships
- Defines the material
- Defines the normal-traction loading
- Defines the element groups
- Subdivides the geometry
- Meshes the geometry
- Creates the `prob37_f.dat` file

► Regenerates the graphics

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob37_2.in` and click Open. Close the Log Window dialog box (which is displayed when the AUI creates the data file). The graphics window should look something like this:

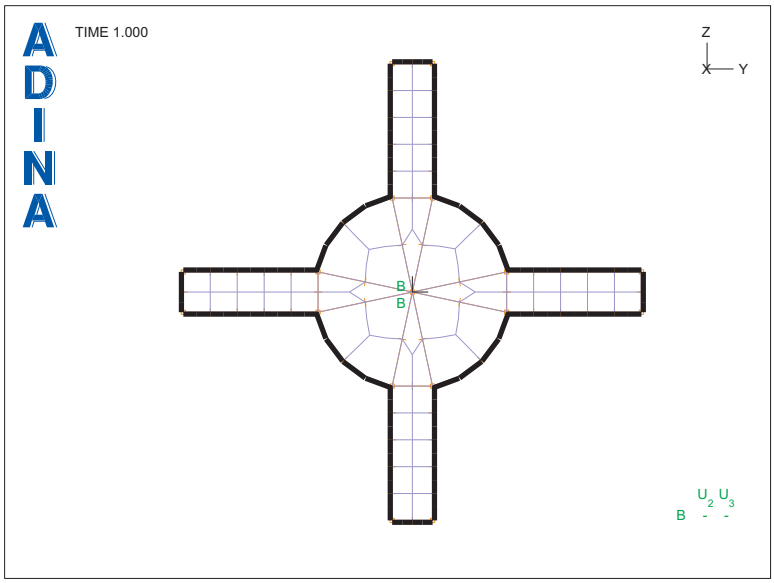


Click the Save icon and save the database to file `prob37_f`.

ADINA Structures model

Click the New icon to begin a new model.

We have prepared a batch file (`prob37_3.in`) that creates the entire ADINA Structures model. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob37_3.in` and click Open. Close the Log Window dialog box (which is displayed when the AUI creates the data file). The graphics window should look something like the figure on the next page.



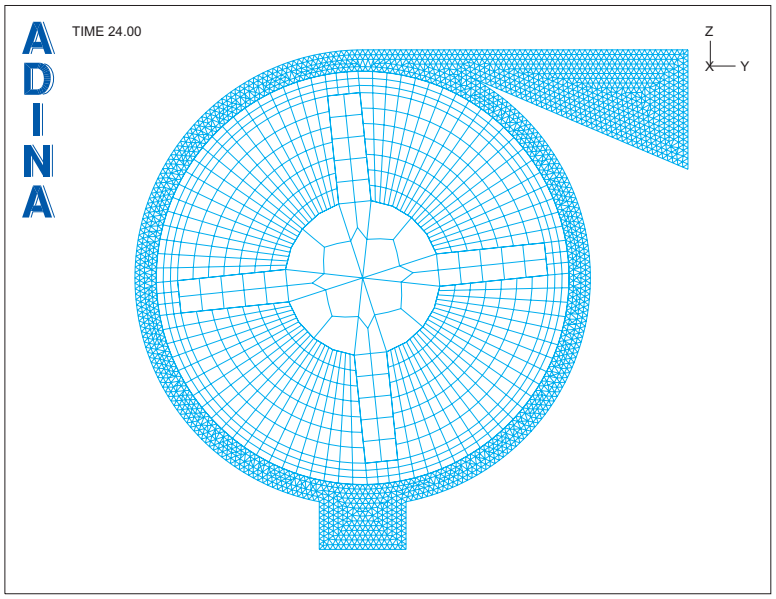
Running ADINA-FSI

Choose Solution→Run ADINA-FSI, click the Start button, select file `prob37_f`, then hold down the Ctrl key and select file `prob37_a`. The File name field should display both file names in quotes. Then click Start.

The ADINA-FSI solution takes 120 steps.

When ADINA-FSI finishes, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob37_f`. Then click the Open icon and open porthole file `prob37_a`. The graphics window should look something like the figure on the next page.

(Your results may be slightly different than ours because free meshing produces different meshes on different platforms.)



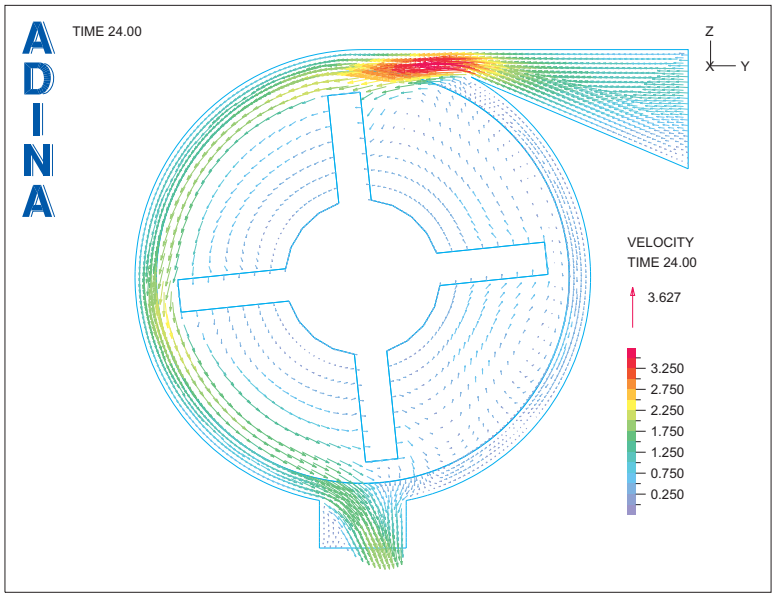
Post-processing

Visualizing the mesh motion: Click the Movie Load Step icon, then the Animate icon. Notice that the mesh surrounding the turbine rotates along with the turbine, and slides relative to the mesh close to the turbine housing. Click the Refresh icon to clear the animation.

Velocity vectors: Click the Model Outline icon, then click the Quick Vector Plot icon. To clear the stress vector plot in the structure, click the Modify Vector Plot icon, make sure that the Vector Quantity is STRESS, click the Delete button, click Yes to confirm, then click OK to close the dialog box. The graphics window should look something like the figure on the next page.

Click the Movie Load Step icon, then the Animate icon to animate the velocity vectors.

Notice that the velocity vectors cross the sliding-mesh boundary. Click the Refresh icon to clear the animation.

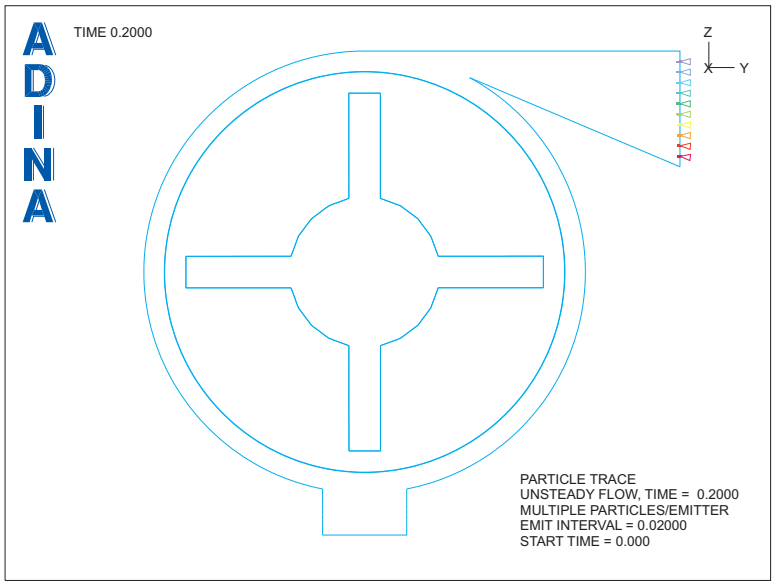


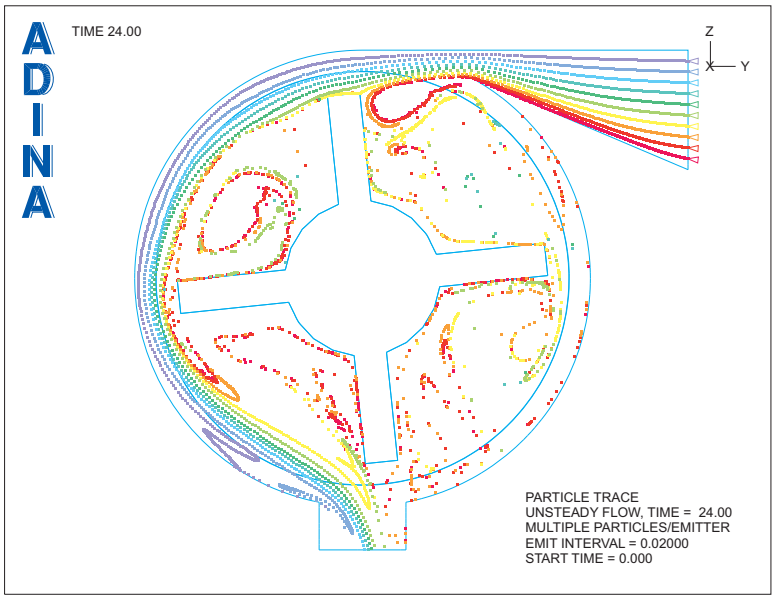
Particle tracing: We can use the particle tracing feature to visualize the fluid motion.

First click the Clear Vector Plot icon to remove the velocity vectors.

We have put the necessary commands for the particle tracing in a batch file (`prob37_1.plo`). Choose File→Open Batch, navigate to the working directory or folder, select the file `prob37_1.plo` and click Open. The graphics window should look something like the top figure on the next page.

At this point, the particle tracing has only been computed for the first time step. Now click the Movie Load Step icon to compute the particle traces for the entire solution. (This calculation may take a long time. Increasing the memory available to the AUI should speed up the calculation.) When the movie is complete, click the Animate icon. The graphics window should look something like the bottom figure on the next page.





Notice how the particles cross over the sliding mesh boundaries.

Click the Refresh icon  to clear the animation. You can also use the icons that change the solution time to view the particle traces at different solution times.

Exiting the AUI: Choose File→Exit (you can discard all changes).

Notes

If there is a slight gap between the two meshes, the sliding-mesh feature will still work. However, during particle tracing, if a particle enters the gap, it is lost and never reenters the model.

The two meshes must be incompatible (that is, they cannot share nodes). One convenient way to generate incompatible meshes is to use separate element groups for the two meshes, then set the Coincidence Checking to Group during meshing of the second element group.

Unsteady particle tracing is very memory-intensive. The memory allocated to the AUI should ideally be set to the amount of RAM (physical memory) on your computer.