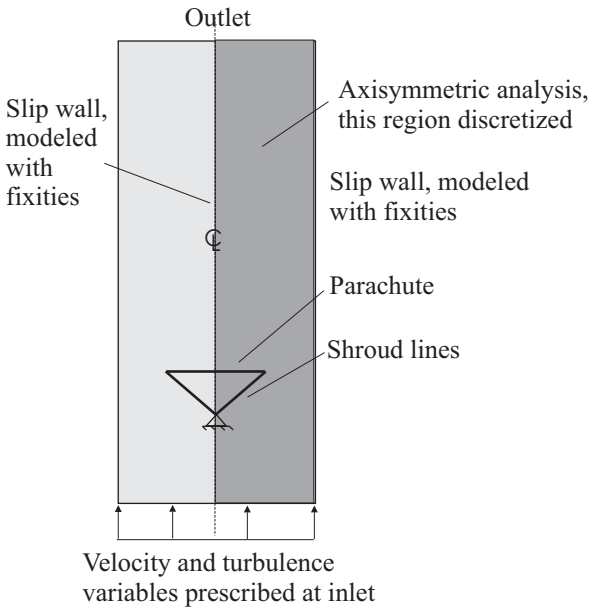
Problem description

Consider the following FSI problem:



The shroud lines are attached to a fixed point and the air flows upwards around the parachute. This simulates the situation in which the parachute floats downwards through the air at a fixed speed.

It is desired to determine the total fluid force applied to the parachute and the final shape of the parachute.

Turbulent flow conditions are assumed and the k-ε turbulence model is used. All input to this problem is given in SI units.

The steady-state solution is determined using a transient analysis in which the loads are held constant.

An axisymmetric analysis is performed. The shroud lines are modeled in a very approximate manner using a truss element. The parachute is also modeled in a very approximate manner using axisymmetric shell elements.

(We also note that we are actually modeling a "parasheet" instead of a parachute. Additionally, our parachute model cannot wrinkle as points in the parachute move towards the centerline; this causes nonphysical compressive hoop stresses in the parachute as the

parachute deforms.)

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

- Using the SAM features in manual mode to refine a fluid mesh in FSI analysis.
- Using the SAM features in automatic mode to refine a fluid mesh in FSI analysis.
- Plotting an axisymmetric mesh using mirror-imaging.

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 include ADINA-FSI.

Much of the input for this problem is stored in the following files:

`prob49m_f_0.in`, `prob49m_a_0.in`, `prob49m1_f.plo`, `prob49m1_a.plo`
`prob49a_f_0.in`, `prob49a_a_0.in`, `prob49a1_f.plo`, `prob49a1_a.plo`

You need to copy these files from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Solution using manual mode

Invoking the AUI and choosing the finite element program

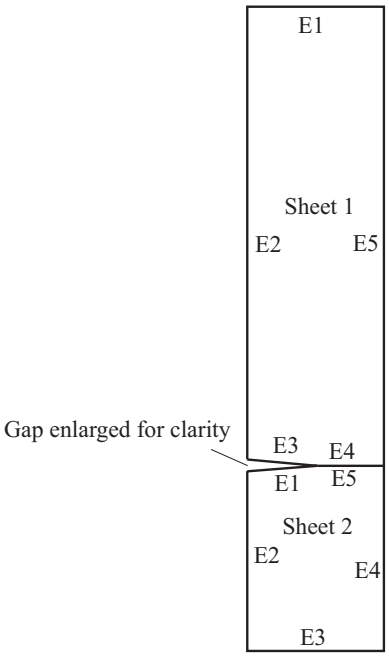
This problem is created using ADINA-M/PS (ADINA-M with Parasolid geometry modeler). The Open Cascade geometry modeler can also be used, but only if the input is modified.

Invoke the AUI and set the Program Module drop-down list to ADINA CFD.

Model definition for fluid model

We have prepared a batch file (`prob49m_f_0.in`) that contains all of the model definition for the fluid model, including the selection of the adaptive CFD feature.

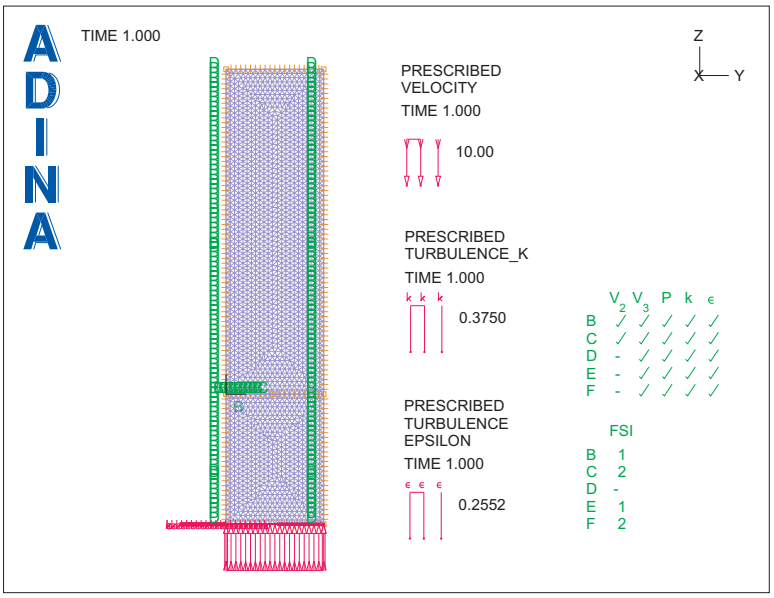
In this model definition, the following geometry is used:



There are several issues regarding the meshing. Because it is anticipated that the meshes near the parachute will be remeshed during the analysis, the meshing needs to be planned to take this remeshing into account. For this model, two sheets are used, so that the remeshing does not create elements that cross the fluid-structure interface boundaries. Also the nodal coincidence parameters in the meshing commands are chosen so that the remeshings can be done in any order (e.g. sheet 2 before sheet 1).

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob49m_f_0.in` and click Open. The graphics window should look something like the figure on the next page.

You can view the file `prob49m_f_0.in` to see the definitions of the geometry, material, boundary conditions, etc.



Selecting steered adaptive meshing in manual mode: Choose Meshing→Steered Adaptive Mesh→Control, set the Steered Adaptive Meshing Mode to Manual and click OK. (Note, since this is the first model, the fields "Restart File from CFD Solution" and "File Containing Geometric Data from Previous Model" are left blank.)

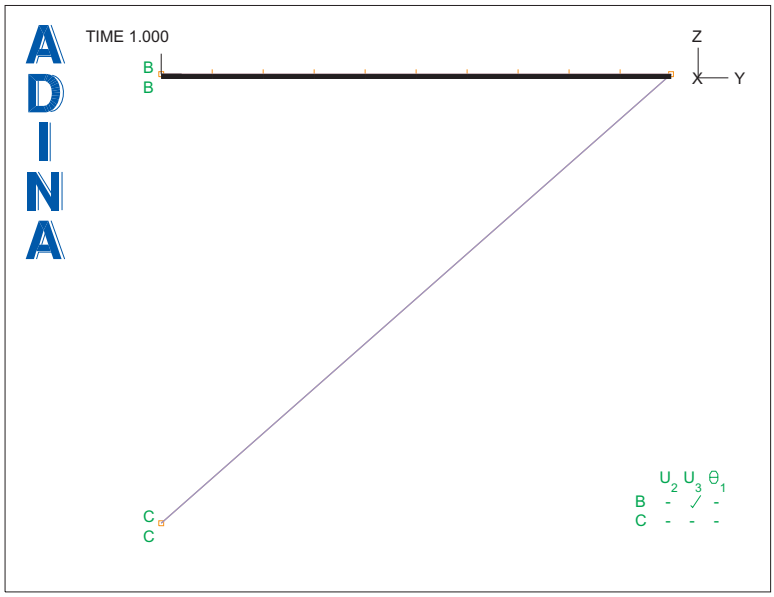
Generating the ADINA CFD data file: First click the Save icon and save the database to file `prob49m_f_0`. Click the Data File/Solution icon, set the file name to `prob49m_f_0`, make sure that the Run Solution button is unchecked and click Save.

Model definition for solid model

We have prepared a batch file (`prob49m_a_0.in`) that contains all of the model definition for the solid model.

Click the New icon (you can discard all changes), then choose File→Open Batch, navigate to the working directory or folder, select the file `prob49m_a_0.in` and click Open. The graphics window should look something like the figure on the next page.

You can view the file `prob49m_a_0.in` to see the definitions of the geometry, material, boundary conditions, etc.



Running ADINA-FSI, loading the porthole file

Choose Solution→Run ADINA-FSI, click Start, select files `prob49m_a_0.dat` and `prob49m_f_0.dat` (you can hold down the Ctrl key to select both files), make sure that the Maximum Memory for Solution is at least 100 MB and click Start.

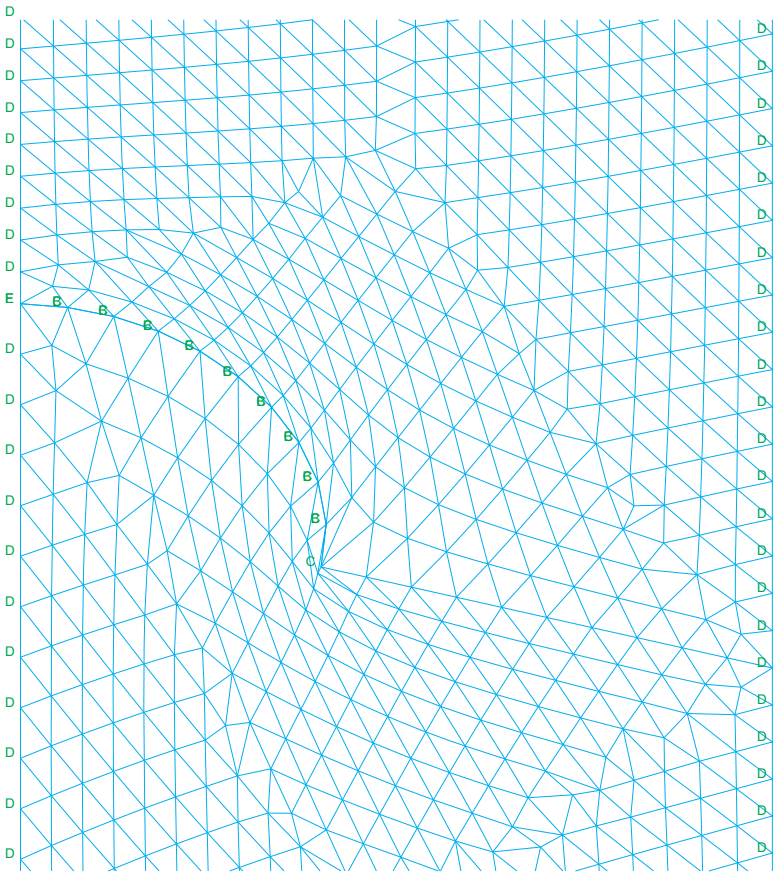
The run ends at time step 23 (time 0.5), with an error code that indicates that an element is overlapped. The last converged step is at time 0.4.

Now close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes) and open file `prob49m_f_0.por`.

Examining the solution

Click the Boundary Plot icon  and enlarge the region near the parachute. The graphics window should look something like the figure on the next page.

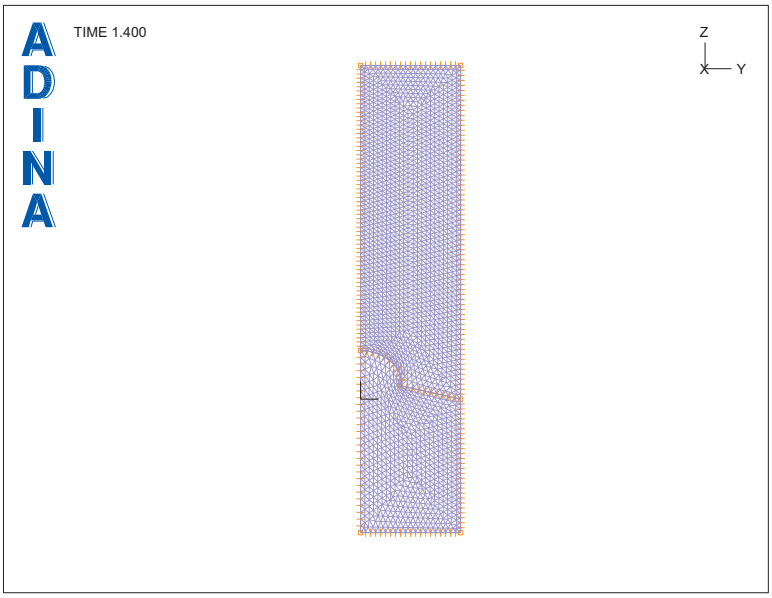
Evidently the elements near the edge of the parachute (near the node marked with a C) are becoming very deformed.



First mesh refinement

Set the Program Module drop-down list to ADINA CFD (you can discard all changes). Do not open file `prob49m_f_0.idb`.

Mesh and results from previous solution: Choose Meshing→Steered Adaptive Mesh→ Control and set the Steered Adaptive Meshing Mode to Manual. Set "Use Mesh in Previous Model at Solution Time" to 0.4. Set the "Restart File from CFD Solution" to `prob49m_f_0.res` using the ... button, and set the "File Containing Geometric Data from Previous Model" to `prob49m_f_0.adp` in a similar way. Click OK, then click the Mesh Plot icon  to show the mesh. The graphics window should look something like the figure on the next page.



This is the mesh at solution time 0.4, except that the solution time in the plot is 1.4 instead of 0.4.

Adaptive meshing criteria: Choose Meshing→Steered Adaptive Mesh→Criterion and add criterion 1. Make sure that the Type is Element Quality and that the Solution Time is 0.4, then set the Minimum Element Quality to 0.8, the Maximum Element Quality to 1.2 and click OK.

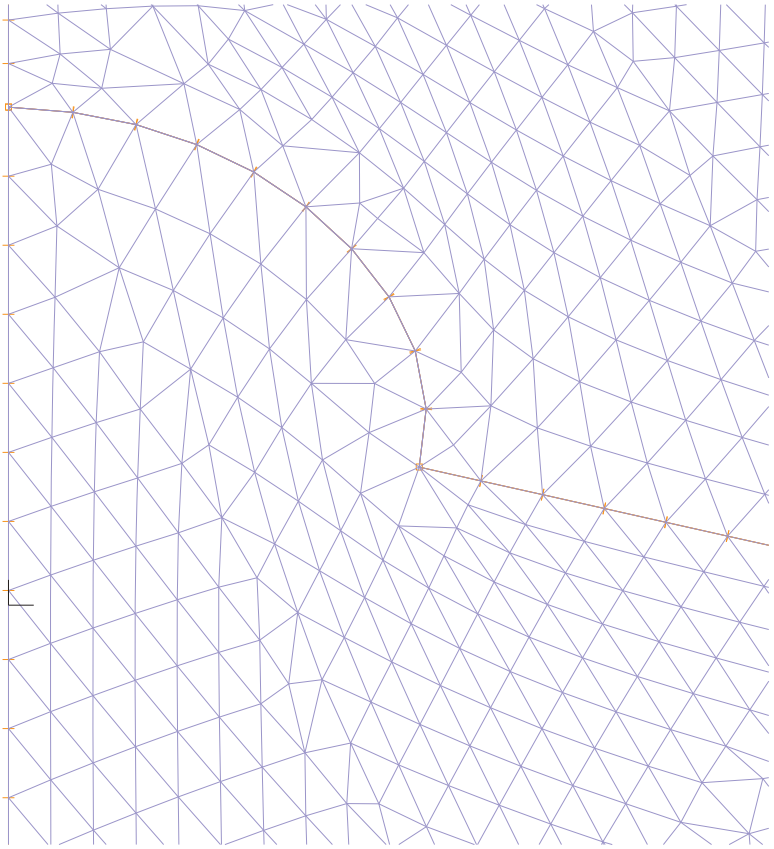
(See notes at the end of this problem for an explanation of this criterion.)

Remesh: Choose Meshing→Steered Adaptive Mesh→Mesh, set "Use Adaptive Mesh Criterion" to 1, and click OK.

Redefine time steps: Choose Control→Time Step, edit the table as follows and click OK.

Number of Steps	Magnitude
1	0.4
16	0.1

Click the Redraw icon and zoom to see the mesh near the parachute. The graphics window should look something like the figure on the next page.



Generating the ADINA CFD data file: First click the Save icon and save the database to file `prob49m_f_1`. Click the Data File/Solution icon, set the file name to `prob49m_f_1`, make sure that the Run Solution button is unchecked and click Save.

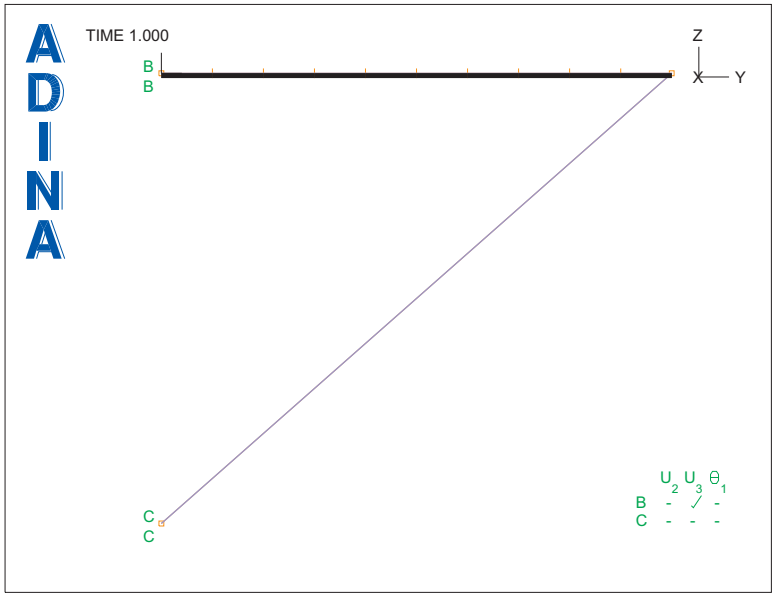
Modifying the solid model

Click the New icon (you can discard all changes), set the Program Module drop-down list to ADINA Structures, click the Open icon and open database file `prob49m_a_0.idb`.

Click the Coupling Options icon , set the Steered Adaptive Meshing Mode to Manual, set "Use Mesh In Previous Model at Solution Time" to 0.4, set the "Restart File from CFD Solution" to `prob49m_f_0.res` and click OK.

Generating the ADINA Structures data file: First choose File→Save As and save the database to file `prob49m_a_1`. Click the Data File/Solution icon  , set the file name to `prob49m_a_1`, make sure that the Run Solution button is unchecked and click Save.

The graphics window should look something like this:



The plot does not show the deformations of the solid model, but these deformations are accounted for.

Running ADINA-FSI, loading the porthole file

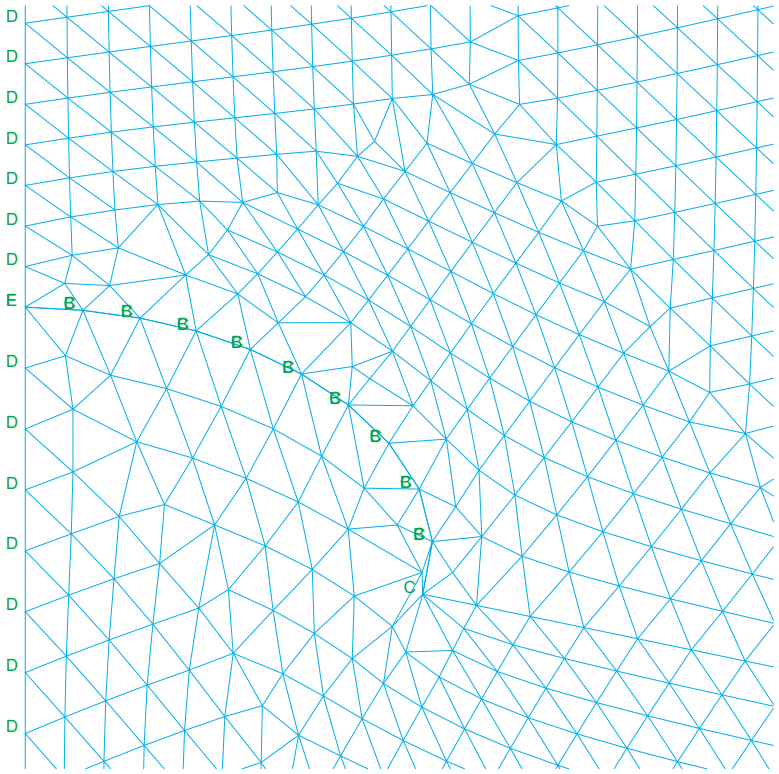
Choose Solution→Run ADINA-FSI, click the Start button, select files `prob49m_a_1.dat` and `prob49m_f_1.dat`, then click Start. The AUI opens a window in which you specify the restart file for the solid model. Enter restart file `prob49m_a_0.res` and click Copy. (Notice that you do not need to specify the restart file for the fluid model.)

The run ends at time step 16 (time 2.0).

Now close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes) and open file `prob49m_f_1.por`.

Examining the solution

Click the Boundary Plot icon and enlarge the region near the parachute. The graphics window should look something like this:



The mesh deformations are not excessive, but the mesh quality is not very good, so it is worthwhile to remesh again.

Second mesh refinement

Set the Program Module drop-down list to ADINA CFD (you can discard all changes). Do not open file `prob49m_f_1.idb`.

Mesh and results from previous solution: Choose Meshing→Steered Adaptive Mesh→ Control and set the Steered Adaptive Meshing Mode to Manual. Set "Use Mesh in Previous Model at Solution Time" to 2.0. Set the "Restart File from CFD Solution" to `prob49m_f_1.res` and set the "File Containing Geometric Data from Previous Model" to `prob49m_f_1.adp`. Click OK, then click the Mesh Plot icon to show the mesh.

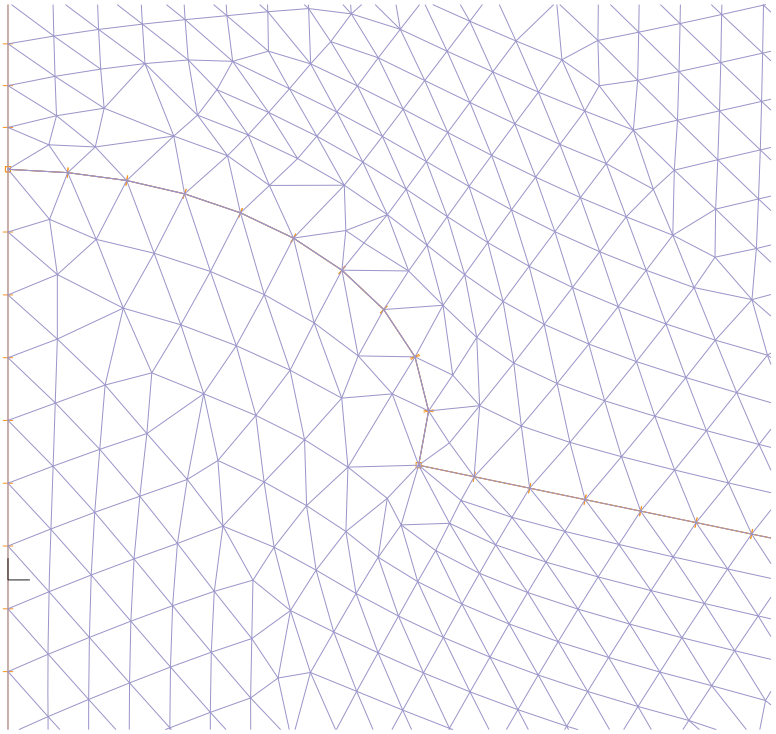
Adaptive meshing criteria: Choose Meshing→Steered Adaptive Mesh→Criterion and add criterion 1. Make sure that the Type is Element Quality and that the Solution Time is 2.0, then set the Minimum Element Quality to 0.8 and the Maximum Element Quality to 1.2 and click OK.

Remesh: Choose Meshing→Steered Adaptive Mesh→Mesh, set "Use Adaptive Mesh Criterion" to 1, and click OK.

Redefine time steps: Choose Control→Time Step, edit the table as follows and click OK.

Number of Steps	Magnitude
1	2.0
8	1.0

Click the Redraw icon  and zoom to see the mesh near the parachute. The graphics window should look something like this.



Generating the ADINA CFD data file: First click the Save icon and save the database to file `prob49m_f_2`. Click the Data File/Solution icon, set the file name to `prob49m_f_2`, make sure that the Run Solution button is unchecked and click Save.

Modifying the solid model

Click the New icon (you can discard all changes), set the Program Module drop-down list to ADINA Structures, click the Open icon and open database file `prob49m_a_1.idb`.

Click the Coupling Options icon, make sure that the Steered Adaptive Meshing Mode is set to Manual, set the "Use Mesh In Previous Model at Solution Time" to 2.0, set the "Restart File from CFD Solution" to `prob49m_f_1.res` and click OK.

Generating the ADINA Structures data file: First choose File→Save As and save the database to file `prob49m_a_2`. Click the Data File/Solution icon, set the file name to `prob49m_a_2`, make sure that the Run Solution button is unchecked and click Save.

The graphics window should look very similar to the window shown for the previous solid models.

Running ADINA-FSI, loading the porthole file

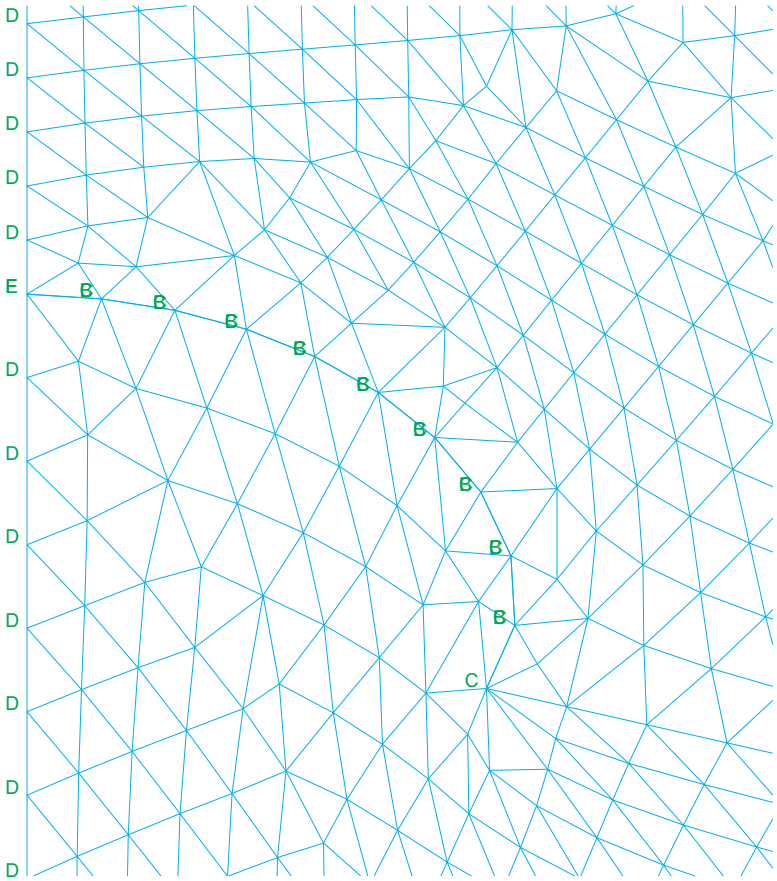
Choose Solution→Run ADINA-FSI, click the Start button, select files `prob49m_a_2.dat` and `prob49m_f_2.dat`, then click Start. The AUI opens a window in which you specify the restart file for the solid model. Enter restart file `prob49m_a_1.res` and click Copy.

The run ends at time step 8 (time 10.0).

Now close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes) and open file `prob49m_f_2.por`.

Examining the solution

Click the Boundary Plot icon and enlarge the region near the parachute. The graphics window should look something like the figure on the next page.



Use the Previous Solution icon ◀, Next Solution icon ▶, … to look at the mesh for different steps. The mesh does not change very much after the first step.

As a check on the model, we will now uniformly refine the mesh and continue the analysis.

Third mesh refinement

Set the Program Module drop-down list to ADINA CFD (you can discard all changes). Do not open file `prob49m_f_2.idb`.

Mesh and results from previous solution: Choose Meshing→Steered Adaptive Mesh→ Control and set the Steered Adaptive Meshing Mode to Manual. Set "Use Mesh in Previous Model at Solution Time" to 10.0. Set the "Restart File from CFD Solution" to `prob49m_f_2.res` and set the "File Containing Geometric Data from Previous Model" to

`prob49m_f_2.adp`. Click OK, then click the Mesh Plot icon to show the mesh.

Adaptive meshing criteria: Choose Meshing→Steered Adaptive Mesh→Criterion and add criterion 1. Set the Type to Element Size and make sure that the Solution Time is 10.0, then set the Minimum Element Size to 3.0, the Maximum Element Size to 1.0, the Preferred Ratio to 0.5 and click OK.

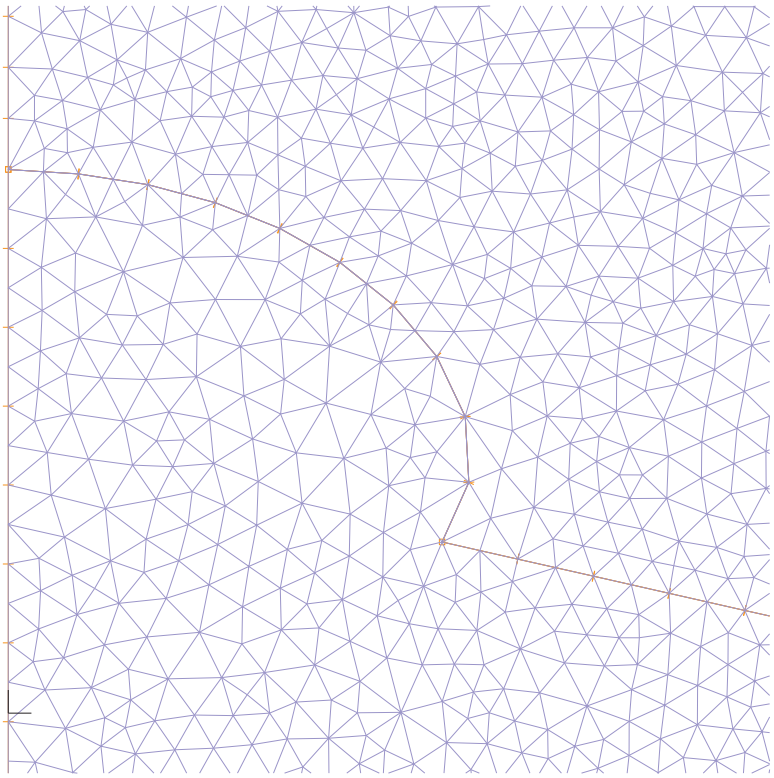
Remesh: Choose Meshing→Steered Adaptive Mesh→Mesh, set "Use Adaptive Mesh Criterion" to 1, and click OK.

Redefine time steps: Choose Control→Time Step, edit the table as follows and click OK:

Number of Steps	Magnitude
1	10.0
3	10.0

Click the Redraw icon and zoom to see the mesh near the parachute. The graphics window should look something like the figure on the next page.

Generating the ADINA CFD data file: First click the Save icon and save the database to file `prob49m_f_3`. Click the Data File/Solution icon, set the file name to `prob49m_f_3`, make sure that the Run Solution button is unchecked and click Save.



Modifying the solid model

Click the New icon (you can discard all changes), set the Program Module drop-down list to ADINA Structures, click the Open icon and open database file `prob49m_a_2.idb`.

Click the Coupling Options icon , make sure that the Steered Adaptive Meshing Mode is set to Manual, set the "Use Mesh In Previous Model at Solution Time" to 10.0, set the "Restart File from CFD Solution" to `prob49m_f_2.res` and click OK.

Generating the ADINA Structures data file: First choose File→Save As and save the database to file `prob49m_a_3`. Click the Data File/Solution icon, set the file name to `prob49m_a_3`, make sure that the Run Solution button is unchecked and click Save.

Running ADINA-FSI, loading the porthole file

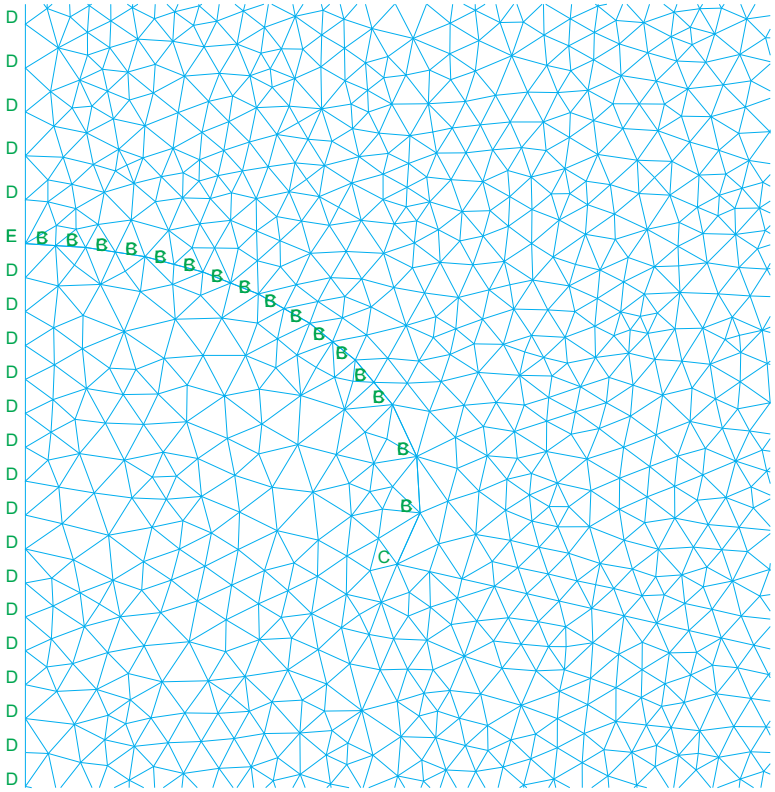
Choose Solution→Run ADINA-FSI, click the Start button, select files `prob49m_a_3.dat` and `prob49m_f_3.dat`, then click Start. The AUI opens a window in which you specify the restart file for the solid model. Enter restart file `prob49m_a_2.res` and click Copy.

The run ends at time step 3 (time 40.0).

Now close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes) and open file `prob49m_f_3.por`.

Examining the solution

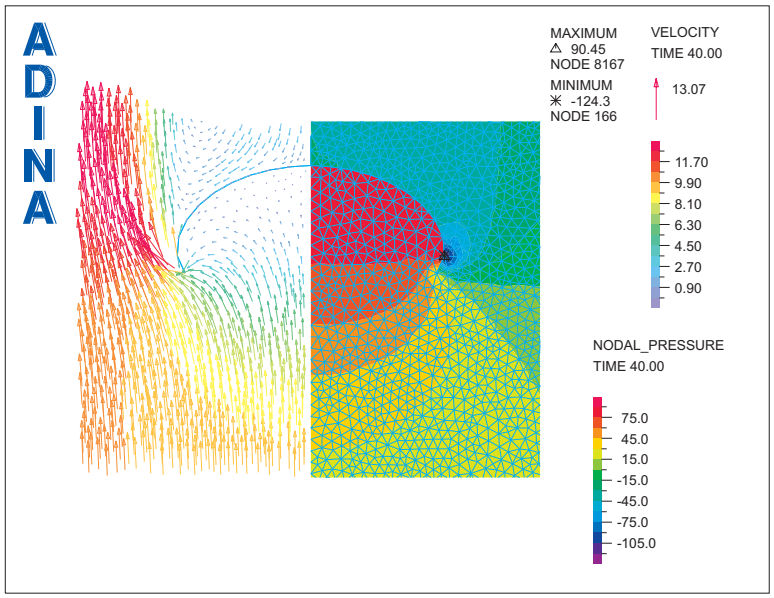
Click the Boundary Plot icon  and enlarge the region near the parachute. The graphics window should look something like this:



Use the Previous Solution icon , Next Solution icon , … to look at the solution for different steps. Again, the solution does not change very much after the first step.

Comparison of fluid solutions

We can load all of the fluid model porthole files together, so that we can examine how the mesh moves and changes during the analysis. We have put the necessary commands in file `prob49m1_f.plo`. Click the New icon (you can discard all changes), choose File→ Open Batch and open file `prob49m1_f.plo`. The graphics window should look something like this:



In this plot, we have plotted the mesh twice, once using mirror-imaging. We have plotted the velocities in the left-hand mesh and the pressure in the right-hand mesh.

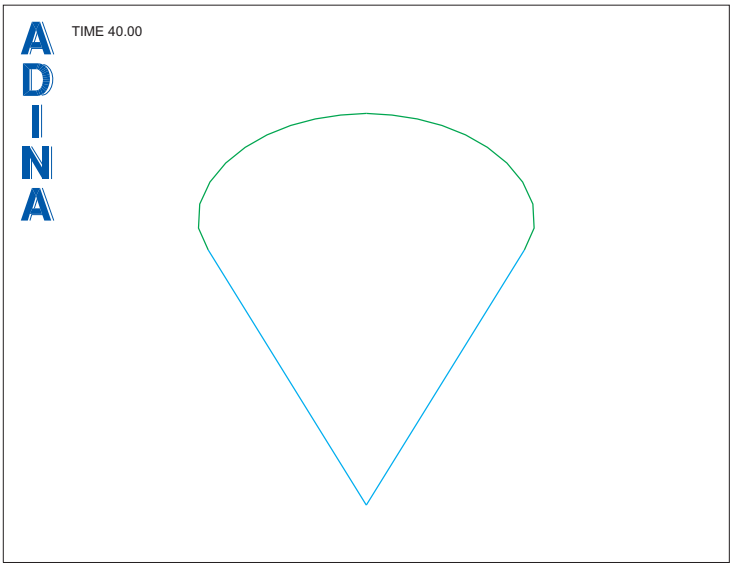
See the notes at the end of this primer problem for a description of how the mirror-imaging is done.

Click on the right-hand mesh using the Pick icon , then click the Previous Solution icon several times, until the topology of the mesh changes. Only the right-hand mesh and its solution is updated. Click the Last Solution icon to display the last mesh and solution, then click on the left-hand mesh and examine its solutions in the same way. For each of the meshes and its solutions, there is very little change for the last few steps.

Now click the Movie Load Step icon to create an animation. Choose Display→Animate, set the Minimum Delay if necessary and click Apply. Both meshes and their solutions are animated.

Parachute deformed shape and force acting on parachute

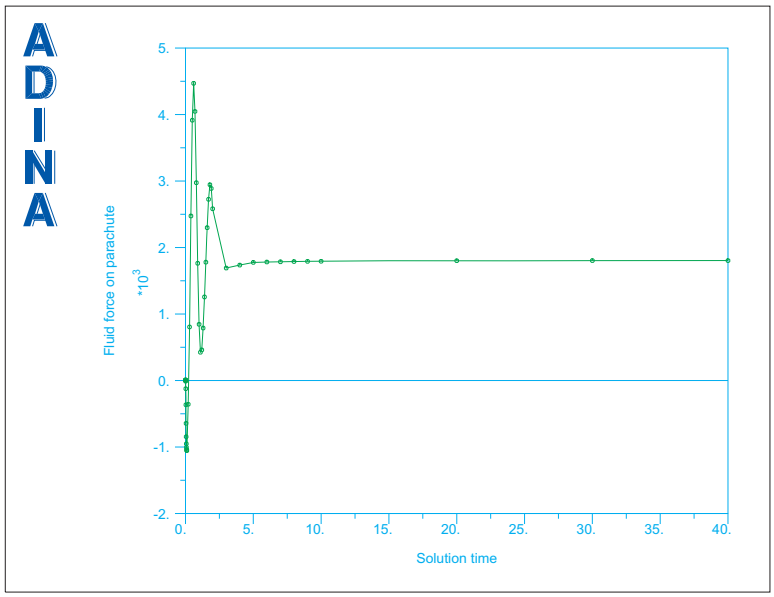
We can load all of the solid model porthole files together, so that we can examine how the parachute moves during the analysis. We have put the necessary commands in file `prob49m1_a.plo`. Click the New icon (you can discard all changes), choose File→ Open Batch and open file `prob49m1_a.plo`. The graphics window should look something like this:



In this plot, we have plotted the mesh twice, once using mirror-imaging.

Click the Movie Load Step icon to create an animation, then click the Animate icon to play the animation. Both meshes are animated.

Now click the Refresh icon, then click the Batch Continue icon. The graphics window should look something like the figure on the next page.



The graph shows the upwards force acting on the parachute for all of the solutions. Choose Graph→List and scroll to the bottom of the dialog box. The force at the last solution time is 1.80032E+03.

Although the graph shows the transient solution as well as the steady-state solution, the transient solution is most likely inaccurate because it is obtained using remeshings. The purpose of the transient analysis is only to obtain a steady-state solution.

Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.

Solution using automatic mode

Recall that the file name format in SAM automatic mode is `*_a_#.in` and `*_f_#.in`. The "*" represents the problem name and the # represents the model number, with 0 for the first model, 1 for the second model, and so on.

In this automatic mode analysis, we will start with a run in which the mesh is repaired when mesh overlap occurs. This run proceeds to solution time 2.0. Then we will restart with an automatic run in which the mesh is refined after every solution step.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA CFD.

Choose a working directory or folder in which the absolute pathname does not contain any spaces, for example `C:\temp` for Windows. The solution using automatic mode will fail if there are spaces in the absolute pathname.

Model definition for fluid model

We have prepared a batch file (`prob49a_f_0.in`) that contains all of the model definition for the fluid model, including the selection of the adaptive CFD feature. This file is the same as the batch file used for manual mode (`prob49m_f_0.in`), except that

1) The time stepping for automatic mode is different. We want to run to time 2.0 in the first part of the automatic mode run.

2) There are two criteria already defined in the automatic mode file. The first criterion is the same as used for the manual mode mesh refinements:

> Element quality, minimum element quality = 0.8, maximum element quality = 1.2, preferred quality = 1.0 (the default)

and the second criterion is a criterion for reducing the sizes of elements:

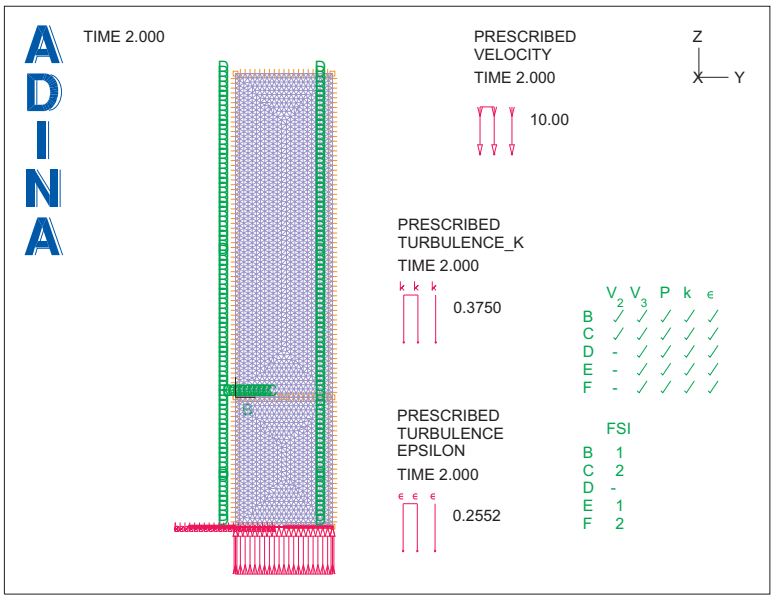
> Element size, minimum element size = 3, maximum element size = 1, preferred ratio = 0.75

We will use the first criterion for the first run (mesh repair) and the second criterion for the second run (mesh refinement).

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob49a_f_0.in` and click Open. The graphics window should look something like the figure on the next page.

Selecting steered adaptive meshing in automatic mode: Choose Meshing→Steered Adaptive Mesh→Control, set the Steered Adaptive Meshing Mode to Automatic, set 'Criterion for Automatic Steered Adaptive Mesh' to 1, set 'Use Mesh in Previous Model at Solution Time' to -1, and click OK.

Generating the ADINA CFD data file: First click the Save icon  and save the database to file `prob49a_f_0`. Click the Data File/Solution icon  , set the file name to `prob49a_f_0`, make sure that the Run Solution button is unchecked and click Save.



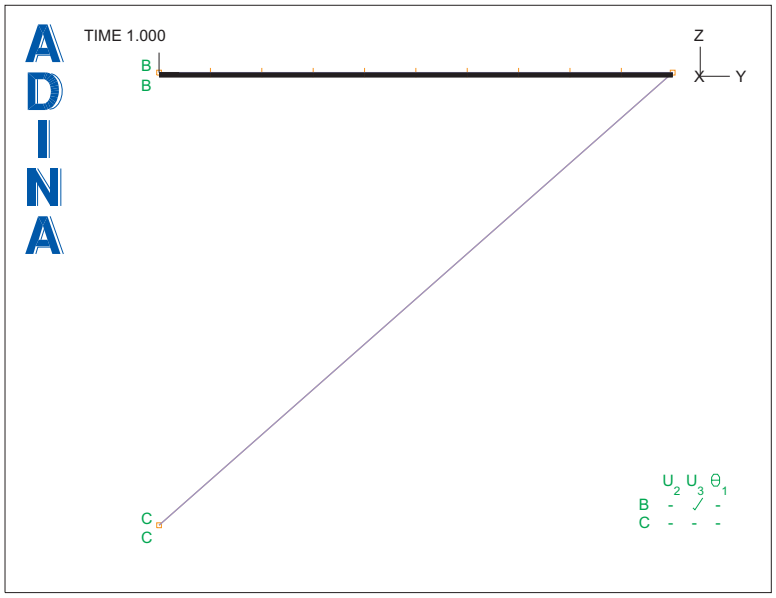
Model definition for solid model

We have prepared a batch file (`prob49a_a_0.in`) that contains all of the model definition for the solid model. This is the same file as is used in the manual mode analysis.

Click the New icon (you can discard all changes), then choose File→Open Batch, navigate to the working directory or folder, select the file `prob49a_a_0.in` and click Open. The graphics window should look something like the figure on the next page.

Selecting steered adaptive meshing in automatic mode: Click the Coupling Options icon , set the Steered Adaptive Meshing Mode to Automatic, set 'Use Mesh in Previous Model at Solution Time' to -1 and click OK.

Generating the ADINA Structures data file: First click the Save icon and save the database to file `prob49a_a_0`. Click the Data File/Solution icon , set the file name to `prob49a_a_0`, make sure that the Run Solution button is unchecked and click Save.



Running ADINA-FSI in automatic mode using the Adaptive Meshing Solution Interface

If you have only one AUI floating license, you need to exit the AUI before using the Adaptive Meshing Solution Interface.

Choose Solution→Run Steered Adaptive and click the Start button. Select file `prob49a_f_0.dat`, then hold down the Ctrl key and select file `prob49a_a_0.dat`. The File name field should display both files in quotes. Set "Maximum Number of Adaptive Steps" to 5, set "Number of Solution Runs " to 1, set "Memory for AUI" to 50 MB, "Max. Memory for Solution" to 100 MB and click Start.

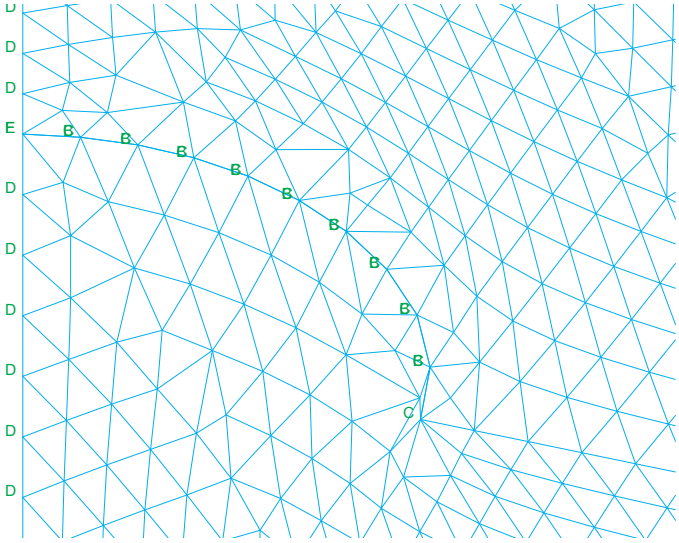
ADINA CFD and the AUI are run consecutively. Eventually the message

```
Finished adaptive run for ... prob49a_0 to ... prob49a_1
```

is displayed.

Now 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 `prob49a_f_1`.

Click the Boundary Plot icon  and enlarge the region near the parachute. The graphics window should look something like the figure on the next page.



Restarting SAM automatic mode

Set the Program Module to ADINA CFD (you can discard all changes). Do not open any database file.

At this point, we would like to continue remeshing, but using a different criterion.

Using a text editor, open file `prob49a_f_adp.in`. You will observe the following text:

```
ADP-CONTROL MESHADAP=MANUAL TSTART= -1.000000000000,
    FRSFILE=prob49a_f_adp.res,
    ADAPTIVE=prob49a_f_adp.adp
*
TIMESTEP DEFAULT
@CLEAR
        10      0.100000000000000E-02
         9      0.100000000000000E-01
        19      0.100000000000000E+00
@
*
```

Change the commands to

```
ADP-CONTROL MESHADAP=MANUAL TSTART= -1.000000000000,
    FRSFILE=prob49a_f_adp.res,
    ADAPTIVE=prob49a_f_adp.adp,
    TIMESTEP=APPEND
*
TIMESTEP DEFAULT
@CLEAR
        10      0.100000000000000E-02
         9      0.100000000000000E-01
        19      0.100000000000000E+00
        1     10.0
@
*
```

(the changed text is highlighted in blue and deleted text is indicated with strikethrough). Then scroll to the bottom of the file, and change the text

```
ADP-MESH CRITERION=            1
```

to

```
ADP-MESH CRITERION=            2
```

Save the file.

Choose Solution→Run Steered Adaptive and click the Start button. Select file `prob49a_f_0.dat`, then hold down the Ctrl key and select file `prob49a_a_0.dat`. The File name file should display both files in quotes. Set " Run Analysis from Adaptive Step" to 2, "Maximum Number of Adaptive Steps" to 3, "Number of Solution Runs" to 3, set "Memory for AUI" to 50 MB, "Max. Memory for Solution" to 100 MB and click Start.

ADINA CFD and the AUI are run consecutively. Eventually the message

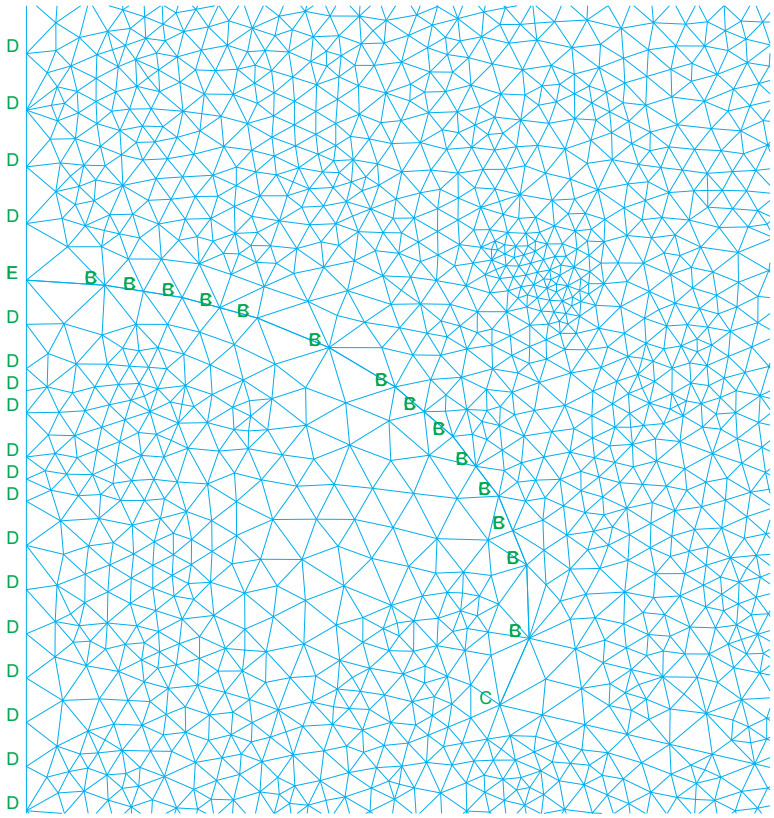
```
Finished adaptive run for ... prob49a_2 to ... prob49a_4
```

is displayed.

Now close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes) and open file `prob49a_f_4.por`.

Examining the solution

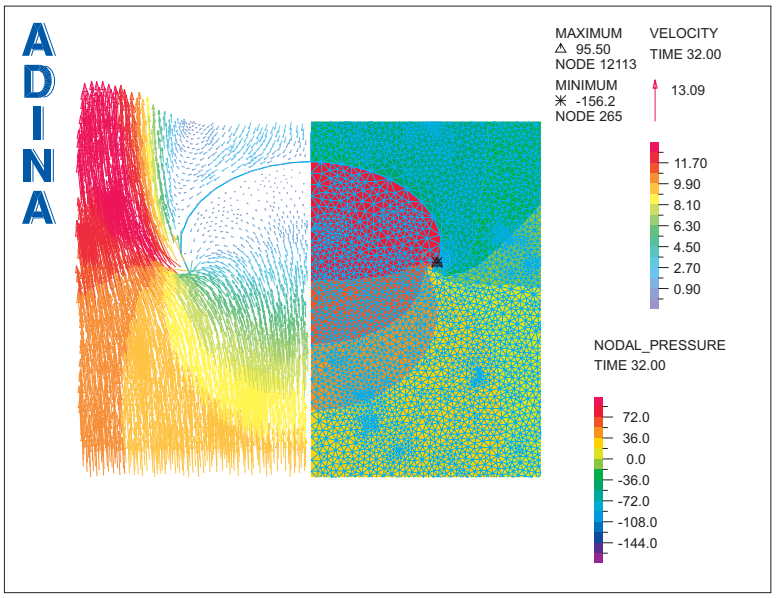
Click the Boundary Plot icon  and enlarge the region near the parachute. The graphics window should look something like the figure on the next page.



Comparison of fluid solutions

We can load all of the fluid model porthole files together, so that we can examine how the mesh moves and changes during the analysis. We have put the necessary commands in file `prob49a1_f.plo`. Click the New icon (you can discard all changes), choose File→ Open Batch and open file `prob49a1_f.plo`. The graphics window should look something like the figure on the next page.

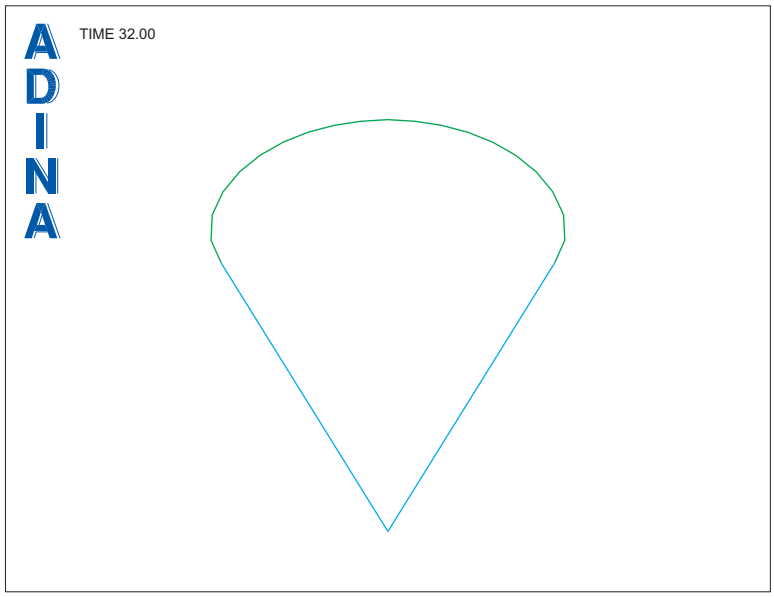
You can animate this plot, as discussed in the manual SAM mode section above.



Parachute deformed shape and force acting on parachute

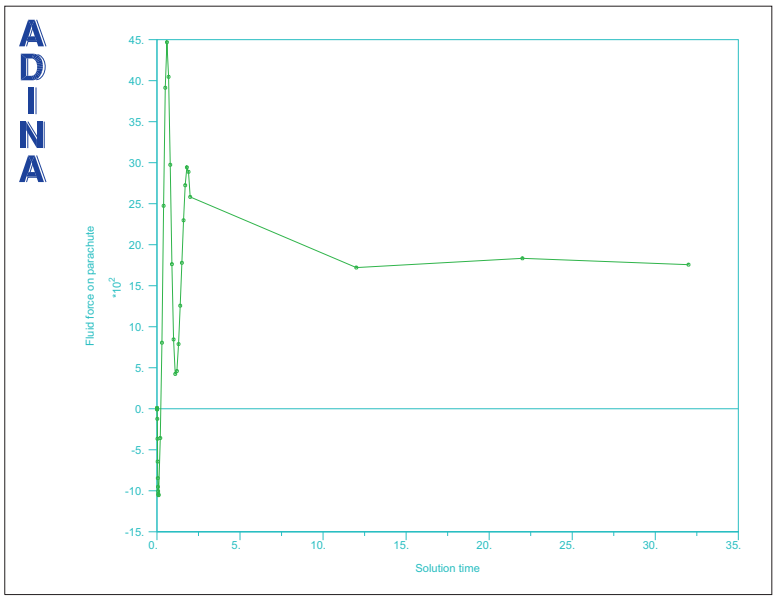
We can load all of the solid model porthole files together, so that we can examine how the parachute moves during the analysis. We have put the necessary commands in file `prob49a1_a.plo`. Click the New icon (you can discard all changes), choose File→ Open Batch and open file `prob49a1_a.plo`. The graphics window should look something like the top figure on the next page.

In this plot, we have plotted the mesh twice, once using mirror-imaging.



You can animate this plot, as discussed in the manual SAM mode section above. Now click the Batch Continue icon 

. The graphics window should look something like this:



The graph shows the upwards force acting on the parachute for all of the solutions. Choose Graph→List and scroll to the bottom of the dialog box. The force at the last solution time is 1.75673E+03.

Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.

Additional notes

Mesh refinements

For all but the last mesh refinement, we use the criterion

> Element quality, minimum element quality = 0.8, maximum element quality = 1.2, preferred quality = 1.0 (the default)

The intent of this criterion is to set the preferred element quality to 1.0, for those elements in which the quality is less than 0.8 or greater than 1.2.

For the last mesh refinement in manual SAM, we use the criterion

> Element size, minimum element size = 3, maximum element size = 1, preferred ratio = 0.5

This sets the preferred element size to 0.5 of the current element size. Since the maximum element size is less than the minimum element size, all elements are affected by this command.

For the last mesh refinement in automatic SAM, we use the criterion

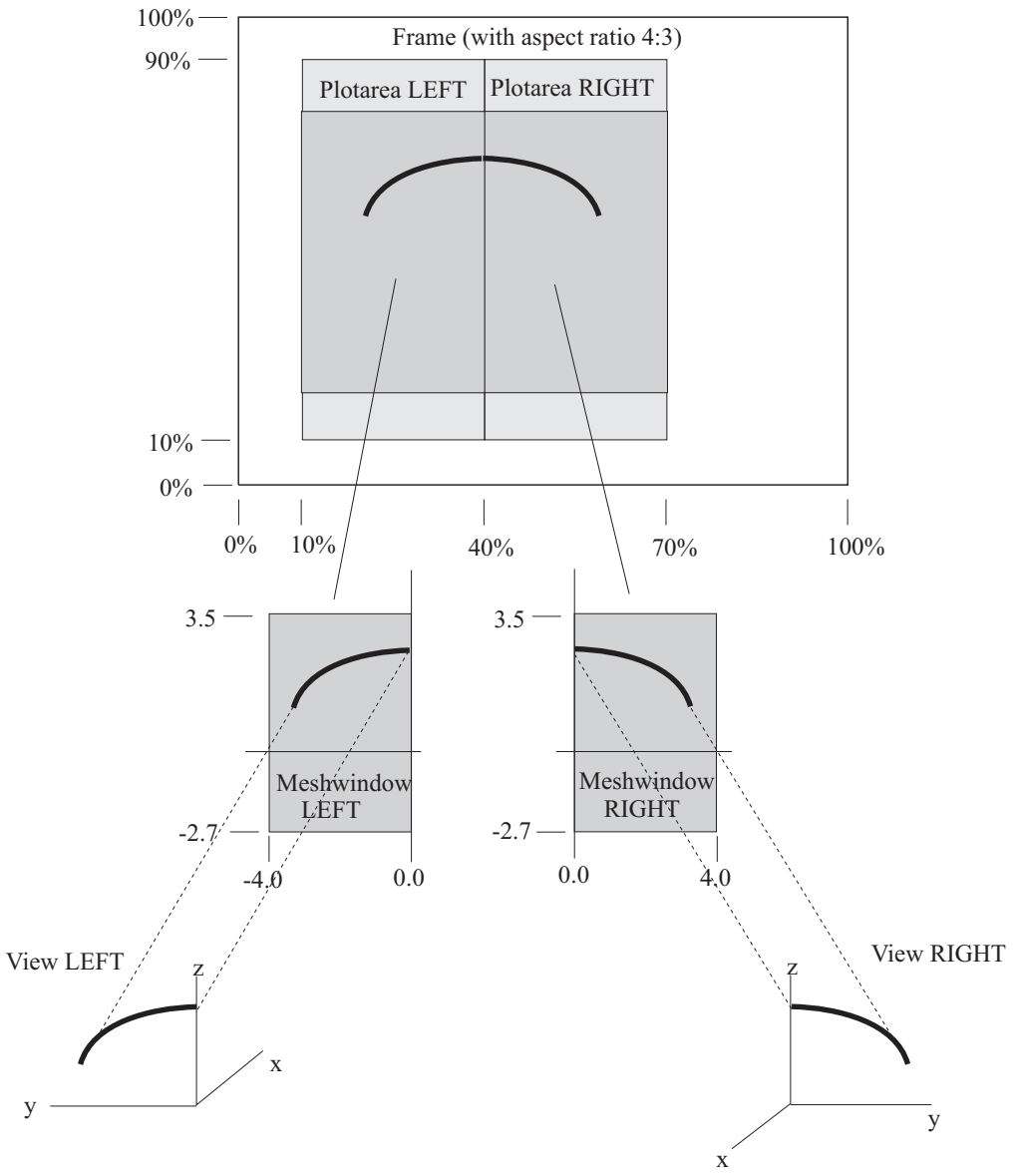
> Element size, minimum element size = 3, maximum element size = 1, preferred ratio = 0.75

which is similar to the one used in manual SAM, but reduces the element size more slowly.

For more information, see the AUI Commands for CFD & FSI manual and the ADINA CFD Theory and Modeling Guide.

Mirror-imaging

The figure on the next page shows how the mirror-imaging is done. The mesh is plotted twice, once to create the right-hand image (without mirror-imaging) and once to create the left-hand image (with mirror-imaging). The view, meshwindow and plotarea for each mesh are chosen as shown in the figure.


100%
90%
10%
0%
Frame (with aspect ratio 4:3)
Plotarea LEFT
Plotarea RIGHT
0%
10%
40%
70%
100%
3.5
-2.7
Meshwindow LEFT
-4.0
0.0
3.5
-2.7
Meshwindow RIGHT
0.0
4.0
View LEFT
z
x
y
View RIGHT
z
y
x

This page intentionally left blank.