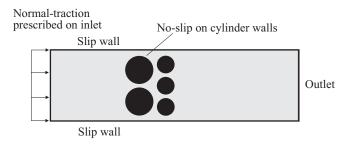
# **Problem description**

Consider the following fluid flow problem:



Steady-state laminar flow conditions are assumed.

It is desired to calculate the inlet mass flux and total fluid force acting on the cylinders using ADINA CFD.

We will use the steered adaptive meshing (SAM) features in order to obtain a refined fluid mesh. We will solve the problem twice, once using the manual mode features of SAM, and the second time using the automatic mode features of SAM.

Recall that a CFD/FSI computation with SAM may have one or more restart runs. Each restart run is referred to as a new model since it has a new mesh for the fluid, and the fluid material properties, loads, initial conditions and boundary conditions can be changed during restarts. The model at the initial time is referred to as the first model, the model at the first restart time is referred to as the second model, and so on.

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.
- Using the SAM features in automatic mode.

#### 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 there are too many nodes in the model.

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

```
prob48m_0.in, prob48m1.plo
prob48a_0.in, prob48a1.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

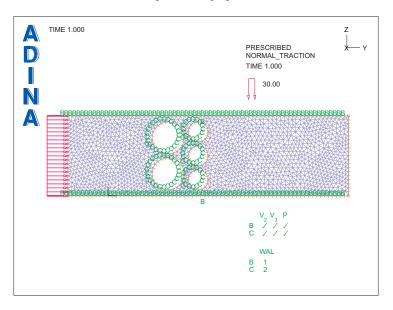
This problem is created using ADINA-M/OC (ADINA-M with OpenCascade geometry modeler). The Parasolid geometry modeler can also be used, but only if the input is modified.

Invoke the AUI in OCC mode (for example, using the command aui 9.3 -occ on Linux platforms) and set the Program Module drop-down list to ADINA CFD.

## **Model definition**

We have prepared a batch file (prob48m\_0.in) that contains all of the model definition except for the selection of the adaptive CFD feature.

Choose File→Open Batch, navigate to the working directory or folder, select the file prob48m\_0.in and click Open. The graphics window should look something like this:



48-2 ADINA Primer

You can view the file prob48m\_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, running ADINA CFD, loading the porthole file

First click the Save icon and save the database to file prob48m\_0. Click the Data File/Solution icon, set the file name to prob48m\_0, make sure that the Run Solution button is checked and click Save.

When ADINA CFD is finished, examine the Log Window. The message "Adaptive mesh file ....adp successfully created." is written. As you will see, this file is used during adaptive meshing.

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 , set the "File Type" field to "ADINA-IN Database Files (\*.idb)", open database file probm48\_0, click the Open icon and open porthole file prob48m\_0.

We first opened the ADINA-IN database, then loaded the porthole file. We did this so that we can use the geometry during the flux and force calculations.

## **Examining the solution**

We have put the commands for calculating the inlet mass flux and force on cylinders into file prob48m1.plo. Choose File—Open Batch, navigate to the working directory or folder, select the file prob48m1.plo and click Open. The results are listed in the Message Window:

Listing for point INLET\_ELEDGESET
TIME FLUX

1.00000E+00 1.97932E+00

```
Listing for point CYLINDERS
TIME FLUID_FORCE

1.00000E+00 1.07457E+02
```

(you might need to use the scrollbar to see these listings). Your results might be slightly different from ours because free-form meshing is used.

You can look at the prob48m1.plo file to see the techniques used to obtain these values. The flux is calculated by integrating the y velocity along the element edges at the inlet (the density is 1.0 in this problem). The element edges at the inlet are selected using the ELEDGESET command using option LINE-EDGE (this option requires the geometry). The fluid force is calculated by summing the reaction forces on all of the nodes on the cylinders. The cylinder nodes are selected using the GNCOMBINATION command with EDGE selections (again, this option requires the geometry).

#### First mesh refinement

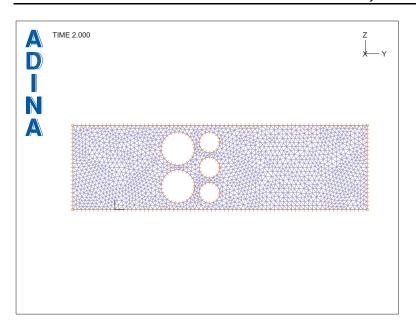
Set the Program Module drop-down list to ADINA CFD (you can discard all changes). Do not open file prob48m\_0.idb.

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 1. Set the "Restart File from CFD Solution" to prob48m\_0.res as follows: click ... to the right of the "Restart File from CFD Solution" field, choose file prob48m\_0 and click Open. Set the "File Containing Geometric Data from Previous Model"

to prob48m\_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 same mesh as was already used, except that the solution time is 2.0 instead of 1.0.

48-4 ADINA Primer

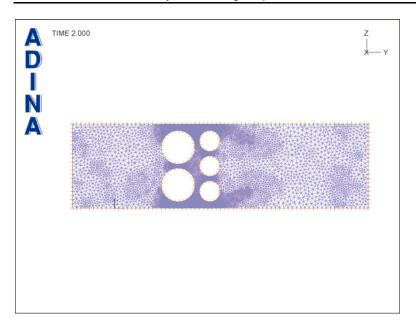


Adaptive meshing criteria: Choose Meshing—Steered Adaptive Mesh—Criterion and add criterion 1. Set the Type to Element Size, the Minimum Element Size to 3 and the Preferred Ratio to 0.8. Add criterion 2, set the Type to Variable Gradient, make sure that the Variable Name is PRESSURE, set the Minimum Element Size to 0 and the Preferred Ratio to 0.5. Click the Copy button, copy to criterion 3, and set the Variable Name to VORTICITY. Add criterion 4, set the Type to Combination, and the Number of Smoothing to 2. Then edit the table as follows and click OK.

Criterion #	Action
1	Append Elements
2	Use Smaller Elements
3	Use Smaller Elements

(see notes at the end of this problem for an explanation of these criteria).

Remesh: Choose Meshing→Steered Adaptive Mesh→Mesh, set "Use Adaptive Mesh Criterion" to 4, and click OK. The graphics window should look something like the figure on the next page.



*Time stepping:* Choose Control→Time Step, add a row to the table so that the table appears as follows, then click OK.

Number of Steps	Magnitude
1	1.0
1	1.0

# Generating the ADINA CFD data file, running ADINA CFD, loading the porthole file

First click the Save icon and save the database to file prob48m\_1. Click the Data File/Solution icon, set the file name to prob48m\_1, make sure that the Run Solution button is checked, make sure that the Maximum Memory for Solution is at least 100 MB and click Save.

When ADINA CFD is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon "File Type" field to "ADINA-IN Database Files (\*.idb)", open database file prob48m\_1, click the Open icon and open porthole file prob48m\_1.

48-6 ADINA Primer

#### **Examining the solution**

We can use the same .plo file as we used earlier to examine the results. Choose File→ Open Batch, navigate to the working directory or folder, select the file prob48m1.plo and click Open. The results are listed in the Message Window:

```
Listing for point INLET_ELEDGESET FLUX

2.00000E+00 2.81502E+00

...

Listing for point CYLINDERS FLUID_FORCE

2.00000E+00 1.19571E+02
```

Both the flux and the fluid force have significantly increased.

#### Second mesh refinement

Set the Program Module drop-down list to ADINA CFD (you can discard all changes). Do not open file prob48m\_1.idb. Choose Edit→Memory Usage and make sure that the ADINA-AUI memory is at least 400 MB.

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. Set the "Restart File from CFD Solution" to prob48m\_1.res, set the "File Containing Geometric Data from Previous Model" to prob48m\_1.adp and click

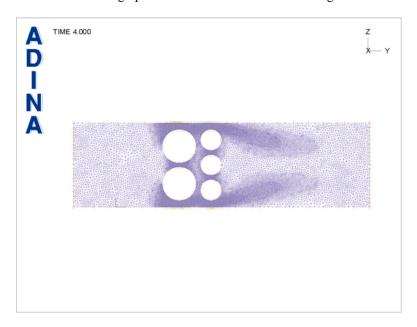
OK. When you click the Mesh Plot icon the graphics window should look like the previous figure, except that the solution time is 4.0 instead of 3.0.

Adaptive meshing criteria, remeshing and time stepping: We use the same commands as before. The instructions are repeated here for convenience.

Choose Meshing—Steered Adaptive Mesh—Criterion and add criterion 1. Set the Type to Element Size, the Minimum Element Size to 3 and the Preferred Ratio to 0.8. Add criterion 2, set the Type to Variable Gradient, make sure that the Variable Name is PRESSURE, set the Minimum Element Size to 0 and the Preferred Ratio to 0.5. Click the Copy button, copy to criterion 3 and set the Variable Name to VORTICITY. Add criterion 4, set the Type to Combination, and the Number of Smoothing to 2. Then edit the table as follows and click OK.

Criterion #	Action
1	Append Elements
2	Use Smaller Elements
3	Use Smaller Elements

Choose Meshing—Steered Adaptive Mesh—Mesh, set "Use Adaptive Mesh Criterion" to 4, and click OK. The graphics window should look something like this:



Choose Control→Time Step, add a row to the table so that the table appears as follows, then click OK.

Number of Steps	Magnitude
1	1.0
1	1.0
1	1.0

# Generating the ADINA CFD data file, running ADINA CFD, loading the porthole file

First click the Save icon and save the database to file prob48m\_2. Click the Data

File/Solution icon , set the file name to prob48m\_2, make sure that the Run Solution button is checked, make sure that the Maximum Memory for Solution is at least 500 MB (preferably at least 1700 MB) and click Save.

48-8 ADINA Primer

When ADINA CFD is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon "File Type" field to "ADINA-IN Database Files (\*.idb)", open database file prob48m\_2, click the Open icon and open porthole file prob48m\_2.

### **Examining the solution**

We can use the same .plo file as we used earlier to examine the results. Choose File→ Open Batch, navigate to the working directory or folder, select the file prob48ml.plo and click Open. The results are listed in the Message Window:

```
Listing for point INLET_ELEDGESET FLUX

3.00000E+00 2.85770E+00

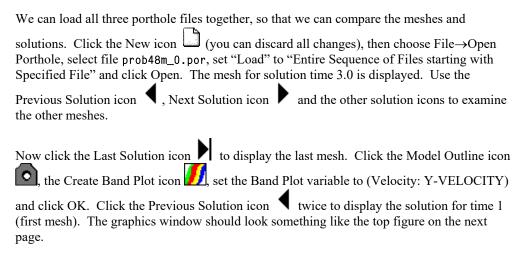
...

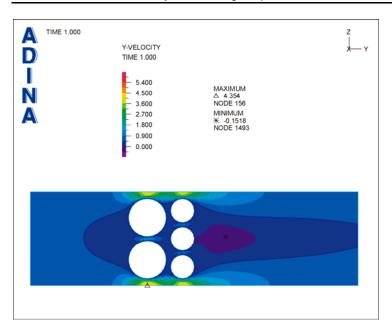
Listing for point CYLINDERS FLUID_FORCE

3.00000E+00 1.19716E+02
```

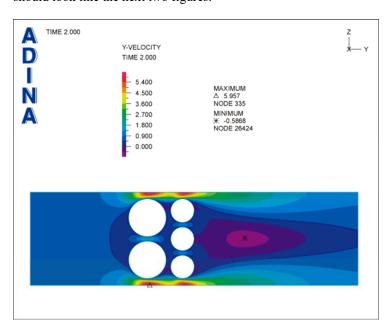
Both the flux and the fluid force have increased, but not as much as between the initial mesh and first mesh refinement.

### Comparison of the meshes and solutions

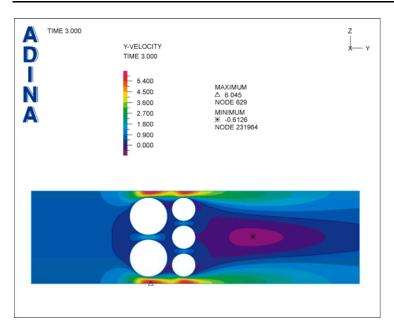




Now click the Next Solution icon twice to display the other solutions. These solutions should look like the next two figures:



**48-10** ADINA Primer



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

# Solution using automatic mode

The setup of the first model in SAM automatic mode is the same as for SAM manual mode except for the setup of SAM itself. Recall that the file name format in SAM automatic mode is \*\_#.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.

### Invoking the AUI and choosing the finite element program

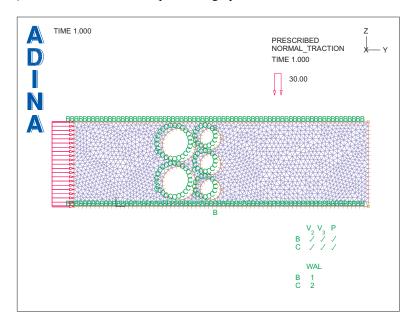
Invoke the AUI in OCC mode (for example, using the command aui 9.3 -occ on Linux platforms) and set the Program Module drop-down list to ADINA CFD.

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**

We have prepared a batch file (prob48a\_0.in) that contains all of the model definition except for the selection of the adaptive CFD feature. We see that the filename prob48a\_0.in conforms to the SAM automatic mode file naming convention for the first model.

Choose File→Open Batch, navigate to the working directory or folder, select the file prob48a\_0.in and click Open. The graphics window should look something like this:



The batch files prob48m\_0.in and prob48a\_0.in are identical, except that the adaptive meshing criteria are already added to prob48a\_0.in. The adaptive meshing criteria in prob48a\_0.in are exactly the same as the criteria used in the manual model analysis. You can view these criteria by choosing Meshing—Steered Adaptive Mesh—Criterion.

### Selecting steered adaptive meshing in automatic mode

Choose Meshing—Steered Adaptive Mesh—Control, set the Steered Adaptive Meshing Mode to Automatic, set the "Criterion for Automatic Steered Adaptive Mesh" to 4, set "Use Mesh in Previous Model at Solution Time" to -1, set "Adaptive Timestep Sequence" to "Appends Original Timestep" and click OK.

*Time stepping:* Choose Control→Time Step and add Time Step Name ADAPTIVE. Make sure that the table appears as follows, then click OK.

Number of Steps	Magnitude
1	1.0

Click OK to close the warning message "Timestep ADAPTIVE is not set to current."

48-12 ADINA Primer

### Generating the ADINA CFD data file

First click the Save icon and save the database to file prob48a\_0. Click the Data File/Solution icon, set the file name to prob48a\_0, make sure that the Run Solution button is unchecked and click Save.

The Log Window displays the messages

```
Adaptive mesh file ... prob48a_0.adp successfully created. Adaptive input file ... prob48a_adp.in successfully created. ADINA-F data input file ... prob48a_0.dat successfully created.
```

### Running ADINA CFD 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 Start. Set the file name to prob48a\_0.dat, set "Maximum Number of Adaptive Steps" to 2, set "Number of Solution Runs" to 3, set "Memory for AUI" to 400 MB, "Max. Memory for Solution" to at least 500 MB and click Start.

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

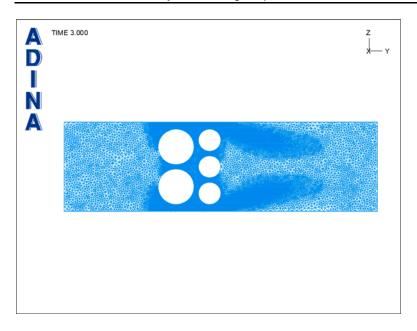
```
Fini shed adaptive run for ... prob48a_0 to ... prob48a_2 is displayed.
```

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 prob48a\_2. The graphics window should look something like the figure on the next page.

This is the same mesh that was generated using manual SAM above.

# **Examining the solution**

With automatic SAM, there is no .idb file corresponding to the refined mesh. Instead the geometry of the refined mesh can be obtained from the .res and .adp files from the refined mesh.



We use this procedure for the post-processing of the automatic SAM results. Choose File—Open Batch, navigate to the working directory or folder, select the file prob48a1.plo and click Open. The results are listed in the Message Window:

```
Listing for point INLET_ELEDGESET FLUX

3.00000E+00 2.85770E+00

...

Listing for point CYLINDERS FLUID_FORCE

3.00000E+00 1.19716E+02
```

These are exactly the same results as obtained using manual SAM.

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

48-14 ADINA Primer

### **Additional notes**

There are many adaptive meshing criteria that can be used in SAM. We use the criteria

- 1: Element size, minimum element size = 3, preferred ratio = 0.8
- 2: Variable gradient, variable PRESSURE, minimum element size = 0, preferred ratio=0.5
- 3: Variable gradient, variable VORTICITY, minimum element size = 0, preferred ratio=0.5
- 4: Combination: smoothing=2

Criterion #	Action
1	Append Elements
2	Use Smaller Elements
3	Use Smaller Elements

The output of the criterion commands is a list of preferred element sizes.

The intent of the first criterion "element size" is to set the preferred element size to PREF times the current element size. The parameter MIN=3 simply ensures that the preferred element size is set for all elements in the model. PREF=0.8 is used to reduce all of the preferred element sizes in the model "slightly", so that overall there is a slight mesh refinement (not considering the effects of the other criteria).

The intent of the second criterion "variable gradient" is to choose preferred element sizes such that the relative variable gradient of the pressure is the same for all elements. Namely,

$$h_{ep} = \frac{\lambda_T \ average \left(h_e \|F_e\|\right)}{\|F_e\|} \text{ where } h_e \text{ is the current element size, } \|F_e\| \text{ is the norm of the}$$

gradient of the pressure in the element,  $h_{ep}$  is the preferred element size and  $\lambda_T$  is the scaling factor PREF. Clearly elements with large pressure gradient will have smaller preferred element sizes. The parameter MIN=0 overrides the default value of the minimum allowed element size. Similarly, the intent of the third criterion is to choose preferred element sizes such that the relative variable gradient of the vorticity is the same for all elements.

The intent of the fourth criterion "combination" is to combine the above criteria such that the smallest preferred element size from each of the criteria is selected. Smooth = 2 smoothes out the resulting mesh twice.

For more information, see the ADINA CFD Theory and Modeling Guide.

48-16 ADINA Primer