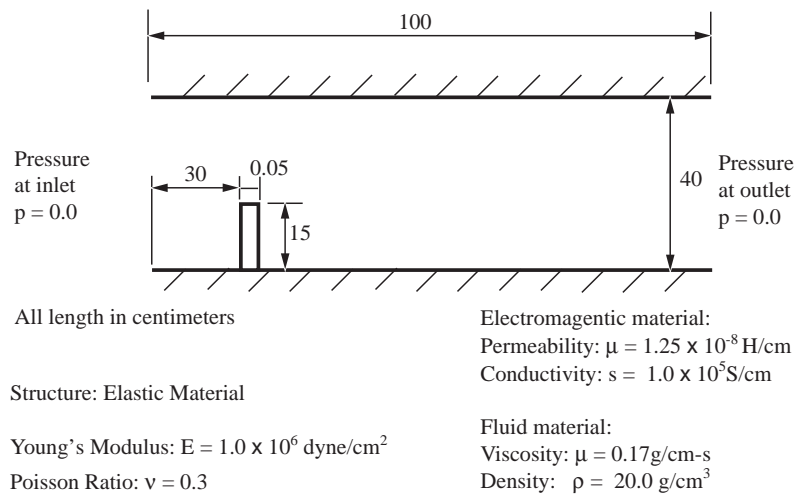


### Problem description

We determine the fluid flow driven by an electromagnetic Lorentz force and the deformation of a very thin flexible structure within a two-dimensional channel, as shown:



In this problem solution, we use an electromagnetic model, coupled with a fluid model for the fluid in the channel, and a solid model for the cantilever structure, which in turn is coupled to the fluid model. The electromagnetic model is entered as an ADINA EM model, the fluid model is entered as an ADINA CFD model and the solid model is entered as an ADINA Structures model. The analysis itself is performed as a fully coupled analysis between ADINA Structures, ADINA CFD and ADINA EM, however, only two data files are generated, since the ADINA CFD and ADINA EM models share the same data file.

This problem is somewhat similar to primer problem 18. Therefore we have put much of the input for this problem into batch files. Please refer to primer problem 18 for an example in which the user interface is used to define the input.

We will demonstrate the following topics in this problem solution:

- performing an electromagnetics/fluid-structure-interaction (EM/FSI) analysis in which separate AUI databases are used for the solid and fluid + electromagnetic models.
- switching between ADINA, ADINA-EM and ADINA-CFD models
- setting up the sparse solver for moving mesh in FSI iteration
- defining a slipping boundary

We assume that you have worked through problems 1 to 55, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

*Problem 56: Electromagnetic driven flow over a cantilever in a channel*

---

**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.

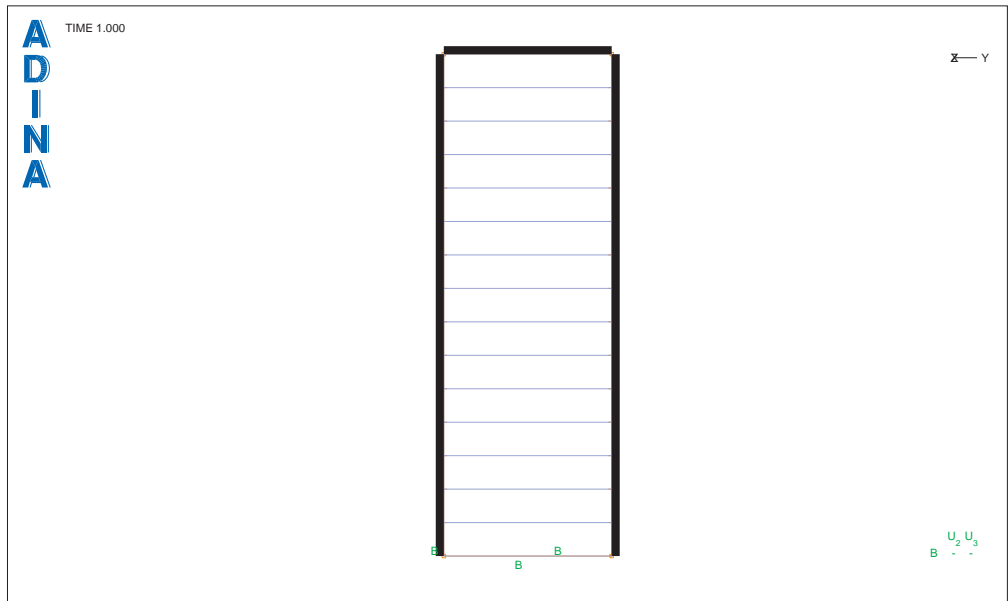
Much of the input for this problem is stored in the following files: prob56\_1.in, prob56\_2.in, prob56\_3.in. You need to copy these files from the folder samples\primer into a working directory or folder before beginning this analysis.

**ADINA Structures model**


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

Invoke the AUI and choose ADINA Structures from the Program Module drop-down list.

We put the entire structural model definition, including the generation of the data file, into batch file prob56\_1.in. Choose File→Open Batch, navigate to the working directory or folder, select the file prob56\_1.in and click Open. The graphics window should look something like this:



## ADINA CFD/ADINA EM model

Click the New icon  to create a new database (you can discard all changes). Choose ADINA CFD from the Program Module drop-down list.

### Defining model control data


*Problem heading:* Choose Control→Heading, enter the heading “Problem 56: electromagnetic driven flow over a cantilever in a channel --- Fluid + EM” and click OK.

*Multiphysics coupling:* Set the Multiphysics Coupling drop-down list to "with Structures & EM". The AUI displays a warning message. Click OK to close the warning message.

*Flow assumptions:* Choose Model→Flow Assumptions, set the Flow Dimension to 2D (in YZ Plane), uncheck the Includes Heat Transfer button and click OK.

*Miscellaneous options :* Choose Control→Miscellaneous Options, set the "Use the Physical Traction Boundary Condition" to "Yes", set the "Background Mesh used to update Moving Mesh" to "Original Mesh", and click OK to close the dialog box.

*Solution iteration:* Choose Control→Solution Process, click the Outer Iteration... button, then click the Advanced Settings... button. In the Outer Iteration Advanced Settings dialog box, set the Equation Residual to All and verify that the Tolerance is 0.0001. Also set the Tolerance for Variable Residual to 0.0001. Set the "Solver for Moving Mesh" to Sparse, and also set "Maximum Iterations In Velocity-Pressure Loop within VPT Loop" to 5. Click OK three times to close all three dialog boxes.


*Fluid structure interaction:* Click the Coupling Options icon , verify that the FSI Solution Coupling is Iterative, set the "Maximum Number of Fluid-Structure Iterations" to 30, and click OK.

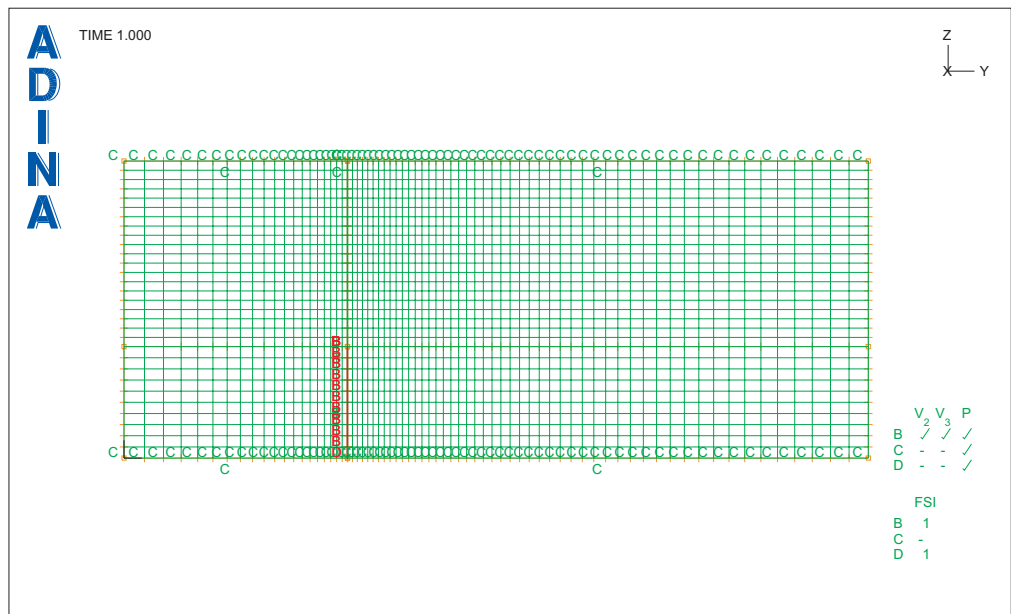
### Defining the ADINA CFD model

We have put much of the ADINA CFD model definition into batch file prob56\_2.in:


```
Time stepping
Time functions
Model geometry
Fluid material
Fluid boundary conditions
Meshing
```

*Problem 56: Electromagnetic driven flow over a cantilever in a channel*

Choose File→Open Batch, navigate to the working directory or folder, select the file prob56\_2.in and click Open. When you click the Color Element Groups icon , the graphics window should look something like this:



Element group 1 is used to model the fluid (including electromagnetic effects) and element group 2 is used to model the electromagnetic effects in the structure. Click the Define

Element Groups icon  and observe that element group 1 has "Fluid Element", "Electric Effects" and "Magnetic Effects" checked, and that element group 2 has just "Electric Effects" and "Magnetic Effects" checked.

**Defining slipping boundary and leader-follower relationships**

To keep good mesh quality, we want the points 2 and 3 moving along the lines 1, 2 and 3. So we define a slipping boundary along the lines 1, 2 and 3, and then define two pairs of leader-follower points.

*Problem 56: Electromagnetic driven flow over a cantilever in a channel*

Choose Meshing→ALE Mesh Constraints→Slipping Boundary, add boundary # 1, enter 1, 2, 3 in the first three rows of the table and click OK. Then choose Meshing→ALE Mesh Constraints→Leader-Follower and, in the table, enter the following leader-follower point pairs and click OK.

Label	Leader Point	Follower Point
1	7	3
2	6	2

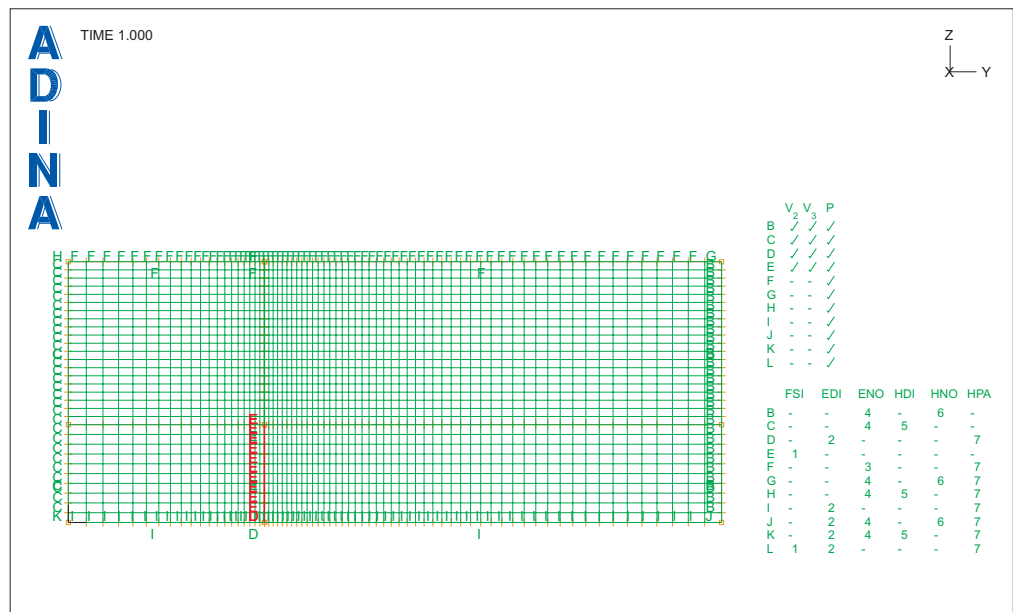
**Defining the ADINA EM model**

*Electromagnetic assumptions:* Choose Model→Electromagnetic→Settings, set the Model Type to "2D E-H model on magnetic plane" and click OK.



We have put much of the ADINA EM model definition into batch file prob56\_3.in:

Electromagnetic material  
Electromagnetic boundary conditions

Choose File→Open Batch, navigate to the working directory or folder, select the file prob56\_3.in and click Open. The boundary conditions table plotted in the graphics window becomes much larger. Use the mouse to move the table entirely into the graphics window. The graphics window should look something like this:




### **Generating the ADINA-CFD and ADINA-EM models in a single data file, saving the ADINA-IN database**



Click the Save icon  and save the database file to file prob56\_e. Click the Data File/Solution icon , set the file name to prob56\_e, make sure that the Run Solution button is unchecked and click Save.




### **Running ADINA FSI/EM**

Choose Solution→Run ADINA FSI/EM, click the Start button, select file prob56\_e, then hold down the Ctrl key and select file prob56\_a. The File name field should display both file names in quotes. Then click Start. When ADINA-FSI/EM finishes, close all open dialog boxes.

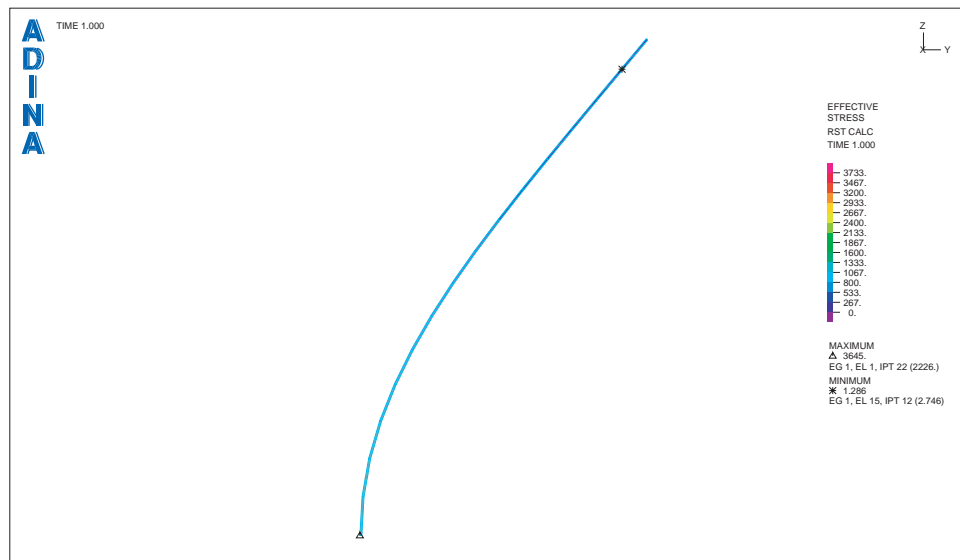
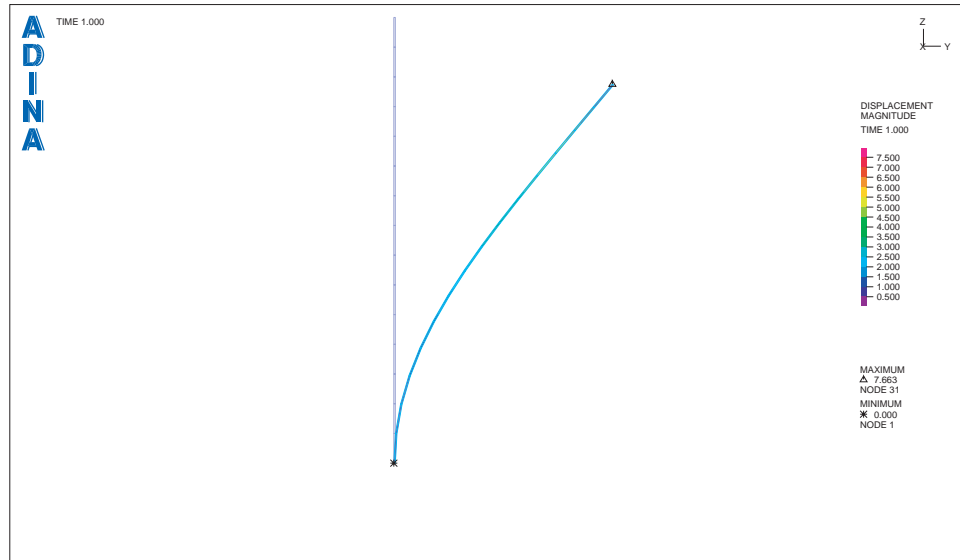
### **Examining the structural solution**

Choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon  and open porthole file prob56\_a.



*Cantilevered structure deflection:* Click the Show Original Mesh icon , then click the Create Band Plot icon , set the Variable to (Displacement: DISPLACEMENT\_MAGNITUDE) and click OK. The graphics window should look something like the top figure on the next page.



*Effective stress in the structure :* Click the Show Original Mesh icon , then click the Clear Band Plot icon  and the Quick Band Plot icon . The graphics window should look something like the bottom figure on the next page.




Problem 56: Electromagnetic driven flow over a cantilever in a channel

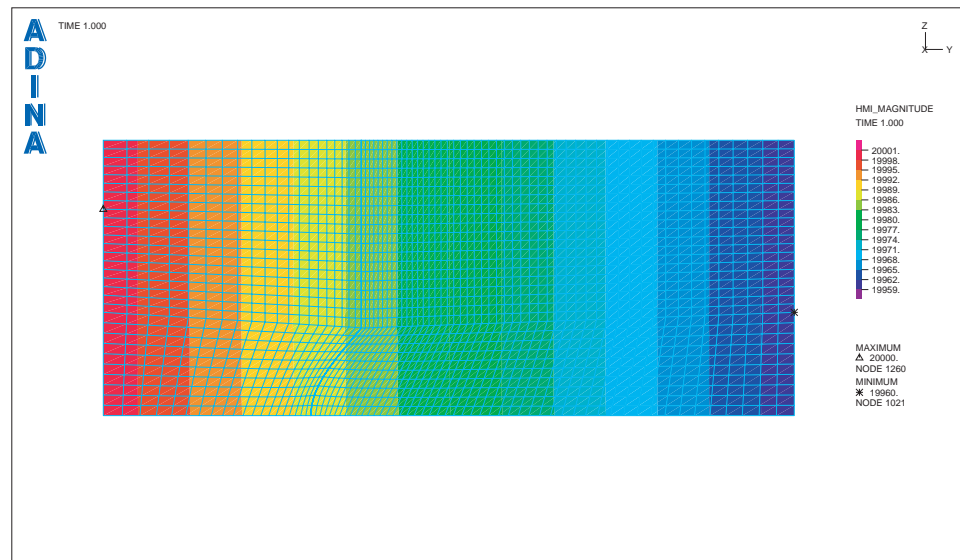





## Examining the electromagnetic and fluid flow solutions

Click the New icon  (you can discard all changes), click the Open icon  and open porthole file prob56\_e.

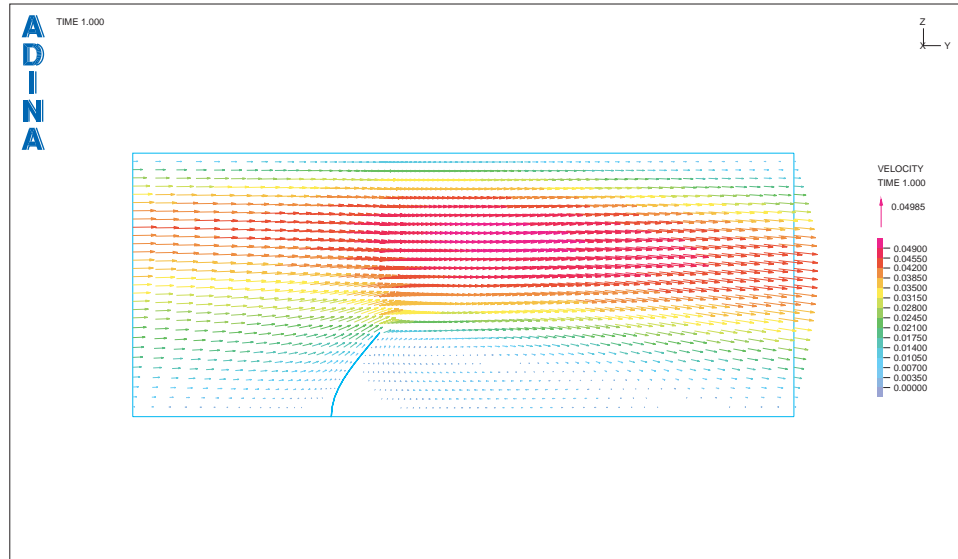
*Electric field:* Click the Model Outline icon , then click the Create Band Plot icon , set the variable to (Electromagnetic: EFI-X) and click OK. The electric field should be constant.



*Magnetic field:* Click the Clear icon  and the Mesh Plot icon . Click the Create Band Plot icon , set the variable to (Electromagnetic: HMI\_MAGNITUDE) and click OK. The graphics window should look something like this:

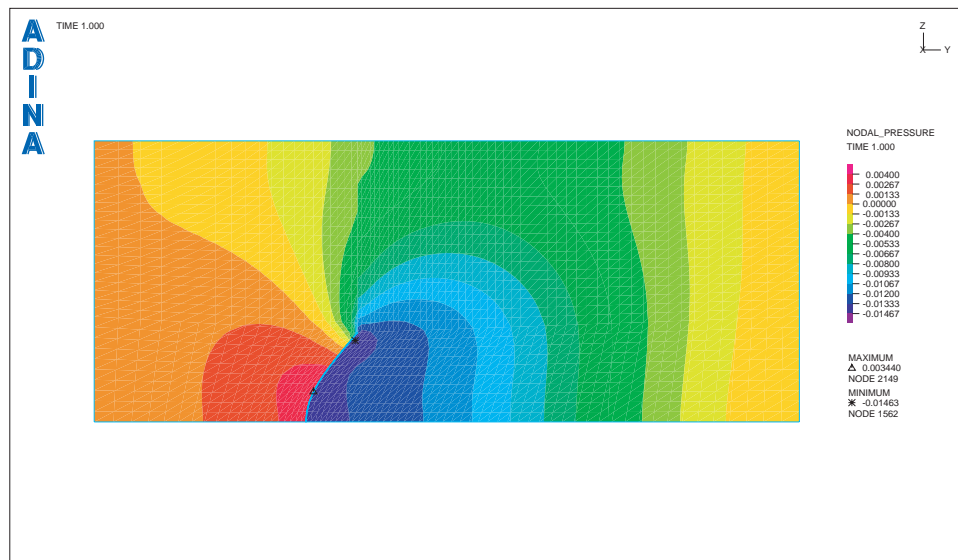


*Velocity vectors:* Click the Clear icon , the Group Outline icon  and the Quick Vector Plot icon . Use the mouse to arrange the graphics until the graphics window looks something like the top figure on the next page.

Problem 56: Electromagnetic driven flow over a cantilever in a channel



Pressure field: Click the Clear Vector Plot icon, , then click the Quick Band Plot icon . Use the mouse to rearrange the graphics until the graphics window looks something like this:



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

*Problem 56: Electromagnetic driven flow over a cantilever in a channel*

---

This page intentionally left blank.