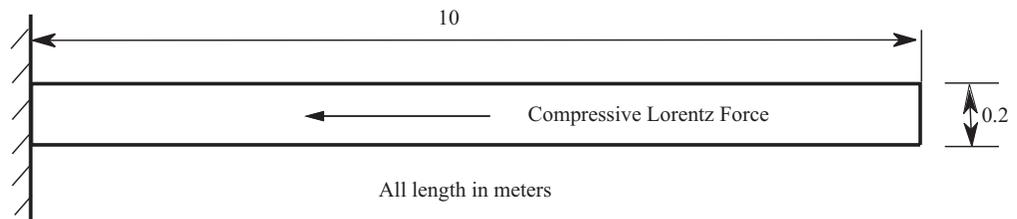


**Problem description**

We determine the buckling load of a bar under an axial Lorentz force. The electromagnetic field coupled with the solid structure will exert an axial distributed load along the bar. The bar will then buckle at some loading when we increase the electromagnetic Lorentz body force along the bar.



Structure: Elastic Material

Young's Modulus:  $E = 1.0 \times 10^8 \text{ N/m}^2$

Poisson Ratio:  $\nu = 0.0$

Electromagnetic material:

Permeability:  $\mu = 1.25 \times 10^{-6} \text{ H/m}$

Conductivity:  $\sigma = 1.0 \times 10^6 \text{ S/m}$

To ease the comparison of the buckling load with the analytical critical solution for distributed axial loading, we make the electric field constant and the magnetic field vary linearly with tiny variation along the whole bar.

In this problem solution, we use an electromagnetic model, coupled with a solid structure model. The electromagnetic model is entered as an ADINA EM model, and the solid model is entered as an ADINA Structures model. The analysis itself is performed as a one-way coupled analysis between ADINA Structures and ADINA EM, similar to one-way FSI coupling. We will demonstrate the following topics that have not been presented in previous problems:

- Performing an electromagnetic-structure coupling analysis in which separate AUI databases are used for the solid and electromagnetic models.
- Running a one-way coupling analysis between ADINA EM and ADINA Structure models.
- Determining the bar buckling load using a load-displacement plot.

We assume that you have worked through problems 1 to 54, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

**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.

### *Problem 55: Bar buckling due to axial Lorentz force*

---

This problem cannot be solved with the 900 nodes version of the ADINA System because the 900 nodes version does not include ADINA Structures/EM.

Much of the input for this problem is stored in the following files: prob55\_1.in, prob55\_2.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 set the Program Module drop-down list to ADINA Structures.

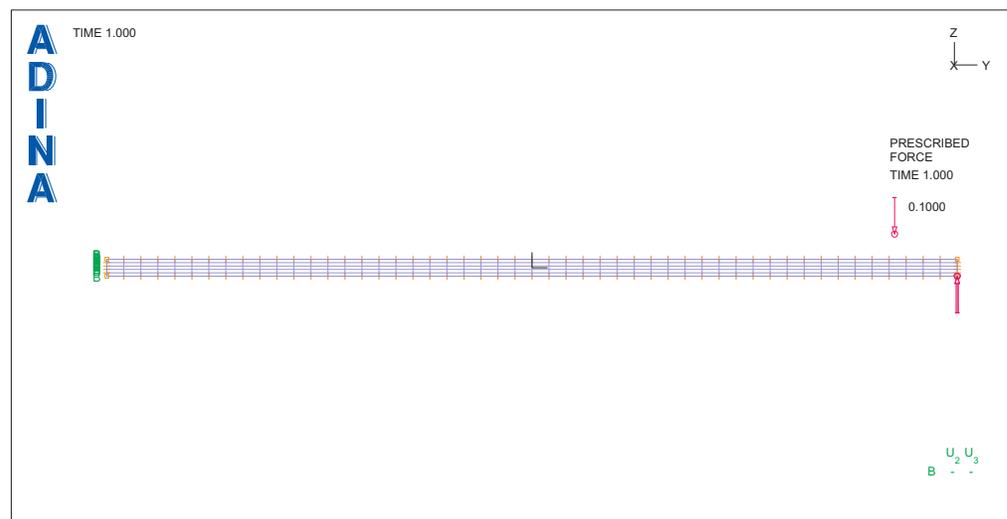
##### **Defining model control data**

*Problem heading:* Choose Control→Heading, enter the heading “Problem 55: Bar buckling under axial Lorentz body force -- Structure” and click OK.

*Multiphysics coupling:* Set the Multiphysics Coupling drop-down list to "with EM".

##### **Model definition**

As this model is similar to other structural models, we put the rest of the structural model definition into batch file prob55\_1.in. Choose File→Open Batch, navigate to the working directory or folder, select the file prob55\_1.in and click Open. The graphics window should look something like this:



The load shown on the plot is a small transverse perturbation force used to trigger the buckling.

We can confirm that the element group has electromagnetic coupling. Click the Element Groups icon , click the Advanced tab and notice that "Includes Electromagnetic Coupling" is checked. Click Cancel to close the dialog box.

### Generating the ADINA Structures data file

Click the Save icon  and save the database to file prob55\_a. Click the Data File/Solution icon , set the file name to prob55\_a, uncheck the Run Solution button and click Save.

### ADINA EM model

#### Choosing the finite element program

Click the New icon  (you can discard all changes), and set the Program Module drop-down list to ADINA EM.

#### Defining model control data

*Problem heading:* Choose Control→Heading, enter the heading "Problem 55: Bar buckling under axial Lorentz body force --- EM" and click OK.

*Analysis assumptions:* Choose Model→Analysis Settings, set the Model Type to "2D E-H model on electric plane", verify that the Analysis Type is Static, set the Tolerance for Residuals to 1E-9, and click OK.

#### Model definition

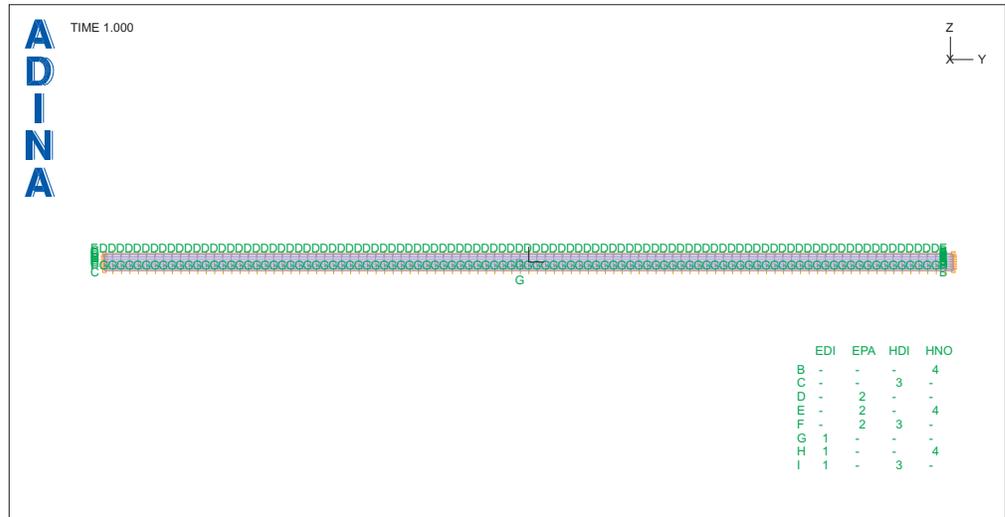
As this model is similar to ADINA EM models shown in previous problems, we put the rest of the EM model definition into batch file prob55\_2.in. Choose File→Open Batch, navigate to the working directory or folder, select the file prob55\_2.in and click Open. The graphics window should look something like the figure on the next page.

#### Generating the ADINA EM model data file

Click the Save icon  and save the database to file prob55\_e. Click the Data File/Solution icon , set the file name to prob55\_e, make sure that the Run Solution button is unchecked and click Save.

*Problem 55: Bar buckling due to axial Lorentz force*

---



**Running ADINA Structures/EM**

Choose Solution→Run ADINA Structures/EM, click the Start button, select file prob55\_e, then hold down the Ctrl key and select file prob55\_a. The File name field should display both file names in quotes. Set the "Run" field to "EM Only", then click Start.

The EM problem runs for 100 steps.

Click Close to close the ADINA Structures/EM dialog box.

Choose Solution→Run ADINA Structures (not Solution→Run ADINA Structures/EM), click the Start button, select file prob55\_a, set "Run" to "One-Way FSI", then click Start.

The structural problem runs for 100 steps.

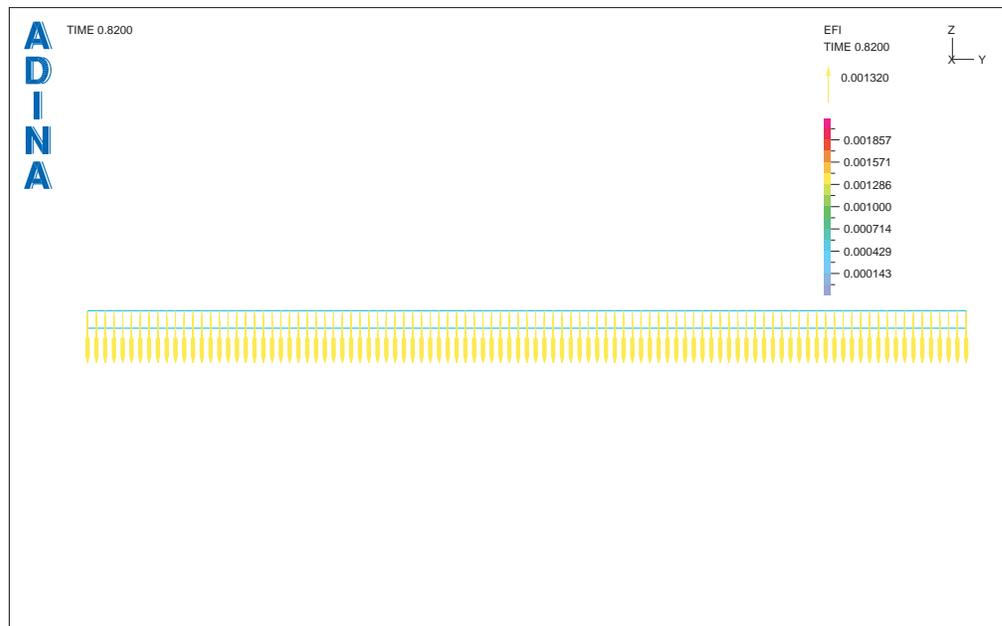
Close all open dialog boxes.

**Examining the electromagnetic solution**

Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob55\_e.

*Save mesh plot defaults:* Click the Model Outline icon  and the Save Mesh Plot Style icon .

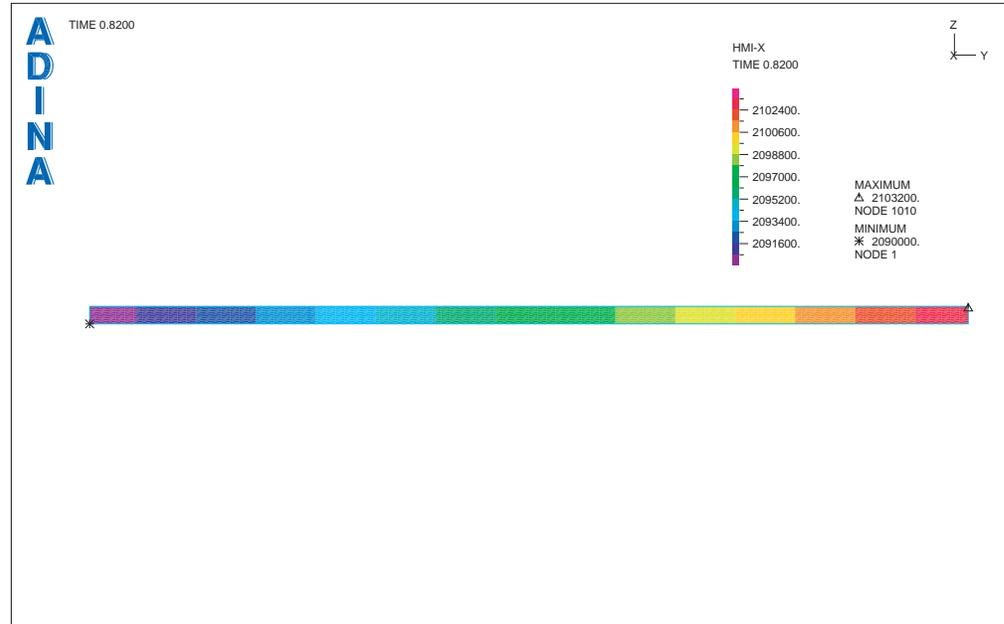
*Electric field:* Click the Create Vector Plot icon , set the Vector Quantity to EFI and click OK. Use the  and  icons to change the solution time to study in detail how the electric field changes as a function of time. Notice that the electric field is constant within the structure for each time. For time 0.82, the graphics window should look something like this:



When you are finished using these icons, click the Last Solution icon  to display the last solution.

*Magnetic field:* Click the Clear Vector Plot icon , click the Create Band Plot icon , set the variable to (Electromagnetic: HMI-X) and click OK. (HMI-X is the x component of the magnetic field intensity.) Use the  and  icons to change the solution time. At time 0.82, the graphics window should look something like the figure on the next page.

*Problem 55: Bar buckling due to axial Lorentz force*



**Examining the structural solution**

Click the New icon  (you can discard all changes), click the Open icon  and open porthole file prob55\_a.

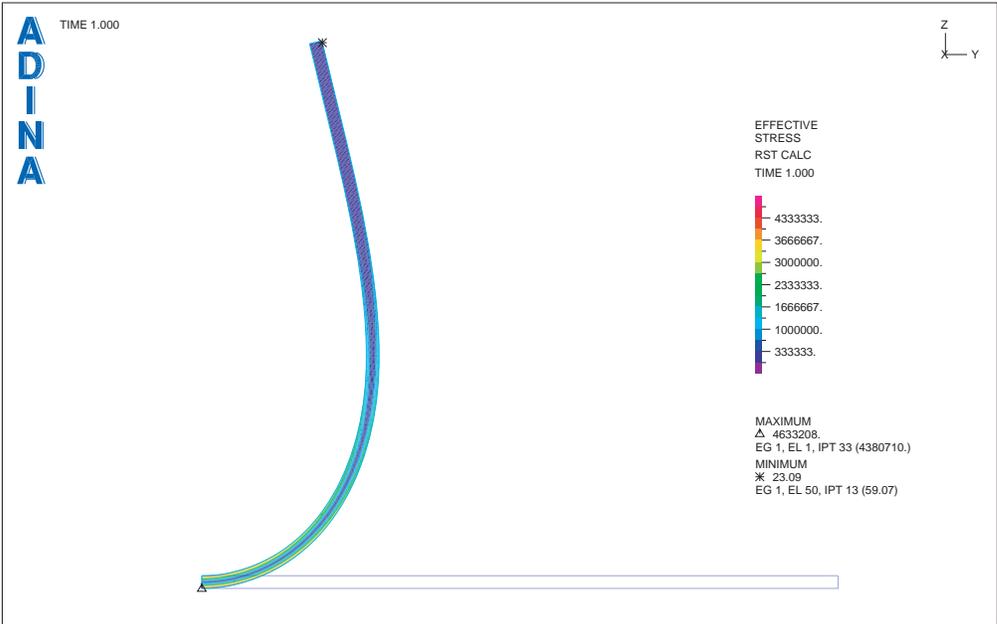
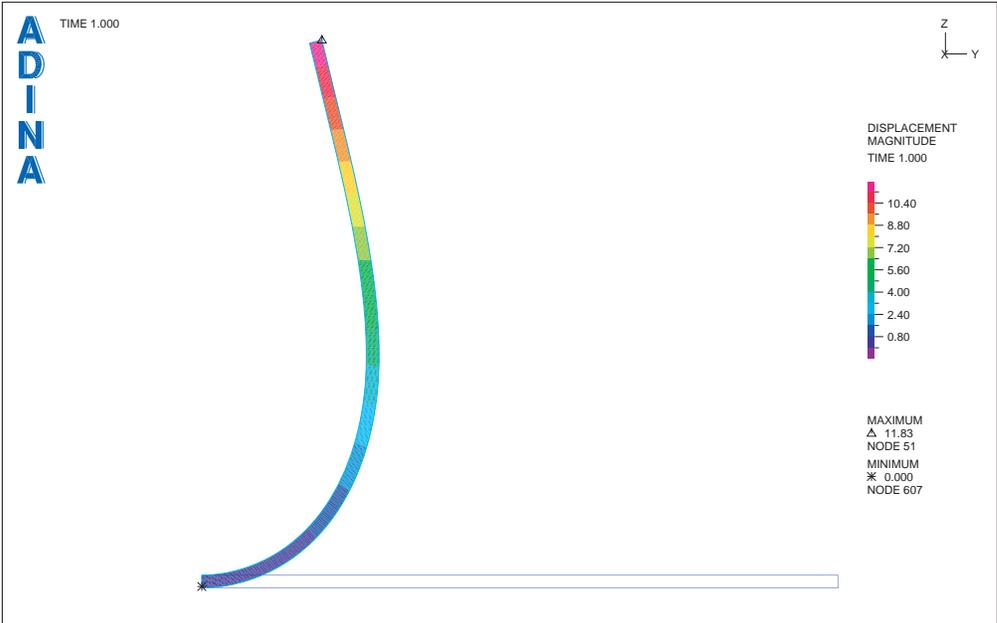
*Bar structure deflection:* Click the Model Outline icon , the Show Original Mesh icon , then click the Create Band Plot icon , set the variable to (Displacement: DISPLACEMENT\_MAGNITUDE) and click OK. The graphics window should look something like the top figure on the next page.

The bar deformation after the buckling is caused by the electromagnetic Lorentz force in the original axial direction of the bar, which does not change direction with deformation.

Use the icons that change the solution time (for example, the  and  icons) to study in detail how the bar displaces from its original position.

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

Problem 55: Bar buckling due to axial Lorentz force



*Problem 55: Bar buckling due to axial Lorentz force*

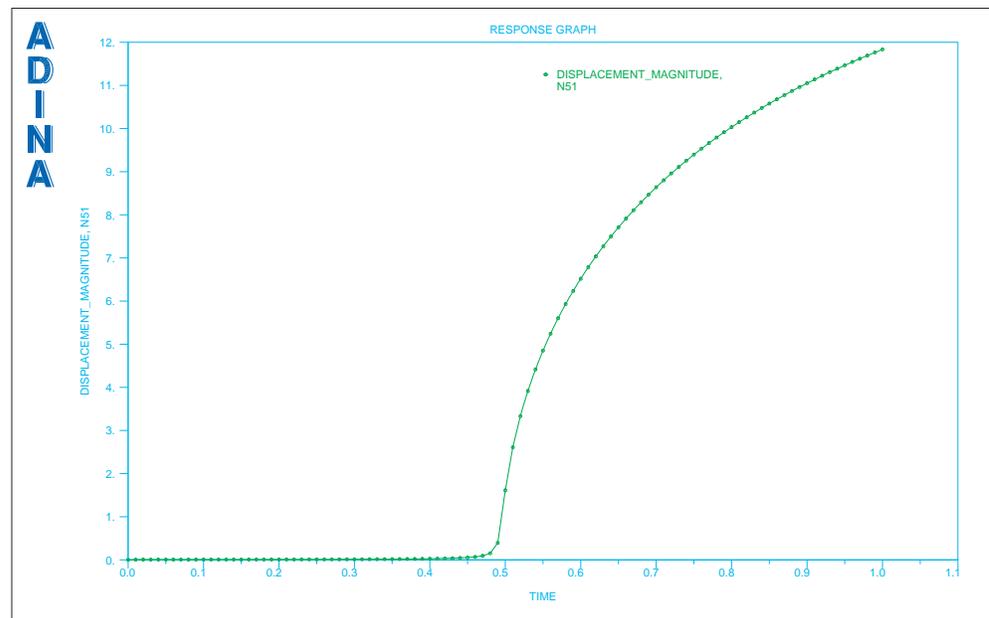
---

**Determining the buckling load**

To show the buckling load, we can graph the displacement of a node at the tip of the bar.

*Define model point:* Choose Definitions→Model Point→Node, add model point N51, set the node # to 51 and click OK.

*Model Point displacement graph:* Click the Clear icon , choose Graph→Response Curve (Model Point), set the Y Coordinate Variable to (Displacement: DISPLACEMENT\_MAGNITUDE), then click OK. The graphics window should look something like this:



The bar buckles around time 0.5, which corresponds to an axial distributed line loading of 104.5 (N/m). This value compares well with the analytical buckling load of 104.49.

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