

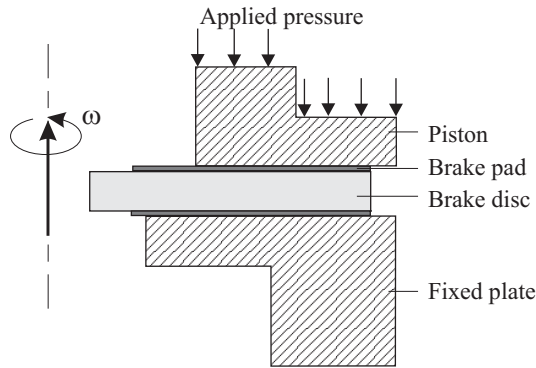
Problem 34

**Thermo-mechanical-coupling
analysis of a disc braking
system**

This page intentionally left blank.

Problem description

The figure shows a disc braking system.



Initially, the brake disc is rotating at 900 rpm and the temperature in the braking system is 25° C. At the time of solution start, a pressure of 2 MPa is applied to the piston and held constant thereafter. During the first 5 seconds of the analysis, the brake disc slows down to 0 rpm.

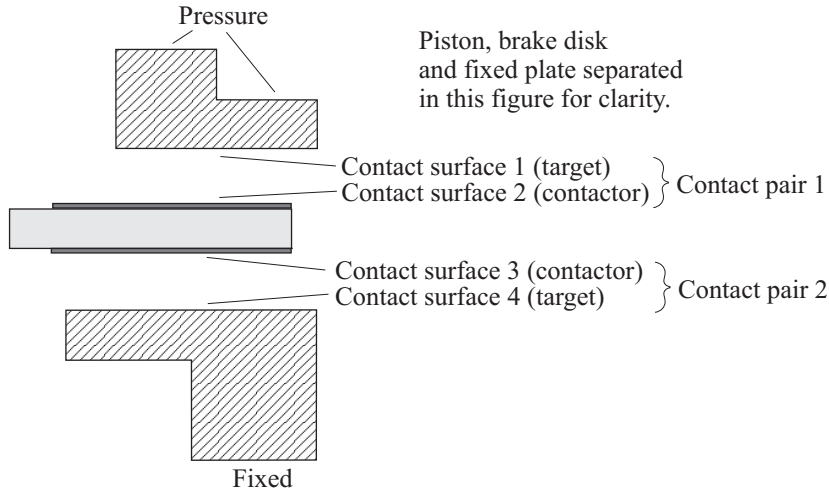
The analysis is fully coupled as follows:

- 1) Heat is generated as a result of the contact between the brake pad, piston and fixed plate, and this heat causes deformation of these components.
- 2) The deformation of these components causes additional contact and therefore additional heating.

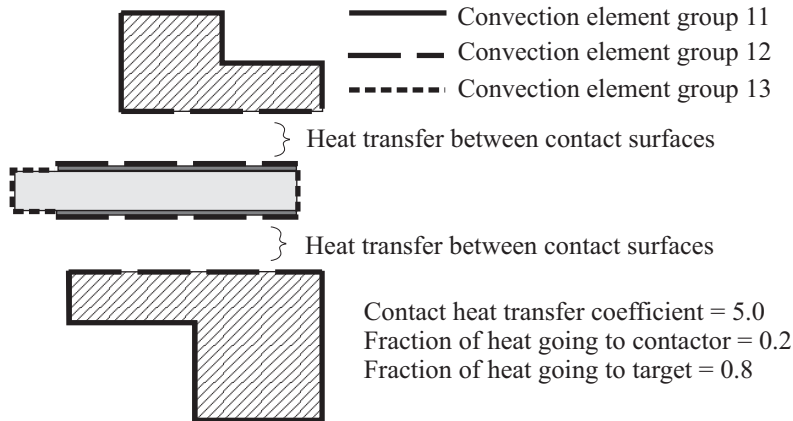
Two finite element models are employed in the single thermo-mechanical analysis. The analysis is a 2D axisymmetric analysis. All of the input data is given in SI units.

(This model can also be solved using the ADINA TMC feature of ADINA Structures, but we do not demonstrate this solution here.)

Structural (ADINA) model: The structural model is shown in the following figure: Both the applied pressure and the disk rotational speed are prescribed in the analysis. The disk rotation is applied to the model using contact-slip loads. The friction coefficient between the contact surfaces is 0.2. Static conditions are assumed (that is, inertial effects are neglected).



Thermal (ADINA-Thermal) model: The thermal model is shown in the following figure. Convection elements are placed where shown in the figure.



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

- Setting up a thermo-mechanical-coupling (TMC) analysis
- Specifying frictional contact
- Specifying contact-slip loads
- Copying a structural model to a thermal model
- Postprocessing the TMC finite element models simultaneously

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 does not contain ADINA-TMC.

Much of the input for this problem is stored in files prob34_1.in, prob34_2.in and prob34_3.in. You need to copy files prob34_1.in, prob34_2.in, prob34_3.in from the folder samples\primer into a working directory or folder before beginning this analysis.


Structural (ADINA) model

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

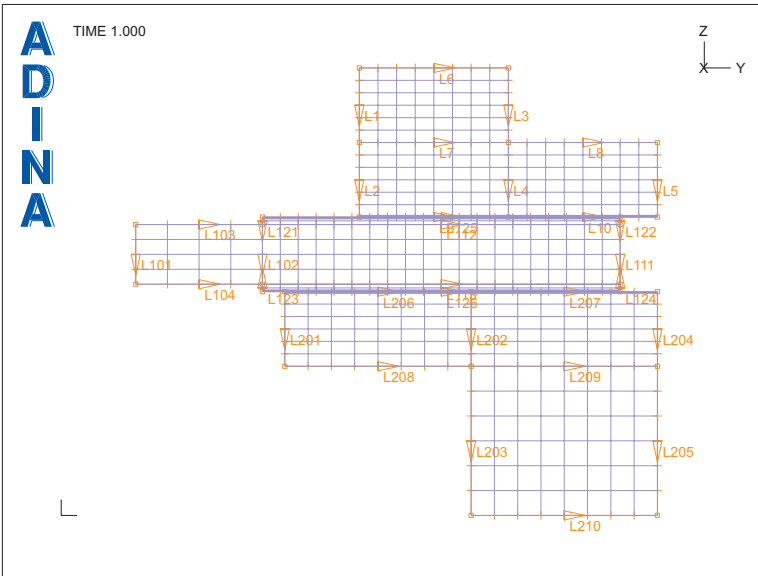
Defining model control data, geometry, subdivision data, boundary conditions, rigid links, displacement loads, material definitions and element groups


We have prepared a batch file (prob34_1.in) which performs the following operations:

- Specifies that the ATS method be used.
- Specifies the initial temperature.
- Defines points, lines and surfaces.
- Subdivides the surfaces.
- Defines materials (thermo-isotropic)
- Defines element groups (axisymmetric 2-D solid elements)
- Generates elements.
- Defines boundary conditions.
- Defines time functions.
- Defines pressure loads.

Click the Open icon , navigate to the working directory or folder, set the 'File type' field to 'ADINA-IN Command Files (*.in)', select the file prob34_1.in and click Open. The AUI processes the commands in the batch file.


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



Contact-slip loads: Click the Apply Load icon , set the Load Type to Contact Slip and click the Define... button to the right of the Load Number field. Add Contact-Slip load 1, set the Angular Velocity to 15.0, the Factor to 6.28318, the “End Position of Axis Vector” to 0.0, 0.0, 1.0 and click OK. In the Apply Load dialog box, enter the following information into the table and click OK:

Site #	Contact Group	Time Function
2	1	2
3	1	2

Generating the ADINA data file

Click the Data File/Solution icon , set the file name to prob34_a, uncheck the Run Solution button and click Save. (We do not want to run the ADINA model by itself.)


Thermal (ADINA-Thermal) model

Set the Program Module to ADINA Thermal (do not create a new database file).

We use the same element layout in the ADINA-Thermal model as in the ADINA model. Choose Meshing→Copy F.E. Model..., set the fields so that the sentence reads “From ADINA Structures Model to ADINA Thermal Model” and click OK. The Log Window should display the messages

```
TWODSOLID element group 1 is copied to TWODCONDUCTION 1
TWODSOLID element group 2 is copied to TWODCONDUCTION 2
TWODSOLID element group 3 is copied to TWODCONDUCTION 3
3 element groups are copied from ADINA to ADINA-T
```


Close the log window.

At this point, both the ADINA and ADINA-Thermal models are displayed. To display just the ADINA-Thermal model, click the Change Zone icon , set the Zone Name to ADINA-T and click OK.

Defining model control data, material definitions and convection elements

We have prepared a batch file (prob34_3.in) which performs the following operations:

- Defines a heading for the thermal model.
- Specifies the initial temperature.
- Defines thermal materials.
- Sets the material numbers for each of the thermal element groups.
- Defines convection materials.
- Defines convection element groups.
- Generates convection elements.
- Defines a constant time function
- Defines convection loads.
- Specifies 200 time steps of size 0.05. (The time step information is specified in the ADINA-Thermal model)
- Specifies a transient analysis.

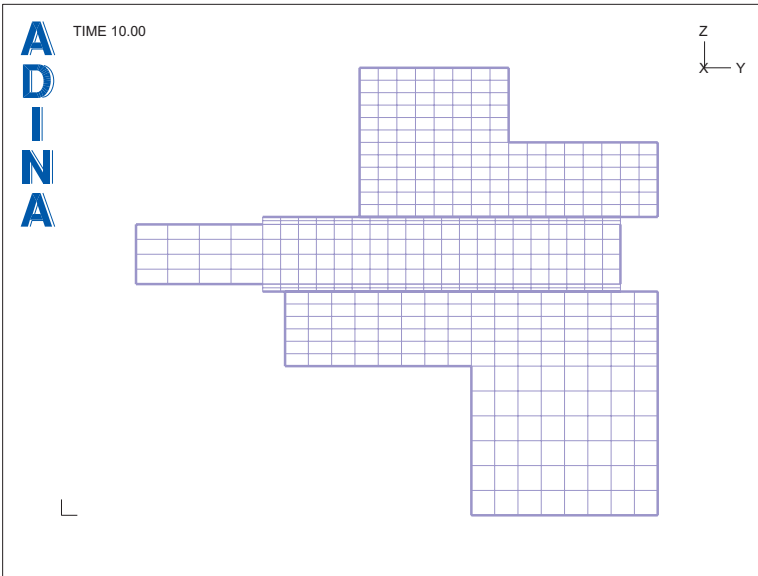
Click the Open icon , navigate to the working directory or folder, set the 'File type' field to 'ADINA-IN Command Files (*.in)', select the file prob34_3.in and click Open. The AUI processes the commands in the batch file.

The graphics window should look something like the top figure on the next page.


Specifying TMC iterations

We would like to perform TMC iterations during this analysis. Choose Control→ Solution Process, set the “TMC Iterations” field to Yes and click OK.

(If you do not set the “TMC Iterations” field to Yes, the analysis is still a TMC analysis, but TMC iterations are not performed. Therefore convergence in displacements and temperatures may not be reached.)





Generating the ADINA-Thermal data file, saving the database

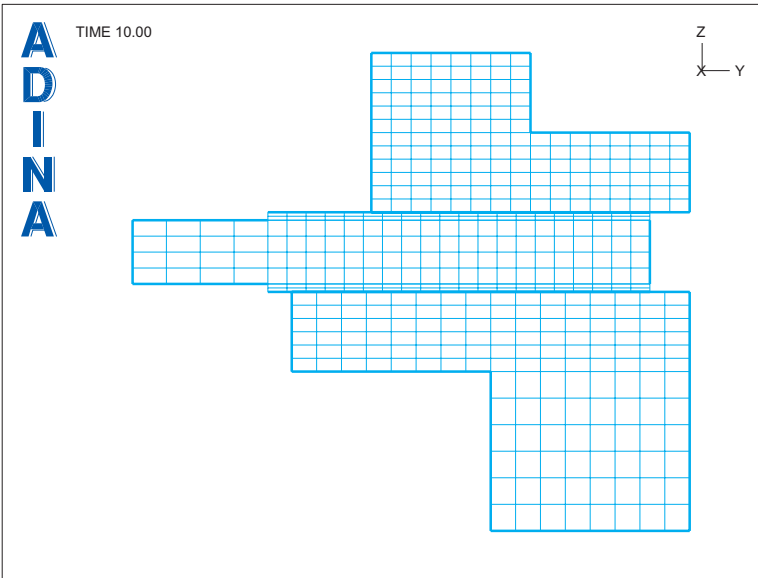
Click the Data File/Solution icon , set the file name to prob34_t, make sure that the Run Solution button is unchecked and click Save. Save the database file to file prob34.

Running ADINA-TMC

Choose Solution→Run ADINA-TMC, click the Start button, select file prob34_a, then hold down the Ctrl key and select file prob34_t. The File name field should display both file names in quotes. Click Start.




The ADINA-TMC run finishes in 200 steps. When ADINA-TMC finishes, close all open dialog boxes. Choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon  and open porthole file prob34_t. Then click the Open icon  and open porthole file prob34_a.


The graphics window should look something like the top figure on the next page. Notice that the convection elements are shown (as thick lines) and the contact segments are shown (as thick lines).



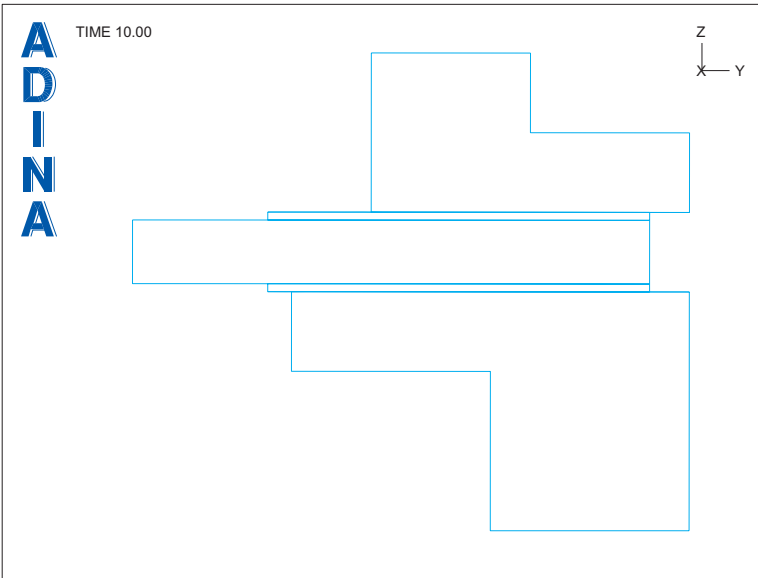
Examining the solution





Because we have loaded both the ADINA and the ADINA-Thermal porthole files, we can display results from both models simultaneously.



We do not want to show the convection elements and the contact segments. Click the Change Zone icon , set the Zone Name to ADINA, then click OK. Click the Group Outline icon . Click the Modify Mesh Plot icon , click the Element Depiction... button, click the “Contact, etc” tab, set the Contact Surface Line Width to 0.0 and click OK twice to close both dialog boxes. The graphics window should look something like the figure on the next page.


Click the Save Mesh Plot Style icon  to set the defaults for the successive mesh plots.



Solution time: Before we plot the solution, we set the default solution time to 3.0. We do this because we anticipate that the maximum range of the solution variables will occur around this time. Choose Definitions→Response, make sure that the Response Name is set to DEFAULT, set the Solution Time to 3.0 and click OK.



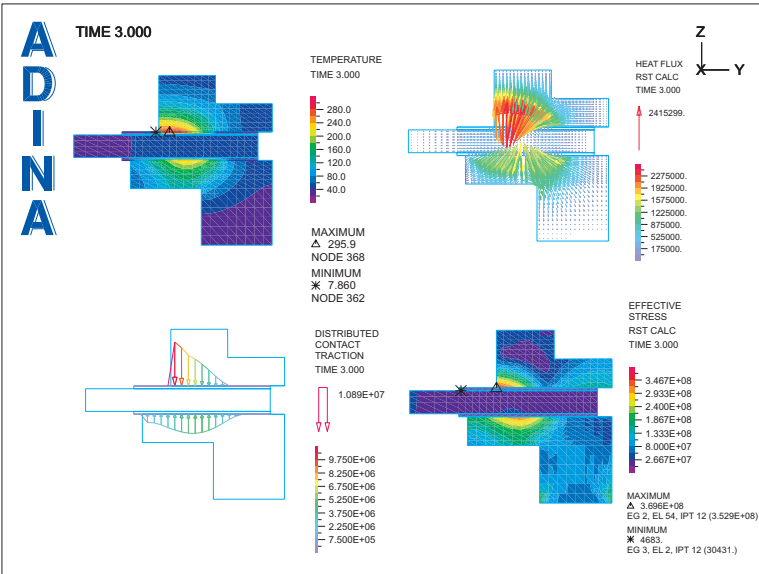
Temperatures: Click the Clear icon  and the Mesh Plot icon . Use the Pick icon  and the mouse to shrink the mesh plot and to move it to the upper-left-hand corner of the graphics window. Click the Create Band Plot icon , set the Band Plot Variable to (Temperature: TEMPERATURE) and click OK. Use the mouse to rearrange the graphics window, as shown in the figure on the next page.



Heat fluxes: Click the Mesh Plot icon . Use the mouse to shrink the mesh plot and to move it to the upper-right-hand corner of the graphics window. Click the Create Vector Plot icon , make sure that the Vector Quantity is set to HEAT_FLUX and click OK. Use the mouse to rearrange the graphics window, as shown in the figure on the next page.

Contact pressures: Click the Mesh Plot icon . Use the mouse to shrink the mesh plot and to move it to the lower-left-hand corner of the graphics window. Choose Display→Reaction Plot→Create, set the Reaction Quantity to DISTRIBUTED_CONTACT_TRACTION and click OK. Use the mouse to rearrange the graphics window, as shown in the figure on the next page.

Effective stresses: Click the Mesh Plot icon . Use the mouse to shrink the mesh plot and to move it to the lower-right-hand corner of the graphics window. Click the Quick Band Plot icon , then use the mouse to rearrange the graphics window until it looks something like the figure on the next page.

Problem 34: Thermo-mechanical-coupling analysis of a disc braking system



Animating the solution: Click the Movie Load Step icon  to create a movie of the solution, then click the Animate icon  to play back the animation.

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