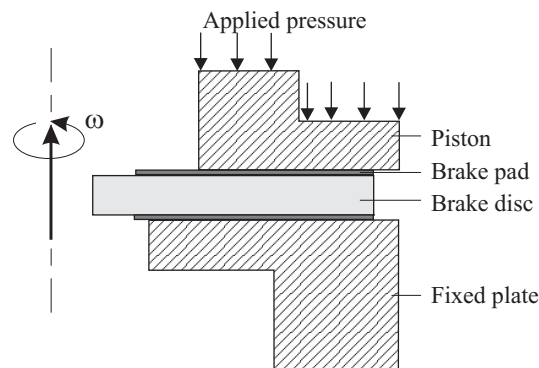## Problem description

The figure shows a disc braking system.

Initially, the brake disc is rotating at 900 rpm and the temperature in the braking system is 25º 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 disk slows down to 0 rpm.
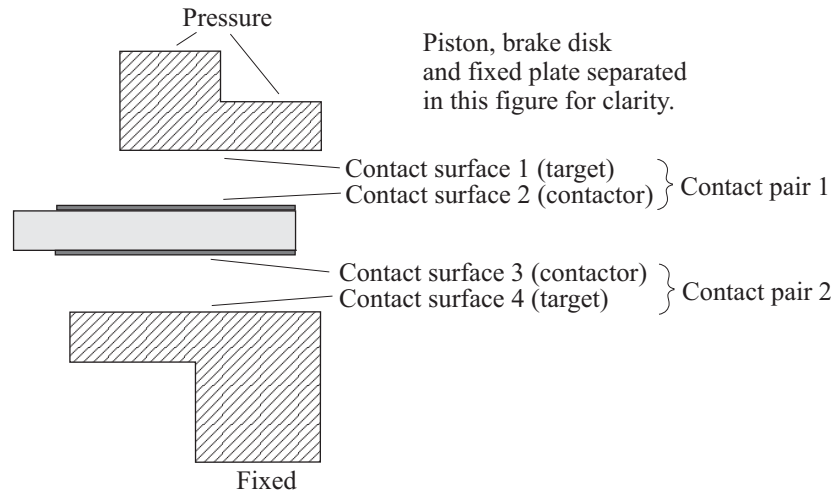
The analysis is fully coupled as follows:

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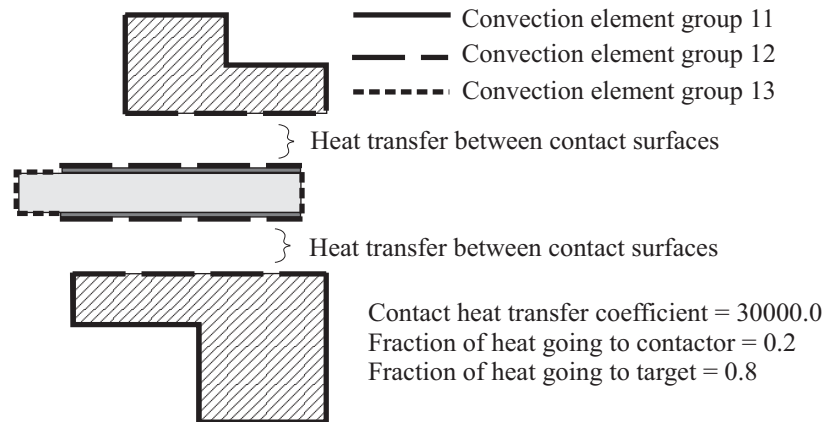
(This model can also be solved using the ADINA TMC feature of ADINA Structures, but we do not demonstrate this solution here.)

Structural (ADINA Structures) 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).

Pressure

Piston, brake disk
and fixed plate separated
in this figure for clarity.

Contact surface 1 (target)
Contact surface 2 (contactor) } Contact pair 1

Contact surface 3 (contactor)
Contact surface 4 (target) } Contact pair 2

Fixed

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

———— Convection element group 11
— · — Convection element group 12
------- Convection element group 13

} Heat transfer between contact surfaces

} Heat transfer between contact surfaces

Contact heat transfer coefficient = 30000.0
Fraction of heat going to contactor = 0.2
Fraction of heat going to target = 0.8

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
- Post-processing 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 Structures) model

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

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`) that 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.

Choose File→Open Batch, navigate to the working directory or folder, 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.
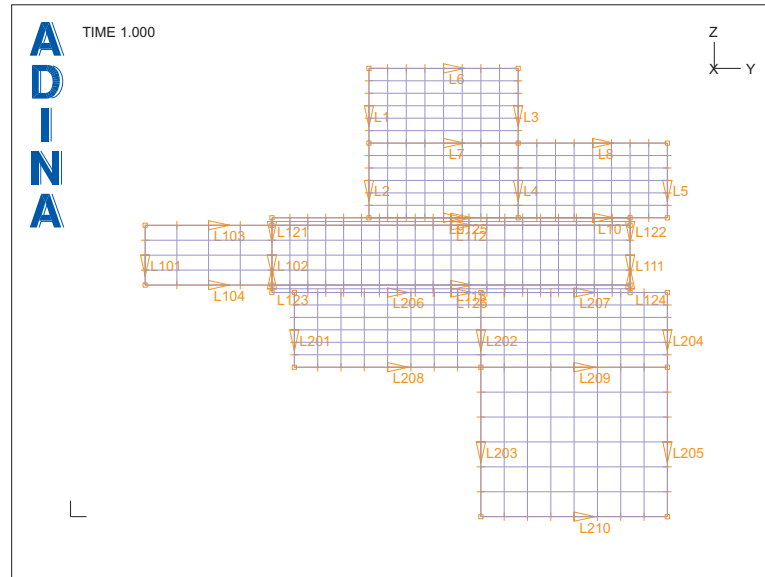
Defining contact conditions

Contact group: Click the Contact Groups icon , add group 1 and set the "Default Coulomb Friction Coefficient" to 0.2. Click the "Node-to-Node, TMC" tab, and, in the "Default Thermo-Mechanical Coupling Settings" box, set the "Heat Transfer Coefficient through Contact" to 30000.0, the "Fraction of Frictional Contact Heat Distributed To Contactor" to 0.2, the "Fraction of Frictional Contact Heat Distributed To Target" to 0.8 and click OK.

Contact surfaces, contact pairs: The contact surface and contact pair definitions are defined in batch file `prob34_2.in`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob34_2.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.
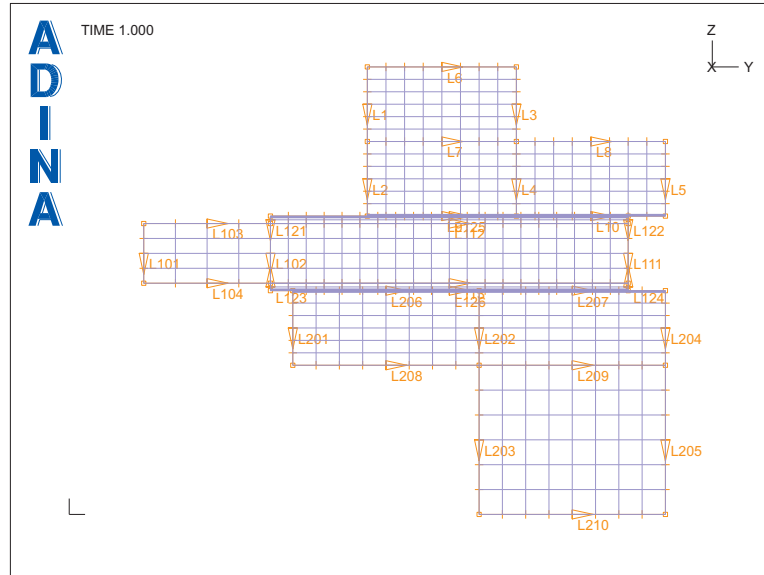
Contact-slip loads: Click the Apply Load icon , set the Load Type to Contact Slip and click the Define… button to the right of the Load Number field. Add Contact-Slip Load 1, set the Angular Velocity to 15.0, the Factor to 6.28318, the "End Position of Axis Vector" to 0.0, 0.0, 1.0 and click OK. In the Apply Load dialog box, enter the following information into the table and click OK:

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

Generating the ADINA Structures 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 Structures model by itself.)

Thermal (ADINA Thermal) model

Set the Program Module drop-down list 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 Structures 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 Structures and ADINA Thermal models are displayed. To display just the ADINA Thermal model, in the Model Tree, click the + next to the "Zone" text, right-click on "2. ADINA-T" and choose Display.

Defining model control data, material definitions and convection elements

We have prepared a batch file (prob34_3.in) that 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.

Choose File→Open Batch, navigate to the working directory or folder, 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 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.
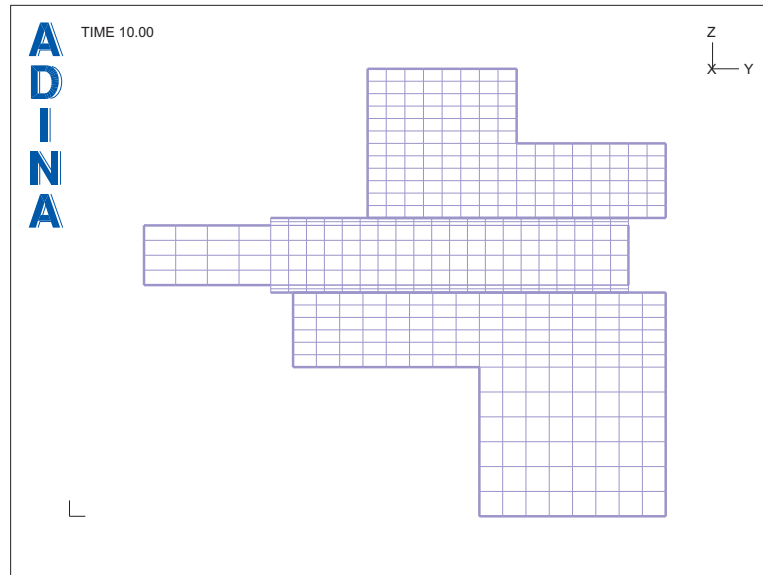
(If you do not set the "TMC Iterations" field to Yes, the analysis is still a TMC analysis, but TMC iterations are not performed. Therefore convergence in displacements and temperatures may not be reached.)

Generating the ADINA Thermal data file, saving the database

Click the Data File/Solution icon ![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. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon ![icon] and open porthole file `prob34_t`. Then click the Open icon ![icon] and open porthole file `prob34_a`.

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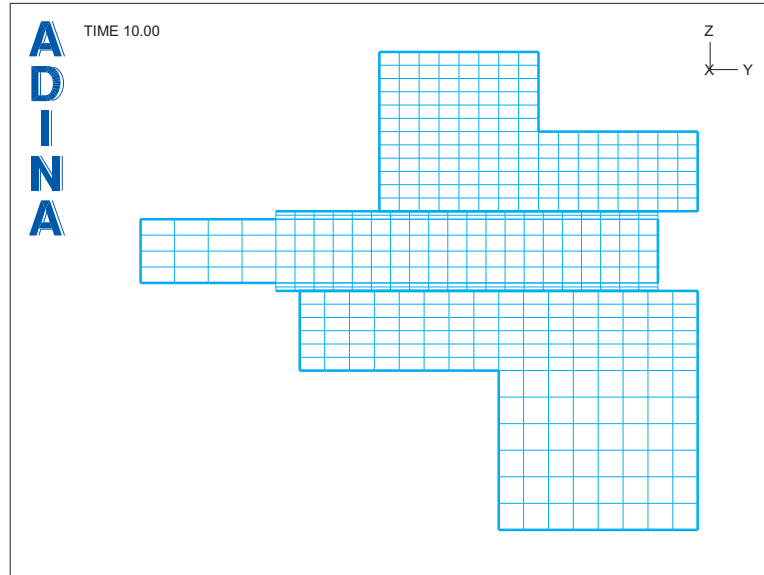
Examining the solution

Because we have loaded both the ADINA Structures 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. In the Model Tree, click the + next to the "Zone" text, right-click on "1. ADINA" and choose Display. 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.
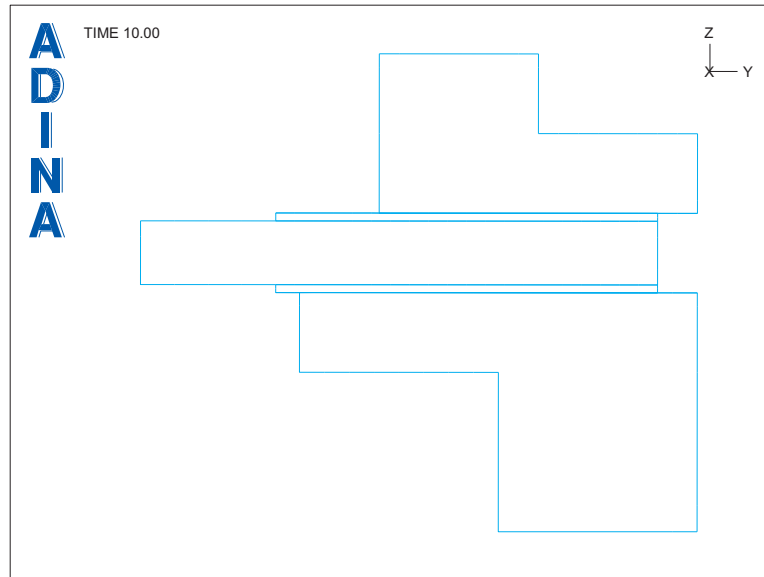
Temperatures: Click the Clear icon and the Mesh Plot icon . Use the Pick icon and the mouse to shrink the mesh plot and to move it to the upper-left-hand corner of the graphics window. Click the Create Band Plot icon , set the Band Plot Variable to (Temperature: TEMPERATURE) and click OK. Use the mouse 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. Click the Create Reaction Plot icon , 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 this:
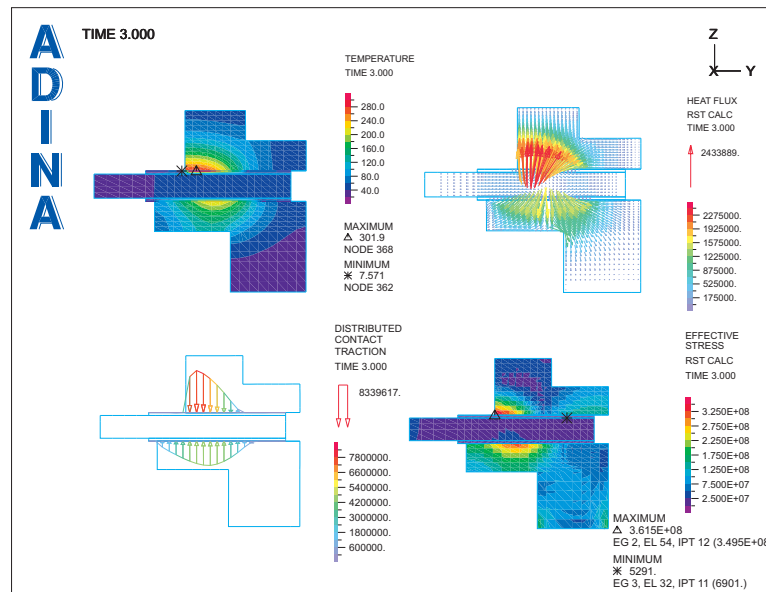
Animating the solution: Click the Movie Load Step icon 📷 to create a movie of the solution, then click the Animate icon ▶ to play back the animation.

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