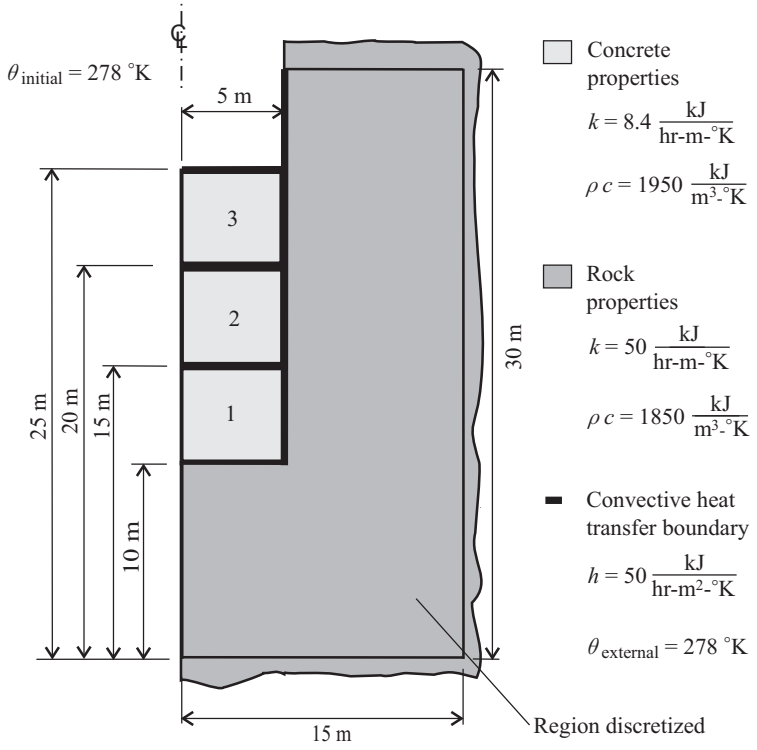
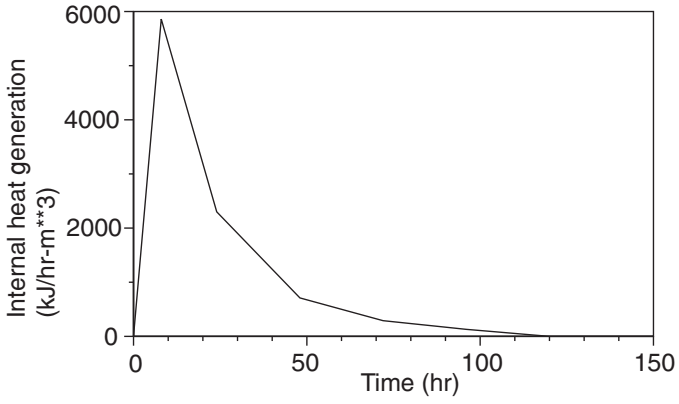
Problem description

During a twelve day period, concrete is added to a hole previously drilled into rock. At the beginning of each 4 day interval, a 5 meter depth of concrete is poured. As the concrete solidifies, internal heat is generated as the water and cement in the concrete react and this heat is conducted into the surrounding rock and convected to the surrounding atmosphere.





In this analysis, we calculate the temperature distribution in the concrete and surrounding rock as a function of time. An axisymmetric analysis is appropriate here. We account for the change in the concrete volume and heat transfer surface area as the concrete is added.

Regarding the units used in this analysis, the time unit is hour, the energy unit is kilojoule, the length unit is meter and the temperature unit is Kelvin. The power unit is therefore kJ/hour, not kilowatt.

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

- Specifying element birth/death
- Specifying internal heat generation loading
- Specifying loads using arrival times
- Making an envelope band plot

We assume that you have worked through problems 1 to 24, 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.

This problem cannot be solved with the 900 nodes version of the ADINA System because there are 971 nodes in the model.

Invoking the AUI and choosing the finite element program

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

Defining model control data

Heading: Choose Control→Heading, enter "Problem 25: Heat transfer from solidifying concrete blocks" and click OK.

Analysis type: Set the Analysis Type to Transient. We will use the Euler backward method for the time integration method. Click the Analysis Options icon  , verify that the Integration Method is Euler Backward Integration and click OK.

Time steps: Choose Control→Time Step, specify 80 time steps of 8 hours each, then click OK.

Time functions: There are a number of different time functions used in this problem. We need a time function to describe the transient behavior of the heat generation from the concrete and we need time functions to describe the environmental temperature of the various convection surfaces. We also need to ensure that the temperatures at the intersections of the convection surfaces are physically realistic. Problems may occur in the calculations of environmental temperatures at nodes that are at the intersection of two or more boundary convection elements. This is because, at present, ADINA Thermal averages the environmental temperature at an intersection according to the environmental temperatures of the connected boundary elements regardless of whether the boundary elements are active. If one or more of the boundary elements are not active at a particular intersection, then the calculated environmental temperature may be artificially low. We circumvent this problem by explicitly setting the environmental temperature at each intersection node.

First let's enter the concrete block time function. This time function (time function 1) gives the variation of internal heat within each block. When we use the time function, we will shift it using the arrival time feature so that the shifted time function is greater than zero when the block is added to the model. Choose Control→Time Function and enter the following information for time function 1:

Time	Value
0	0
8	5860
24	2300
48	710
72	290
96	130
120	0
10000	0

Click Save to store this definition. Now we enter the time functions for the environmental temperature of the boundary convection elements. Define time function 2 as

Time	Value
0	278
95.9	278
96.0	0
10000	0

time function 3 as

Time	Value
0	0
95.9	0
96	278
191.9	278
192	0
10000	0

time function 4 as

Time	Value
0	278
191.9	278
192	0
10000	0

time function 5 as

Time	Value
0	0
191.9	0
192	278
10000	278

and time function 6 as

Time	Value
0	278
10000	278

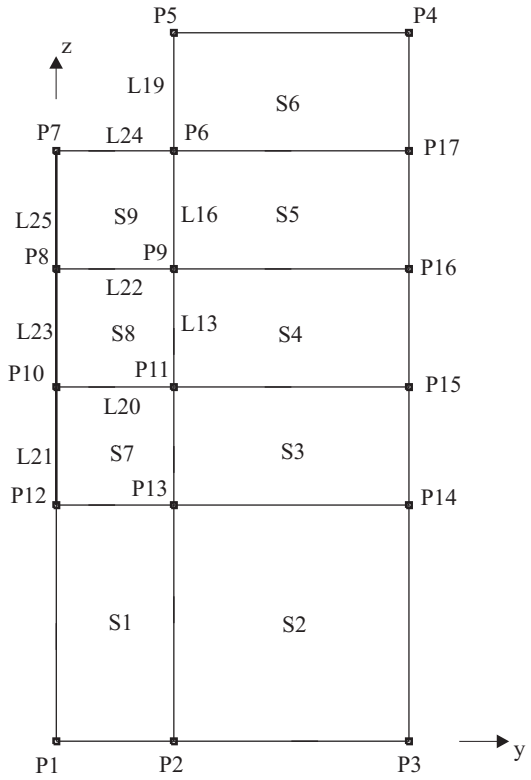
Click OK to close the dialog box.

Initial conditions: Choose Control→Analysis Assumptions→Default Temperature Settings, set the Default Initial Temperature to 278 and click OK.

Defining the geometry

The figure on the next page shows the key geometry used in defining the model.

Points: We need to enter enough geometry points to describe the geometry and to make it easy to define the surfaces used for meshing. Click the Define Points icon  , enter the following data given in the table on the next page and click OK.



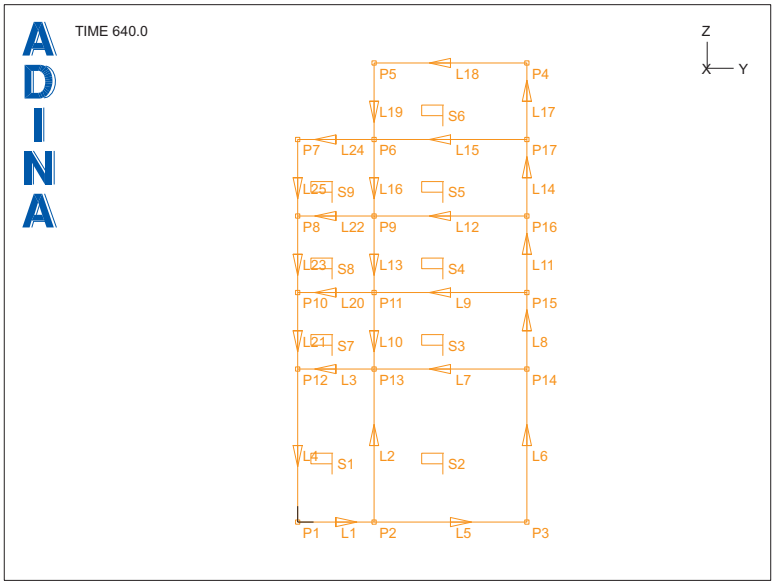
Point #	X2	X3
1	0	0
2	5	0
3	15	0
4	15	30
5	5	30
6	5	25
7	0	25
8	0	20
9	5	20
10	0	15
11	5	15
12	0	10
13	5	10
14	15	10
15	15	15
16	15	20
17	15	25

Click the Point Labels icon to display the point numbers.

Surfaces: Click the Define Surfaces icon, create surfaces 1 to 9 using the following data and click OK.

Surface number	Type	Point 1	Point 2	Point 3	Point 4
1	Vertex	1	2	13	12
2	Vertex	2	3	14	13
3	Vertex	13	14	15	11
4	Vertex	11	15	16	9
5	Vertex	9	16	17	6
6	Vertex	6	17	4	5
7	Vertex	12	13	11	10
8	Vertex	10	11	9	8
9	Vertex	8	9	6	7

When you click the Line/Edge Labels icon and the Surface/Face Labels icon, the graphics window should look something like this:



Defining the material data

We need to define the physical properties of the concrete and the base rock. Click the Manage Materials icon  and click the "k isotropic, c constant" button. In the Define Constant Isotropic Material dialog box, add material number 1, set the Thermal Conductivity to 8.4, the Heat Capacity to 1950 and click Save. Now add material number 2, set the Thermal Conductivity to 50, the Heat Capacity to 1850 and click OK. (Do not close the Manage Material Definitions dialog box yet.)

Click the Convection Constant button. In the Define Constant Convection Material dialog box, add material number 3, set the Convection Coefficient to 50 and click OK.

Click Close to close the Manage Material Definitions dialog box.

Defining the birth and death times

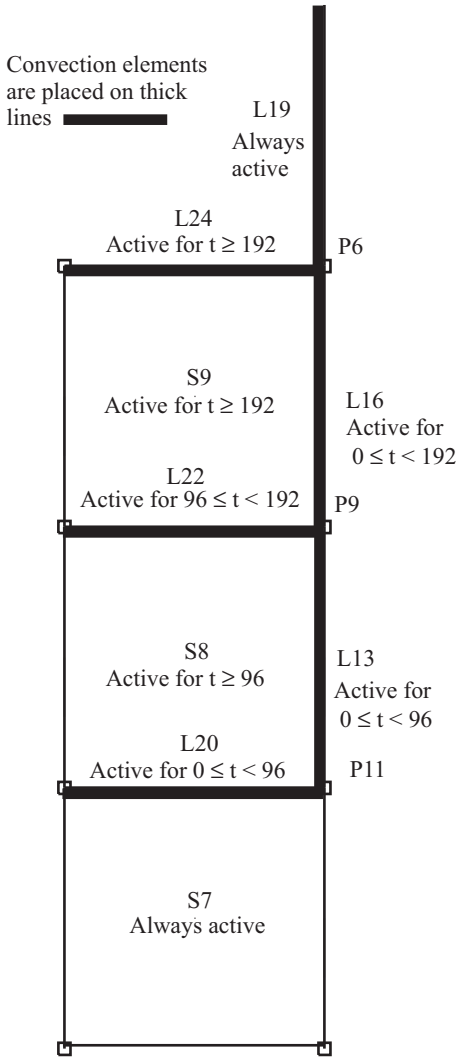
The figure on the next page shows the modeling of the concrete and the convection boundary conditions.

The only two surfaces for which the elements have birth times are surfaces 8 and 9, corresponding to the second and third concrete blocks. Choose Model→ Element Properties→2-D Conduction. In the first row of the table, set the Surface # to 8 and the Birth Time to 95.9, and, in the second row of the table, set the Surface # to 9 and the Birth Time to 191.9. Click OK to close the dialog box.

(Please note that we enter a birth time that is slightly earlier than the solution time for which we want the elements to be active. This avoids possible round-off error.)

Now we will define the birth and death times of the boundary convection elements attached to the concrete blocks. Choose Model→Element Properties→2-D Convection, add the following information and click OK.

Line number	Birth time	Death time
20	0	95.9
13	0	95.9
22	95.9	191.9
16	0	191.9
24	191.9	0



Defining the loads

Internal heat generation: We will define the internal heat generation on the surfaces corresponding to the concrete elements. Load application 1 defines the internal heat load for the first concrete block, load application 2 defines the internal heat load for the second concrete block and load application 3 defines the internal heat load for the third concrete block. Each load application uses the same load definition, but each load application is shifted in time by 96 hours.

Click the Apply Load icon  , set the Load Type to Internal Heat and click the Define... button to the right of the Load Number field. In the Define Internal Heat dialog box, add Internal Heat Number 1, set the Heat Generation/Volume to 1 and click OK. In the Apply Load dialog box, set the "Apply to" field to Surface, and in the first three rows of the table, set the Surface # to 7, 8, 9 and set the Arrival Time to 0, 96, 192. Click OK to close the dialog box.

Environmental temperatures: We will assign the environmental temperatures to the boundary convection elements. In the following, we also assign environmental temperatures to the boundary convection element intersection points in order to override the averaging calculations described above.

Click the Apply Load icon  , set the Load Type to Convection and click the Define... button to the right of the Load Number field. In the Define Convection dialog box, add Convection Number 1, set the Environment Temperature to 1 and click OK. In the Apply Load dialog box, set the "Apply to" field to Line and define the environmental temperatures on lines using the following data:

Line #	Time function
20	2
13	2
22	3
16	4
24	5
19	6

Now set the "Apply to" field to Point and define the environmental temperatures on points using the following data:

Point #	Time function
11	2
9	4
6	6

Click OK to close the dialog box.

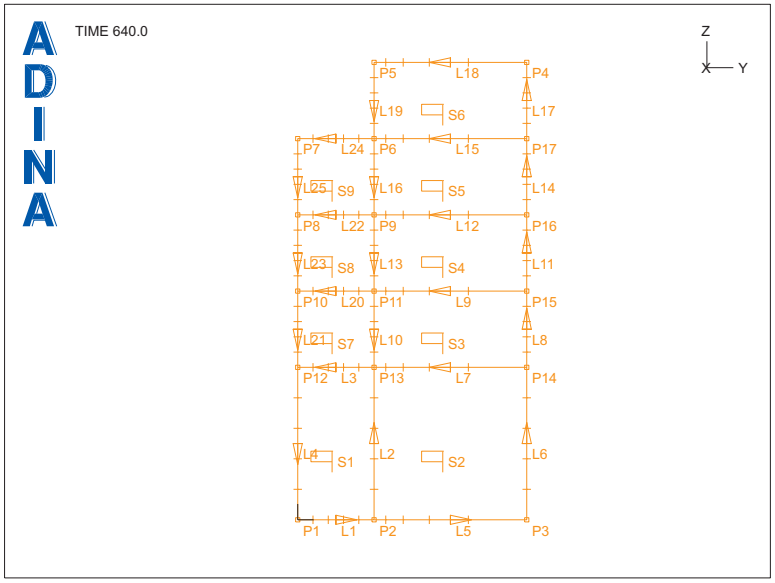
Defining the element groups

We need three element groups, group 1 for the concrete elements, group 2 for the rock elements and group 3 for the boundary convection elements. Click the Element Groups icon  and add element group 1. Set the Type to 2-D Conduction, verify that the Element Sub-

Type is Axisymmetric, make sure that the Default Material is set to 1 and click Save. Now add element group 2, set the Type to 2-D Conduction, verify that the Element Sub-Type is Axisymmetric, set the Default Material to 2 and click Save. Finally add element group 3, set the Type to Boundary Convection, set the Element Sub-Type to Axisymmetric, set the Default Material to 3 and click OK.

Defining the subdivision data

We will set the subdivisions of the surfaces explicitly. Click the Subdivide Surfaces icon  , set the Surface Number to 1, set the number of subdivisions in the u and v directions to 5, enter 7, 8, 9 in the first three rows of the table and click Save. Now set the Surface Number to 2, set the number of subdivisions in the u and v directions to 5, set the "Length Ratio of Element Edges" for the u direction to 0.2, enter 3, 4, 5, 6 in the first four rows of the table and click OK. The graphics window should look something like this:



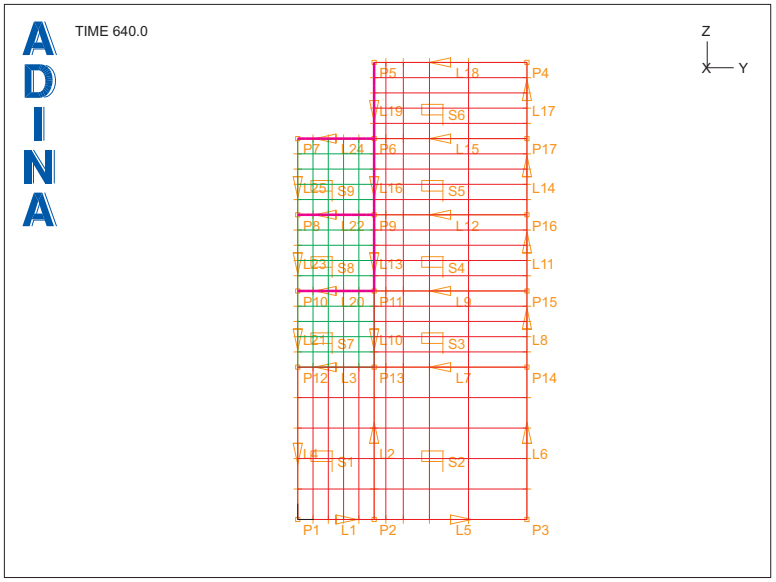
Defining the finite elements and nodes

Concrete elements: Click the Mesh Surfaces icon  , set the Type to 2-D Conduction, make sure that the Element Group is set to 1, enter surface numbers 7, 8, 9 in the first three rows of the table and click Apply.

Rock elements: Set the Element Group to 2, enter surface numbers 1 to 6 in the first six rows of the table and click OK.

Convection elements: Click the Mesh Lines icon  , enter lines 20, 13, 22, 16, 24, 19 in the first six rows of the table and click OK.

When you click the Color Element Groups icon  , the graphics window should look something like this:



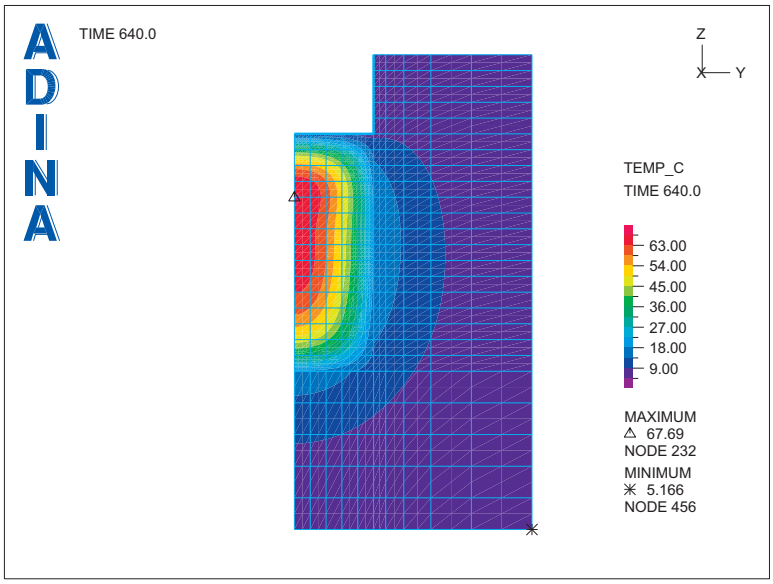
Generating the data file, running ADINA Thermal, loading the porthole file

Click the Save icon  and save the database to file `prob25`. Click the Data File/Solution icon  , set the file name to `prob25`, make sure that the Run Solution button is checked and click Save. When ADINA Thermal 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  and open porthole file `prob25`.

Displaying the temperatures

To display bands of temperature for the last solution time, click the Quick Band Plot icon  . We would rather display the temperatures in degrees C, so choose Definitions→ Variable→Resultant, add resultant TEMP_C, define it as `TEMPERATURE-273` and click OK. Then click the Modify Band Plot icon  , set the Band Plot Variable to (User Defined: TEMP_C) and click OK.

The graphics window should look something like this:



Now we reduce the number of colors used in the band plot (we do this so that the animation discussed below is processed more quickly). Click the Modify Band Plot icon, click the Band Table... button, set the Number of Colors to 4 and click OK twice to close both dialog boxes.

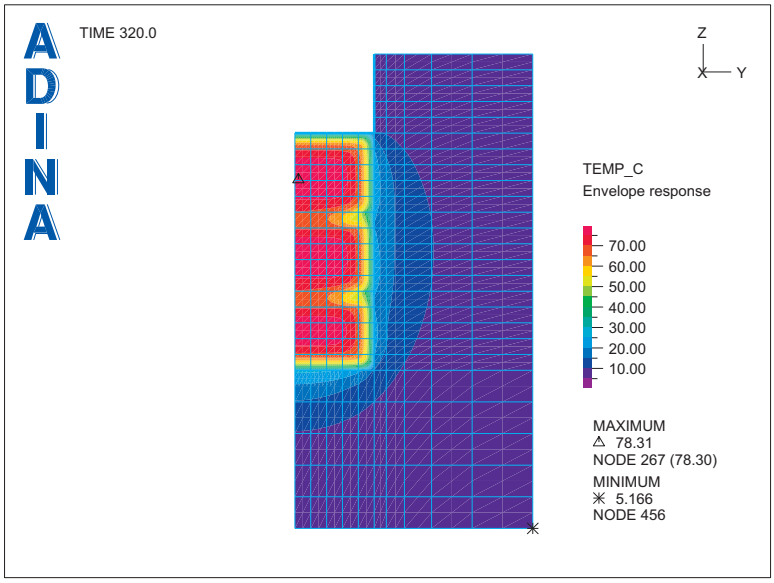
To create an animation showing the temperature as a function of time, click the Movie Load Step icon. The AUI creates the animation frame-by-frame and displays each frame after it has been computed. You can see the concrete blocks as they are added.

When the movie shoot is finished, display the animation by clicking the Animate icon. To display the animation more slowly, choose Display→Animate, set the Minimum Delay greater than 0 and click OK. Click the Refresh icon to remove the last frame of the animation from the display when you are finished viewing the animation.

Making a band plot of the envelope of the temperatures

We will plot the temperatures using an envelope. At each point in the model, the AUI determines the maximum temperature over the entire range of solution times, then the AUI plots the results as bands.

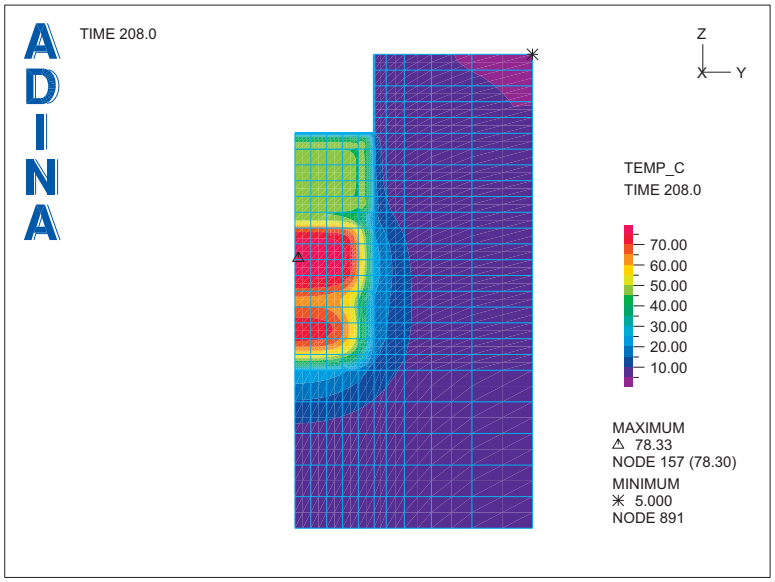
Click the Clear icon  , then the Mesh Plot icon  . To prepare for plotting the temperatures using an envelope, choose Definitions→Response, set the Response Name to DEFAULT, set the Type to Envelope and click OK. Then click the Create Band Plot icon  , set the Band Plot Variable to (User Defined:TEMP_C) and click OK. The graphics window should look something like this:



Determining the maximum temperature at the nodes, plotting the results for the corresponding solution time

Choose List→Extreme Values→Zone, in the Extreme Values box, set the Number to 5, set Variable 1 to (User Defined:TEMP_C) and click Apply. The AUI lists the maximum temperature of 7.82978E+01 (degrees C) at time 2.08000E+02 (hours) at node 128. The AUI also lists the next four maximum temperatures. The fourth maximum temperature is 7.82960E+01 at node 267, and from the plot, node 267 is the location of the band plot close maximum. Click Close to close the dialog box.

To plot the entire temperature field at time 208, choose Definitions→Response, set the Response Name to DEFAULT, set the Type to Load Step, set the Solution Time to 208 and click OK. Now click the Clear icon  and the Mesh Plot icon  , then click the Create Band Plot icon  , set the Band Plot Variable to (User Defined:TEMP_C) and click OK. The graphics window should look something like the figure on the next page.



Note: the plots show slightly larger temperatures than the listings, because the plots consider temperatures within the elements as well as nodal temperatures. Because quadratic elements are used, the maximum temperatures occur within elements.

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