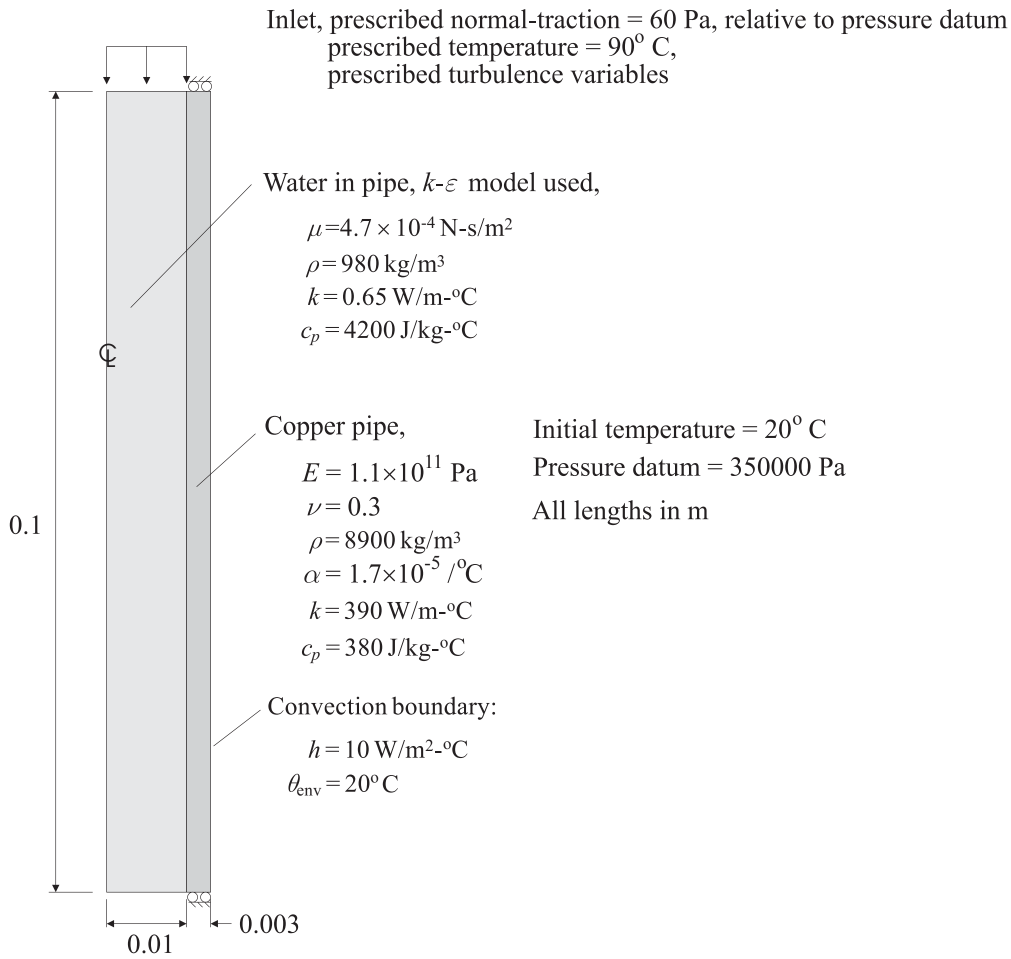
Problem description

Consider a copper pipe containing water. Initially the water is at rest, with a pressure of 350000 Pa, and the temperature of the pipe and water is 20° C. Then water at 90° C flows into the pipe with a pressure drop of 60 Pa.

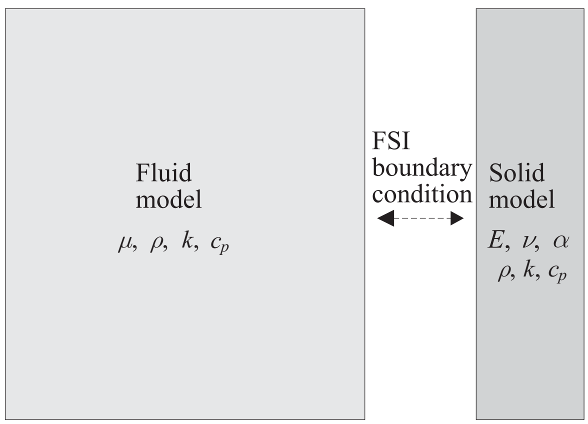


We want to compute the stresses in the pipe.

The analysis is considered to be transient in the fluid and heat transfer analysis, but static in the stress analysis.

The pressure datum feature is used in the fluid model. In this feature, the pressure datum is only added to the FSI stresses passed to the structure, as well as to the output. Internally, the fluid flow does not include the pressure datum.

This problem will be solved using two-way thermal FSI (TFSI), with pressure, velocity and temperature coupling between the fluid and the solid.



The analysis is fully coupled. In addition to the usual pressure and displacement coupling on the FSI boundary, the heat fluxes are also coupled on the FSI boundary.

The analysis consists of two parts:

Part 1: Analysis to obtain static initial condition

The pressure datum in the fluid is set to 350000 Pa and the inlet temperature is set to 20^o C.

Part 2: Transient analysis

The inlet pressure is suddenly increased to 60 Pa (relative to the pressure datum) and the inlet pressure is suddenly increased to 90^o C. The pressure drop of 60 Pa is chosen to give a fluid velocity on the order of 1 m/s.

Both of these analyses are performed in the same solution run. Part 1 is solved for using a time step of 100 sec, and part 2 is solved for using 30 time steps of size 0.1 sec.

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

• Performing a TFSI analysis
• Using the pressure datum feature in the fluid model
• Using the corner node temperature interpolation feature in the structures model

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 of the ADINA System does not contain ADINA-FSI.

Much of the input for this problem is stored in the following files: `prob51_1.in`, `prob51_2.in`. You need to copy these files from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

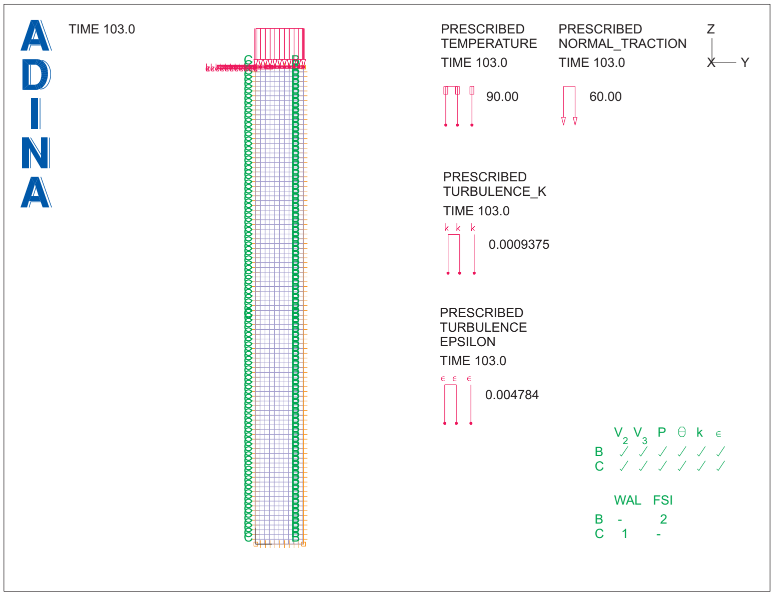
Invoke the AUI and set the Program Module drop-down list to ADINA CFD.

Model definition - fluid model

We have prepared a batch file (`prob51_1.in`) that defines most of the fluid model:

► Transient analysis, FCBI-C elements, turbulence analysis, FSI analysis, iteration tolerances.
► Geometry points, lines, surfaces.
► Material model.
► Boundary conditions.
► Initial conditions for the temperature.
► Time steps, time functions, inlet boundary conditions. The turbulence boundary condition is defined in terms of a velocity of 1.0 m/s and a length of 0.02 m (the pipe diameter). The normal traction boundary condition is specified as 60 Pa (inlet pressure relative to the pressure datum).
► Element groups and meshing.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob51_1.in` and click Open. The graphics window should look something like the figure on the next page.



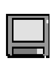
Thermal FSI: Choose Model→Flow Assumptions and set "Thermal Coupling" to "Through Fluid-Structure Interface Boundary" (do not close the dialog box yet).

Pressure datum: We have not yet defined the pressure datum. In the Specify Flow Assumptions dialog box, click the Pressure Datum... button. In the Specify Pressure Datum dialog box, click the ... button to the right of the Time Function field. In the Define Time Function dialog box, add time function 4, define it according to the following table and click OK.

Time	Value
0	0
100	1
1E20	1

In the Specify Pressure Datum dialog box, set "Prescribed on" to Model, the Multiplier to 350000, the Time Function to 4 and click OK twice to close both dialog boxes.

Generating the ADINA CFD data file

Click the Save icon  and save the database to file `prob51_f`. Click the Data File/Solution icon , set the file name to `prob51_f`, uncheck the Run Solution button and click Save.

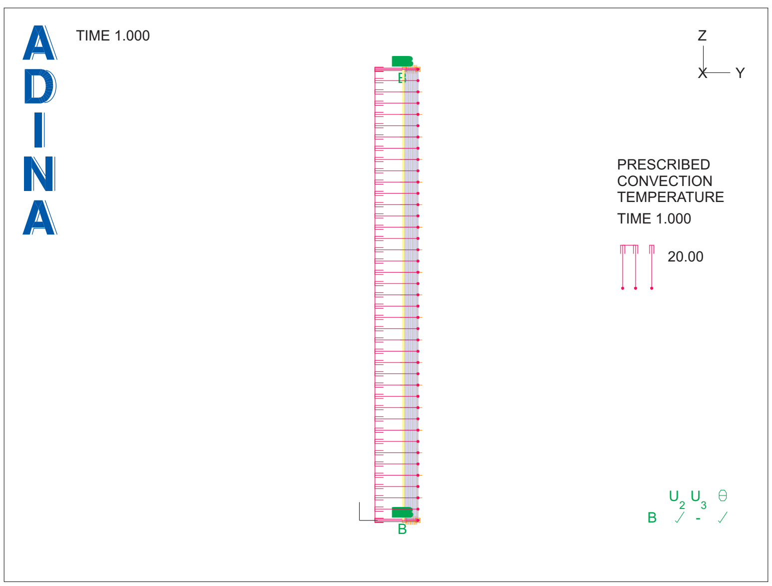
Model definition - solid model

We have prepared a batch file (`prob51_2.in`) that defines the solid model:

- ► New database.
- ► Current FE program set to ADINA Structures.
- ► FSI analysis
- ► Transient TMC analysis with iterative coupling
- ► Geometry points, lines, surfaces.
- ► Boundary conditions, including an FSI boundary.
- ► Initial conditions for the temperature.
- ► Structural material model. An elastic material with coefficient of thermal expansion is used.
- ► Thermal material model.
- ► Element group and meshing. 9-node elements are used in the solid model.
- ► Convection load

Notice that there is no time stepping information defined in the solid model.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob51_2.in` and click Open. The graphics window should look something like this:



Corner node temperature interpolations: Because the structural analysis uses 9-node elements, we need to use only the corner nodes of these elements for the thermal FSI analysis. Choose Control→TMC Model and click the ... button to the right of the "Type of Solution" field. In the Heat Transfer Analysis Control dialog box, check the "Use Corner Nodes for Heat Flow Solution" field and click OK twice to close both dialog boxes.

Saving thermal strains: We would like to examine the thermal strains during post-processing. Choose Control→Porthole→Select Element Results, add Result Selection 1, set Thermal to All, then click OK.

Generating the ADINA Structures data file

Choose File→Save As and save the database to file `prob51_a`. Click the Data File/Solution icon, set the file name to `prob51_a`, uncheck the Run Solution button and click Save.

Running ADINA-FSI

Choose Solution→Run ADINA-FSI, click the Start button, select file `prob51_f`, then hold down the Ctrl key and select file `prob51_a`. The File name field should display both file names in quotes. Then click Start.

ADINA-FSI runs for 31 time steps. When ADINA-FSI is finished, close all open dialog boxes, and set the Program Module drop-down list to Post-Processing (you can discard all changes). Click the Open icon and open porthole file `prob51_f`, then click the Open icon and open porthole file `prob51_a`.

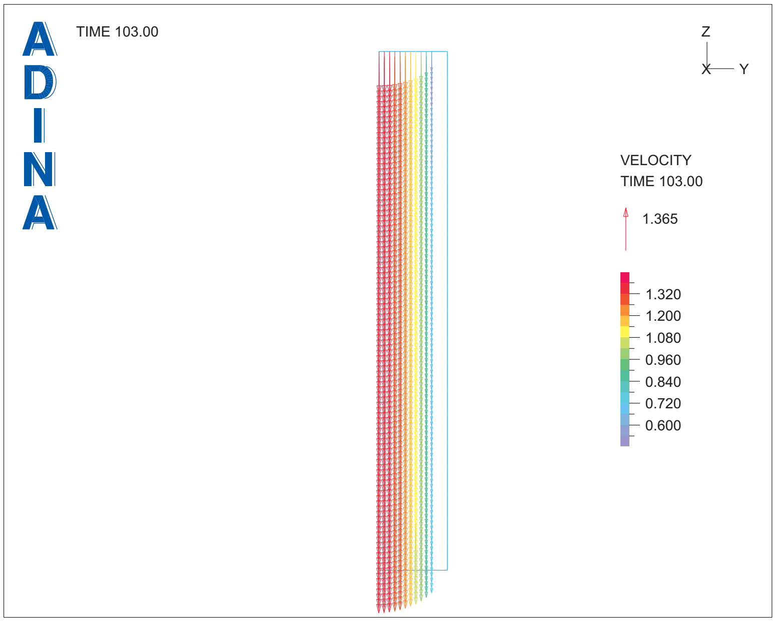
Post-processing - velocities

Click the Model Outline icon, then click the Save Mesh Plot Style icon.

Click the Create Vector Plot icon, set the Vector Quantity to VELOCITY and click OK. The graphics window should look something like the figure on the next page.

The velocity field is fully developed, and the maximum velocity is comparable to the velocity used in the turbulence load specifications.

Now click the First Solution icon to observe the initial velocity field. The velocities are all zero, as expected. As you click the Next Solution icon repeatedly, you can see the velocity field developing.

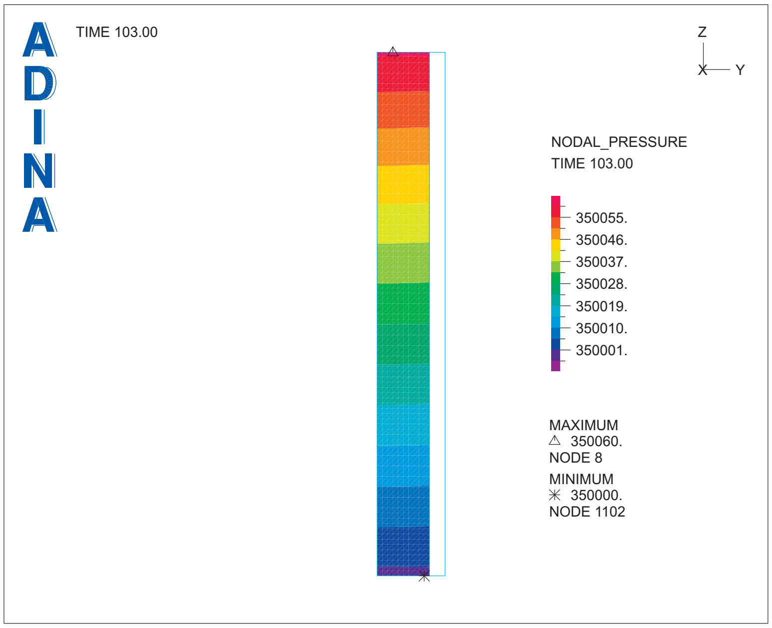


Post-processing - fluid pressures

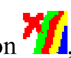
Click the Last Solution icon, click the Clear Vector Plot icon, then click the Create Band Plot icon, set the Band Plot Variable to (Stress: NODAL_PRESSURE) and click OK. The graphics window should look something like the figure on the next page.

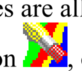
The fluid pressure includes both the pressure datum effect and the prescribed inlet normal traction.

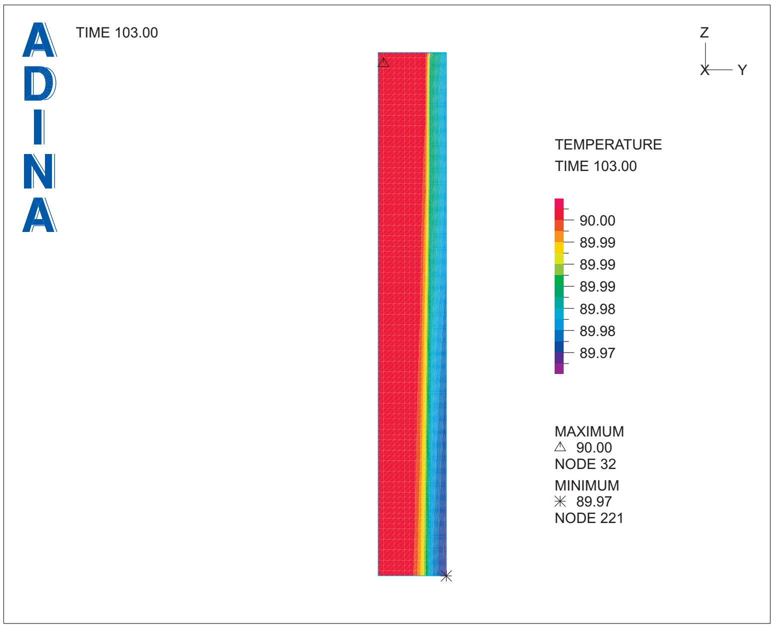
Now click the First Solution icon to observe the initial pressure field. The pressure in the fluid is equal to the pressure datum. When you click the Next Solution icon, notice that the pressure drop is immediately established, and as you continue to click the Next Solution icon, notice that the pressure drop changes very little.

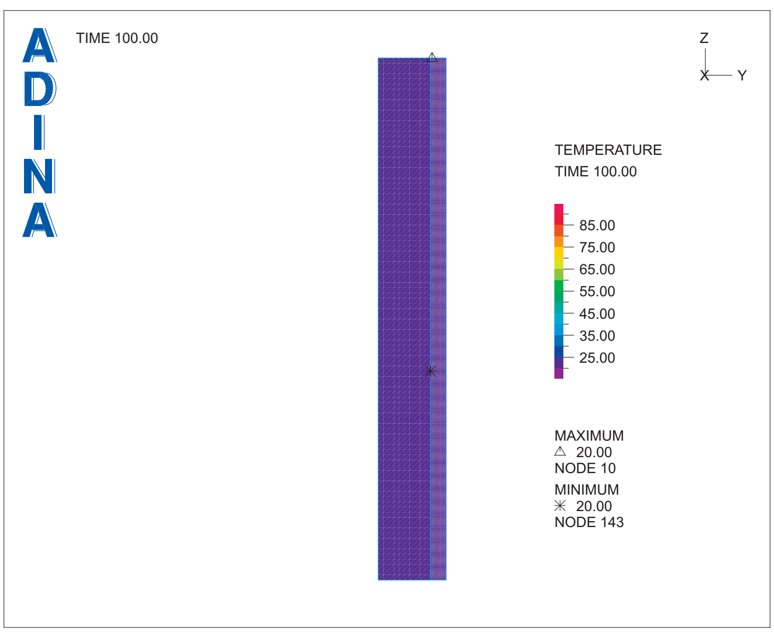


Post-processing - temperatures

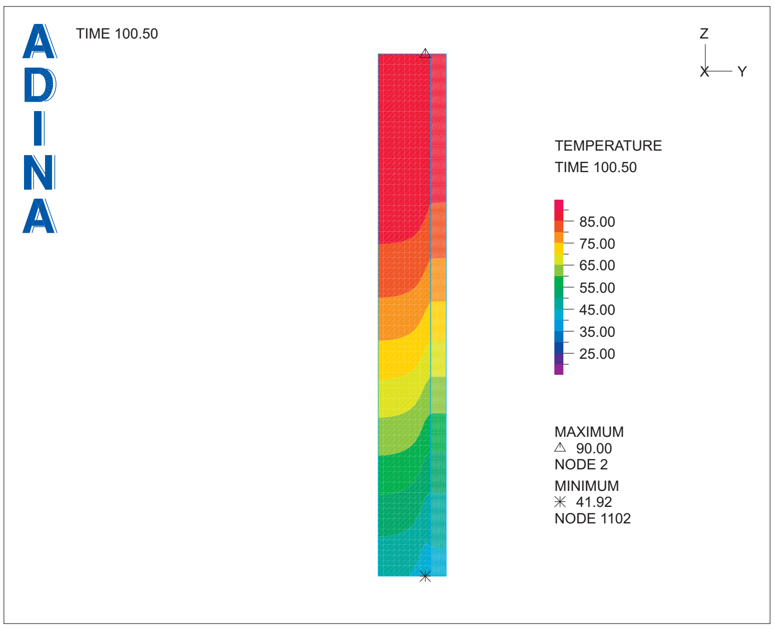
Click the Last Solution icon , click the Clear Band Plot icon , then click the Create Band Plot icon , set the Band Plot Variable to (Temperature: TEMPERATURE) and click OK. The graphics window should look something like the top figure on the next page.

Since the temperatures are all near 90º, the bands do not show lower temperatures. Click the Modify Band Plot icon , click the Band Table ... button, set the Minimum to 20 and click OK twice to close both dialog boxes. When you click the First Solution icon , the graphics window should look something like the bottom figure on the next page.


ADINA
TIME 103.00
Z
X
Y
TEMPERATURE
TIME 103.00
90.00
89.99
89.99
89.99
89.98
89.98
89.97
MAXIMUM
△ 90.00
NODE 32
MINIMUM
✳ 89.97
NODE 221


ADINA
TIME 100.00
Z
X
Y
TEMPERATURE
TIME 100.00
85.00
75.00
65.00
55.00
45.00
35.00
25.00
MAXIMUM
△ 20.00
NODE 10
MINIMUM
✳ 20.00
NODE 143

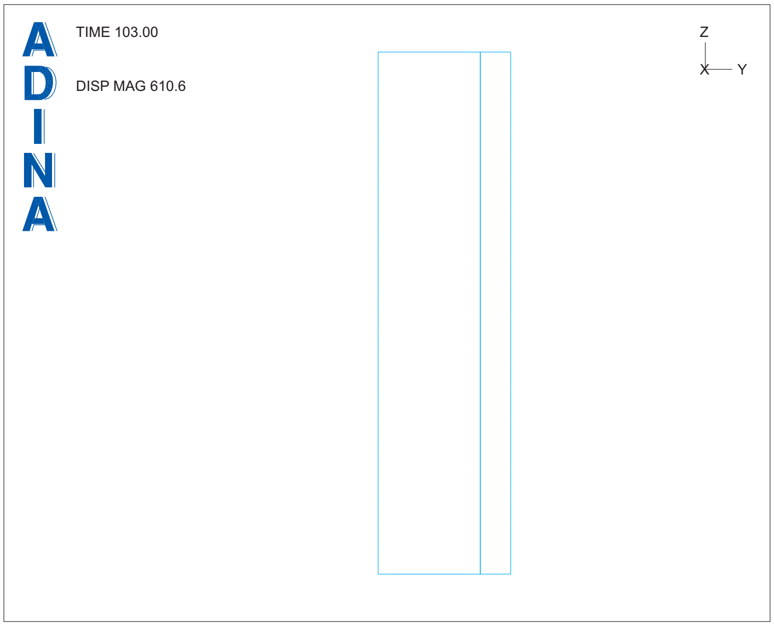
The fluid and structure temperatures are all 20º, as expected. When you click the Next Solution icon repeatedly, notice that the temperatures rise rapidly in the fluid and structure. At first, the temperature rises more rapidly in the structure (the thermal conductivity is higher in the structure), but then the temperature rises more rapidly in the fluid (due to convection of the fluid). For example, the solution for time 100.5 should look something like this:

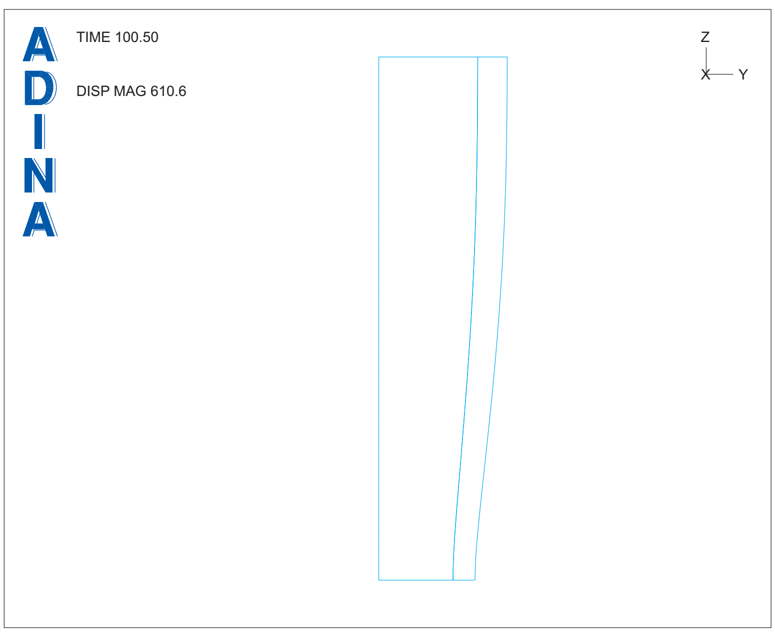


Post-processing - displacements

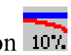
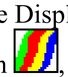
Click the Last Solution icon , then click the Clear Band Plot icon and the Scale Displacements icon . The graphics window should look something like the top figure on the next page.

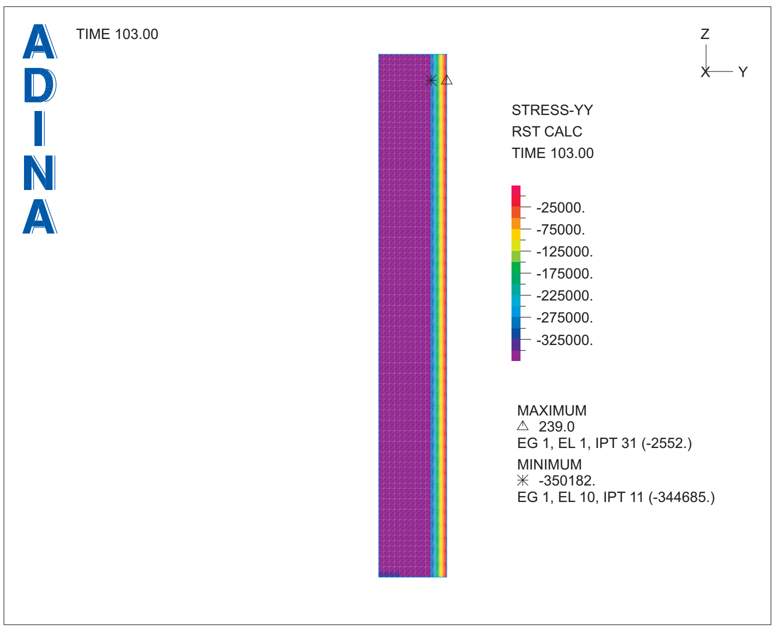
The pipe wall has moved outwards, as expected due to both the pressure and thermal effects. Click the First Solution icon to examine the displacements due to just the pressure effects. Clearly the pressure effects alone do not cause significant displacement of the pipe wall. When you click the Next Solution icon repeatedly, notice that the pipe wall moves outwards starting from the top of the model; this corresponds to the temperature rise in the pipe wall. For example, the solution for time 100.5 should look something like the bottom figure on the next page.


ADINA
TIME 103.00
DISP MAG 610.6
Z
X
Y


ADINA
TIME 100.50
DISP MAG 610.6
Z
X
Y

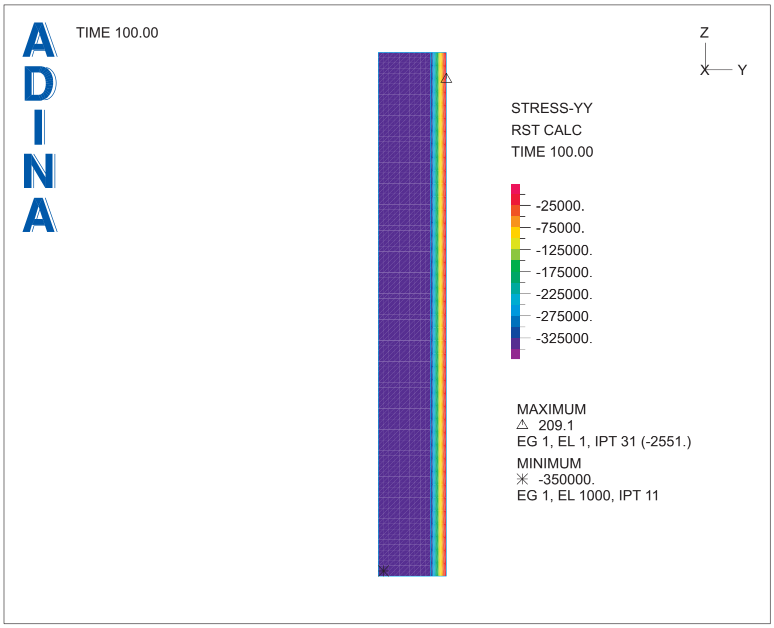
Post-processing - radial stress

Click the Last Solution icon  , then click the Scale Displacements icon  (to unscale the displacements). Now click the Create Band Plot icon , set the Band Plot Variable to (Stress: STRESS-YY) and click OK. The graphics window should look something like this:



In the pipe wall, STRESS-YY varies from -350000 (at the fluid-structure boundary) to 0 (at the outer surface), as expected. Click the First Solution icon  to examine the radial stress due to just the pressure effects. The graphics window should look something like the figure on the next page.

The radial stress field corresponding to just the pressure effects is very similar to the radial stress field at the end of the analysis.

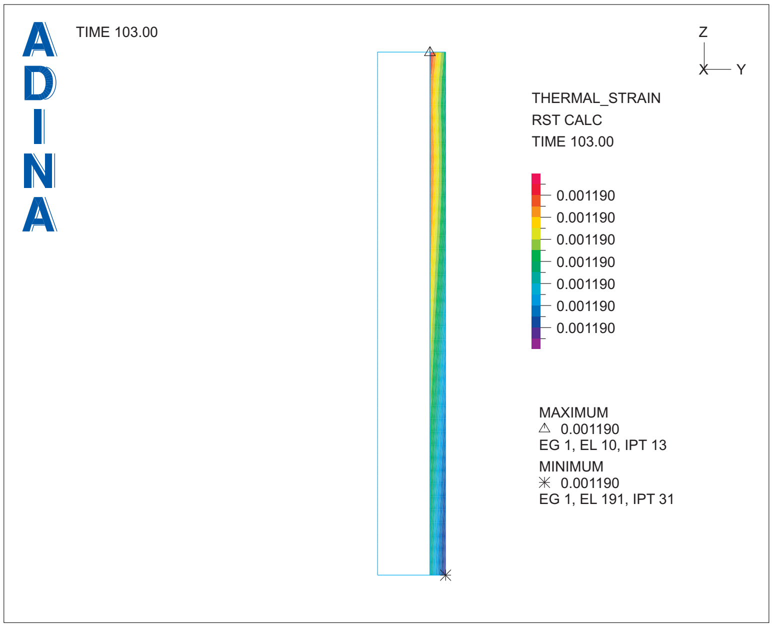


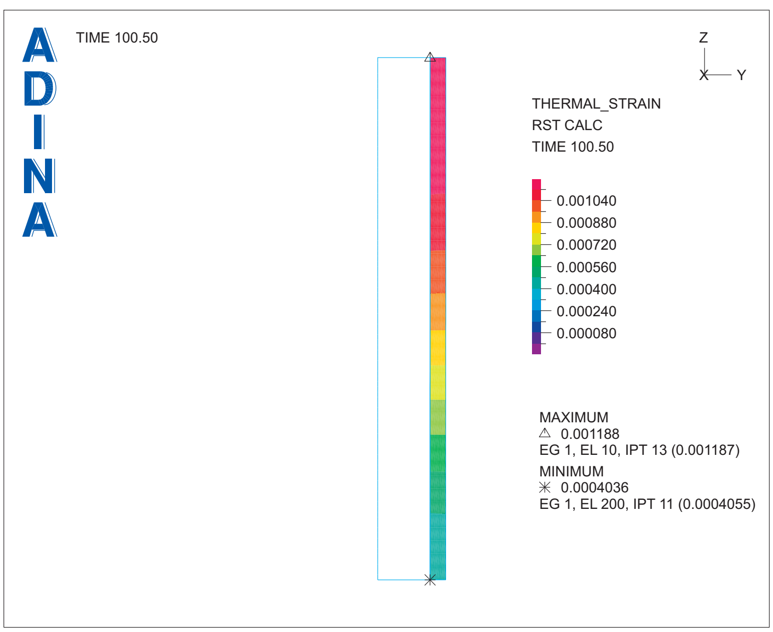
Post-processing - thermal strain

Click the Last Solution icon , click the Clear Band Plot icon , then click the Create Band Plot icon , set the Band Plot Variable to (Strain: THERMAL_STRAIN) and click OK. The graphics window should look something like the top figure on the next page.

The thermal strain is nearly constant in the pipe wall, and corresponds to the value obtained from the formula $\alpha(\theta - {}^0\theta) = 1.7\times10^{-5}(90-20) = 0.00119$.

Since the thermal strains are all near 0.00119, the bands do not show lower thermal strains. Click the Modify Band Plot icon , click the Band Table ... button, set the Minimum to 0 and click OK twice to close both dialog boxes. When you click the First Solution icon , notice that the thermal strains are zero, then when you click the Next Solution icon repeatedly, notice that the thermal strain rapidly increases, corresponding to the temperature increase. For example, the solution for time 100.5 should look something like the bottom figure on the next page.


ADINA
TIME 103.00
Z
X Y
THERMAL_STRAIN
RST CALC
TIME 103.00
0.001190
0.001190
0.001190
0.001190
0.001190
0.001190
0.001190
MAXIMUM
△ 0.001190
EG 1, EL 10, IPT 13
MINIMUM
✳ 0.001190
EG 1, EL 191, IPT 31


ADINA
TIME 100.50
Z
X Y
THERMAL_STRAIN
RST CALC
TIME 100.50
0.001040
0.000880
0.000720
0.000560
0.000400
0.000240
0.000080
MAXIMUM
△ 0.001188
EG 1, EL 10, IPT 13 (0.001187)
MINIMUM
✳ 0.0004036
EG 1, EL 200, IPT 11 (0.0004055)

Post-processing - detailed stress analysis in the pipe wall

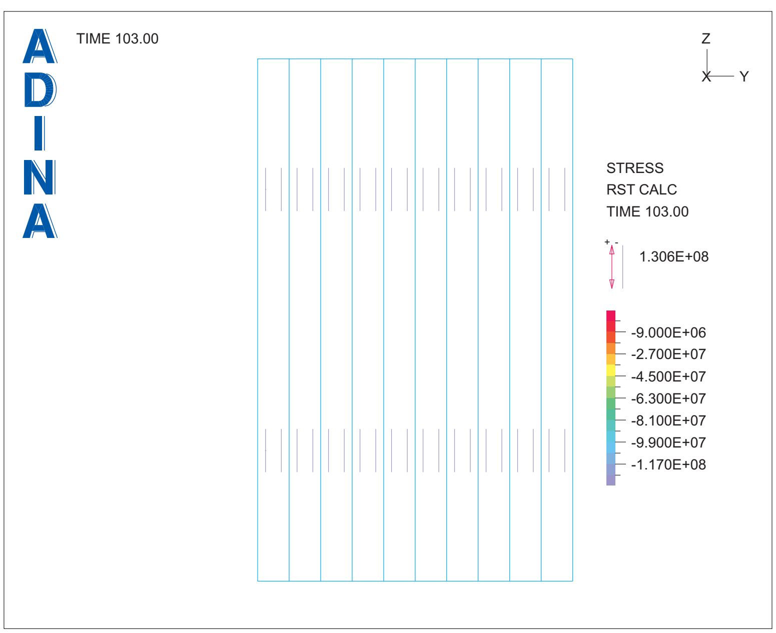
We would now like to examine the stresses at the top of the pipe wall. Click the Last Solution icon , the Reset Mesh Plot Style icon and the Clear icon .

Zone: Click the Display Zone icon , click the ... button to the right of the Zone Name field and add zone name PIPE_WALL_TOP. Click the Edit button, type, in the first row of the table:

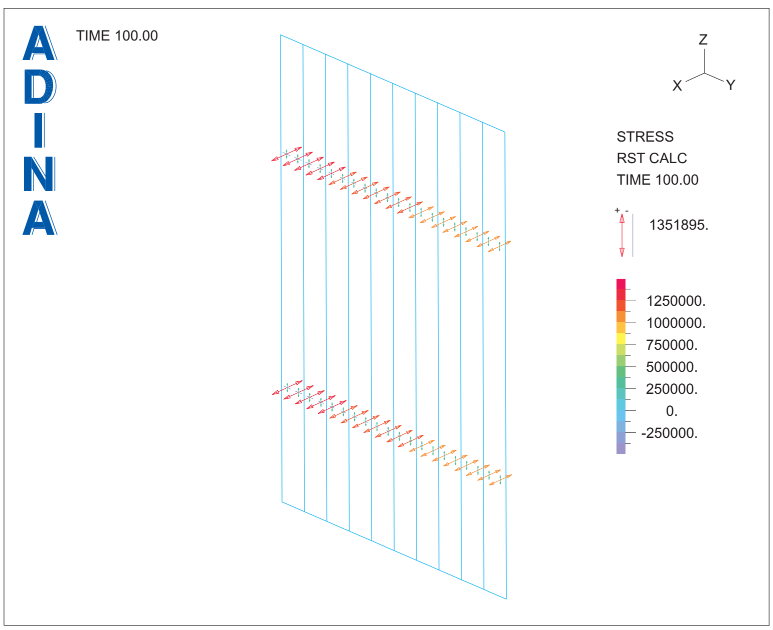
```
ELEMENTS 1 TO 10 OF PROGRAM ADINA
```

and click OK. In the Create Mesh Plot with Specified Zone dialog box, make sure that the Zone Name is set to PILE_WALL_TOP and click OK.

Stress vectors: Click the Quick Vector Plot icon . The graphics window should look something like this:



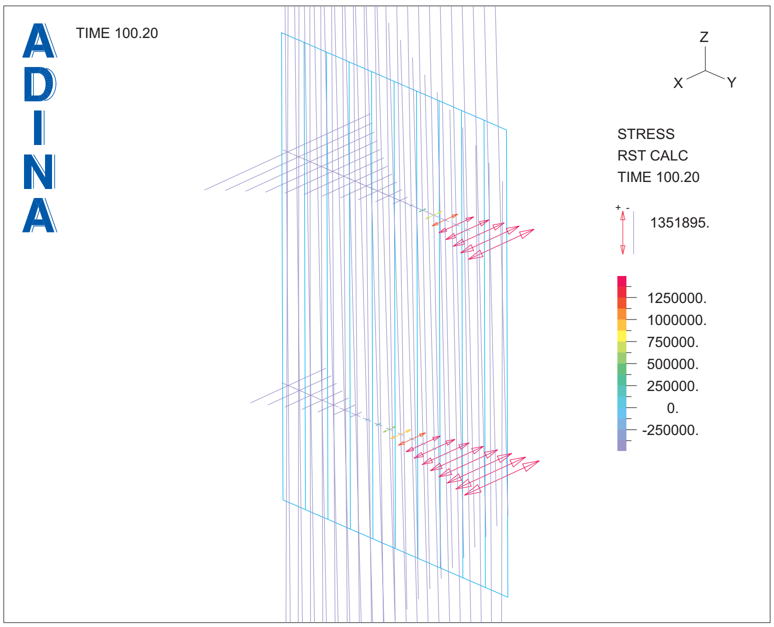
Evidently the stress field is dominated by compressive axial stresses, which is expected since the pipe wall is constrained in the axial direction. When you click the First Solution icon , the stresses are apparently much smaller. To see these stresses, click the Clear Vector Plot icon , then click the Quick Vector Plot icon . In this view, the hoop stresses are not visible. Use the mouse to rotate the mesh plot out-of-plane until the graphics window should look something like this:

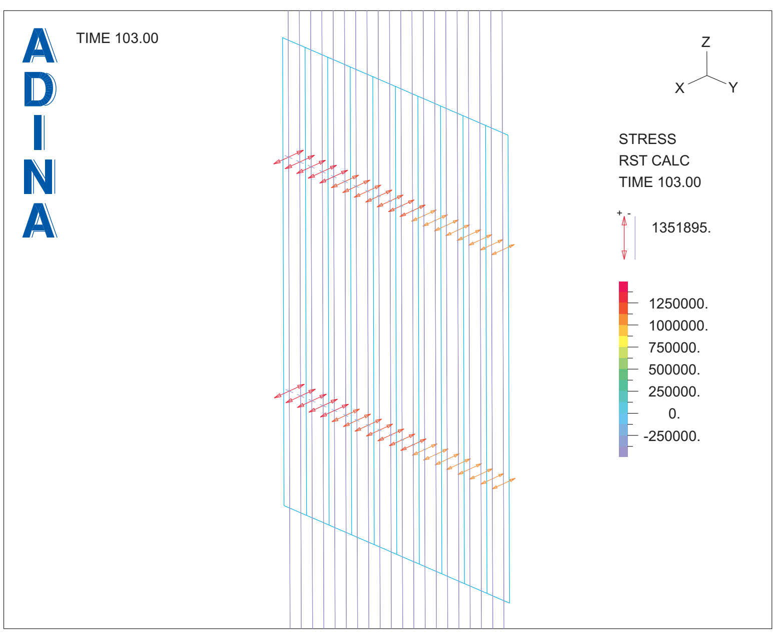


Initially the stress field is dominated by the hoop stresses. When you click the Next Solution icon repeatedly, notice that the stress field is highly influenced by the thermal effects. For example, the solution for time 100.2 should look something like the top figure on the next page.

Now click the Last Solution icon . The graphics window should look something like the bottom figure on the next page.

Comparing the steady-state solution with thermal stresses to the solution without thermal effects, it is seen that the only difference is in the axial stresses.


ADINA
TIME 100.20
Z
X
Y
STRESS
RST CALC
TIME 100.20
1351895.
1250000.
1000000.
750000.
500000.
250000.
0.
-250000.


ADINA
TIME 103.00
Z
X
Y
STRESS
RST CALC
TIME 103.00
1351895.
1250000.
1000000.
750000.
500000.
250000.
0.
-250000.

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

Notes

1) The pressure datum feature is very useful for incompressible flow problems in which the nominal fluid pressure is non-zero. However, the pressure datum feature cannot be used for general compressible flows.

2) In the zone definition, it is necessary to use the string '`OF PROGRAM ADINA`' because both structural and fluid results are loaded. The string '`OF PROGRAM ADINA`' specifies that the zone contains only ADINA Structures elements. To select only ADINA CFD elements, use the string '`OF PROGRAM ADINA-F`'.

3) The analytical thermal stresses due to only a uniform thermal expansion are $\tau_{xx} = \tau_{yy} = 0$, $\tau_{zz} = -E\varepsilon^t = -1.1\text{E+}11 \cdot 0.00119 = -1.309\text{E+}08$. This is why the only difference between the initial stress state and the steady-state thermal stress state is the axial stress. (Recall that the stress analysis is linear, so the principle of superposition applies.)