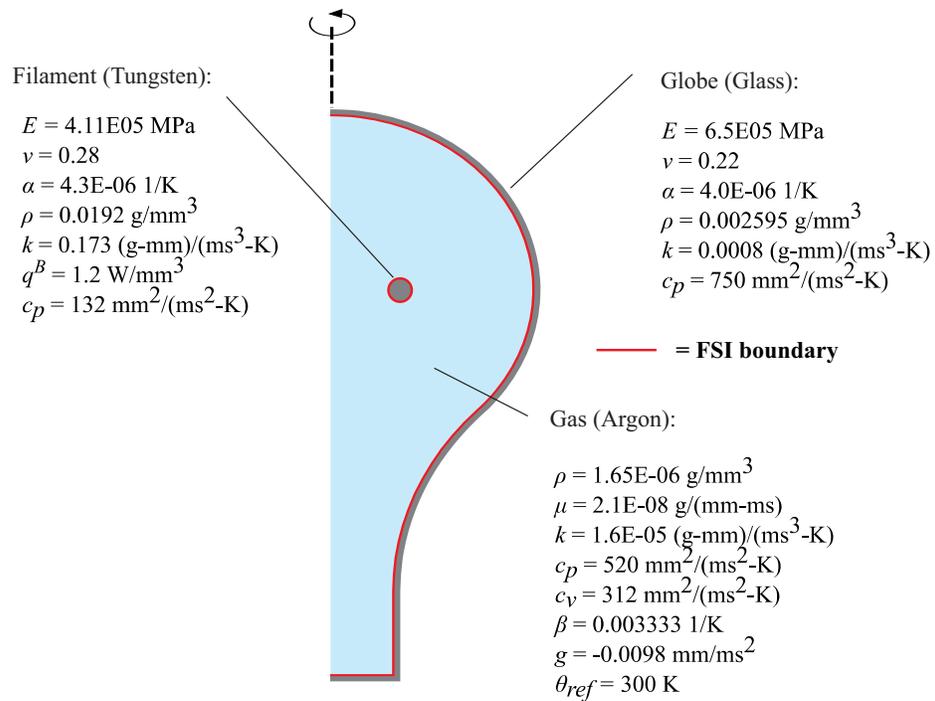


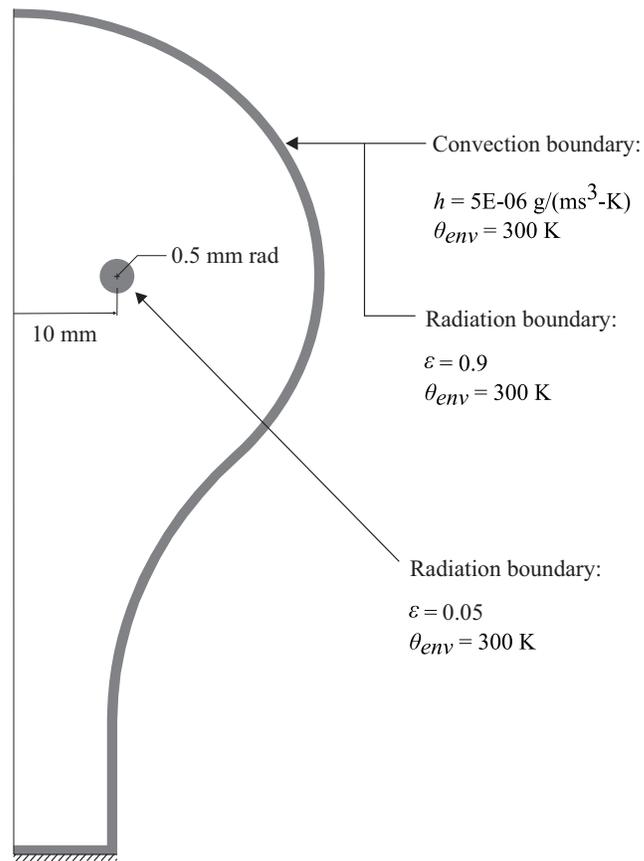
Problem description

This tutorial demonstrates the use of ADINA for analyzing the fluid flow and heat transfer in a lightbulb using the Thermal Fluid-Structure Interaction (TFSI) features of ADINA. The 2D model under consideration is shown schematically in the figures below.



Problem 65: Free convection in a lightbulb

Solid model



The internal heat generation in the filament is calculated as follows:

$$q^B = q/V$$

where q is the total heat generation (in W) and V is the filament volume (in mm^3). For this problem, $V = (2\pi R)(\pi r^2)$ where R is the radial distance from the centerline to the center of the filament and r is the filament radius. Substituting $q = 60 \text{ W}$ and the filament dimensions into the above gives $q^B = 1.216 \text{ W}/\text{mm}^3$.

The FCBI-C element is used in the fluid part of the model.

In this tutorial, we will demonstrate the following topics that have not been presented in previous problems.

- Specifying internal heat generation loading in the solid 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 TFSI is not supported in the 900 nodes version.

Much of the input for this problem is stored in files `prob65_1a.in`, `prob65_2a.in`, `prob65_3a.in`, `prob65_1f.in`, and `prob65_ees.txt`. 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 Structures. Set the Analysis Type to "Statics" and set the Multiphysics Coupling drop-down list to "with CFD".

Model definition for the structural model

Defining model control data

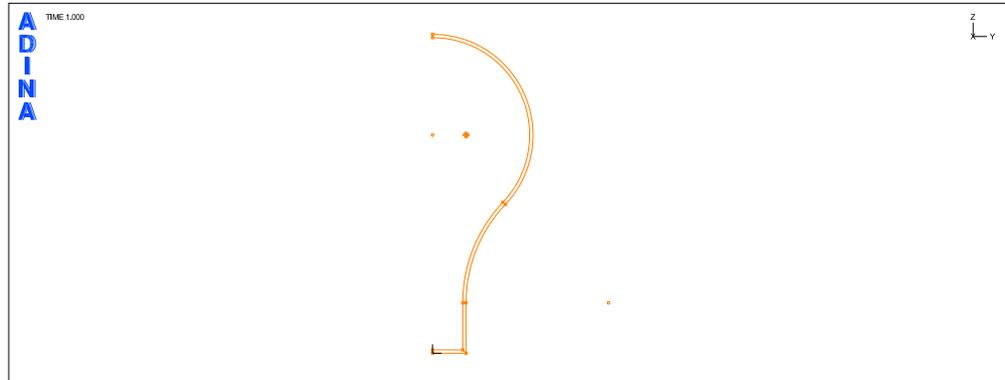
Problem heading: Choose Control→Heading and set the Problem Heading to "Free Convection in a Lightbulb" and click OK.

Thermal analysis: Choose Control→TMC Model, set the "Type of Solution" to "TMC Direct Coupling", click the ... button to the right of that field, check the "Use Corner Nodes for Heat Flow Solution" button, then click OK twice to close both dialog boxes.

Geometry definitions

The included batch file `prob65_1a.in` contains commands for generating the structural model's geometry. To run this batch file, choose File→Open Batch, select file `prob65_1a.in` and click Open. The graphics window should look something like the figure on the top of the next page.

Problem 65: Free convection in a lightbulb



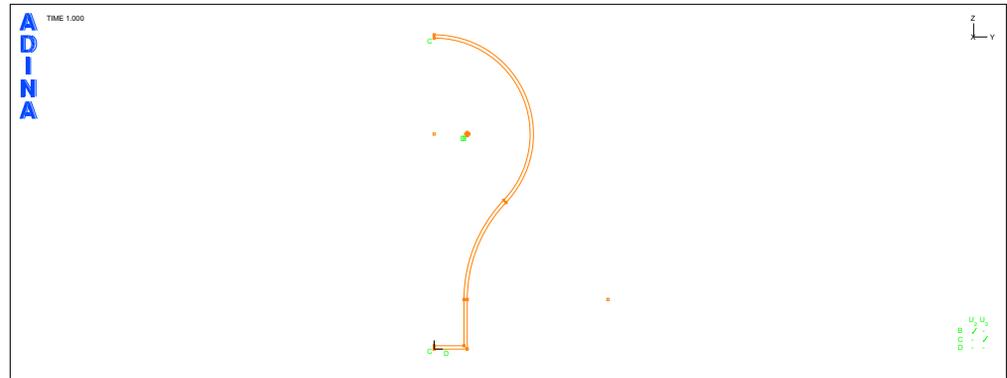
Defining the material properties

Click the Manage Materials icon  and click the “Elastic Isotropic” button. In the Define Isotropic Linear Elastic Material dialog box, add material 1. Set the Description field to “Tungsten”, the Young’s Modulus to $4.11E5$, the Poisson’s Ratio to 0.28, the Density to 0.0192, and the Coefficient of Thermal Expansion to $4.3E-06$, then click Save. Add material 2, and set the Description to “Glass”. Set the Young’s Modulus to $6.5E5$, the Poisson’s Ratio to 0.22, the Density to 0.002595, and the Coefficient of Thermal Expansion to $4E-06$. Click OK to close the Define Isotropic Linear Elastic Material dialog box.

In the Manage Material Definitions dialog box, click the TMC Material button and click the “k isotropic, c constant” button. In the Define Constant Isotropic Material dialog box, add material 1 and set the Description to “Tungsten”. Set the Thermal Conductivity to 0.173, the Heat Capacity/Mass to 132, and the Density to 0.0192. Click Save, add material 2 and set the Description to “Glass”. Set the Thermal Conductivity to 0.0008, the Heat Capacity/Mass to 750 and the Density to 0.002595. Click OK, and click Close twice to close both dialog boxes.

Defining boundary conditions

The fixities and FSI boundary conditions are stored in the batch file `prob65_2a.in`. To run this batch file, choose File→Open Batch, select file `prob65_2a.in`, and click Open. The graphics window should look something like the figure on the top of the next page.



Defining loadings

We will define convection and radiation loadings on the globe, and radiation and internal heat generation loadings on the filament.

Convection load

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 Load dialog box, add Convection Number 1, set the Environment Temperature to 300, then click the ... button to the right of the Convection Property field. In the Define Convection Property dialog box, add Property Number 1, set the Convection Coefficient to 5E-6 and click OK. In the Define Convection Load dialog box, set the Convection Property to 1 and click OK. Finally, in the Apply Load dialog box, set the 'Apply to' field to Line, fill in the table as follows, and click Apply (do not close the dialog box yet).

Line {p}
1
2
3
4

Radiation load on the globe

In the Apply Load dialog box, change the Load Type to Radiation, then click the Define... button to the right of the Load Number field. In the Define Radiation Load dialog box, add Radiation Number 1 and set the Environment Temperature to 300, then click the ... button to the right of the Radiation Property field. In the Define Radiation Property dialog box, add Property Number 1, set the Temperature Unit to Kelvin, the Emissivity Coefficient to 0.9, the Stefan-Boltzmann Constant to 5.66E-14 and click OK. In the Define Radiation Load dialog box, set the Radiation Property to 1 and click OK. Finally, in the Apply Load dialog box, set

Problem 65: Free convection in a lightbulb

the 'Apply to' field to Line, and fill in the table as follows, and click Apply (do not close the dialog box yet):

Line {p}
1
2
3
4

Radiation load on the filament

In the Apply Load dialog box, click the Define... button to the right of the Load Number field. In the Define Radiation Load dialog box, add Radiation Number 2 and set the Environment Temperature to 300, then click the ... button to the right of the Radiation Property field. In the Define Radiation Property dialog box, copy Property Number 1 to Property Number 2, set the Emissivity Coefficient to 0.05 and click OK to close the dialog box.

In the Define Radiation Load dialog box, set the Radiation Property to 2 and click OK. In the Apply Load dialog box, set the Load Number to 2, make sure that the 'Apply to' field is set to Line, and fill in the table as follows and click Apply (do not close the dialog box yet):

Line {p}
9
10
11
12

Internal heat generation load on the filament

In the Apply Load dialog box, 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 1.216 and click OK. In the Apply Load dialog box, set the 'Apply to' field to Surface, fill in the table as follows and click OK.

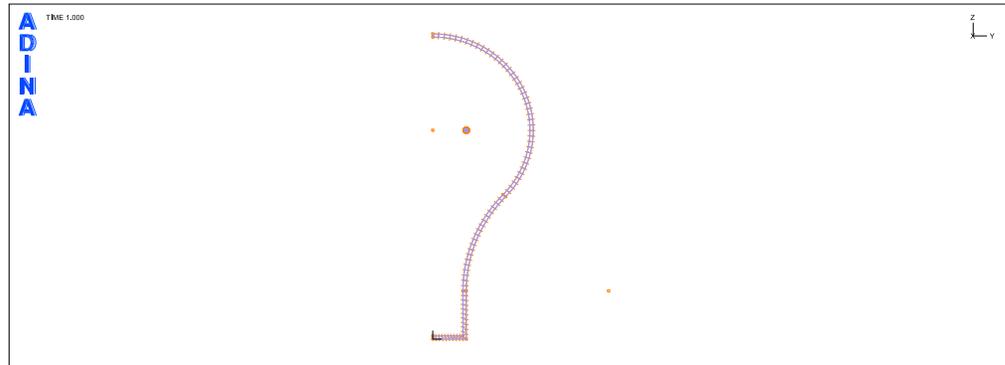
Surface {p}
5
6
7
8

Defining element groups

Click the Element Groups icon , and add Group Number 1. Set the Type to 2-D Solid, make sure that the Element Sub-Type is set to Axisymmetric, and make sure the Default and Thermal Materials are set to 1. Now add Group Number 2, set the Default and Thermal Materials to 2 and click OK.

Defining the initial conditions and mesh

The included batch file prob65_3a.in contains commands for generating the mesh and defining the initial condition. To run this batch file, choose File→Open Batch, select file prob65_3a.in and click Open. Remove the highlighting of the FSI boundaries by clicking the Show Fluid Structure Boundary icon . The graphics window should look something like this:



Generating the ADINA Structures data file and saving the AUI database

Click the Data File/Solution icon , set the file name to prob65_a, uncheck the “Run Solution” button, and click Save. Now click the Save icon  and save the database to file prob65_a.

ADINA-CFD model

Click the New icon  to create a new database, and set the Program Module drop-down list to “ADINA CFD”.

Defining model control data

FSI analysis: Set the Analysis Type to “Transient” and set the Multiphysics Coupling drop-down list to “with Structures”.

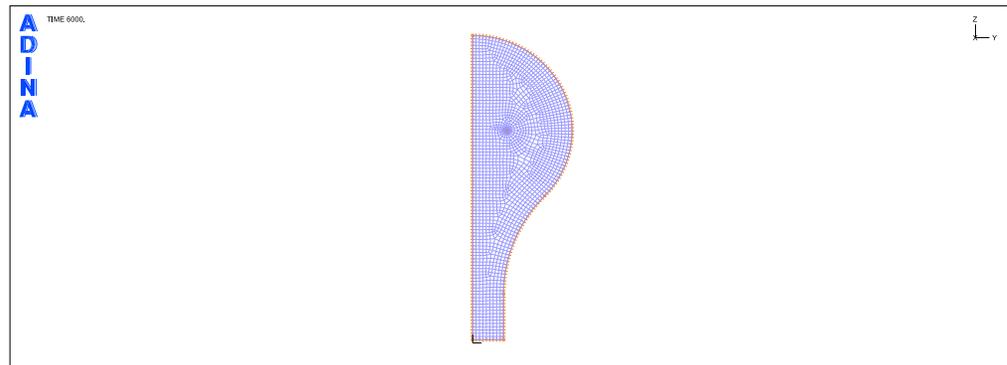
Problem heading: Choose Control→Heading, set the Problem Heading to “Free Convection in a Lightbulb” and click OK.

Flow assumptions: Choose Model→Flow Assumptions, set the Flow Dimension to 2D (in YZ plane), make sure that the Flow Type is set to Incompressible, and click OK.

Model definition

The included batch file prob65_1f.in contains the commands for generating the fluid model’s geometry, element groups and associated mesh, material settings, boundary and initial conditions, time steps, and solution process settings. To run this batch file, choose File→Open Batch, select file prob65_1f.in and click Open.

The graphics window should look something like this:



Generating the ADINA CFD data file, saving the AUI database

Click the Data File/Solution icon , set the file name to prob65_f, uncheck “Run Solution”, and click Save. Now click the Save icon  and save the database to file prob65_f.

Running ADINA-FSI

Choose Solution→Run ADINA-FSI..., click the Start button, select file prob65_f.dat, then hold down the Ctrl key and select file prob65_a.dat. The File name field should display both filenames in quotes. Click Start.

The ADINA FSI analysis takes 600 solution steps.

We will examine the output file as part of the post-processing for this model. Therefore you might not want to close the ADINA FSI window yet.

When the model is finished, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob65_a. Repeat this to open prob65_f.

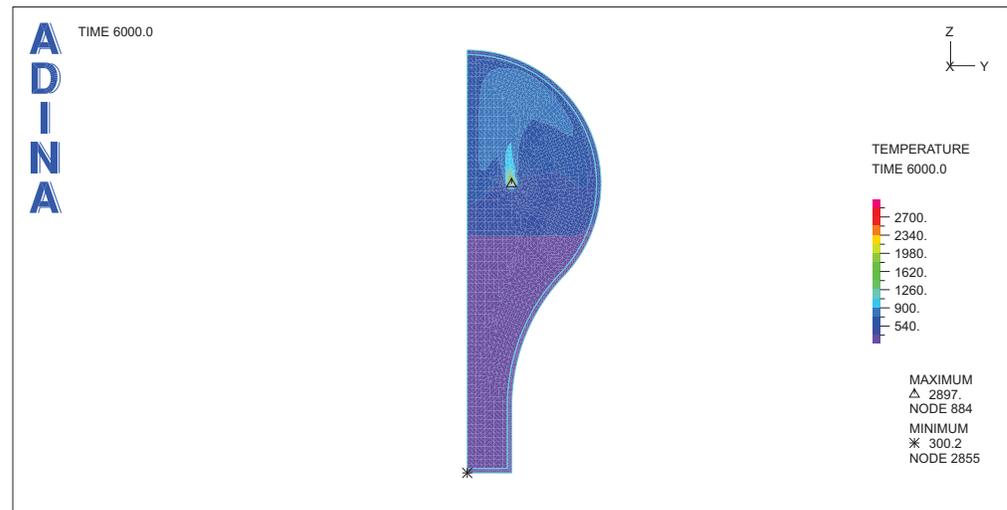
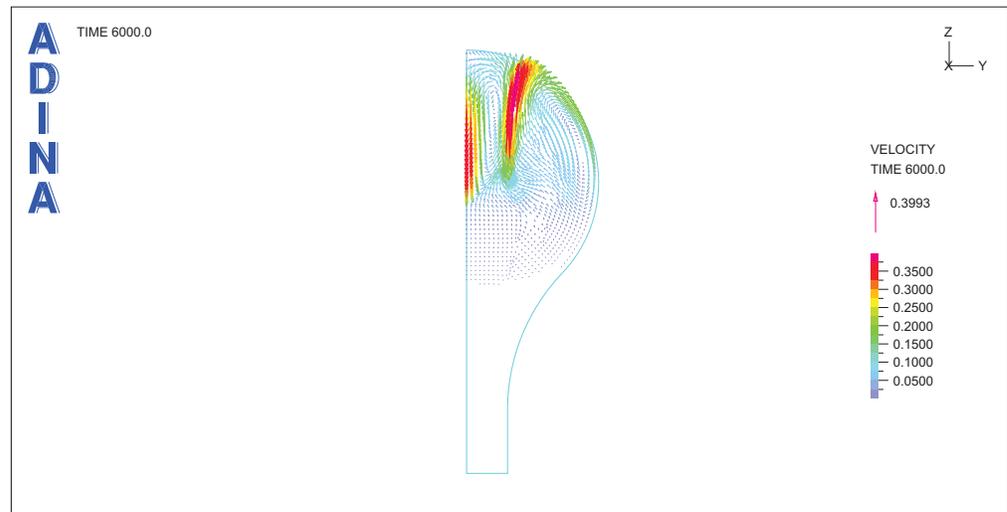
Post-Processing

In the Model Tree, expand the Zone list, right click on 2. ADINA-F, and choose Display.

Click the Model Outline icon  and click the Quick Vector Plot icon . The velocity vector plot should appear as shown on the top figure on the next page.

Click the Clear icon  and the Model Outline icon . Click the Create Band Plot icon , set the Band Plot Variable to (Temperature:TEMPERATURE) and click OK. The temperature plot should appear as shown on the bottom figure on the next page.

Problem 65: Free convection in a lightbulb



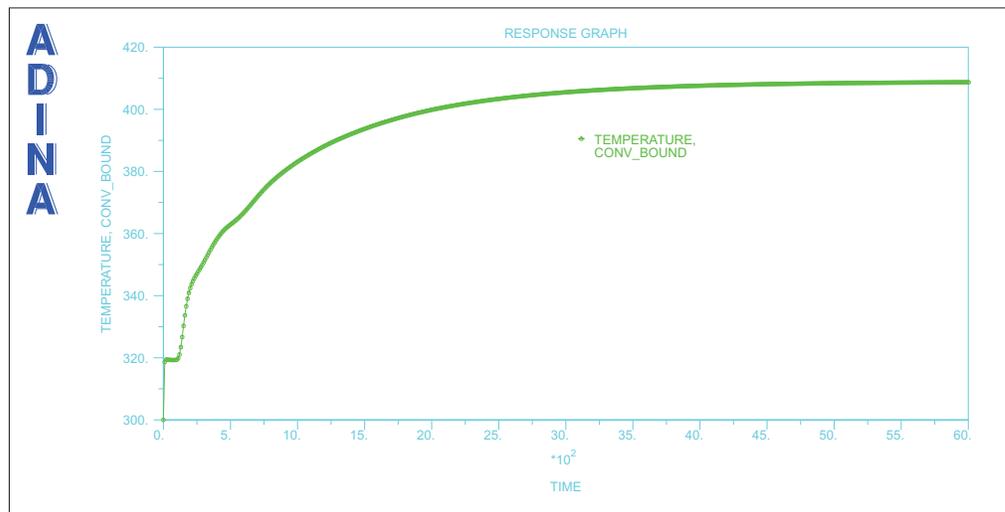
The highest temperature is at the filament (as expected), with a value of about 2897 K.

We will plot the surface temperature of the globe as a function of time. First, we need to create an element edge set corresponding to the elements along the globe. In the Model Tree, right click 1. ADINA and choose Display. Set the FE Model drop-down list to “ADINA

Structures.” Click the Element Edge Set icon  and add element edge-set number 1. Click Import..., select file prob65_ees.txt, then click OK. In the Define Element Edge Set dialog

box, click Save. The outer surface of the globe should be highlighted. Click OK to close the dialog box.

Choose Definitions→Model Point (Special) →Mesh Integration. Add name CONV_BOUND, set the “Integration Type” to Averaged, and the “Integrate Over” field to Lines. Click OK to close the dialog box. Click the Clear icon , choose Graph→Response Curve (Model Point). In the Display Response Curve (Model Point) dialog box, set the Y Coordinate Variable to (Temperature:TEMPERATURE), and click OK. The graph should look something like this:



Choose Graph→List and scroll to the bottom of the list. The temperature at the last solution time should be 4.08695+02 (your results may be slightly different because free-form meshing was used in the fluid domain).

Examining the heat flux into and out of the fluid domain

We will examine the ADINA FSI output file to determine the heat fluxes into and out of the fluid domain. Click the View Output button in the ADINA FSI window, then click the prob65_f.out tab, or open file prob65_f.out using a text editor. Scroll to the bottom of the file and scroll backwards until you see the following text:

Problem 65: Free convection in a lightbulb

```

-----
FORCE AND HEATFLOW ON WALL AND STRUCTURE AT TIME 0.60000000E+04
      integration of <pressure>          integration of <-shear><dot><n>
BC      Fp = (Fpx,Fpy,Fpz)                Fs = (Fsx,Fsy,Fsz)
-----
      4 ( 0.0000E+00 -0.6440E-21  0.2310E-35) ( 0.0000E+00 -0.6128E-24  0.5154E-38)
      1 ( 0.0000E+00  0.6247E-03 -0.8667E-04) ( 0.0000E+00  0.3939E-05 -0.1807E-05)
      2 ( 0.0000E+00 -0.4451E-06  0.7042E-06) ( 0.0000E+00 -0.2278E-06  0.1064E-05)
-----
Total ( 0.0000E+00  0.6243E-03 -0.8596E-04) ( 0.0000E+00  0.3711E-05 -0.7432E-06)

-----
      Ft = Fp + Fs          <-heatflux><dot><n>
-----
( 0.0000E+00 -0.6446E-21  0.2316E-35)  0.1098E-63
( 0.0000E+00  0.6286E-03 -0.8847E-04)  0.3195E+01
( 0.0000E+00 -0.6728E-06  0.1768E-05) -0.3204E+01
-----
( 0.0000E+00  0.6280E-03 -0.8671E-04) -0.8298E-02

```

(we have folded this output in the above so that it fits onto the page).

From this printout, we see:

- The heat flux into boundary condition 1 is 0.3195E+01 (W/radian).
- The heat flux into boundary condition 2 is -0.3204E+01 (W/radian)

Positive values correspond to heat entering the fluid domain, negative values correspond to heat leaving the fluid domain. These values are nearly equal and opposite, indicating that steady-state conditions are nearly reached.

Because this is an axisymmetric analysis, the dimensions of the heat flux are in W/radian, therefore multiplying the heat flux into boundary condition 1 by 2π gives a heat flux of 20.07 W into the fluid domain.

Since the total internal heat generation in the filament is 60 W, the remainder of the heat is seen to be transmitted by radiation directly from the filament. We can check this using the formula for heat transfer by radiation:

$$q^R = \sigma \varepsilon A (\theta^4 - \theta_{env}^4)$$

where $\sigma = 5.66 \times 10^{-14}$ is the Stefan-Boltzmann constant, $\varepsilon = 0.05$ is the emissivity, $A = 2\pi R(2\pi r)$ is the filament area where R and r are defined above, and in which now we are assuming 2π radians of the filament, θ is the filament temperature and θ_{env} is the environmental temperature. Substituting numerical data (with the filament temperature of 2897 K) gives the heat transfer by radiation equal to 39.34 W. Summing the heat flux by radiation and the heat flux into the fluid domain, we obtain 59.41 W, which closely matches the specified wattage (60 W) of the lightbulb.

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