Fluid Damper

Introduction

Fluid dampers are used in military devices for shock isolation and in civil structures for suppressing earthquake-induced shaking and wind-induced vibrations, among many other applications. Fluid dampers work by dissipating the mechanical energy into heat (Ref. 1). This example shows the phenomenon of viscous heating and consequent temperature increase in a fluid damper.Viscous heating is also important in microflow devices, where a small cross-sectional area and large length of the device can generate significant heating and affect the fluid flow consequently (Ref. 2).

Model Definition

The structural elements of a fluid damper are relatively few. Figure 1 depicts a schematic of the fluid damper modeled herein with its main components: damper cylinder housing, piston rod, piston head, and viscous fluid in the chamber. There is a small annular space between the piston head and the inside wall of the cylinder housing. This acts as an effective channel for the fluid. As the piston head moves back and forth inside the damper cylinder, fluid is forced to pass through the annular channel with large shear rate, which leads to significant heat generation. The heat is transferred in both the axial and radial directions. In the radial direction, the heat is conducted through the cylinder house wall and convected to the air outside the damper, which is modeled using the Newton's convective cooling law.

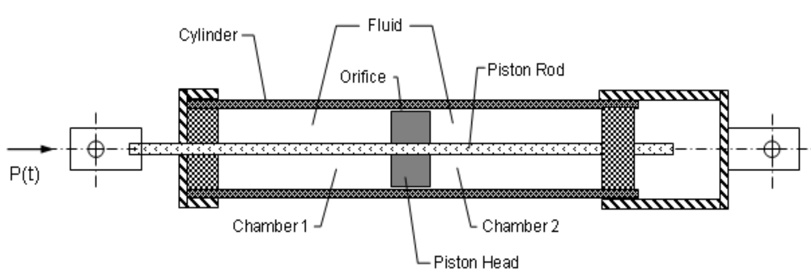

Figure 1: A sketch of a typical fluid damper with its major components

You make use of the axially symmetric nature of the fluid damper and model it in a 2D-axisymmetric geometry as shown in Figure 2. The geometric dimensions and other parameters of the damper are taken according to Ref. 1 to represent the smaller, 15 kip damper experimentally studied therein. Thus, the piston head has a diameter of 8.37 cm, the piston rod diameter is 2.83 cm, and the gap thickness is about 1/100 of the piston head diameter. The damper has the maximum stroke U_0 of 0.1524 m. The damper solid parts are made of steel, and the damping fluid is silicone oil.

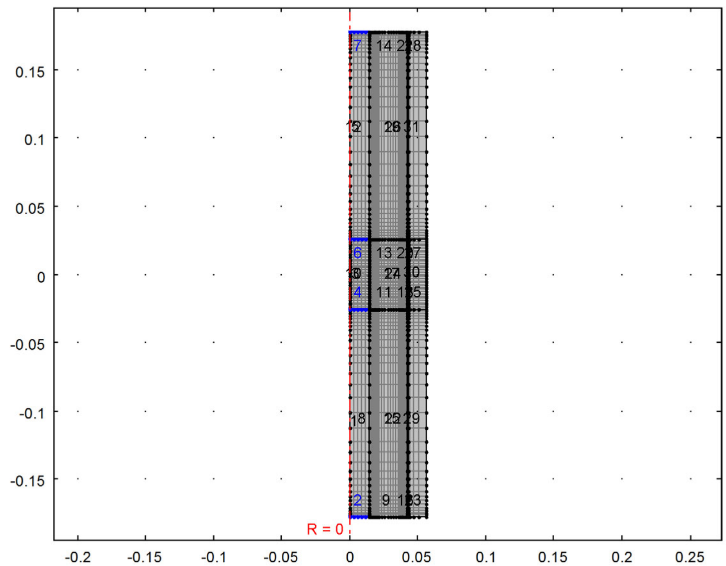

Figure 2: Geometry and mesh. The subdomains (from left to right) represent: piston rod, piston head and damping fluid space, the damper outer wall.

FLUID FLOW

The fluid flow in the fluid damper is described by the weakly compressible Navier-Stokes equations, solving for the velocity field $\mathbf{u} = (u, w)$ and the pressure p:

$$\rho\frac{\partial \mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \nabla\cdot\left(\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \left(\frac{2}{3}\eta - \kappa_{\mathrm{dv}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

The density is assumed independent of the temperature, while the temperature dependence of the fluid viscosity is taken into account as:

$$\mu = \mu_0 - \alpha(T - T_0) \tag{1}$$

The reference material properties of silicone oil are used.

No-slip wall boundary conditions are applied for both ends of the damper cylinder and on the inner wall of the damper cylinder house. Moving/sliding wall with the given velocity is applied on the boundaries of the piston head and on the piston rod.

CONJUGATE HEAT TRANSFER

The conjugate heat transfer is solved both in the fluid domain and the damper cylinder house wall: heat transfer by convection and conduction in the fluid domain, heat transfer by conduction only in the solid domain, and the temperature field is continuous between the fluid and solid domains. In the fluid domain, the viscous heating is activated and pressure work can be included when the slight compressibility of the damper fluid needs to be considered:

$$\rho C\mathrm{p}\frac{\partial T}{\partial t} + \nabla\cdot(-k\nabla T) + \rho C\mathrm{p}\mathbf{u}\cdot\nabla T = Q - \frac{T}{\rho}\frac{\partial\rho}{\partial T}\bigg|_p\frac{\partial p}{\partial t} + \eta\left[\nabla\mathbf{u} + (\nabla\mathbf{u})^T - \frac{2}{3}(\nabla\cdot\mathbf{u})\mathbf{I}\right]:\nabla\mathbf{u}$$

where the second term and the last term on the right-hand side represent the heat source from pressure work and viscous dissipation, respectively. Hence, the problem is a fully coupled fluid-thermal interaction problem.

In the solid domain of the cylinder house wall, this equation reduces to conductive heat transfer equation without any heating source.

The heat flux boundary condition based on the Newton's cooling law is applied on the outside boundaries of the cylinder house wall. The temperature field is continuous between the fluid and solid domains. The ends of the damper connected to the structures outside are kept at constant temperature.

The piston head movement is provided as harmonic oscillations with given amplitude and frequency, $z = a_0\sin(2\pi f)$. The motion is modeled using the arbitrary Lagrangian-Eulerian (ALE) deformed mesh. The ALE method handles the dynamics of the deforming geometry and the moving boundaries with a moving grid. The Navier-Stokes equations for fluid flow and heat equations for temperature variation are formulated in these moving coordinates.

Results and Discussion

The modeled loading has the amplitude of 0.127 m, and the excitation frequency is 0.4 Hz. This represents the long-stroke loading experiment performed in Ref. 1. The loading time period is 40 s.

Note that the simulation results for the temperature are presented in degrees Fahrenheit for the sake of easier comparison with the experimental measurements.

Figure 3 gives the temperature field in the damper at the end the loading. It also shows a typical streamline configuration for the flow induced in the damping fluid.

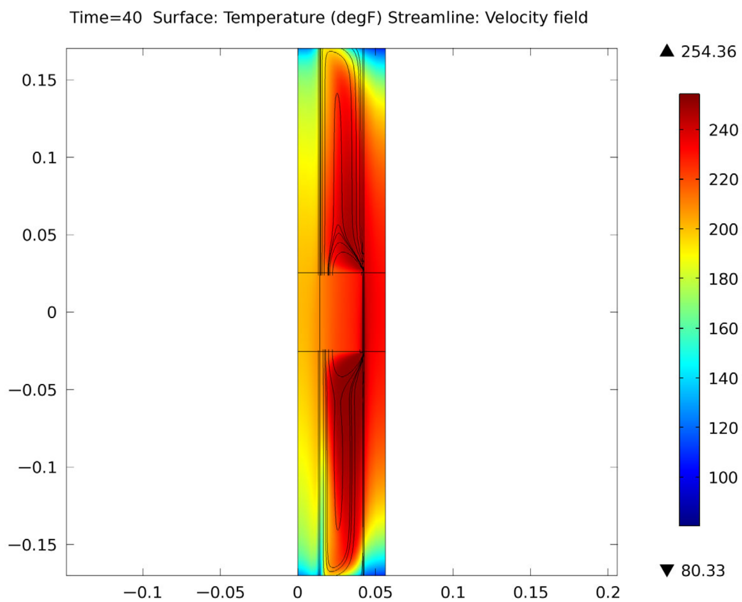

Figure 3: Temperature field in the damper at the end of simulation.

Figure 4 shows the temperature of the inner wall of the damper at the end-of-stroke position $z = U_0$. This corresponds to the internal probe position under experiments performed in Ref. 1. The simulation results show very good agreement with the experimental measurements (see Fig. 9 in Ref. 1).

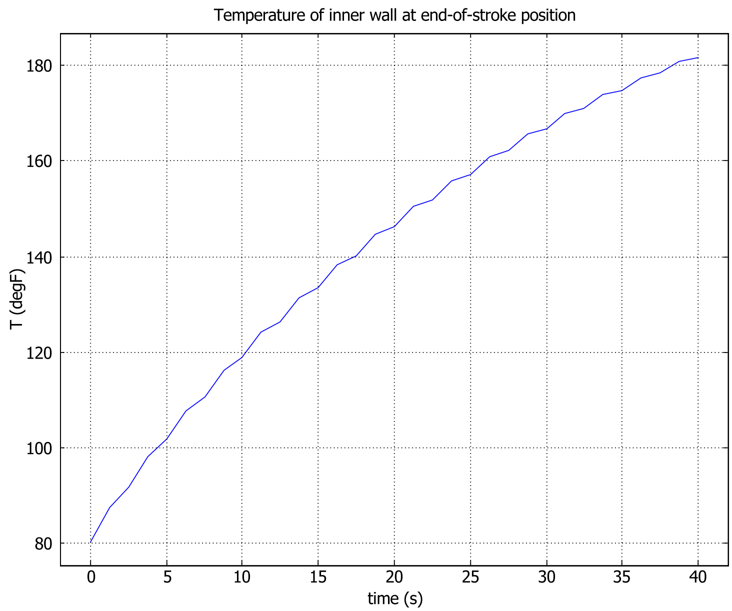

Figure 4: Temperature at the probe position.

Figure 5 shows the temperature variation along the inner wall of the damper after 10 s and 40 s of loading. It clearly shows that the temperature at the probe position does not represent the maximum temperature within the damper. This supports the conclusion drawn in Ref. 1, where the choice of the probe positioning was limited by the construction of the outer shell of the damper. Figure 5 also shows that the temperature near the center of the damper increases by more than 100 degrees already after few loading cycles.

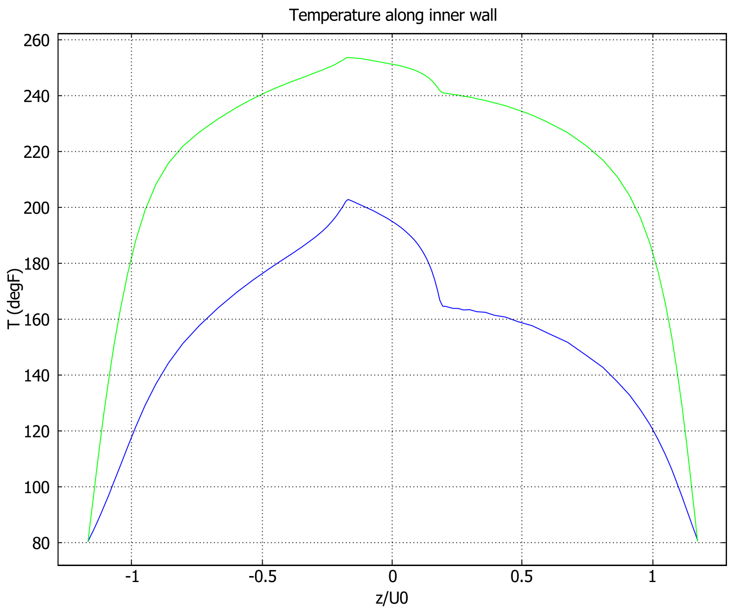

Figure 5: Temperature of the damper inner wall. The probe position corresponds to $z/U_0 = 1$.

Notes About the COMSOL Implementation

You decompose the computational domain into several parts and mesh the domains with mapped meshes to resolve the very thin annular space. For the moving mesh you prescribe the displacement of the mesh in each domain so that their alignment remains unchanged with a zero displacement at the top and the bottom of the damper cylinder housing connecting to the high-performance seal, and the displacement equal to that of the piston head is used for the domain lined up with the piston head. This is achieved by specifying the mesh displacement field as a linear function of the deformed mesh frame coordinate and the reference (material) frame coordinate.

The steel material needed for the damper solid parts is available in the built-in material library. You create a user-defined material for the silicone oil. Such damping fluids are typically characterized by the density, kinematic viscosity at the temperature 25° C, and so-called *viscosity temperature coefficient*, VTC = 1–(viscosity at 98.9° C)/(viscosity at 37.8° C). Using this parameters, you create the linear correlation for the dynamic viscosity given by Equation 1.

References

1. C. J. Black and N. Makris, "Viscous Heating of Fluid Dampers under Small and large Amplitude Motions: Experimental Studies and Parametric Modeling," *J. Eng. Mech.*, vol. 133, pp. 566—577, 2007.

2. G. L. Morini, "Viscous Heating in Liquid Flows in micro-channels," *Int. J. Heat Mass Transfer*, vol. 48, pp. 3637—3647, 2005.

Model Library path: `CFD_Module/Verification_Models/fluid_damper`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow (nitf)**.

5 Click **Add Selected**.

6 In the **Add Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

7 Click **Add Selected**.

8 Click **Next**.

9 In the **Studies** tree, select **Preset Studies for Selected Physics>Time Dependent**.

10 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 Go to the **Settings** window for Parameters.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `fluid_damper_parameters.txt`.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, right-click **Model 1>Definitions** and choose **Variables**.

2 Go to the **Settings** window for Variables.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `fluid_damper_variables.txt`.

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, right-click **Model 1>Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type `Dr/2`.

4 In the **Height** edit field, type `2*Ld`.

5 Locate the **Position** section. In the **z** edit field, type `-Ld`.

Rectangle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type `Dp/2`.

4 In the **Height** edit field, type `2*Ld`.

5 Locate the **Position** section. In the **z** edit field, type `-Ld`.

Rectangle 3

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type `Dd/2-Hw`.

4 In the **Height** edit field, type `2*Ld`.

5 Locate the **Position** section. In the **z** edit field, type `-Ld`.

Rectangle 4

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type `Dd/2`.

4 In the **Height** edit field, type `2*Ld`.

5 Locate the **Position** section. In the **z** edit field, type `-Ld`.

Rectangle 5

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 Go to the **Settings** window for Rectangle.

3 Locate the **Size** section. In the **Width** edit field, type `Dd/2`.

4 In the **Height** edit field, type `2*Lp`.

5 Locate the **Position** section. In the **z** edit field, type `-Lp`.

Union 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Union**.

2 Select the objects **r1**, **r2**, **r3**, **r4**, and **r5** only.

3 In the **Model Builder** window, right-click **Geometry 1** and choose **Build All**.

4 Click the **Zoom Extents** button on the Graphics toolbar.

The model geometry is now complete. It should look similar to that shown in Figure 2.

CONJUGATE HEAT TRANSFER

From the **Model Builder** window's **View Menu**, choose **Show More Options**.

Fluid 1

1 In the **Model Builder** window, right-click **Model 1>Conjugate Heat Transfer** and choose **Fluid**.

2 Select Domains 4 and 6–9 only.

3 Go to the **Settings** window for Fluid.

4 Locate the **Model Inputs** section. From the p list, select **User defined**. In the associated edit field, type `p0`.

5 Locate the **Thermodynamics** section. From the χ list, select **User defined**. In the **Model Builder** window, right-click **Fluid 1** and choose **Viscous Heating**.

Fluid 1

In the **Model Builder** window, right-click **Fluid 1** and choose **Pressure Work**.

MATERIALS

1 In the **Model Builder** window, right-click **Model 1>Materials** and choose **Open Material Browser**.

2 Go to the **Material Browser** window.

3 Locate the **Materials** section. In the **Materials** tree, select **Built-In>Steel AISI 4340**.

4 Right-click and choose **Add Material to Model** from the menu.

In the following steps, you create a new material for the damping fluid, Silicone oil.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Select Domains 4 and 6–9 only.

3 In the **Model Builder** window, right-click **Material 2** and choose **Rename**.

4 Go to the **Rename Material** dialog box and type `Silicone oil` in the **New name** edit field.

5 Click **OK**.

Silicone oil

1 In the **Model Builder** window, click **Silicone oil>Basic**.

2 Go to the **Settings** window for Property Group.

3 Locate the **Output Properties and Model Inputs** section. In the **Quantities** tree, select **Model Inputs>Temperature**.

4 Click **Add**.

5 Locate the **Local Parameters** section. In the **Local parameters** table, enter the following settings:

PARAMETER	EXPRESSION
nu_25C	0.0125[m^2/s]
VTC	0.6[1]

6 In the **Model Builder** window, click **Silicone oil**.

7 Go to the **Settings** window for Material.

8 Locate the **Material Contents** section. In the **Material Contents** table, enter the following settings:

PROPERTY	NAME	VALUE
Density	rho	950
Dynamic viscosity	mu	nu_25C*rho*(1-VTC*(T-311[K])/(61[K]))/(1+VTC*0.2107)

PROPERTY	NAME	VALUE
Thermal conductivity	k	`22.5`
Heat capacity at constant pressure	Cp	`2e3`

MOVING MESH

1 In the **Model Builder** window, click **Model 1>Moving Mesh**.
2 Go to the **Settings** window for Moving Mesh.
3 Click to expand the **Frame Settings** section.
4 From the **Geometry shape order** list, select **1**.

Prescribed Deformation 1

Right-click **Model 1>Moving Mesh** and choose **Prescribed Deformation**.

Prescribed Deformation 2

1 In the **Model Builder** window, right-click **Moving Mesh** and choose **Prescribed Deformation**.
2 Right-click **Moving Mesh** and choose **Prescribed Deformation**.

Prescribed Deformation 1

1 In the **Model Builder** window, click **Prescribed Deformation 1**.
2 Select Domains 2, 5, 8, and 11 only.
3 Go to the **Settings** window for Prescribed Deformation.
4 Locate the **Prescribed Mesh Displacement** section. In the *dz* edit-field array, type `zp` on the 2nd row.

Prescribed Deformation 2

1 In the **Model Builder** window, click **Prescribed Deformation 2**.
2 Select Domains 1, 4, 7, and 10 only.
3 Go to the **Settings** window for Prescribed Deformation.
4 Locate the **Prescribed Mesh Displacement** section. In the *dz* edit-field array, type `zlin1` on the 2nd row.

Prescribed Deformation 3

1 In the **Model Builder** window, click **Prescribed Deformation 3**.
2 Select Domains 3, 6, 9, and 12 only.
3 Go to the **Settings** window for Prescribed Deformation.

4 Locate the **Prescribed Mesh Displacement** section. In the *dz* edit-field array, type `zlin2` on the 2nd row.

CONJUGATE HEAT TRANSFER

Wall 2

1 In the **Model Builder** window, right-click **Model 1>Conjugate Heat Transfer** and choose the boundary condition **Laminar Flow>Wall**.

2 Select Boundaries 11 and 13 only.

3 Go to the **Settings** window for Wall.

4 Locate the **Boundary Condition** section. From the **Boundary condition** list, select **Moving wall**.

5 Specify the $\mathbf{u}_w$ vector as

0	r
d(zp,t)	z

Wall 3

1 In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the boundary condition **Laminar Flow>Wall**.

2 Select Boundaries 8, 12, and 17 only.

3 Go to the **Settings** window for Wall.

4 Locate the **Boundary Condition** section. From the **Boundary condition** list, select **Sliding wall**.

5 In the U_w edit field, type `d(zp,t)`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 Go to the **Settings** window for Initial Values.

3 Locate the **Initial Values** section. In the **p** edit field, type `p0`.

4 In the **T** edit field, type `T0`.

Temperature 1

1 In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the boundary condition **Heat Transfer>Temperature**.

2 Select Boundaries 2, 7, 9, 14, 16, 21, 23, and 28 only.

3 Go to the **Settings** window for Temperature.

4 Locate the **Temperature** section. In the T_0 edit field, type `T0`.

Heat Flux 1

1 In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the boundary condition **Heat Transfer>Heat Flux**.
2 Select Boundaries 29–31 only.
3 Go to the **Settings** window for Heat Flux.
4 Locate the **Heat Flux** section. Click the **Inward heat flux** button.
5 In the T_{ext} edit field, type `T0`.
6 In the h edit field, type `hwall`.

Because the damper is a closed container, you need to pin-point the pressure level within. To achieve that, use the point constraint as follows.

Pressure Point Constraint 1

1 In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose **Points>Pressure Point Constraint**.
2 Select Vertex 12 only.
3 Go to the **Settings** window for Pressure Point Constraint.
4 Locate the **Pressure Constraint** section. In the p_0 edit field, type `p0`.

MESH 1

Mapped 1

In the **Model Builder** window, right-click **Model 1>Mesh 1** and choose **Mapped**.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 23, 25, 27, and 28 only.
3 Go to the **Settings** window for Distribution.
4 Locate the **Distribution** section. From the **Distribution properties** list, select **Predefined distribution type**.
5 In the **Number of elements** edit field, type `4`.
6 In the **Element ratio** edit field, type `4`.
7 From the **Distribution method** list, select **Geometric sequence**.
8 Select the **Reverse direction** check box.

Mapped 1

1 In the **Model Builder** window, click **Mapped 1**.
2 Select Domains 1–12 only.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 1, 5, 8, 12, 15, 19, 22, 26, 29, and 31 only.
3 Go to the **Settings** window for Distribution.
4 Locate the **Distribution** section. From the **Distribution properties** list, select **Predefined distribution type**.
5 In the **Number of elements** edit field, type `32`.
6 In the **Element ratio** edit field, type `8`.
7 From the **Distribution method** list, select **Geometric sequence**.
8 Select the **Symmetric distribution** check box.

Distribution 3

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 9, 11, 13, and 14 only.
3 Go to the **Settings** window for Distribution.
4 Locate the **Distribution** section. From the **Distribution properties** list, select **Predefined distribution type**.
5 In the **Number of elements** edit field, type `30`.
6 In the **Element ratio** edit field, type `10`.
7 From the **Distribution method** list, select **Geometric sequence**.
8 Select the **Symmetric distribution** check box.

Distribution 4

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 16, 18, 20, and 21 only.
3 Go to the **Settings** window for Distribution.
4 Locate the **Distribution** section. In the **Number of elements** edit field, type `8`.

Distribution 5

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 3, 10, 17, 24, and 30 only.
3 Go to the **Settings** window for Distribution.

4 Locate the **Distribution** section. In the **Number of elements** edit field, type `32`.

Distribution 6

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 2, 4, 6, and 7 only.
3 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

The mesh is now complete. It should look similar to that shown in Figure 2.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.
2 Go to the **Settings** window for Time Dependent.
3 Locate the **Study Settings** section. In the **Times** edit field, type `range(0,tstep,tmax)`.
4 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.
5 Expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Time-Dependent Solver 1**.
2 Go to the **Settings** window for Time-Dependent Solver.
3 Click to expand the **Time Stepping** section.

   You can gain extra accuracy in time if you restrict the BDF solver to be of second order. To compensate for the loss in robustness by doing so, a maximum time step is also specified.
4 From the **Minimum BDF order** list, select **2**.
5 Select the **Initial step** check box.
6 In the associated edit field, type `1e-5`.
7 Select the **Maximum step** check box.
8 In the associated edit field, type `tstep/20`.
9 Click to expand the **Results While Solving** section.
10 Select the **Plot** check box.

Before computing the solution, create several plots. One of them will be displayed and updated while solving.

RESULTS

Data Sets

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Solution 1**.
2 Go to the **Settings** window for Solution.
3 Locate the **Solution** section. From the **Frame** list, select **Spatial (r, phi, z)**.
4 In the **Model Builder** window, right-click **Data Sets** and choose **Cut Point 2D**.
5 Go to the **Settings** window for Cut Point 2D.
6 Locate the **Point Data** section. In the **r** edit field, type `Dd/2-Hw`.
7 In the **z** edit field, type `U0`.

2D Plot Group 1

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.
2 Right-click **Results>2D Plot Group 1** and choose **Surface**.
3 Go to the **Settings** window for Surface.
4 Locate the **Expression** section. From the **Unit** list, select **degF**.
5 In the **Model Builder** window, right-click **2D Plot Group 1** and choose **Streamline**.
6 Go to the **Settings** window for Streamline.
7 In the upper-right corner of the **Expression** section, click **Replace Expression**.
8 From the menu, choose **Conjugate Heat Transfer (Heat Transfer)>Velocity field (nitf.ur, nitf.uz)**.

1D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2 Right-click **Results>1D Plot Group 2** and choose **Line Graph**.
3 Select Boundaries 22, 24, and 26 only.
4 Go to the **Settings** window for Line Graph.
5 Locate the **Y-Axis Data** section. From the **Unit** list, select **degF**.
6 Locate the **X-Axis Data** section. From the **Parameter** list, select **Expression**.
7 In the **Expression** edit field, type `z/U0`.

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.
2 Right-click **Results>1D Plot Group 3** and choose **Point Graph**.
3 Go to the **Settings** window for Point Graph.
4 Locate the **Data** section. From the **Data set** list, select **Cut Point 2D 1**.
5 Locate the **Expression** section. From the **Unit** list, select **degF**.
6 In the **Model Builder** window, click **1D Plot Group 3**.
7 Go to the **Settings** window for 1D Plot Group.
8 Click to expand the **Axis** section.
9 Locate the **Plot Settings** section. Select the **Title** check box.
10 In the associated edit field, type `Temperature of inner wall at end-of-stroke position`.
11 Select the **x-axis label** check box.
12 In the associated edit field, type `time (s)`.
13 Select the **y-axis label** check box.
14 In the associated edit field, type `T (degF)`.

1D Plot Group 2

1 In the **Model Builder** window, click **Results>1D Plot Group 2**.
2 Go to the **Settings** window for 1D Plot Group.
3 Locate the **Plot Settings** section. Select the **Title** check box.
4 In the associated edit field, type `Temperature along inner wall`.
5 Select the **x-axis label** check box.
6 In the associated edit field, type `z/U0`.
7 Select the **y-axis label** check box.
8 In the associated edit field, type `T (degF)`.

STUDY 1

Now return to the Study branch to set up the visualization and compute the solution.

Solver 1

1 In the **Model Builder** window, click **Study 1>Solver Configurations>>Solver 1>Time-Dependent Solver 1**.
2 Go to the **Settings** window for Time-Dependent Solver.

3 Locate the **Results While Solving** section. From the **Plot group** list, select **1D Plot Group 3**.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

During the solution time, a plot of the temperature at the probe position will be displayed and updated following the solver time steps.

2D Plot Group 1

When the solution is finished, click the **Zoom Extents** button on the Graphics toolbar. The plot in the **Graphics** window shows the temperature field and the flow streamlines within the damper, which should appear similar to that shown in Figure 3.

1D Plot Group 2

1 In the **Model Builder** window, click **Results>1D Plot Group 2**.

2 Go to the **Settings** window for 1D Plot Group.

3 Locate the **Data** section. From the **Time selection** list, select **From list**.

4 In the **Times** list, select **10** and **40**.

5 Click the **Plot** button.

This shows the temperature distribution along the damper inner wall at times 10 s and 40 s, it should look similar to that shown in Figure 5.

1D Plot Group 3

In the **Model Builder** window, right-click **Results>1D Plot Group 3** and choose **Plot**.

This shows the temperature variation at the probe position after the complete loading time period, it should look similar to that shown in Figure 4.