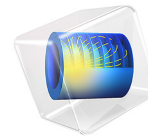
Created in COMSOL Multiphysics 5.4



Viscous Heating in a 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.

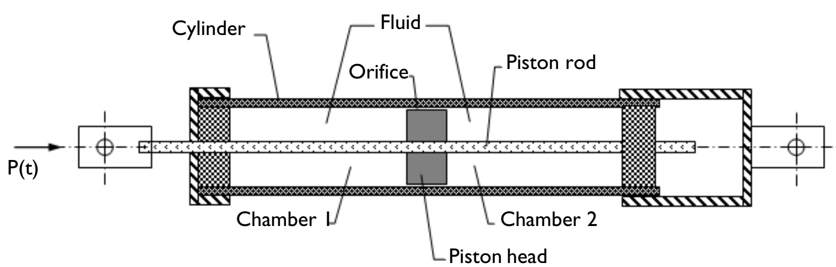


Figure 1: A sketch of a typical fluid damper with its major components.

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.

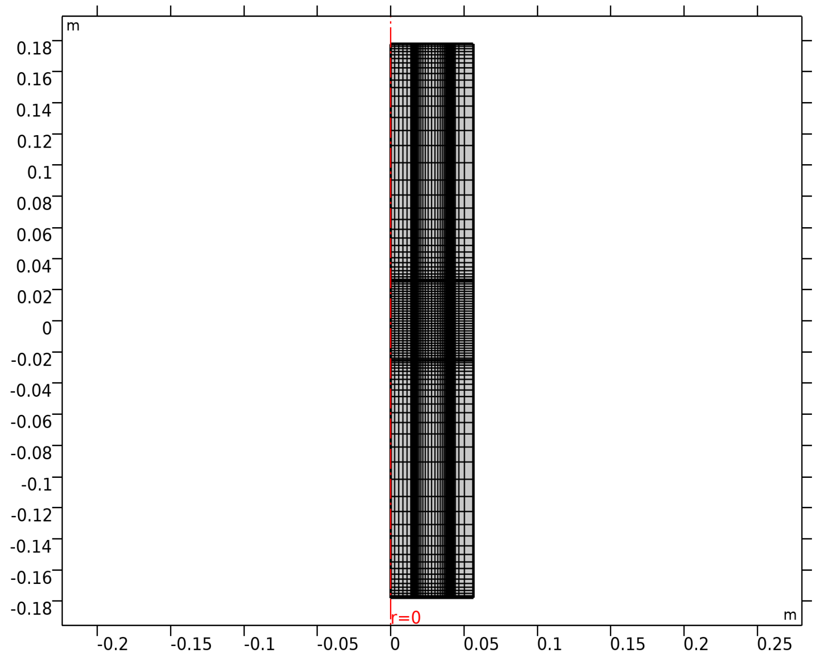


Figure 2: Geometry and mesh. The domains (from left to right) represent: piston rod, piston head and damping fluid space, and the damper outer wall.

FLUID FLOW

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

$$\rho\frac{\partial\mathbf{u}}{\partial t} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}})]$$

$$\rho\nabla\cdot\mathbf{u} = 0$$

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. A 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 dissipation is activated:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + \mu[\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}]:\nabla \mathbf{u}$$

where the second term on the right-hand side represent the heat source from viscous dissipation. 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, $z = a_0 \sin(2\pi f t)$, the piston head starting the simulation in its middle position. 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.

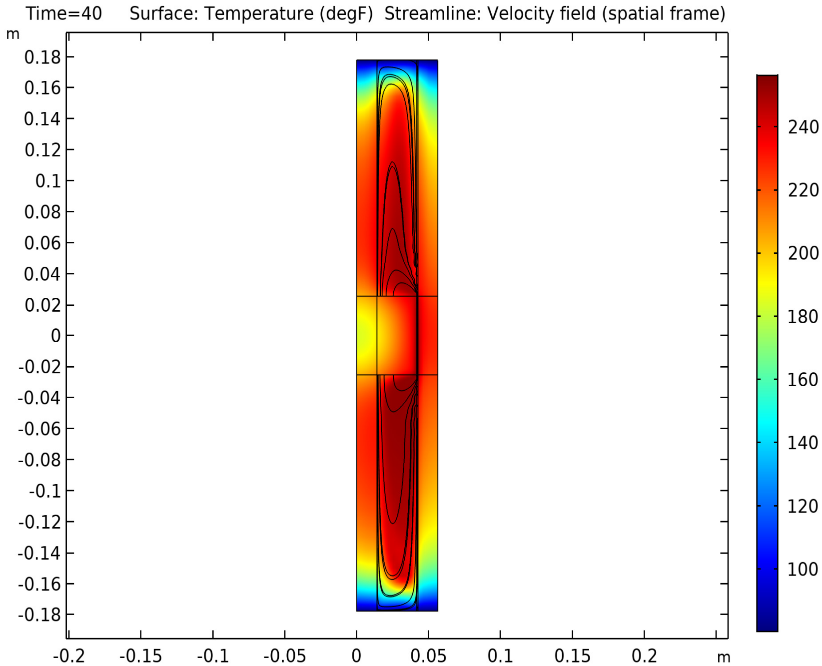


Figure 3: Temperature field in the damper at the end of simulation.

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).

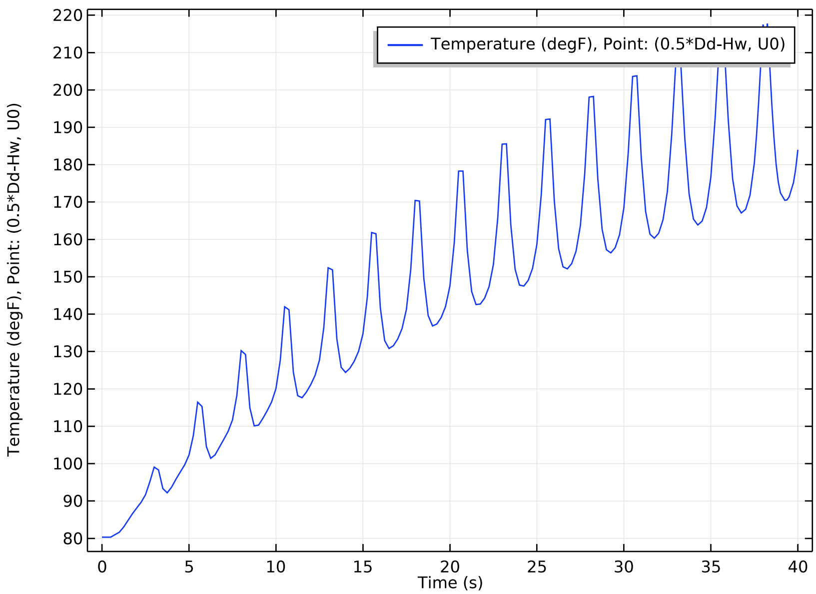


Figure 4: Temperature at the probe position.

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 about 100 degrees already after few loading cycles.

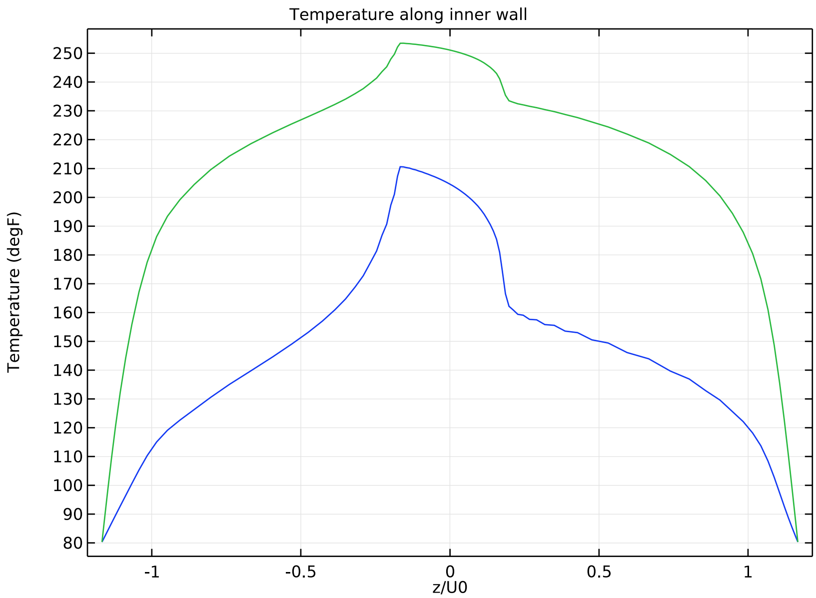


Figure 5: Temperature of the damper inner wall. The probe position corresponds to $z/U_0 = 1$.

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 25º C, and so-called *viscosity temperature coefficient*, VTC = 1–(viscosity at 98.9º C)/(viscosity at 37.8º 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.

Application Library path: `Heat_Transfer_Module/Buildings_and_Constructions/fluid_damper`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model’s Application Libraries folder and double-click the file `fluid_damper_parameters.txt`.

5 In the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

Analytic 1 (an1)

1 In the **Model Builder** window, under **Global Definitions** click **Analytic 1 (an1)**.

2 In the **Settings** window for **Analytic**, locate the **Definition** section.

3 In the **Expression** text field, type `a0*sin(2*pi*f*t)`.

4 In the **Arguments** text field, type `t`.

5 Locate the **Units** section. In the **Arguments** text field, type `s`.

6 In the **Function** text field, type `m`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
t	0	40

8 Click **Plot**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

10 In the **Function name** text field, type `zp`.

11 Right-click **Global Definitions>Analytic 1 (an1)** and choose **Rename**.

12 In the **Rename Analytic** dialog box, type `Piston displacement` in the **New label** text field.

13 Click **OK**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

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

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dr/2`.

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

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

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dp/2`.

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

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

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dd/2-Hw`.

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

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

Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dd/2`.

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

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

Rectangle 5 (r5)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Dd/2`.

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

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

6 In the **Geometry** toolbar, click **Build All**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

The model geometry is now complete.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domains 4 and 6–9 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

In the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer in Solids and Fluids (ht)**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.
2 Select Domains 4 and 6–9 only.
3 In the **Settings** window for **Fluid**, locate the **Thermodynamics, Fluid** section.
4 From the γ list, choose **User defined**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-In>Steel AISI 4340**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

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

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Silicone Oil` in the **Label** text field.
3 Select Domains 4 and 6–9 only.

Silicone Oil (mat2)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Silicone Oil (mat2)** node, then click **Basic (def)**.
2 In the **Settings** window for **Property Group**, locate the **Model Inputs** section.
3 Click **Select Quantity**.
4 In the **Physical Quantity** dialog box, type `temperature` in the text field.
5 Click **Filter**.
6 In the tree, select **General>Temperature (K)**.
7 Click **OK**.

8 In the **Settings** window for **Property Group**, locate the **Local Properties** section.

9 In the **Local properties** table, enter the following settings:

Name	Expression	Unit	Description
nu_25C	0.0125[m^2/s]	m^2/s	
VTC	0.6[1]		

10 In the **Model Builder** window, click **Silicone Oil (mat2)**.

11 In the **Settings** window for **Material**, locate the **Material Contents** section.

12 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	22.5	W/(m·K)	Basic
Density	rho	950	kg/m^3	Basic
Heat capacity at constant pressure	Cp	2e3	J/(kg·K)	Basic
Dynamic viscosity	mu	nu_25C*rho*(1-VTC*(T-311[K])/(61[K]))/(1+VTC*0.2107)	Pa·s	Basic

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Solids and Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, type `T0` in the T text field.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

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

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T0`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundaries 29–31 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 Click the **Convective heat flux** button.

5 In the T_{ext} text field, type `T0`.

6 In the h text field, type `hwall`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

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 **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 12 only.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 22, 24, and 26 only.

3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 From the **Translational velocity** list, choose **Manual**.

MULTIPHYSICS

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Nonisothermal Flow 1 (nitf1)**.

2 In the **Settings** window for **Nonisothermal Flow**, locate the **Flow Heating** section.

3 Select the **Include viscous dissipation** check box.

4 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

DEFINITIONS

Prescribed Deformation 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Moving Mesh** click **Prescribed Deformation 1**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Deformation** section.

3 Specify the *dx* vector as

0	R
zp(t)	Z

4 Select Domains 2, 5, 8, and 11 only.

Prescribed Deformation 2

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Deformation** section.

3 Specify the *dx* vector as

0	R
zlin1	Z

4 Select Domains 1, 4, 7, and 10 only.

Prescribed Deformation 3

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Deformation** section.

3 Specify the *dx* vector as

0	R
zlin2	Z

4 Select Domains 3, 6, 9, and 12 only.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundaries 23, 25, 27, and 28 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 From the **Distribution type** list, choose **Predefined**.

6 In the **Number of elements** text field, type 4.

7 In the **Element ratio** text field, type 4.

8 From the **Growth formula** list, choose **Geometric sequence**.

9 Select the **Reverse direction** check box.

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 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 32.

6 In the **Element ratio** text field, type 8.

7 From the **Growth formula** list, choose **Geometric sequence**.

8 Select the **Symmetric distribution** check box.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 9, 11, 13, and 14 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 30.

6 In the **Element ratio** text field, type 10.

7 From the **Growth formula** list, choose **Geometric sequence**.

8 Select the **Symmetric distribution** check box.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 16, 18, 20, and 21 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 8.

Distribution 5

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 3, 10, 17, 24, and 30 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 32.

Distribution 6

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 2, 4, 6, and 7 only.

3 In the **Model Builder** window, click **Mesh 1**.

4 In the **Settings** window for **Mesh**, click **Build All**.

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

5 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

DEFINITIONS

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

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Domain Point Probe 1**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **r** to `Dd/2-Hw`.

4 In row **Coordinates**, set **z** to `U0`.

5 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

6 In the **Settings** window for **Point Probe Expression**, type `temppr` in the **Variable name** text field.

7 Locate the **Expression** section. From the **Table and plot unit** list, choose **degF**.

STUDY 1

Step 1: Time Dependent

The simulation starts when the piston is in the lowest position consistent with the steady flow initial conditions. In addition, a finer sampling is defined on the last cycle to obtain a better plot of the velocity.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,tstep,(ncycle-1)/f) range((ncycle-1)/f,tstep/2,tmax)`.

4 Click to expand the **Results While Solving** section. From the **Update at** list, choose **Times stored in output**.

5 In the **Home** toolbar, click **Compute**.

RESULTS

Temperature, 3D (ht)

Change the unit of the temperature results to degrees Fahrenheit for the sake of easier comparison with the experimental measurements.

Surface 1

1 In the **Model Builder** window, expand the **Temperature, 3D (ht)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **degF**.

4 In the **Temperature, 3D (ht)** toolbar, click **Plot**.

Contour

1 In the **Model Builder** window, expand the **Results>Isothermal Contours (ht)** node, then click **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 From the **Unit** list, choose **degF**.

4 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

Velocity (spf)

To produce a plot of the temperature field and the flow streamlines within the damper, modify the default plot. The plot should appear similar to that shown in Figure 3.

1 In the **Model Builder** window, under **Results** right-click **Velocity (spf)** and choose **Rename**.

2 In the **Rename 2D Plot Group** dialog box, type `Temperature Surface and Velocity Streamlines` in the **New label** text field.

3 Click **OK**.

Surface

1 In the **Model Builder** window, expand the **Results> Temperature Surface and Velocity Streamlines** node.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `T`.

4 From the **Unit** list, choose **degF**.

Temperature Surface and Velocity Streamlines

In the **Model Builder** window, expand the **Surface** node.

Streamline 1

1 Right-click **Results>Temperature Surface and Velocity Streamlines** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Laminar Flow>Velocity and pressure>u,w - Velocity field (spatial frame)**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Starting point controlled**.

Probe Plot Group 6

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 6**.

2 In the **Settings** window for **1D Plot Group**, type `Inner Wall Temperature at End-of-Stroke Position` in the **Label** text field.

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

1D Plot Group 7

Now set up the plot that shows the temperature distribution along the damper inner wall at times 8 s and 38 s, it should look similar to that shown in Figure 5.

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature along Inner Wall` in the **Label** text field.

Line Graph 1

1 Right-click **Temperature along Inner Wall** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 From the **Unit** list, choose **degF**.

4 Select Boundaries 22, 24, and 26 only.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z/U0`.

Temperature along Inner Wall

1 In the **Model Builder** window, under **Results** click **Temperature along Inner Wall**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Temperature along inner wall`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `z/U0`.

7 Select the **y-axis label** check box.

8 Locate the **Data** section. From the **Time selection** list, choose **From list**.

9 In the **Times (s)** list, choose **10** and **40**.

10 In the **Temperature along Inner Wall** toolbar, click **Plot**.

Derived Values

Finally, plot the average velocity within the damper over the last cycle.

Surface Average 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

2 Select Domains 4 and 6–9 only.

3 In the **Settings** window for **Surface Average**, locate the **Data** section.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, choose **38.75**, **38.875**, **39**, **39.125**, **39.25**, **39.375**, **39.5**, **39.625**, **39.75**, **39.875**, and **40**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`spf.U`	`m/s`	`Velocity magnitude`

7 Click **Evaluate**.

1D Plot Group 8

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Average velocity over the last cycle` in the **Label** text field.

Table Graph 1

1 Right-click **Average velocity over the last cycle** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Table** list, choose **Table 2**.

4 In the **Average velocity over the last cycle** toolbar, click **Plot**.