

# 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 2Daxisymmetric 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 of 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 25°C, and so-called *viscosity temperature coefficient*, VTC =  $1-(viscosity at 98.9^{\circ}C)/(viscosity at 37.8^{\circ}C)$ . Using this parameter, 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: CFD\_Module/Nonisothermal\_Flow/fluid\_damper

# Modeling Instructions

From the File menu, choose New.

#### NEW

In the New window, click 🔗 Model Wizard.

## MODEL WIZARD

- I 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 M Done.

### **GLOBAL DEFINITIONS**

#### Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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.

### Piston Displacement

I In the Home toolbar, click f(X) Functions and choose Global>Analytic.

- 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 table, enter the following settings:

Argument	Unit
t	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	Unit
t	0	40	S

8 Click 💽 Plot.

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

**IO** In the **Function name** text field, type zp.

II In the Label text field, type Piston Displacement.

## DEFINITIONS

Variables I

- I In the Home toolbar, click a= 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 I

#### Rectangle 1 (r1)

- I In the Geometry toolbar, click 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)

- I In the **Geometry** toolbar, click **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)

- I In the **Geometry** toolbar, click **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)

- I In the **Geometry** toolbar, click **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)

- I In the **Geometry** toolbar, click **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)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- **2** Select Domains 4 and 6–9 only.

#### HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

#### Fluid I

- I In the Model Builder window, under Component I (compl)> Heat Transfer in Solids and Fluids (ht) click Fluid I.
- 2 Select Domains 4 and 6–9 only.
- 3 In the Settings window for Fluid, locate the Thermodynamics, Fluid section.
- 4 From the Fluid type list, choose Gas/Liquid.
- **5** From the  $\gamma$  list, choose **User defined**.

#### ADD MATERIAL

- I 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.

Silicone Oil

- I In the Model Builder window, under Component I (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.
- 4 In the Model Builder window, expand the Component I (compl)>Materials> Silicone Oil (mat2) node, then click Basic (def).
- 5 In the Settings window for Basic, locate the Model Inputs section.
- 6 Click + Select Quantity.
- 7 In the Physical Quantity dialog box, type temperature in the text field.
- 8 Click 🔫 Filter.
- 9 In the tree, select General>Temperature (K).

IO Click OK.

II In the Settings window for Basic, locate the Local Properties section.

12 In the Local properties table, enter the following settings:

Name	Expression	Unit	Description
nu_25C	0.0125[m^2/s]	m²/s	
VTC	0.6[1]		

I3 In the Model Builder window, under Component I (compl)>Materials click Silicone Oil (mat2).

14 In the Settings window for Material, locate the Material Contents section.

**I5** In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Ср	2e3	J/(kg·K)	Basic
Density	rho	950	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	22.5	W/(m·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

- In the Model Builder window, under Component I (compl)> Heat Transfer in Solids and Fluids (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the T text field, type T0.

## Temperature I

- I In the Physics toolbar, click Boundaries and choose Temperature.
- 2 Select Boundaries 2, 7, 9, 14, 16, 21, 23, and 28 only. These are the upper and lower boundaries of the cylinder.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the  $T_0$  text field, type T0.

## Heat Flux 1

- I 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 From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type hwall.
- **6** In the  $T_{\text{ext}}$  text field, type T0.

## LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) 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 pinpoint the pressure level within. To achieve that, use the point constraint as follows.

Pressure Point Constraint I

- I In the Physics toolbar, click 💭 Points and choose Pressure Point Constraint.
- **2** Select Point 12 only.

#### Wall 2

- I 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.

## COMPONENT I (COMPI)

Prescribed Deformation 1

- I In the Definitions toolbar, click Moving Mesh and choose Domains> Prescribed Deformation.
- **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

- I In the Definitions toolbar, click Moving Mesh and choose Domains> 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

- I In the Definitions toolbar, click Moving Mesh and choose Domains> Prescribed Deformation.
- **2** In the **Settings** window for **Prescribed Deformation**, locate the **Prescribed Deformation** section.
- **3** Specify the *dx* vector as



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

## MESH I

Mapped I

In the **Mesh** toolbar, click **Mapped**.

Distribution I

- I Right-click Mapped I and choose Distribution.
- **2** Select Boundaries 23, 25, 27, and 28 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 4.
- 6 In the Element ratio text field, type 4.
- 7 From the Growth rate list, choose Exponential.

8 Select the Reverse direction check box.

#### Distribution 2

- I In the Model Builder window, right-click Mapped I 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 rate list, choose Exponential.
- 8 Select the Symmetric distribution check box.

## Distribution 3

- I Right-click Mapped I 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 rate list, choose Exponential.
- 8 Select the Symmetric distribution check box.

#### Distribution 4

- I Right-click Mapped I 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

- I Right-click Mapped I 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

I Right-click Mapped I and choose Distribution.

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

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

## DEFINITIONS

## Domain Point Probe 1

I In the Definitions toolbar, click probes and choose Domain Point Probe.

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

- 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.

Point Probe Expression 1 (ppb1)

- I In the Model Builder window, expand the Domain Point Probe I node, then click Point Probe Expression I (ppb1).
- 2 In the Settings window for Point Probe Expression, type temppr in the Variable name text field.
- 3 Locate the Expression section. From the Table and plot unit list, choose degF.

### STUDY I

#### 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.

- I In the Model Builder window, under Study I click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Output 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

- I In the Model Builder window, expand the Temperature, 3D (ht) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degF.

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. Start with creating additional Cut Plane datasets for plotting the streamlines.

#### Cut Plane 1

- I In the **Results** toolbar, click **Cut Plane**.
- 2 In the Settings window for Cut Plane, locate the Plane Data section.
- 3 From the Plane type list, choose General.
- **4** From the **Plane entry method** list, choose **Point and normal vector**.
- 5 Find the Normal vector subsection. In the x text field, type 1.
- **6** In the **z** text field, type **0**.
- **7** Click **I** Plot to visualize the orientation of the cut plane.

## Cut Plane 2

- I Right-click Cut Plane I and choose Duplicate.
- 2 In the Settings window for Cut Plane, locate the Plane Data section.
- 3 Find the Normal vector subsection. In the y text field, type 1.
- 4 Click 💿 Plot.

## Temperature and Velocity Streamlines

- I In the Model Builder window, under Results click Temperature, 3D (ht).
- 2 In the Settings window for 3D Plot Group, type Temperature and Velocity Streamlines in the Label text field.

#### Streamline Surface 1

- I In the **Temperature and Velocity Streamlines** toolbar, click **More Plots** and choose **Streamline Surface**.
- 2 In the Settings window for Streamline Surface, locate the Data section.

- 3 From the Dataset list, choose Cut Plane I.
- 4 From the Solution parameters list, choose From parent.
- 5 Click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>Laminar Flow>Velocity and pressure>u,v,w Velocity field (spatial frame).
- 6 Locate the Streamline Positioning section. From the Positioning list, choose Uniform density.
- 7 In the Separating distance text field, type 0.025.
- 8 Locate the Coloring and Style section. Find the Point style subsection. From the Color list, choose Black.

Streamline Surface 2

- I Right-click Streamline Surface I and choose Duplicate.
- 2 In the Settings window for Streamline Surface, locate the Data section.
- 3 From the Dataset list, choose Cut Plane 2.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 In the **Temperature and Velocity Streamlines** toolbar, click **O** Plot.

Inner Wall Temperature at End-of-Stroke Position

- I In the Model Builder window, under Results click Probe Plot Group 6.
- 2 In the Settings window for ID Plot Group, type Inner Wall Temperature at End-of-Stroke Position in the Label text field.
- 3 Locate the Legend section. Clear the Show legends check box.

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.

#### Temperature Along Inner Wall

Now set up the plot that shows the temperature distribution along the damper inner wall at times 10 s and 40 s, it should look like that shown in Figure 5.

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Temperature Along Inner Wall in the Label text field.

Line Graph I

- I 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** In the **Expression** text field, type z/U0.
- 4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- **5** In the **Expression** text field, type T.
- 6 From the Unit list, choose degF.
- 7 Select Boundaries 22, 24, and 26 only.
- 8 Click to expand the Legends section. Select the Show legends check box.

#### Temperature Along Inner Wall

- I In the Model Builder window, click Temperature Along Inner Wall.
- 2 In the Settings window for ID 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 Select the y-axis label check box. In the associated text field, type Adimensional height (1).
- 7 Locate the Data section. From the Time selection list, choose From list.
- 8 In the Times (s) list, choose 10 and 40.
- 9 In the Temperature Along Inner Wall toolbar, click 🗿 Plot.

#### Surface Average 1

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

- I In the Model Builder window, expand the Results>Derived Values node.
- 2 Right-click Results>Derived Values and choose Average>Surface Average.
- **3** Select Domains 4 and 6–9 only.
- 4 In the Settings window for Surface Average, locate the Data section.
- 5 From the Time selection list, choose From list.
- 6 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.
- 7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
spf.U	m/s	Velocity magnitude

8 Click **= Evaluate**.

## Average Velocity Over the Last Cycle

- I In the **Results** toolbar, click  $\sim$  **ID** Plot Group.
- 2 In the Settings window for ID Plot Group, type Average Velocity Over the Last Cycle in the Label text field.
- 3 Locate the Title section. From the Title type list, choose Manual.
- 4 In the Title text area, type Temperature along inner wall.

## Table Graph 1

- I 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 **I** Plot.

