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 $p$:

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

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

The density is assumed independent of the temperature, while the temperature dependence of the fluid viscosity is taken into account as:
\[ \mu = \mu_0 - \alpha(T - T_0) \] (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:

\[
\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) + \rho C_p u \cdot \nabla T = Q - \frac{T}{\rho \frac{\partial \rho}{\partial T}} \left| \frac{\partial \rho}{\partial T} + \eta \left[ \nabla u + (\nabla u)^T \right] - 3\eta (\nabla \cdot u) I \right| \nabla u
\]

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 of the loading. It also shows a typical streamline configuration for the flow induced in the damping fluid.

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, $VTC = 1 - (\text{viscosity at } 98.9° \text{ C})/(\text{viscosity at } 37.8° \text{ C})$. Using this parameter, you create the linear correlation for the dynamic viscosity given by Equation 1.
References


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.
5. Click Add Selected.
6. In the Add Physics tree, select Mathematics>Deformed Mesh>Moving Mesh (ale).
7. Click Add Selected.
8. Click Next.
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 \(-L_d\).

**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 \( D_d/2 \).
4 In the **Height** edit field, type \( 2*L_p \).
5 Locate the **Position** section. In the \( z \) edit field, type \(-L_p\).

**Union 1**
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Union**.
2 Select the objects \( r_1 \), \( r_2 \), \( r_3 \), \( r_4 \), and \( r_5 \) 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 \( p_0 \).
5 Locate the **Thermodynamics** section. From the \( \chi \) 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:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>EXPRESSION</th>
</tr>
</thead>
<tbody>
<tr>
<td>nu_25C</td>
<td>0.0125 (m^2/s)</td>
</tr>
<tr>
<td>VTC</td>
<td>0.6 [1]</td>
</tr>
</tbody>
</table>

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:

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>NAME</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>rho</td>
<td>950</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>mu</td>
<td>(nu_25C*\rho*(1-VTC*(T-311[K])/(61[K])/((1+VTC*0.2107)))</td>
</tr>
</tbody>
</table>
Solved with COMSOL Multiphysics 4.1

---

**FLUID DAMPER**

**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 3*

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.

---

<table>
<thead>
<tr>
<th>PROPERTY</th>
<th>NAME</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal conductivity</td>
<td>k</td>
<td>22.5</td>
</tr>
<tr>
<td>Heat capacity at constant pressure</td>
<td>Cp</td>
<td>2e3</td>
</tr>
</tbody>
</table>
4 Locate the **Prescribed Mesh Displacement** section. In the $dz$ edit-field array, type $z_{11,n2}$ 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 $u_w$ vector as

<table>
<thead>
<tr>
<th>0</th>
<th>r</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d(zp,t)$</td>
<td>z</td>
</tr>
</tbody>
</table>

*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 $p_0$.
4 In the $T$ edit field, type $T_0$.

**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 $T_0$.

**Heat Flux**

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 $T_0$.

6 In the $h$ edit field, type $h_{wall}$.

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 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 $p_0$.

**Mesh**

**Mapped**

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

**Distribution**

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 \(\text{range}(0, \text{tstep}, \text{tmax})\).
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 \(\text{tstep}/20\).
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.