Created in COMSOL Multiphysics 5.6

# Lumped Model of a Human Body

## Introduction

Several mass–spring–damper models have been developed to study the response of a human body. In such models, the lumped elements represent the mass of different body parts, and stiffness and damping properties of various tissues.

In this example, a lumped model of a human body having five degrees of freedom is analyzed. The model includes the shoe-ground interaction with the human body. The **Mass**, **Spring**, and **Damper** nodes of the Lumped Mechanical System interface are used to model the body including shoe and ground. First, an eigenfrequency study is performed to find out the natural frequencies of the system and then a frequency response analysis is performed to compute the system response for a specified base excitation. The data for this model is taken from Ref. 1.

## Model Definition

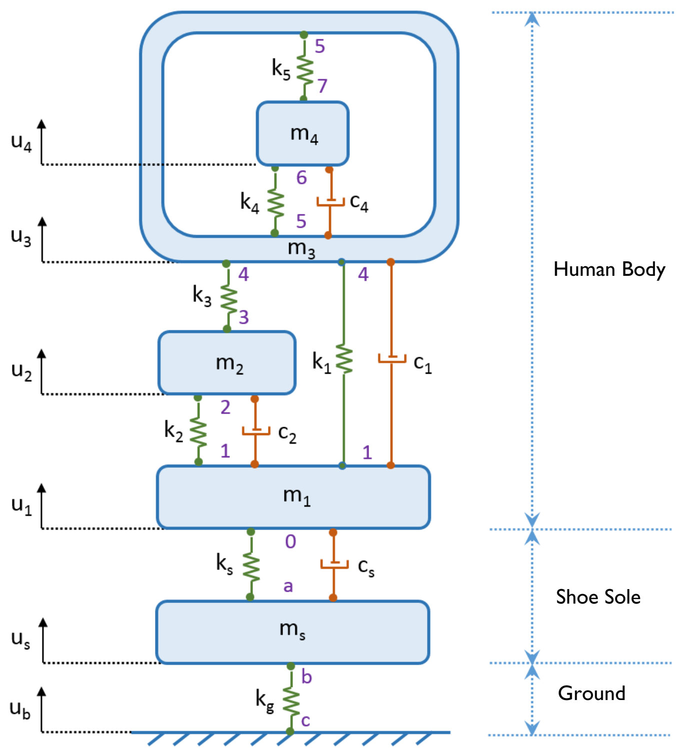


*Figure 1: The lumped 5-dof model of a human body including shoe-ground interaction. The five degrees of freedom of the system as well as the node numbers of the systems are also shown.*

The lumped model of the human body is shown in Figure 1. This model has three main components:

- Human body (4 dofs: $u_4$, $u_2$, $u_3$, $u_4$)
- Shoe sole (1 dof: $u_s$)
- Ground (0 dof)

### HUMAN BODY MODELING

The four-body model, also called LN model, is one of most commonly used lumped representation of a human body. This model consists of four masses ($m_1$, $m_2$, $m_3$, $m_4$), five springs ($k_1$, $k_2$, $k_3$, $k_4$, $k_5$), and three dampers ($c_1$, $c_2$, $c_4$). In this model, the entire human body is divided into four parts:

- Lower rigid mass ($m_1$)
- Lower wobbling mass ($m_2$)
- Upper rigid mass ($m_3$)
- Upper wobbling mass ($m_4$)

Here the wobbling mass includes the mass of all nonrigid parts such as muscles, skin, blood vessels, and so on.

### SHOE MODELING

A shoe is modeled as a one-body model having mass ($m_s$), spring ($k_s$), and damper ($c_s$) elements. In practice the force between the shoe and foot is a nonlinear function of spring deformation. In this model however it is assumed to be a linear function.

### GROUND MODELING

The ground is modeled with a spring ($k_g$) element representing the stiffness of the soil. Three different values of ground stiffness is used in this model to compare the response of the system:

- Soft soil (99 kN/m)
- Hard soil (359 kN/m)
- Very hard soil (880 kN/m)

For the frequency response analysis, a base excitation ($u_b$) is prescribed at the ground.

### VIBRATION TRANSMISSIBILITY

A vertical displacement transmissibility is defined as the ratio of displacement at one of the masses to the base excitation. As an example, the displacement transmissibility at the second mass can be defined as:

$$H_{m2} = \frac{u_{m2}}{u_b}$$

The transmissibility is computed in the frequency response analysis and can be plotted as a function of frequency.

### MODEL PARAMETERS

The parameters used in the model are given in the table below:

TABLE 1: MODEL PARAMETERS

| DESCRIPTION | NAME | EXPRESSION |
|---|---|---|
| Mass 1 | m1 | 6.15 kg |
| Mass 2 | m2 | 6 kg |
| Mass 3 | m3 | 12.58 kg |
| Mass 4 | m4 | 50.34 kg |
| Spring constant, spring 1 | k1 | 6 kN/m |
| Spring constant, spring 2 | k2 | 6 kN/m |
| Spring constant, spring 3 | k3 | 10 kN/m |
| Spring constant, spring 4 | k4 | 10 kN/m |
| Spring constant, spring 5 | k5 | 18 kN/m |
| Damping coefficient, damper 1 | c1 | 0.3 kN-s/m |
| Damping coefficient, damper 2 | c2 | 0.65 kN-s/m |
| Damping coefficient, damper 4 | c4 | 1.9 kN-s/m |
| Mass, shoe sole | ms | 0.3 kg |
| Spring constant, shoe sole | ks | 403 kN/m |
| Damping coefficient, shoe sole | cs | 2170 kN-s/m |
| Spring constant, ground | kg | 880 kN/m |
| Base excitation | ub | 10 mm |

## *Results and Discussion*

The model considered here has five degrees of freedom and hence in total it can predict five natural frequencies. The first two natural frequencies of the system are given in Table 2:

TABLE 2: EIGENFREQUENCIES

| EIGENFREQUENCY | VALUE |
|---|---|
| First | 1.765+0.611i Hz |
| Second | 55.4+11.024i Hz |

Figure 2 and Figure 3 show the displacement amplitude and phase of various masses at different frequencies. It can be seen that some of the masses have higher displacement near the two natural frequencies computed in the eigenfrequency analysis. It can also be noticed that the lower body masses ($m_1, m_2$) have comparatively higher displacement at the second natural frequency whereas the upper body masses ($m_3, m_4$) have higher displacement at the first natural frequency.

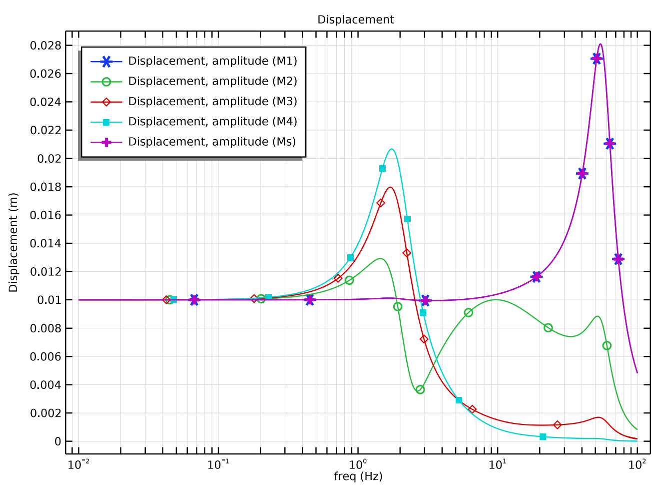


*Figure 2: Variation of displacement amplitude with frequency.*

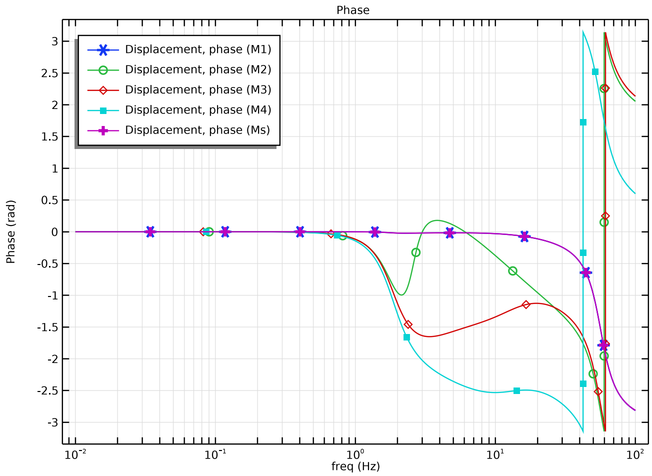


*Figure 3: Variation of displacement phases with frequency.*

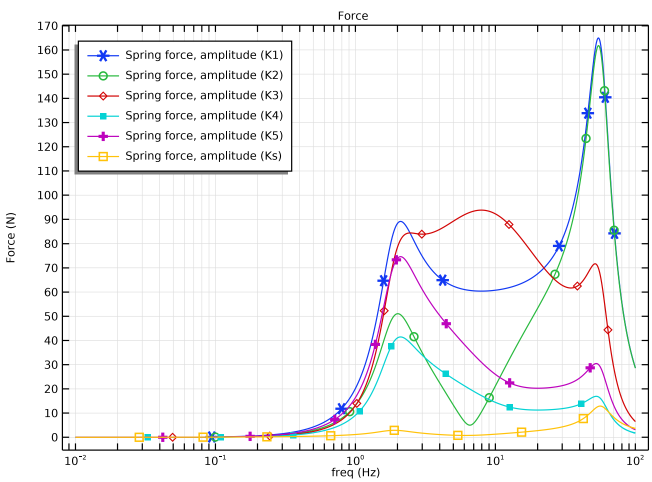


*Figure 4: Variation of amplitude of various spring forces with frequency.*

Figure 4 and Figure 5 show the amplitude and phase of various spring forces at different frequencies. It can be noticed that amplitude of the spring forces are much higher at higher frequencies compared to lower frequencies.

Figure 6 shows the frequency spectrum of the transmissibility of the second mass. The plot compares the transmissibility for three different soils having different hardness values. It is seen that the first peak corresponding to the first natural frequency is almost unaffected. The second peak, however, shifts toward lower frequencies as the soil stiffness is reduced.

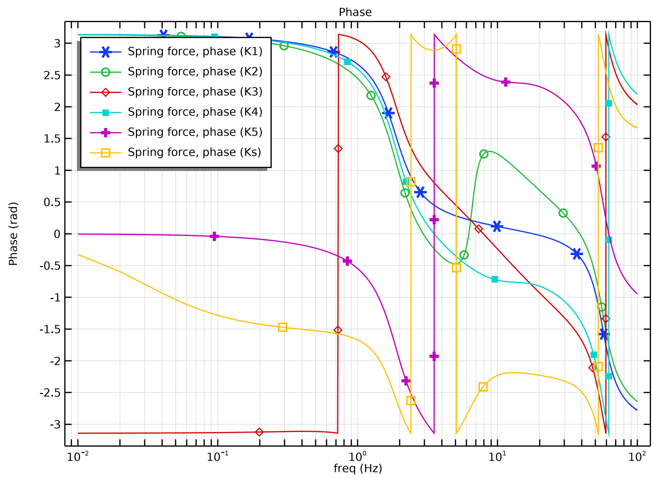


*Figure 5: Variation of phase of various spring forces with frequency.*

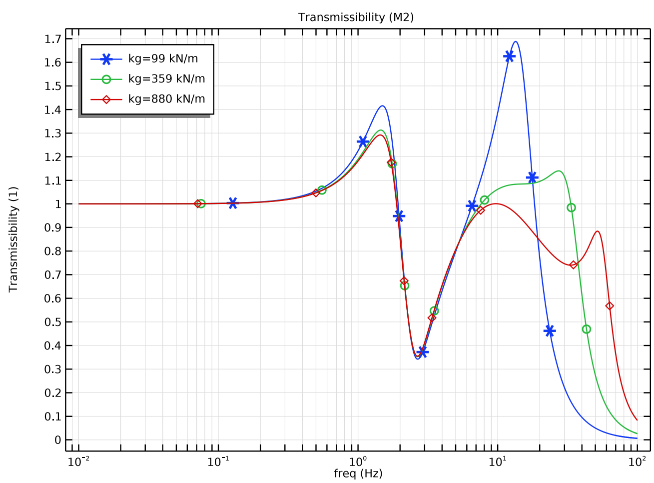


*Figure 6: Frequency variation of vertical displacement transmissibility of second mass.*

## *Notes About the COMSOL Implementation*

- **Fixed Node** is the default node in the **Lumped Mechanical System** interface. It can be disabled if none of the nodes of the system are fixed.
- Lumped models in general have very few degrees of freedom compared to FEM models. Thus, for the eigenfrequency computation, **All (filled matrix)** can be used in **Eigenfrequency search method**.
- Default plots from the **Lumped Mechanical System** interface can be customized by selecting the appropriate options in the **Results** section of various features.

## *Reference*

1. A.A. Nikooyan, A.A. Zadpoor, “Mass-spring-damper modelling of the human body to study running and hopping-an overview,” *Journal of Engineering in Medicine*, vol. 225, no. 12, pp. 1121–35, 2011.

**Application Library path:** `Multibody_Dynamics_Module/Biomechanics/lumped_human_body`

## *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 **0D**.
2. In the **Select Physics** tree, select **Structural Mechanics>Lumped Mechanical System (lms)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Eigenfrequency**.
6. Click **Done**.

## GLOBAL DEFINITIONS

*Parameters 1*

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 `lumped_human_body_parameters.txt`.

## LUMPED MECHANICAL SYSTEM (LMS)

*Fixed Node 1 (fix1)*

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Component 1 (comp1)>Lumped Mechanical System (lms)>Fixed Node 1 (fix1)** and choose **Disable**.

Define the lumped model of a human body and choose spring forces, and mass displacements as default plots.

*Mass 1 (M1)*

1 In the **Model Builder** window, right-click **Lumped Mechanical System (lms)** and choose **Mass**.

2 In the **Settings** window for **Mass**, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p1 | 0 |
| p2 | 1 |

4 Locate the **Component Parameters** section. In the $m$ text field, type `m1`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

*Spring 1 (K1)*

1 In the **Physics** toolbar, click **Global** and choose **Spring**.

2 In the **Settings** window for **Spring**, locate the **Node Connections** section.

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

| Label | Node names |
| --- | --- |
| p1 | 1 |
| p2 | 4 |

**4** Locate the **Component Parameters** section. In the *k* text field, type `k1`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

*Damper 1 (C1)*

**1** In the **Physics** toolbar, click **Global** and choose **Damper**.

**2** In the **Settings** window for **Damper**, locate the **Node Connections** section.

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

| Label | Node names |
| --- | --- |
| p1 | 1 |
| p2 | 4 |

**4** Locate the **Component Parameters** section. In the *c* text field, type `c1`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

**6** Clear the **Displacement** check box.

*Spring 2 (K2)*

**1** In the **Physics** toolbar, click **Global** and choose **Spring**.

**2** In the **Settings** window for **Spring**, locate the **Node Connections** section.

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

| Label | Node names |
| --- | --- |
| p1 | 1 |
| p2 | 2 |

**4** Locate the **Component Parameters** section. In the *k* text field, type `k2`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

*Damper 2 (C2)*

**1** In the **Physics** toolbar, click **Global** and choose **Damper**.

**2** In the **Settings** window for **Damper**, locate the **Node Connections** section.

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

| Label | Node names |
|---|---|
| p1 | 1 |
| p2 | 2 |

**4** Locate the **Component Parameters** section. In the *c* text field, type `c2`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

**6** Clear the **Displacement** check box.

*Mass 2 (M2)*

**1** In the **Physics** toolbar, click **Global** and choose **Mass**.

**2** In the **Settings** window for **Mass**, locate the **Node Connections** section.

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

| Label | Node names |
|---|---|
| p1 | 2 |
| p2 | 3 |

**4** Locate the **Component Parameters** section. In the *m* text field, type `m2`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

*Spring 3 (K3)*

**1** In the **Physics** toolbar, click **Global** and choose **Spring**.

**2** In the **Settings** window for **Spring**, locate the **Node Connections** section.

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

| Label | Node names |
|---|---|
| p1 | 3 |
| p2 | 4 |

**4** Locate the **Component Parameters** section. In the *k* text field, type `k3`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

*Mass 3 (M3)*

1 In the **Physics** toolbar, click **Global** and choose **Mass**.

2 In the **Settings** window for **Mass**, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p1 | 4 |
| p2 | 5 |

4 Locate the **Component Parameters** section. In the *m* text field, type `m3`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

*Spring 4 (K4)*

1 In the **Physics** toolbar, click **Global** and choose **Spring**.

2 In the **Settings** window for **Spring**, locate the **Node Connections** section.

3 In the table, enter the following settings:

| Label | Node names |
|---|---|
| p1 | 5 |
| p2 | 6 |

4 Locate the **Component Parameters** section. In the *k* text field, type `k4`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

*Damper 3 (C3)*

1 In the **Physics** toolbar, click **Global** and choose **Damper**.

2 In the **Settings** window for **Damper**, type `C4` in the **Name** text field.

3 Locate the **Node Connections** section. In the table, enter the following settings:

| Label | Node names |
|---|---|
| p1 | 5 |
| p2 | 6 |

4 Locate the **Component Parameters** section. In the *c* text field, type `c4`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

**6** Clear the **Displacement** check box.

*Mass 4 (M4)*

**1** In the **Physics** toolbar, click **Global** and choose **Mass**.

**2** In the **Settings** window for **Mass**, locate the **Node Connections** section.

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

| Label | Node names |
| --- | --- |
| p1 | 6 |
| p2 | 7 |

**4** Locate the **Component Parameters** section. In the *m* text field, type `m4`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

*Spring 5 (K5)*

**1** In the **Physics** toolbar, click **Global** and choose **Spring**.

**2** In the **Settings** window for **Spring**, locate the **Node Connections** section.

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

| Label | Node names |
| --- | --- |
| p1 | 7 |
| p2 | 5 |

**4** Locate the **Component Parameters** section. In the *k* text field, type `k5`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

After defining the human body model, now define the lumped model for the shoe and ground.

*Spring 6 (K6)*

**1** In the **Physics** toolbar, click **Global** and choose **Spring**.

**2** In the **Settings** window for **Spring**, type `Ks` in the **Name** text field.

**3** Locate the **Node Connections** section. In the table, enter the following settings:

| Label | Node names |
| --- | --- |
| p1 | 0 |
| p2 | a |

**4** Locate the **Component Parameters** section. In the $k$ text field, type `ks`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

*Damper 5 (C5)*

**1** In the **Physics** toolbar, click **Global** and choose **Damper**.

**2** In the **Settings** window for **Damper**, type `Cs` in the **Name** text field.

**3** Locate the **Node Connections** section. In the table, enter the following settings:

| Label | Node names |
| --- | --- |
| p1 | 0 |
| p2 | a |

**4** Locate the **Component Parameters** section. In the $c$ text field, type `cs`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

**6** Clear the **Displacement** check box.

*Mass 5 (M5)*

**1** In the **Physics** toolbar, click **Global** and choose **Mass**.

**2** In the **Settings** window for **Mass**, type `Ms` in the **Name** text field.

**3** Locate the **Node Connections** section. In the table, enter the following settings:

| Label | Node names |
| --- | --- |
| p1 | a |
| p2 | b |

**4** Locate the **Component Parameters** section. In the $m$ text field, type `ms`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

*Spring 6a (K6)*

**1** In the **Physics** toolbar, click **Global** and choose **Spring**.

**2** In the **Settings** window for **Spring**, type `Kg` in the **Name** text field.

**3** Locate the **Node Connections** section. In the table, enter the following settings:

| Label | Node names |
|---|---|
| p1 | b |
| p2 | c |

**4** Locate the **Component Parameters** section. In the $k$ text field, type `kg`.

**5** Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Force** check box.

**6** Clear the **Displacement** check box.

Next, define a base excitation at the ground for the frequency domain analysis.

*Displacement Node 1 (disp1)*

**1** In the **Physics** toolbar, click **Global** and choose **Displacement Node**.

**2** In the **Settings** window for **Displacement Node**, locate the **Node Connections** section.

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

| Label | Node name |
|---|---|
| p1 | c |

**4** Locate the **Terminal Parameters** section. In the $u_{p10}$ text field, type `ub`.

## STUDY 1

*Step 1: Eigenfrequency*

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

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

**3** From the **Eigenfrequency search method** list, choose **All (filled matrix)**.

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

## RESULTS

*Evaluation Group 1*

In the **Evaluation Group 1** toolbar, click **Evaluate**.

## ADD STUDY

**1** In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

**2** Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

## STUDY 2

### *Parametric Sweep*

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| kg (Spring constant, ground) | `99 359 880` | `kN/m` |

### *Step 1: Frequency Domain*

1 In the **Model Builder** window, click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(0.01,0.01,100)`.

4 In the **Study** toolbar, click **Compute**.

## RESULTS

### *Displacement, Amplitude (lms) 1*

Follow the instructions below to modify the default plots of mass displacements and spring forces as shown in Figure 2, Figure 3, Figure 4, and Figure 5 respectively.

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Displacement (m)`.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 In the **Displacement, Amplitude (lms) 1** toolbar, click **Plot**.

6 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

### *Displacement, Phase (lms) 1*

1 In the **Model Builder** window, click **Displacement, Phase (lms) 1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Phase (rad)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 In the **Displacement, Phase (lms) 1** toolbar, click **Plot**.

7 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

*Force, Amplitude (lms)*

1 In the **Model Builder** window, click **Force, Amplitude (lms)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Force (N)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 In the **Force, Amplitude (lms)** toolbar, click **Plot**.

7 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

*Force, Phase (lms)*

1 In the **Model Builder** window, click **Force, Phase (lms)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Phase (rad)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 In the **Force, Phase (lms)** toolbar, click **Plot**.

7 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.

Follow the instructions below to plot the displacement transmissibility at mass-2 as shown in Figure 6.

*Transmissibility (M2)*

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

2 In the **Settings** window for **1D Plot Group**, type `Transmissibility (M2)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 1 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Global 1*

1 Right-click **Transmissibility (M2)** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Lumped Mechanical System>Two port components>M2>lms.M2_uAmp - Displacement, amplitude (M2) - m**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| comp1.lms.M2_uAmp/ub | 1 | Transmissibility |

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

6 In the **Transmissibility (M2)** toolbar, click **Plot**.

7 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.