Created in COMSOL Multiphysics 5.6

Biomechanical Model of the Human Body in a Sitting Posture

Introduction

In this example, a biomechanical model of the human body is developed for evaluating the dynamic response to the vertical vibrations in a sitting posture. It demonstrates how the Multibody Dynamics interface in COMSOL Multiphysics can be used to model various parts and connections in a human body and study the whole body vibrations (WBV). The problem statement and the model parameters are taken from Ref. 1.

The dynamic response of a human body in any vibration environment can be predicted using this biomechanical model. In the automobile industry, this model can be used in ride quality simulations as well as when designing vibration isolators such as the seats.

Model Definition

GEOMETRY AND CONNECTIONS

The biomechanical model of a human body consists of different body parts such as head, torso, pelvis, thighs, viscera, and legs, as shown in Figure 1. All the body parts are treated as lumped masses and defined as rigid bodies. The values of mass and moment of inertia of each body part about its center of mass are given in Table 1.

TABLE 1: MASS AND MOMENT OF INERTIA OF DIFFERENT BODY PARTS

BODY PART	MASS (kg)	MOMENT OF INERTIA (kg-m^2)
Head	7.24	0.411
Torso	19.90	1.627
Pelvis	11.01	0.692
Thighs	20.35	1.180
Viscera	12.92	-
Legs	-	-

The viscera are not allowed to rotate, and the legs are not allowed to translate or rotate. Hence corresponding values of mass and moment of inertia are not required in the above table.

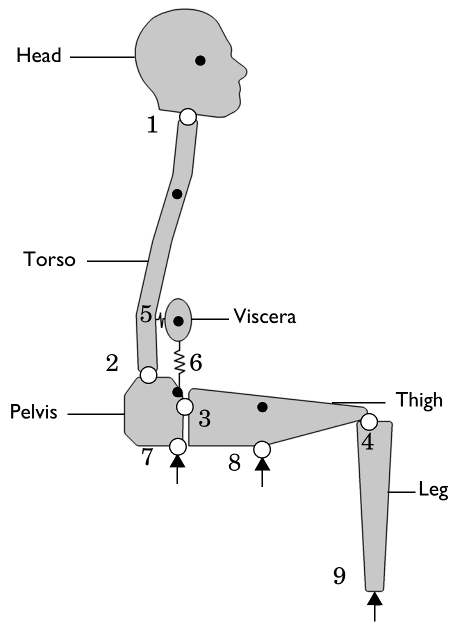

Figure 1: Model geometry showing different body parts and connections between them. The center of mass of each body part and the locations of vibration excitation are also shown.

The connections between different body parts can be approximated using translational and rotational springs and dampers, applied on the relative motion between the two body parts. This type of connection is modeled using the elastic version of fixed joint. Once the elastic version of a joint is used, the translational and rotational stiffness and damping values between the two connected parts can be provided.

In this model, the source of vibration (seat) is not modeled explicitly and instead of that a base motion node is used. Here the input excitation is of $1\ \mathrm{m/s^2}$ in the vertical direction at three different locations. The body parts, which are directly in touch with the vibrating seat, are legs, thighs, and pelvis. The connections between these body parts and the vibrating seat are also modeled using fixed joint. The elasticity on these joints is included wherever it is required to model the cushioning effect of the seat.

The values of stiffness and damping coefficients, translational as well as rotational, for all the connections in the model are provided in table below.

TABLE 2: JOINT ELASTICITY DETAILS

FIXED JOINT	TRANSLATIONAL STIFFNESS (KN/M)	TRANSLATIONAL DAMPING COEFFICIENT (KN-S/M)	ROTATIONAL STIFFNESS (KN-M/RAD)	ROTATIONAL DAMPING COEFFICIENT (KN-M-S/RAD)
Head-torso	113.7, 113.7	0.066, 0.066	0.915	0.340
Torso-pelvis	0.299, 0.299	1.79, 1.79	0.328	0.724
Pelvis-thigh	6.40, 6.40	0.061, 0.061	0.162	0.030
Thigh-leg	23.55, 23.55	0.154, 0.154	0.220	0.104
Viscera-torso	1.93, 0	0.079, 0	0	0
Viscera-pelvis	0, 18.37	0, 0.197	0	0
Seat-pelvis	0.905, 121.3	0.015, 0.047	0	0
Seat-thigh	0.614, 16.71	0.014, 8.01	0	0
Seat-leg	-	-	-	-

The legs are connected to the seat with the rigid version of fixed joint and hence it doesn't need joint elasticity parameters.

VIBRATION TRANSMISSIBILITY

In this example, first an eigenfrequency analysis is performed to determine the damped and undamped natural frequencies of vibration. Secondly a frequency response analysis is carried out around the natural frequencies to find out the vertical transmissibility (ratio of vertical acceleration of the head to the input acceleration of the seat), the rotational transmissibility (ratio of angular acceleration of the head to the input acceleration the seat) and the apparent mass (ratio of the force at the seat to the input acceleration of the seat).

$$H_{\text{vert}} = \frac{(\ddot{y})_{\text{head}}}{(\ddot{y})_{\text{seat}}},\ H_{rot} = \frac{(\ddot{\phi})_{\text{head}}}{(\ddot{y})_{\text{seat}}},\ M_a = \frac{(F)_{\text{seat}}}{(\ddot{y})_{\text{seat}}}$$

The transmissibilities and apparent mass are directly related to the comfort feeling. Especially the vertical and rotational transmissibility affect the ride comfort and the vision.

Results and Discussion

Figure 2 shows one of the rotational eigenmodes of the undamped biomechanical model. In this mode, rotational movement of the head and torso segments can be seen. The viscera, pelvis, thighs and legs do not have considerable movement as compared to the other two.

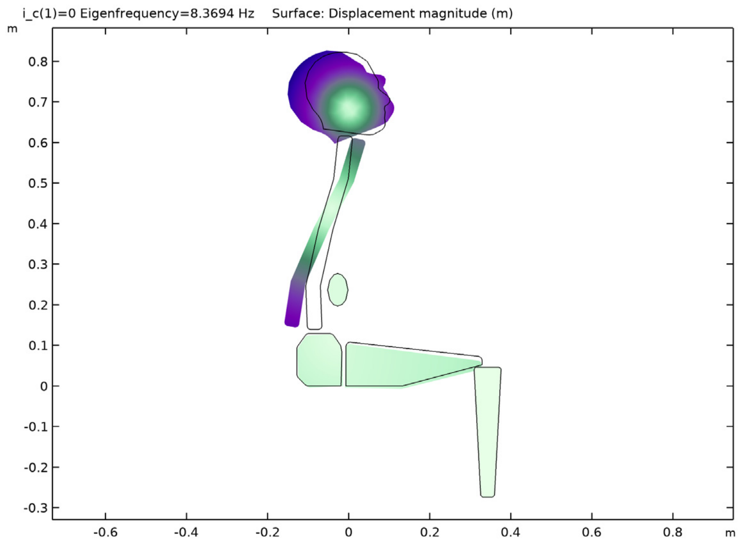

Figure 2: One of the rotational eigenmodes of the undamped biomechanical model.

The first major translational eigenmode of the damped biomechanical model is shown in Figure 3. In this mode, there is a downward movement of head, pelvis, and viscera whereas other body parts do not move much.

Figure 4 shows the second major translational eigenmode. In this mode, body parts like head, torso, and pelvis move downward whereas viscera move upward.

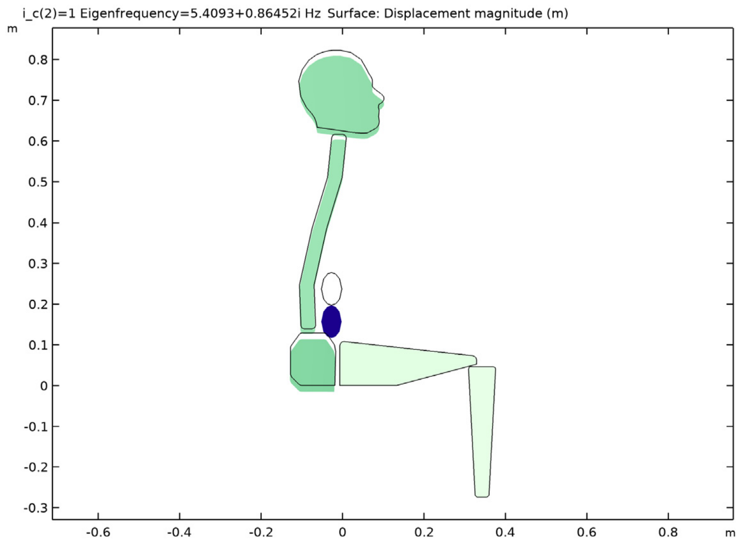

Figure 3: First major translational eigenmode of the damped biomechanical model.

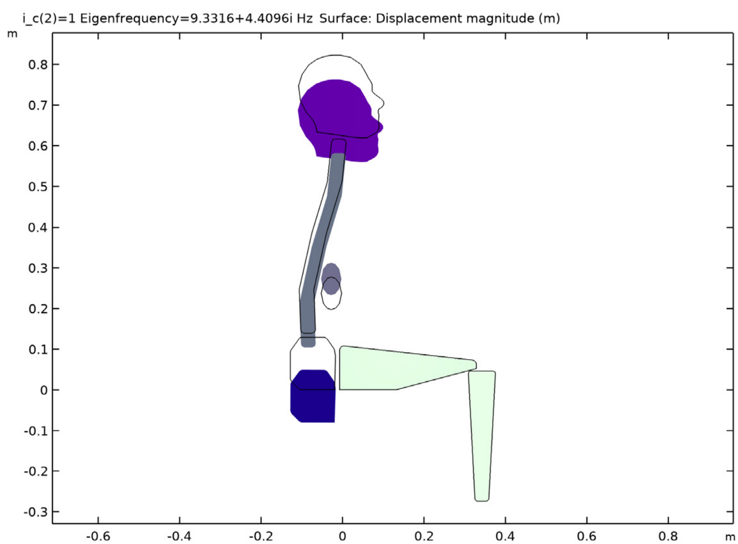

Figure 4: Second major translational eigenmode of the damped biomechanical model.

[Figure 5] shows the variation of vertical transmissibility with the excitation frequency. The primary and secondary resonance are visible in the range of 4-6 Hz and 8-10 Hz respectively. The eigenmodes for these two translational modes are shown in [Figure 3] and [Figure 4].

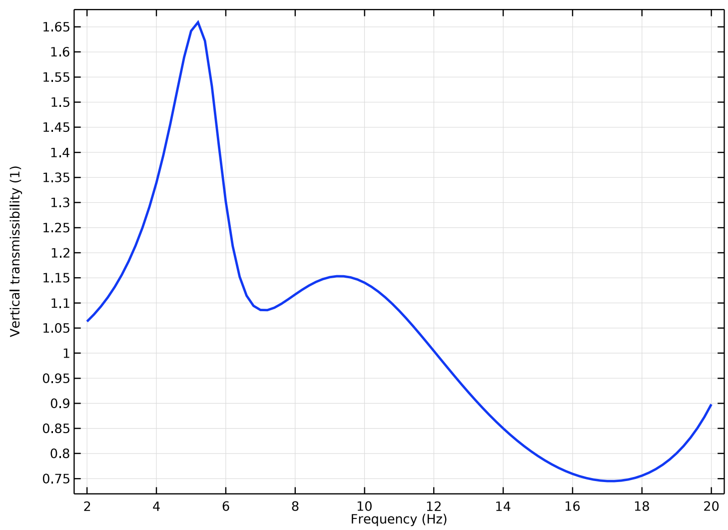

Figure 5: Vertical transmissibility versus excitation frequency.

[Figure 6] shows the variation of rotational transmissibility with the excitation frequency. A high value of rotational transmissibility is not recommended as apart from reducing the comfort level it also directly affects the vision.

While vertical and rotation transmissibility depicts the endpoint characteristics of the model, apparent mass conveys the driving point characteristics, relating the force and the motion at the seat. [Figure 7] shows the variation of apparent mass of the system with the excitation frequency.

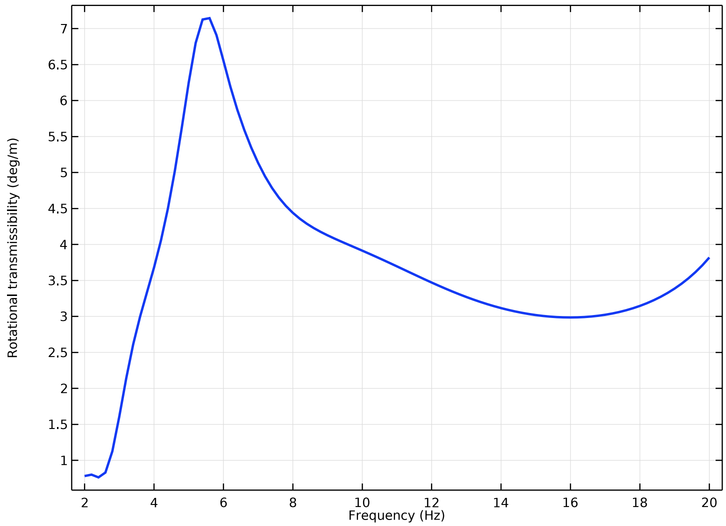

Figure 6: Rotational transmissibility versus excitation frequency.

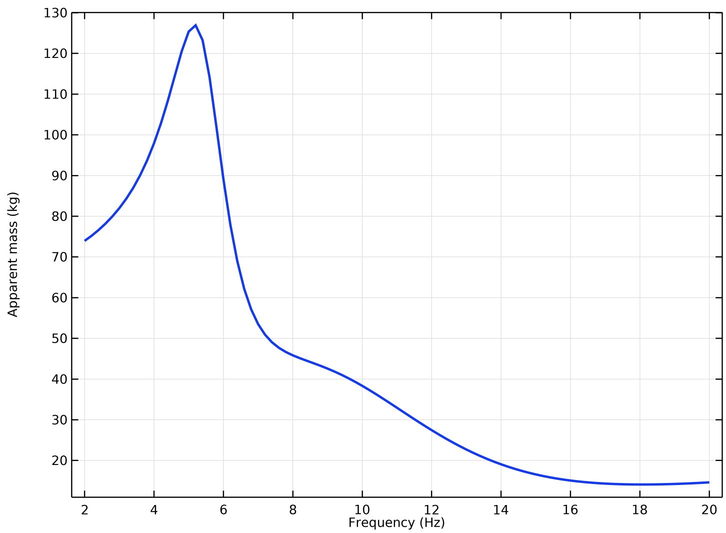

Figure 7: Apparent mass versus excitation frequency.

Notes About the COMSOL Implementation

- To model a lumped mass, use **Mass and Moment of Inertia** subnode of the **Rigid Domain** node and enter the inertia properties given at a certain point. Also make the density zero in the dummy domain.
- To model a bushing, use **Joint Elasticity** node of the **Fixed Joint** node and enter the stiffness and damping properties of the joint.
- Use **Base Motion** node to excite the system instead of modeling an actual vibrating base (seat).
- The connections set up in the model and net system DOFs can be reviewed in the **Joints Summary** and **Rigid Body DOF Summary** sections at the physics node.

Reference

1. Tae-Hyeong Kim, Young-Tae Kim, and Yong-San Yoon, "Development of a biomechanical model of the human body in a sitting posture with vibration transmissibility in the vertical direction," *Int. J. Industrial Ergonomics*, vol. 35, pp. 817–829, 2005.

Application Library path: `Multibody_Dynamics_Module/Biomechanics/seated_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 **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Multibody Dynamics (mbd)**.

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 `seated_human_body_parameters.txt`.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Click **Build Selected**.

MULTIBODY DYNAMICS (MBD)

Do as follows to generate Rigid Domain nodes for all body parts.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Multibody Dynamics (mbd)**.

2 In the **Settings** window for **Multibody Dynamics**, locate the **Automated Model Setup** section.

3 Select the **Include mass and moment of inertia node** check box. This automatically sets the density of all rigid domains to zero and adds a Mass and Moment of Inertia subnode to each Rigid Domain node.

4 Click **Physics Node Generation** in the upper-right corner of the **Automated Model Setup** section. From the menu, choose **Create Rigid Domains**.

Rigid Domain: Pelvis

1 In the **Model Builder** window, expand the **Rigid Domains (All)** node, then click **Rigid Domain 1**.

2 In the **Settings** window for **Rigid Domain**, type `Rigid Domain: Pelvis` in the **Label** text field.

Mass and Moment of Inertia 1

1 In the **Model Builder** window, expand the **Rigid Domain: Pelvis** node, then click **Mass and Moment of Inertia 1**.

2 In the **Settings** window for **Mass and Moment of Inertia**, locate the **Mass and Moment of Inertia** section.

3 In the m text field, type `m_pelvis`.

4 In the I_z text field, type `I_pelvis`.

5 Locate the **Center of Mass** section. From the list, choose **Centroid of selected entities**.

6 From the **Entity level** list, choose **Point**.

Center of Mass: Point 1

1 In the **Model Builder** window, click **Center of Mass: Point 1**.

2 Select Point 11 only.

Rigid Domains

Similarly, model the other four body parts by assigning other Rigid Domain nodes in the group**Rigid Domains (All)** and resetting the inputs using the information given in the table below.

Name	**Selection**	**Mass**	**Moment of Inertia**	**Center of Mass (Point)**
Rigid Domain: Head	2	m_head	I_head	46
Rigid Domain: Torso	3	m_torso	I_torso	46
Rigid Domain: Viscera	4	m_viscera	0	55
Rigid Domain: Thigh	5	m_thigh	I_thigh	62

The rotational motion of the viscera is not included in the model. Therefore constrain the rotation of **Rigid Domain: Viscera** by using the **Prescribed Displacement/Rotation 1** subnode.

Rigid Domain: Viscera

In the **Model Builder** window, click **Rigid Domain: Viscera**.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Displacement/Rotation**.

2 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Rotation** section.

3 From the **By** list, choose **Constrained rotation**.

Use the **Rigid Domain 6** node to model the legs.

Rigid Domain: Leg

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Rigid Domains (All)** click **Rigid Domain 6**.

2 In the **Settings** window for **Rigid Domain**, type `Rigid Domain: Leg` in the **Label** text field.

Add a Base Motion node to model the effects of a vibrating seat.

Base Motion: Seat

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

2 In the **Settings** window for **Base Motion**, type `Base Motion: Seat` in the **Label** text field.

3 Locate the **Base Motion** section. From the **Base motion type** list, choose **Acceleration**.

4 Specify the $\mathbf{a}_{\mathrm{b}}$ vector as

0	x
vtt_in	y

Fixed Joint: Head-Torso

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

2 In the **Settings** window for **Fixed Joint**, type `Fixed Joint: Head-Torso` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Rigid Domain: Head**.

4 From the **Destination** list, choose **Rigid Domain: Torso**.

5 Locate the **Center of Joint** section. From the **Entity level** list, choose **Point**.

6 Locate the **Joint Elasticity** section. From the list, choose **Elastic joint**.

Center of Joint: Point 1

1 In the **Model Builder** window, click **Center of Joint: Point 1**.

2 Select Point 24 only.

Joint Elasticity 1

1 In the **Model Builder** window, click **Joint Elasticity 1**.

2 In the **Settings** window for **Joint Elasticity**, locate the **Spring** section.

3 From the list, choose **Diagonal**.

4 In the $\mathbf{k}_u$ table, enter the following settings:

k1	0
0	k1

5 In the k_θ text field, type `kr1`.

6 Locate the **Viscous Damping** section. From the list, choose **Diagonal**.

7 In the $\mathbf{c}_u$ table, enter the following settings:

if(i_c==1,c1,0)	0
0	if(i_c==1,c1,0)

8 In the c_θ text field, type `if(i_c==1,cr1,0)`.

Fixed Joints

Similarly create eight more fixed joints between the different body parts by duplicating **Fixed Joint: Head-Torso** and resetting the inputs using the information given in the table below.

Name	**Source**	**Destination**	**Center of joint selection (points)**
Fixed joint: Torso-Pelvis	Rigid Domain: Torso	Rigid Domain: Pelvis	39
Fixed joint: Pelvis-Thigh	Rigid Domain: Pelvis	Rigid Domain: Thigh	14
Fixed joint: Thigh-Leg	Rigid Domain: Thigh	Rigid Domain: Leg	64
Fixed joint: Viscera-Torso	Rigid Domain: Viscera	Rigid Domain: Torso	55
Fixed joint: Viscera-Pelvis	Rigid Domain: Viscera	Rigid Domain: Pelvis	55

Name	Source	Destination	Center of joint selection (points)
Fixed joint: Seat-Pelvis	Base motion: Seat	Rigid Domain: Pelvis	13
Fixed joint: Seat-Thigh	Base motion: Seat	Rigid Domain: Thigh	61
Fixed joint: Seat-Leg	Base motion: Seat	Rigid Domain: Leg	69, 70

Joint Elasticity

Enter the joint elasticity parameters using the following table:

Name	KU (Diagonal Values)	KTH	CU (Diagonal Values)	CTH
Fixed joint: Torso-pelvis	k2, k2	kr2	If(i_c==I, c2, 0), If(i_c==I, c2, 0)	If(i_c==I, cr2, 0)
Fixed joint: Pelvis-thigh	k3, k3	kr3	If(i_c==I, c3, 0), If(i_c==I, c3, 0)	If(i_c==I, cr3, 0)
Fixed joint: Thigh-leg	k4, k4	kr4	If(i_c==I, c4, 0), If(i_c==I, c4, 0)	If(i_c==I, cr4, 0)
Fixed joint: Viscera-torso	kh5, 0	0	If(i_c==I, ch5, 0), 0	0
Fixed joint: Viscera-pelvis	0, kv6	0	0, If(i_c==I, cv6, 0)	0
Fixed joint: Seat-pelvis	kh7, kv7	0	If(i_c==I, ch7, 0), If(i_c==I, cv7, 0)	0
Fixed joint: Seat-thigh	kh8, kv8	0	If(i_c==I, ch8, 0), If(i_c==I, cv8, 0)	0

In this model, the legs are directly mounted to the vibrating base. Therefore, disable the joint elasticity of the Seat-Leg joint.

Fixed Joint: Seat-Leg

1 In the **Model Builder** window, click **Fixed Joint: Seat-Leg**.

2 In the **Settings** window for **Fixed Joint**, locate the **Joint Elasticity** section.

3 From the list, choose **Rigid joint**.

Enable the joint force computation for the Seat-Pelvis and the Seat-Thigh joints to compute the apparent mass.

Fixed Joint: Seat-Thigh

1 In the **Model Builder** window, click **Fixed Joint: Seat-Thigh**.
2 In the **Settings** window for **Fixed Joint**, locate the **Joint Forces and Moments** section.
3 From the list, choose **Computed using weak constraints**.

Fixed Joint: Seat-Pelvis

1 In the **Model Builder** window, click **Fixed Joint: Seat-Pelvis**.
2 In the **Settings** window for **Fixed Joint**, locate the **Joint Forces and Moments** section.
3 From the list, choose **Computed using weak constraints**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3 From the **Element size** list, choose **Coarser**.
4 Click **Build All**.

STUDY 1: EIGENFREQUENCY

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Study 1: Eigenfrequency` in the **Label** text field.

Add a parametric sweep to find the undamped and damped natural frequencies of the system by changing the value of the damping controller parameter.

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
i_c (Damping controller)	0 1	

Step 1: Eigenfrequency

1 In the **Model Builder** window, click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `12`.

5 In the **Search for eigenfrequencies around** text field, type `0`.

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

RESULTS

Mode Shape (mbd)

Follow the instructions below to plot the eigenmodes shown in Figure 2, Figure 3, and Figure 4.

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

2 From the **Eigenfrequency (Hz)** list, choose **5.4093+0.86452i**.

3 In the **Mode Shape (mbd)** toolbar, click **Plot**.

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

5 From the **Eigenfrequency (Hz)** list, choose **9.3316+4.4096i**.

6 In the **Mode Shape (mbd)** toolbar, click **Plot**.

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

8 From the **Parameter value (i_c)** list, choose **0**.

9 From the **Eigenfrequency (Hz)** list, choose **8.3694**.

10 In the **Mode Shape (mbd)** toolbar, click **Plot**.

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

Add a new study to carry out the frequency response analysis of this system.

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: FREQUENCY RESPONSE

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2: Frequency Response` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2: Frequency Response** 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(2,0.2,20)`.

Define the transmissibility variables to use them in the postprocessing.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
H_vert	abs(mbd.rd2.u_tty)/vtt_in		Vertical transmissibility
H_rot	abs(mbd.rd2.th_ttz)/vtt_in	rad/m	Rotational transmissibility
M_a	abs(mbd.fxj7.Fy+ mbd.fxj8.Fy)/vtt_in	kg	Apparent mass

STUDY 2: FREQUENCY RESPONSE

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

Follow the instructions below to plot the vertical transmissibility shown in Figure 5

RESULTS

Vertical Transmissibility

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Frequency Response/ Solution 5 (sol5)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

6 In the associated text field, type `Frequency (Hz)`.

Global 1

1 Right-click **Vertical Transmissibility** 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)>Definitions>Variables>H_vert - Vertical transmissibility**.

3 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 In the **Vertical Transmissibility** toolbar, click **Plot**.

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

Duplicate the vertical transmissibility plot to create the rotational transmissibility plot shown in Figure 6.

Rotational Transmissibility

1 In the **Model Builder** window, right-click **Vertical Transmissibility** and choose **Duplicate**.

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

Global 1

1 In the **Model Builder** window, expand the **Rotational Transmissibility** node, then click **Global 1**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`H_rot`	`deg/m`	`Rotational transmissibility`

4 In the **Rotational Transmissibility** toolbar, click **Plot**.

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

Duplicate the rotational transmissibility plot to create the apparent mass plot shown in Figure 7.

Apparent Mass

1 In the **Model Builder** window, right-click **Rotational Transmissibility** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Apparent Mass` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Apparent Mass** node, then click **Global 1**.

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)>Definitions>Variables>M_a - Apparent mass - kg**.

3 In the **Apparent Mass** toolbar, click **Plot**.

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