Created in COMSOL Multiphysics 6.1

Vibration in a Washing Machine Assembly

Introduction

Vibration and noise, due to nonuniform distribution of clothes, is a common problem in washing machines.

This model simulates a simplified multibody dynamics model of a horizontal-axis portable washing machine. An eigenfrequency analysis is performed to compute the natural frequencies and mode shapes of the entire assembly. Transient analysis is performed to find out the vibrations induced in the housing during the spinning cycle. The housing is modeled as a flexible shell.

Note: This model requires the Multibody Dynamics Module and the Structural Mechanics Module.

Model Definition

The geometry of the washing machine assembly is shown in Figure 1.

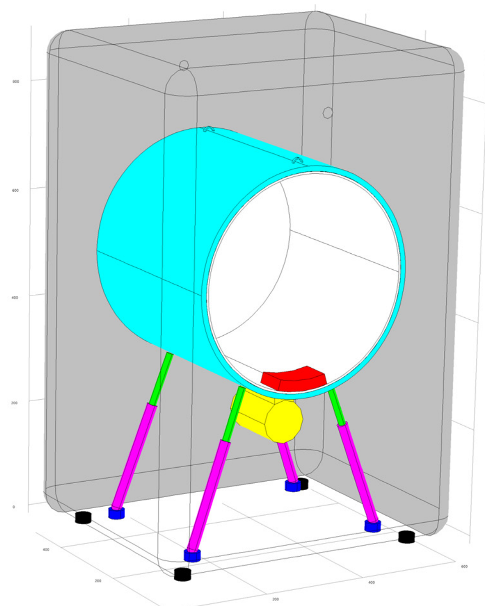


Figure 1: Model geometry (top, bottom, front, and left panels of the housing are hidden)

A key to the coloring of the parts constituting the washing machine is given below.

TABLE 1: IDENTIFICATION OF VARIOUS PARTS OF WASHING MACHINE ASSEMBLY

Part	**Color in figure**
Clothes	Red
Drum	White
Tub	Cyan
Motor	Yellow
Pistons	Green
Cylinders	Magenta
Mountings	Blue
Base supports	Black
Housing	Gray

The following assumptions are used:

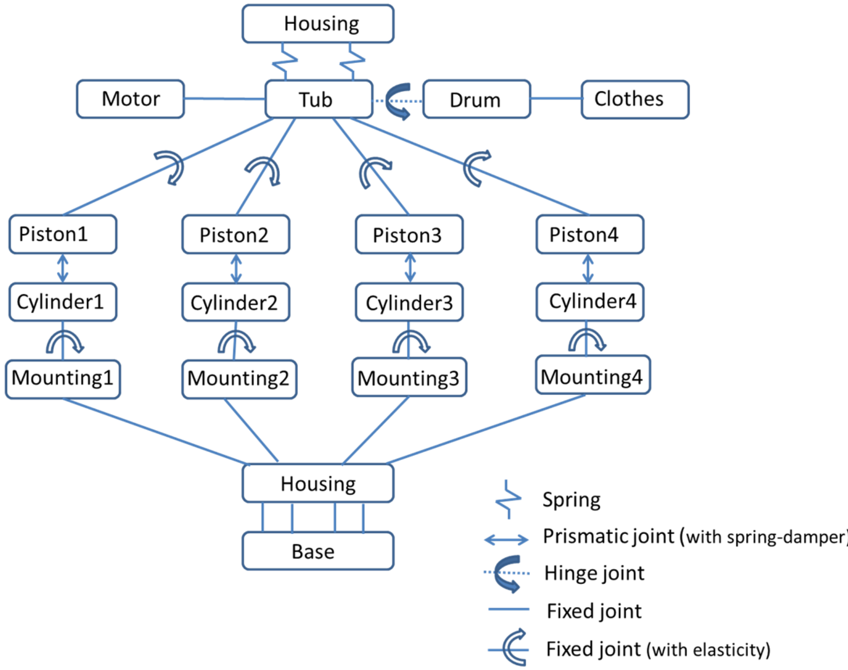
- The housing is modeled using flexible shell elements.
- The other parts are modeled by rigid solids.
- The clothes do not move relative to the drum.

Connection Details

The connections of the housing to the remaining parts are as follows:

- Connection with the base, at four locations on the bottom surface, using fixed joints.
- Connection with the mountings, at four locations on the bottom surface, using fixed joints.
- Connection with the tub, at front and back surfaces, using stabilizing springs. These springs are not shown in the geometry (Figure 1) or in the model.

The details of all the connections in the assembly are shown below:



Results and Discussion

The mode shapes of the washing machine assembly can be seen in Figure 2 and Figure 3. In these figures, one of the modes showing the translation of the tub whereas the other one shows the rotation of the tub about the vertical axis. The corresponding housing deformation can also be seen. The magnitude of deformation of housing is very small compared to the motion of the tub, so a different color table is used for better clarity.

Figure 4 and Figure 5 shows the tub displacement magnitude with the drum rotation or the position of unbalanced clothes for the full time duration. The color of the trajectory has the time information representing red as the initial time and blue as final time.

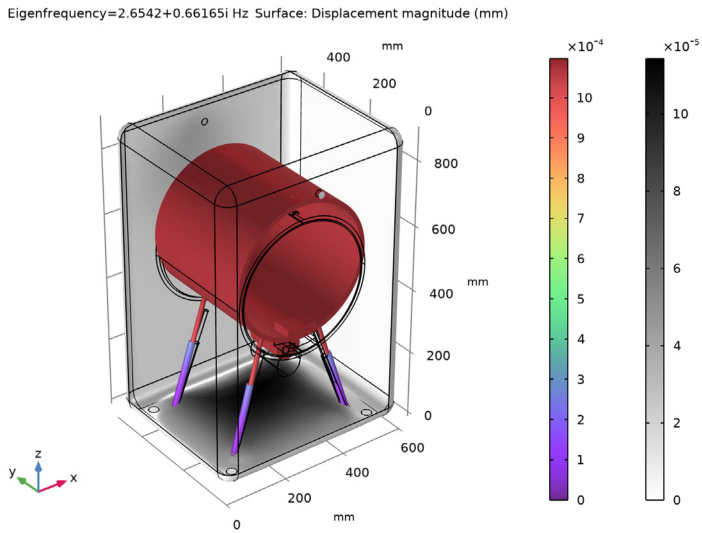


Figure 2: One of the eigenmodes of the washing machine assembly (with the tub translating along vertical axis).

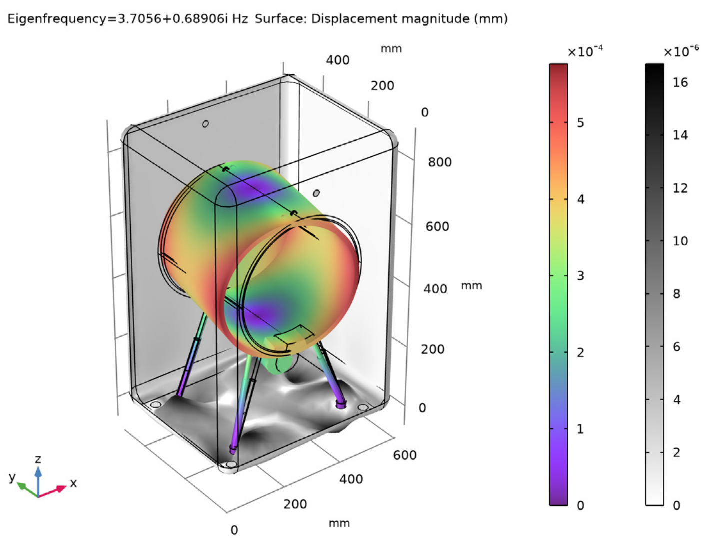


Figure 3: One of the eigenmodes of the washing machine assembly (with the tub rotating about vertical axis).

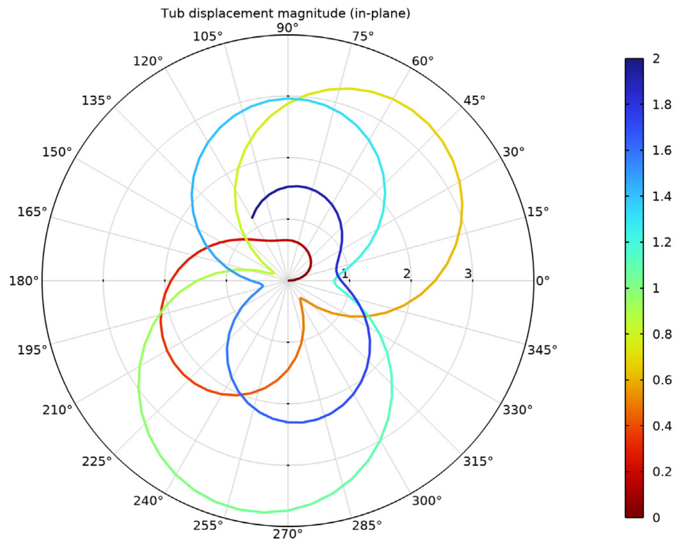


Figure 4: In-plane (x-z) displacement magnitude of the tub with the position of unbalanced clothes.

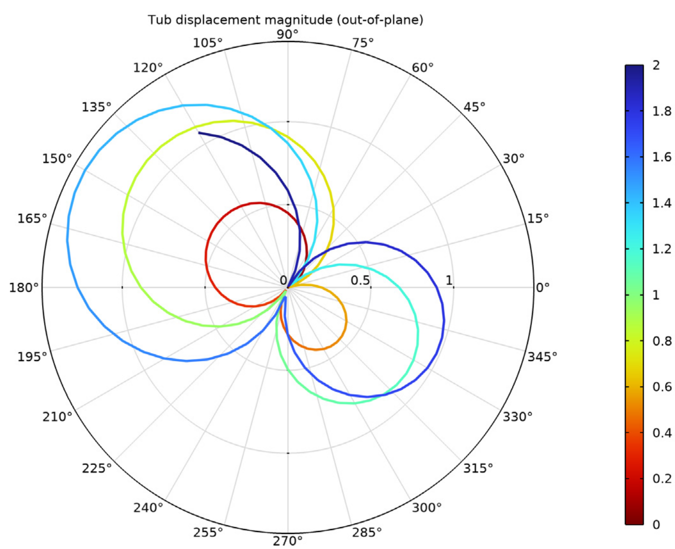


Figure 5: Out-of-plane (y) displacement magnitude of the tub with the position of unbalanced clothes.

Tub rotation about all three axes with the rotation of the drum is shown in Figure 6 below.

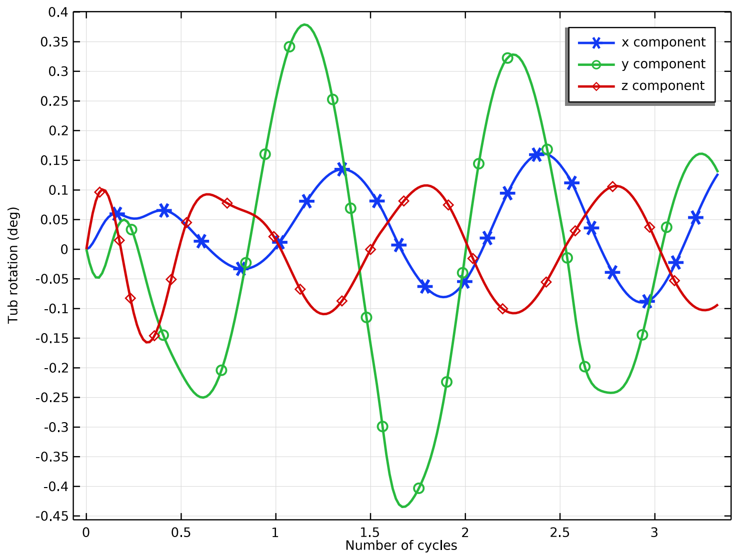


Figure 6: Tub rotation about all three axes.

Figure 7 shows the extension in the front and back stabilizing springs with the rotation of the drum.

The relative displacement between the piston and the cylinder for the different struts is shown in Figure 8. The energy dissipation in the struts with the rotation of the drum is plotted in Figure 9.

The deformation of the housing in the vertical direction with the drum rotation at the locations where mountings are attached is shown in Figure 10.

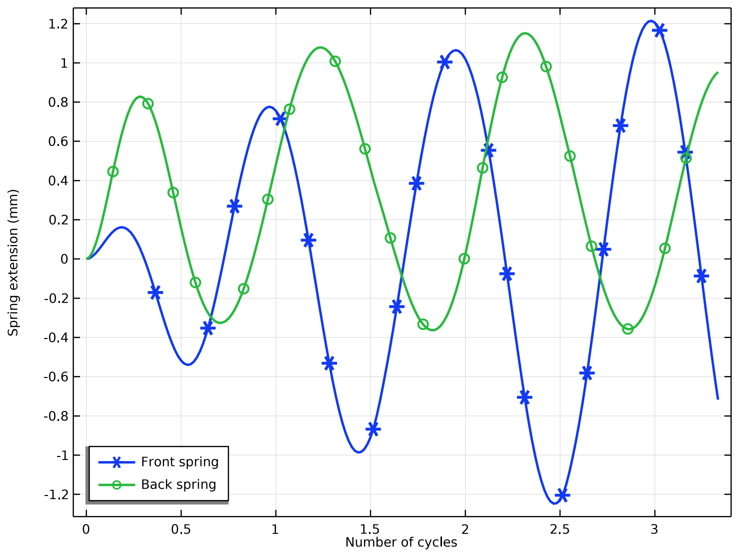


Figure 7: Extension of the stabilizing springs.

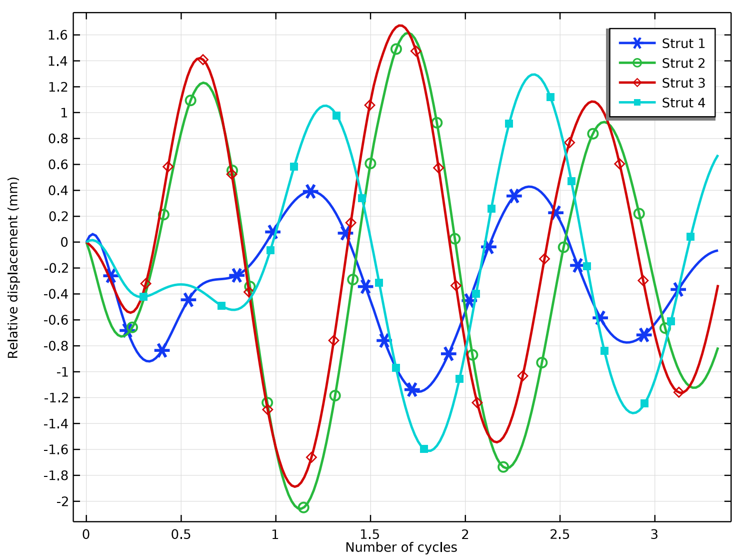


Figure 8: Relative displacement between the piston and the cylinder of different struts.

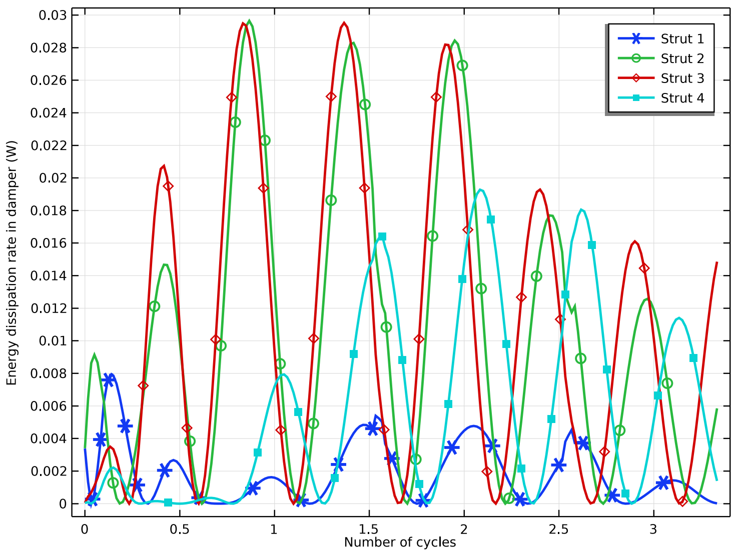


Figure 9: Energy dissipation rate in different struts.

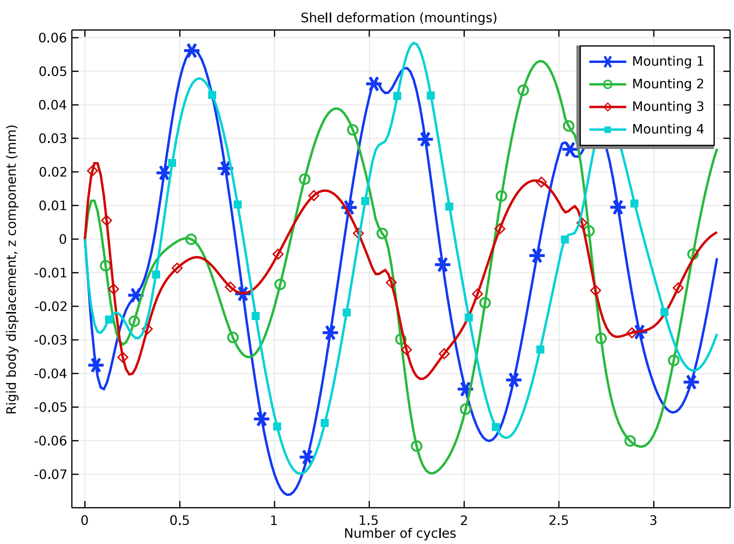


Figure 10: Vertical deformation of the housing at the mounting locations.

Figure 11 shows the deformation of the housing at a point on the right side wall.

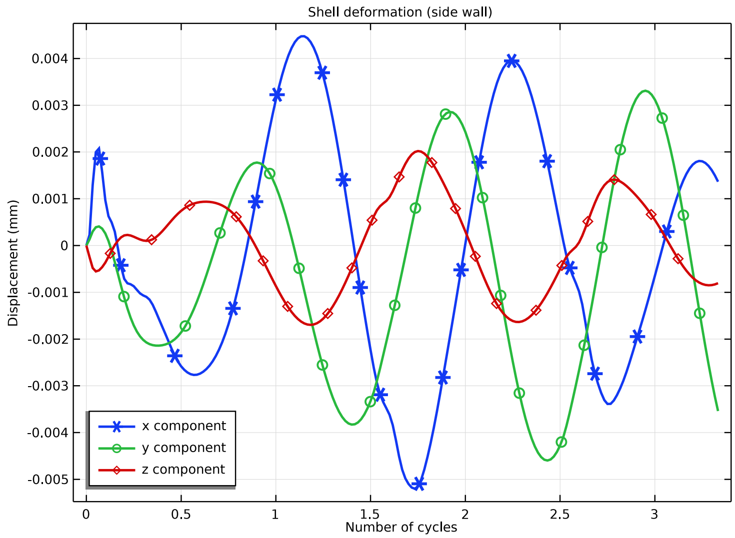


Figure 11: Deformation of the housing in different directions at a point on the side wall.

Notes About the COMSOL Implementation

- Use the **Mass and Moment of Inertia** subnode of the **Rigid Material** node to enter the inertia properties given at a certain point.
- The connections set up in the model and the details of the system DOF and constraints can be seen in the **Joints Summary** and **Rigid Body DOF Summary** sections of the **Multibody Dynamics** node.
- Use the **Attachment** boundary condition in the Shell interface to establish the connection to the solid objects through joints and springs.
- The numbering used in the model for the piston, cylinder, mounting, and base is such that 1, 2, 3, and 4 corresponds to front-left, front-right, back-right, and back-left locations respectively.

Application Library path: `Multibody_Dynamics_Module/Machinery_and_Robotics/washing_machine_vibration`

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 **3D**.

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

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Eigenfrequency**.

8 Click **Done**.

GLOBAL DEFINITIONS

Start by importing the model parameters and geometry.

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 `washing_machine_vibration_parameters.txt`.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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 `washing_machine_vibration.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 Clear the **Create pairs** check box.

5 In the **Home** toolbar, click **Build All**.

DEFINITIONS

Hiding the front panels of the geometry will make it more convenient to set up the model.

View 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **View 1**.

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

3 Select the **Wireframe rendering** check box.

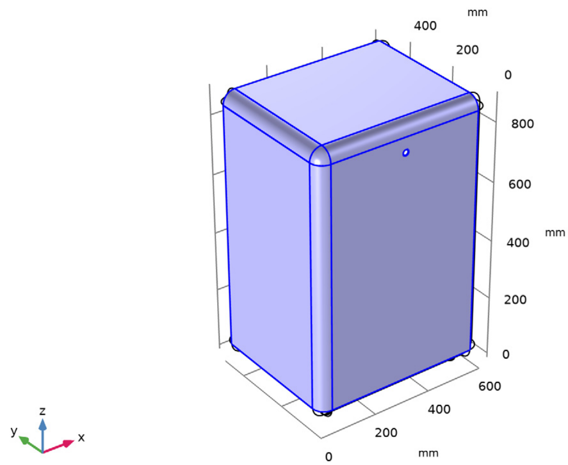
Hide for Geometry 1

1 Right-click **View 1** and choose **Hide for Geometry**.

2 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 1, 2, 5, 6, 10, 12, and 14 only.



Define some selections to be used later.

Housing

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Housing` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 36 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

5 Select the **Group by continuous tangent** check box.

Strut axis 1

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 228 only.

5 In the **Label** text field, type `Strut axis 1`.

Strut axis 2

1 Right-click **Strut axis 1** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Strut axis 2` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Clear Selection**.

4 Select Edge 444 only.

Strut axis 3

1 Right-click **Strut axis 2** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Strut axis 3` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Clear Selection**.

4 Select Edge 484 only.

Strut axis 4

1 Right-click **Strut axis 3** and choose **Duplicate**.

2 In the **Settings** window for **Explicit**, type `Strut axis 4` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Clear Selection**.

4 Select Edge 268 only.

Define various components of the washing machine assembly.

MULTIBODY DYNAMICS (MBD)

Clothes

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Multibody Dynamics (mbd)** and choose **Material Models>Rigid Material**.

2 In the **Settings** window for **Rigid Material**, type `Clothes` in the **Label** text field.

3 Select Domain 13 only.

Set the density of the selected domain to zero. Use **Mass and Moment of Inertia** subnode instead to specify the mass of the domain.

4 Locate the **Density** section. From the ρ list, choose **User defined**.

Mass and Moment of Inertia 1

1 In the **Physics** toolbar, click **Attributes** and choose **Mass and Moment of Inertia**.

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 `Mc`.

Use the **Applied Force** subnode to account for the gravitational force.

Clothes

In the **Model Builder** window, click **Clothes**.

Applied Force 1

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Force**.
2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.
3 Specify the **F** vector as

0	x
0	y
-Mc*g_const	z

Drum

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.
2 In the **Settings** window for **Rigid Material**, type `Drum` in the **Label** text field.
3 Select Domain 8 only.

   Select **Force initial values** in the **Consistent initialization** list, to enforce the parts, which are connected to the drum such as clothes, to rotate with the same angular speed.

4 Locate the **Initial Values** section. From the list, choose **Locally defined**.
5 From the **Consistent initialization** list, choose **Force initial values**.
6 Select the **Translation along first axis** check box.
7 Select the **Translation along second axis** check box.
8 Select the **Translation along third axis** check box.
9 Select the **Total rotation** check box.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Center of Rotation** section.
3 From the list, choose **Centroid of selected entities**.
4 From the **Entity level** list, choose **Point**.
5 Locate the **Initial Values: Rotational** section. Specify the ω vector as

0	x
omega	y
0	z

Center of Rotation: Point 1

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

2 Select Points 141 and 151 only.

Tub

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.

2 In the **Settings** window for **Rigid Material**, type `Tub` in the **Label** text field.

3 Select Domains 5, 15, and 16 only.

Motor

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.

2 In the **Settings** window for **Rigid Material**, type `Motor` in the **Label** text field.

3 Select Domain 14 only.

4 Locate the **Density** section. From the ρ list, choose **User defined**.

Mass and Moment of Inertia 1

1 In the **Physics** toolbar, click **Attributes** and choose **Mass and Moment of Inertia**.

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 `Mm`.

4 In the $\mathbf{I}$ text field, type `Im`.

Piston 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.

2 In the **Settings** window for **Rigid Material**, type `Piston 1` in the **Label** text field.

3 Select Domain 11 only.

Rigid Materials

Similarly create more components by duplicating **Piston 1** and resetting the inputs using the information given in the table below.

Name	Selection
Piston 2	17
Piston 3	18
Piston 4	12
Cylinder 1	9
Cylinder 2	19

Name	Selection
Cylinder 3	20
Cylinder 4	10
Mounting 1	6
Mounting 2	21
Mounting 3	22
Mounting 4	7
Base	1-4

Base

In the **Model Builder** window, click **Base**.

Fixed Constraint 1

In the **Physics** toolbar, click **Attributes** and choose **Fixed Constraint**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho0	kg/m³	Basic

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Aluminum**.

4 Click **Add to Component** in the window toolbar.

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

MATERIALS

Aluminum (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **Housing**.

SHELL [HOUSING]

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

2 In the **Settings** window for **Shell**, type `Shell [Housing]` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Housing**.

Use linear elements for the shell to reduce the computation time.

4 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

As the amplitude of vibration is going to be small in the shell, select the **Force linear strains** option in the **Linear Elastic Material 1** node. This will reduce the analysis time.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell [Housing] (shell)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d_0 text field, type `0.001[m]`.

Linear Elastic Material 1

1 In the **Model Builder** window, click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.

3 Select the **Geometrically linear formulation** check box.

Define the attachments in the Shell interface to couple it with the Multibody Dynamics interface.

Front spring

1 In the **Physics** toolbar, click **Edges** and choose **Attachment**.

2 In the **Settings** window for **Attachment**, type `Front spring` in the **Label** text field.

3 Select Edges 45, 46, 49, and 50 only.

Shell Attachments

Similarly, create more attachments by duplicating **Front Spring** and resetting the inputs using the information given in the table below:

Name	Selection (edge)
Back Spring	`47, 48, 51, 52`
Mounting 1	`37, 38, 41, 42`
Mounting 2	`53, 54, 57, 58`
Mounting 3	`55, 56, 59, 60`
Mounting 4	`39, 40, 43, 44`
Base 1	`29, 30, 33, 34`
Base 2	`61, 62, 65, 66`
Base 3	`63, 64, 67, 68`
Base 4	`31, 32, 35, 36`

Use the **Spring-Damper** node to model the stabilizing springs.

MULTIBODY DYNAMICS (MBD)

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

Housing-tub (front)

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

2 In the **Settings** window for **Spring-Damper**, type `Housing-tub (front)` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Front spring (shell)**.

4 Locate the **Spring-Damper** section. In the *k* text field, type `kt`.

Destination Point 1

1 In the **Model Builder** window, expand the **Housing-tub (front)** node, then click **Destination Point 1**.

2 Select Points 246 and 248 only.

Housing-tub (back)

1 In the **Model Builder** window, right-click **Housing-tub (front)** and choose **Duplicate**.

2 In the **Settings** window for **Spring-Damper**, type `Housing-tub (back)` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Back spring (shell)**.

Destination Point 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Housing-tub (back)** node, then click **Destination Point 1**.

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

3 Click **Clear Selection**.

4 Select Points 258 and 260 only.

Define the connection between the drum and the tub using a **Hinge Joint**.

Tub-drum

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

2 In the **Settings** window for **Hinge Joint**, type `Tub-drum` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Tub**.

4 From the **Destination** list, choose **Drum**.

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

6 Locate the **Axis of Joint** section. Specify the $\mathbf{e}_0$ vector as

0	x
1	y
0	z

Center of Joint: Point 1

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

2 Select Points 141 and 151 only.

Tub-drum

In the **Model Builder** window, click **Tub-drum**.

Prescribed Motion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Prescribed Motion**.

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

3 From the **Prescribed motion through** list, choose **Angular velocity**.

4 In the ω_p text field, type `omega`.

Define **Fixed Joints** between motor-tub and drum-clothes.

Motor-tub

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

2 In the **Settings** window for **Fixed Joint**, type `Motor-tub` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Motor**.

4 From the **Destination** list, choose **Tub**.

Center of Joint: Boundary 1

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

2 Select Boundaries 168 and 171 only.

Drum-clothes

1 In the **Model Builder** window, right-click **Motor-tub** and choose **Duplicate**.

2 In the **Settings** window for **Fixed Joint**, type `Drum-clothes` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Drum**.

4 From the **Destination** list, choose **Clothes**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Drum-clothes** node, then click **Center of Joint: Boundary 1**.

2 In the **Settings** window for **Center of Joint: Boundary**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 157 only.

Model the struts using **Prismatic Joints**.

Cylinder 1-piston 1

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

2 In the **Settings** window for **Prismatic Joint**, type `Cylinder 1-piston 1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Cylinder 1**.

4 From the **Destination** list, choose **Piston 1**.

5 Locate the **Axis of Joint** section. From the list, choose **Select a parallel edge**.

Center of Joint: Boundary 1

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

2 Select Boundary 144 only.

Joint Axis 1

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

2 In the **Settings** window for **Joint Axis**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Strut axis 1**.

Cylinder 1-piston 1

In the **Model Builder** window, click **Cylinder 1-piston 1**.

Spring and Damper 1

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

2 In the **Settings** window for **Spring and Damper**, locate the **Spring and Damper: Translational** section.

3 In the k_u text field, type `ks`.

4 In the c_u text field, type `cs`.

Prismatic Joints

Create the other three prismatic joints by duplicating **Cylinder 1-piston 1** and resetting the inputs using the information given in the table below:

Name	Source	Destination	Center of Joint (boundary)	Joint axis selection (edge)
Cylinder 2-piston 2	Cylinder 2	Piston 2	195	Strut axis 2
Cylinder 3-piston 3	Cylinder 3	Piston 3	201	Strut axis 3
Cylinder 4-piston 4	Cylinder 4	Piston 4	150	Strut axis 4

Tub-piston 1

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

2 In the **Settings** window for **Fixed Joint**, type `Tub-piston 1` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Tub**.

4 From the **Destination** list, choose **Piston 1**.

5 Locate the **Axis of Joint** section. From the list, choose **Select a parallel edge**.

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

Center of Joint: Boundary 1

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

2 Select Boundary 149 only.

Joint Axis 1

1. In the **Model Builder** window, click **Joint Axis 1**.
2. In the **Settings** window for **Joint Axis**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Strut axis 1**.

Joint Elasticity 1

1. In the **Model Builder** window, click **Joint Elasticity 1**.
2. In the **Settings** window for **Joint Elasticity**, locate the **Elastic Degrees of Freedom** section.
3. Clear the **First axis** check box.
4. Clear the **Second axis** check box.
5. Clear the **Third axis** check box.
6. Locate the **Spring** section. In the $\mathbf{k}_\theta$ text field, type `kbr`.
7. Locate the **Viscous Damping** section. In the $\mathbf{c}_\theta$ text field, type `cbr`.

Fixed Joints (with Elasticity)

Similarly create seven additional bushings by duplicating **Tub 1-piston 1** and resetting the inputs using the information given in the table below:

Name	Source	Destination	Center of Joint (boundary)	Joint axis selection (edge)
Tub-piston 2	Tub	Piston 2	190	Strut axis 2
Tub-piston 3	Tub	Piston 3	196	Strut axis 3
Tub-piston 4	Tub	Piston 4	155	Strut axis 4
Cylinder 1-mounting 1	Cylinder 1	Mounting 1	74	Strut axis 1
Cylinder 2-mounting 2	Cylinder 2	Mounting 2	247	Strut axis 2
Cylinder 3-mounting 3	Cylinder 3	Mounting 3	257	Strut axis 3
Cylinder 4-mounting 4	Cylinder 4	Mounting 4	84	Strut axis 4

Cylinder 2-piston 2

1. In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** click **Cylinder 2-piston 2**.
2. In the **Settings** window for **Prismatic Joint**, locate the **Axis of Joint** section.
3. Select the **Reverse direction** check box.

Cylinder 3-piston 3

1 In the **Model Builder** window, click **Cylinder 3-piston 3**.

2 In the **Settings** window for **Prismatic Joint**, locate the **Axis of Joint** section.

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

Tub-piston 2

1 In the **Model Builder** window, click **Tub-piston 2**.

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

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

Tub-piston 3

1 In the **Model Builder** window, click **Tub-piston 3**.

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

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

Cylinder 2-mounting 2

1 In the **Model Builder** window, click **Cylinder 2-mounting 2**.

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

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

Cylinder 3-mounting 3

1 In the **Model Builder** window, click **Cylinder 3-mounting 3**.

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

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

Mounting 1-housing

1 In the **Model Builder** window, right-click **Cylinder 4-mounting 4** and choose **Duplicate**.

2 In the **Settings** window for **Fixed Joint**, type `Mounting 1-housing` in the **Label** text field.

3 Locate the **Attachment Selection** section. From the **Source** list, choose **Mounting 1**.

4 From the **Destination** list, choose **Mounting 1 (shell)**.

5 Locate the **Axis of Joint** section. From the list, choose **Specify direction**.

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

7 Locate the **Center of Joint** section. From the list, choose **Centroid of destination**.

Fixed Joints (Rigid)

Similarly, create seven additional connections by duplicating **Mounting 1-housing** and resetting the inputs using the information given in the table below:

Name	Source	Destination
Mounting 2-housing	Mounting 2	Mounting 2(shell)
Mounting 3-housing	Mounting 3	Mounting 3(shell)
Mounting 4-housing	Mounting 4	Mounting 4(shell)
Housing-base 1	Base 1 (shell)	Base
Housing-base 2	Base 2 (shell)	Base
Housing-base 3	Base 3 (shell)	Base
Housing-base 4	Base 4 (shell)	Base

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 **Fine**.

4 Click **Build All**.

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 In the **Search for eigenfrequencies around** text field, type `2`.

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

5 In the **Settings** window for **Study**, locate the **Study Settings** section.

6 Clear the **Generate default plots** check box.

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

Use the instructions below to reproduce the mode shape shown in Figure 2.

RESULTS

Mode Shape

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape` in the **Label** text field.

Surface 1

1 Right-click **Mode Shape** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.

5 Click **OK**.

Deformation 1

Right-click **Surface 1** and choose **Deformation**.

Surface 2

1 In the **Model Builder** window, right-click **Mode Shape** and choose **Surface**.

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

3 In the **Expression** text field, type `shell.disp`.

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

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

9 From the **Color table transformation** list, choose **Reverse**.

Deformation 1

1 Right-click **Surface 2** and choose **Deformation**.

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

3 In the **X component** text field, type `u2`.

4 In the **Y component** text field, type `v2`.

5 In the **Z component** text field, type `w2`.

Mode Shape

1 In the **Model Builder** window, under **Results** click **Mode Shape**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **2.6542+0.66165i**.

4 In the **Mode Shape** toolbar, click **Plot**.

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

Change the eigenfrequency to obtain the eigenmode as shown in Figure 3.

6 From the **Eigenfrequency (Hz)** list, choose **3.7056+0.68906i**.

7 In the **Mode Shape** toolbar, click **Plot**.

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

Before performing a transient analysis, define additional variables to use them in 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
`uin_tub`	`sqrt(mbd.rd3.u^2+ mbd.rd3.w^2)`	m	`Tub displacement magnitude (in-plane)`
`uout_tub`	`abs(mbd.rd3.v)`	m	`Tub displacement magnitude (out-of-plane)`
`th_drum`	`mbd.hgj1.th`	rad	`Drum rotation`
`n_cycle`	`th_drum/360[deg]`		`Number of cycles`

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> Time Dependent**.

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

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

STUDY 2

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,0.01,2)`.

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

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

5 Clear the **Generate default plots** check box.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

Modify the default solver settings in such a way that both physics are solved together.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 Click to expand the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

6 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** and choose **Fully Coupled**.

7 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

8 From the **Jacobian update** list, choose **On every iteration**.

9 In the **Maximum number of iterations** text field, type `15`.

10 In the **Model Builder** window, under **Study 2>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver 1** click **Advanced**.

11 In the **Settings** window for **Advanced**, locate the **General** section.

12 From the **Null-space function** list, choose **Orthonormal**.

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

Duplicate the mode shape plot and change the dataset to plot the displacement obtained in the transient analysis.

RESULTS

Displacement

1 In the **Model Builder** window, right-click **Mode Shape** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

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

4 From the **Time (s)** list, choose **1**.

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

6 In the **Model Builder** window, expand the **Displacement** node.

Deformation 1

1 In the **Model Builder** window, expand the **Results>Displacement>Surface 1** node, then click **Deformation 1**.

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

3 Select the **Scale factor** check box. In the associated text field, type `1`.

Deformation 1

1 In the **Model Builder** window, expand the **Results>Displacement>Surface 2** node, then click **Deformation 1**.

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

3 Select the **Scale factor** check box. In the associated text field, type `1`.

4 In the **Displacement** toolbar, click **Plot**.

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

Follow the instructions below to get the polar plots of the tub displacement shown in Figure 4 and Figure 5.

Tub displacement magnitude (in-plane)

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

2 In the **Settings** window for **Polar Plot Group**, type `Tub displacement magnitude (in-plane)` in the **Label** text field.

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

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

Global 1

1 Right-click **Tub displacement magnitude (in-plane)** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **r-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>uin_tub - Tub displacement magnitude (in-plane) - m**.

3 Locate the θ **Angle Data** section. From the **Parameter** list, choose **Expression**.

4 Click **Replace Expression** in the upper-right corner of the θ **Angle Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>th_drum - Drum rotation - rad**.

5 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

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

Color Expression 1

1 Right-click **Global 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Solver>t - Time - s**.

3 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

4 In the **Tub displacement magnitude (in-plane)** toolbar, click **Plot**.

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

Tub displacement magnitude (out-of-plane)

1 In the **Model Builder** window, right-click **Tub displacement magnitude (in-plane)** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Tub displacement magnitude (out-of-plane)` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Tub displacement magnitude (out-of-plane)** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **r-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>uout_tub - Tub displacement magnitude (out-of-plane) - m**.

3 In the **Tub displacement magnitude (out-of-plane)** toolbar, click **Plot**.

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

Use the instructions below to generate the plot of the tub rotation and stabilizing springs extension shown in Figure 6 and Figure 7 respectively.

Tub rotation

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

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

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

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

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Number of cycles`.

7 Select the **y-axis label** check box. In the associated text field, type `Tub rotation (deg)`.

Global 1

1 Right-click **Tub rotation** 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)>Multibody Dynamics>Rigid domains>Tub>Rigid body rotation (spatial frame) - rad>mbd.rd3.thx - Rigid body rotation, x component**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Rigid domains>Tub>Rigid body rotation (spatial frame) - rad>mbd.rd3.thy - Rigid body rotation, y component**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Rigid domains>Tub>Rigid body rotation (spatial frame) - rad>mbd.rd3.thz - Rigid body rotation, z component**.

Change the units to degrees from radians.

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

Expression	Unit	Description
`mbd.rd3.thx`	`deg`	`Rigid body rotation, x component`
`mbd.rd3.thy`	`deg`	`Rigid body rotation, y component`
`mbd.rd3.thz`	`deg`	`Rigid body rotation, z component`

6 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>n_cycle - Number of cycles**.

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 From the **Positioning** list, choose **Interpolated**.

10 In the **Number** text field, type `20`.

11 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
`x component`
`y component`
`z component`

13 In the **Tub rotation** toolbar, click **Plot**.

To plot the strut displacement and the energy dissipation rate in the struts as shown in Figure 8 and Figure 9 respectively, follow the instructions below.

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

Stabilizing spring extension

1 In the **Model Builder** window, right-click **Tub rotation** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Stabilizing spring extension` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **y-axis label** check box.

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

Global 1

1 In the **Model Builder** window, expand the **Stabilizing spring extension** 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)>Multibody Dynamics>Spring-Dampers>Housing-tub (front)>mbd.spd1.dl - Spring extension - m**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Spring-Dampers>Housing-tub (back) >mbd.spd2.dl - Spring extension - m**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Front spring`
`Back spring`

5 In the **Stabilizing spring extension** toolbar, click **Plot**.

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

Strut displacement

1 In the **Model Builder** window, right-click **Stabilizing spring extension** and choose **Duplicate**.

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

3 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Global 1

1 In the **Model Builder** window, expand the **Strut displacement** 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)>Multibody Dynamics>Prismatic joints>Cylinder 1-piston 1>mbd.prj1.u - Relative displacement - m**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Prismatic joints>Cylinder 2-piston 2>mbd.prj2.u - Relative displacement - m**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Prismatic joints>Cylinder 3-piston 3>mbd.prj3.u - Relative displacement - m**.

5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Multibody Dynamics>Prismatic joints>Cylinder 4-piston 4>mbd.prj4.u - Relative displacement - m**.

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

Legends
`Strut 1`
`Strut 2`
`Strut 3`
`Strut 4`

7 In the **Strut displacement** toolbar, click **Plot**.

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

Energy dissipation rate

1 In the **Model Builder** window, right-click **Strut displacement** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Energy dissipation rate` in the **Label** text field.

Global 1

1 In the **Model Builder** window, expand the **Energy dissipation rate** 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
`mbd.prj1.Qdamper`	`W`	`Energy dissipation rate in damper`
`mbd.prj2.Qdamper`	`W`	`Energy dissipation rate in damper`
`mbd.prj3.Qdamper`	`W`	`Energy dissipation rate in damper`
`mbd.prj4.Qdamper`	`W`	`Energy dissipation rate in damper`

4 In the **Energy dissipation rate** toolbar, click **Plot**.

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

Use the instructions below to plot the shell deformation components at the mountings and the side wall as shown in Figure 10 and Figure 11 respectively.

Shell deformation (mountings)

1 In the **Model Builder** window, right-click **Energy dissipation rate** and choose **Duplicate**.

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

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 In the **Model Builder** window, expand the **Shell deformation (mountings)** 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
`shell.att3.w`	`mm`	`Rigid body displacement, z component`
`shell.att4.w`	`mm`	`Rigid body displacement, z component`
`shell.att5.w`	`mm`	`Rigid body displacement, z component`
`shell.att6.w`	`mm`	`Rigid body displacement, z component`

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Mounting 1`
`Mounting 2`
`Mounting 3`
`Mounting 4`

5 In the **Shell deformation (mountings)** toolbar, click **Plot**.

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

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.

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

4 Locate the **Point Data** section. In the **X** text field, type `600`.

5 In the **Y** text field, type `250`.

6 In the **Z** text field, type `450`.

7 From the **Snapping** list, choose **Snap to closest boundary**.

8 Click **Plot**.

Shell deformation (side wall)

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

2 In the **Settings** window for **1D Plot Group**, type `Shell deformation (side wall)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 Locate the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Number of cycles`.

7 Select the **y-axis label** check box. In the associated text field, type `Displacement (mm)`.

8 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Point Graph 1

1 Right-click **Shell deformation (side wall)** and choose **Point Graph**.

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

3 In the **Expression** text field, type `u2`.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>n_cycle - Number of cycles**.

5 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **Interpolated**.

8 In the **Number** text field, type `20`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
x component

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type v2.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
y component

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

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

3 In the **Expression** text field, type w2.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
z component

5 In the **Shell deformation (side wall)** toolbar, click **Plot**.

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