Vibration in a Washing Machine Assembly

Introduction

Vibration and noise, due to non-uniform 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.

Model Definition

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

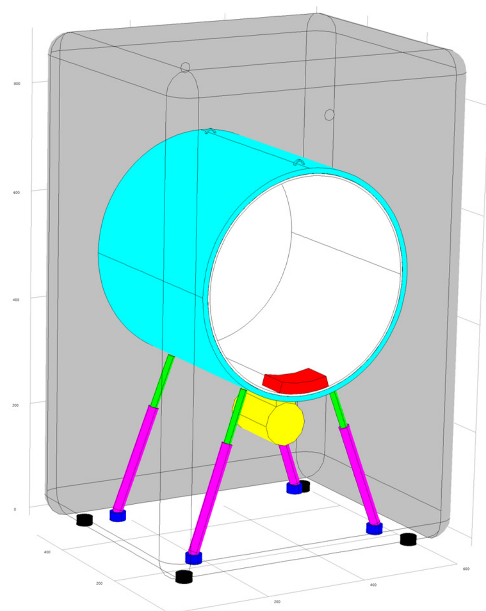


Figure 1: Model geometry (top, bottom, front, and left panels of the housing are hidden)

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:

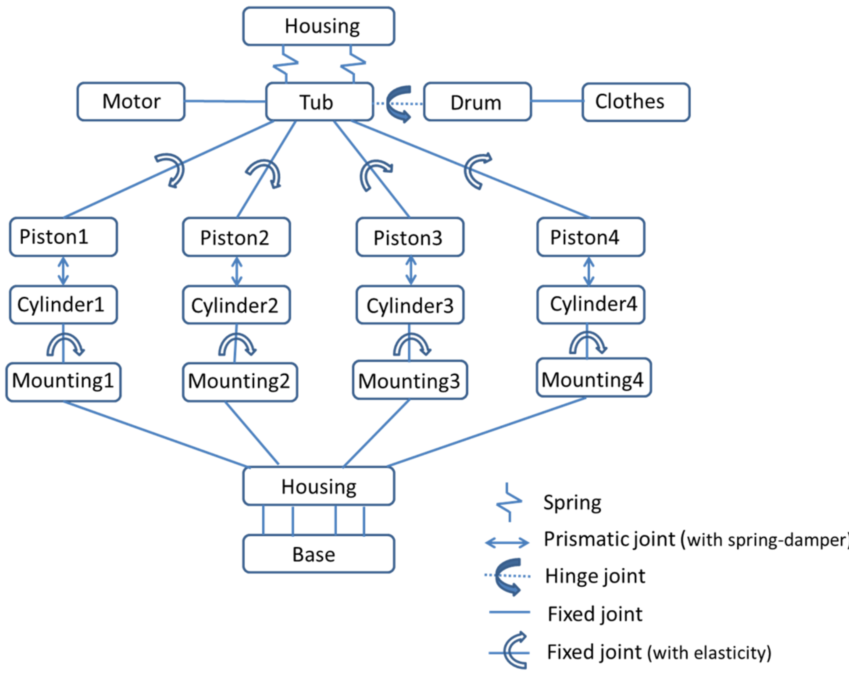
- The housing is modeled using flexible shell elements.
- The other parts are modeled by rigid solids.
- The clothes do not move relative to the drum.

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 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 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.

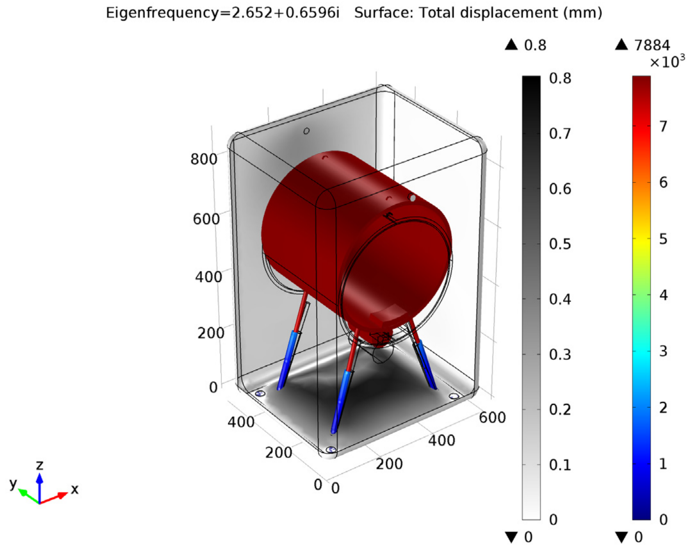


Figure 2: One of the eigenmodes of the washing machine assembly (with tub translating along vertical axis).

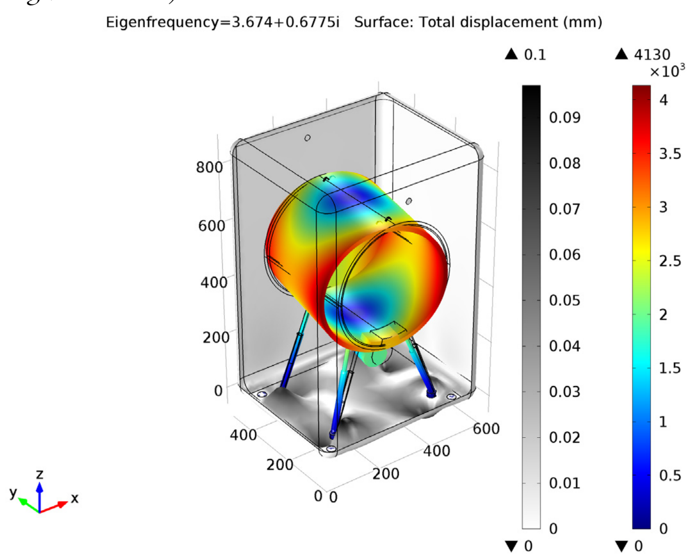


Figure 3: One of the eigenmodes of the washing machine assembly (with tub rotating about vertical axis).

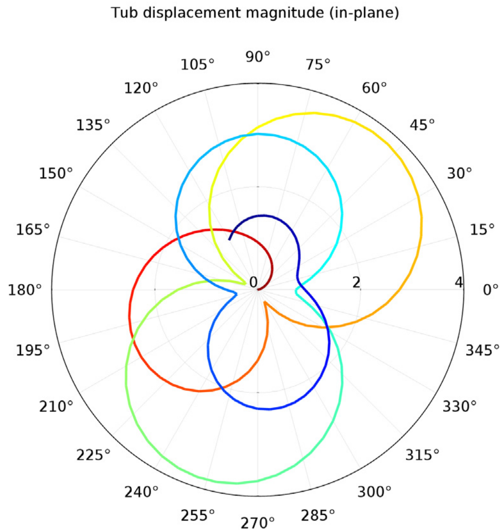


Figure 4: In-plane (x-z) displacement magnitude of the tub with the position of unbalanced clothes.

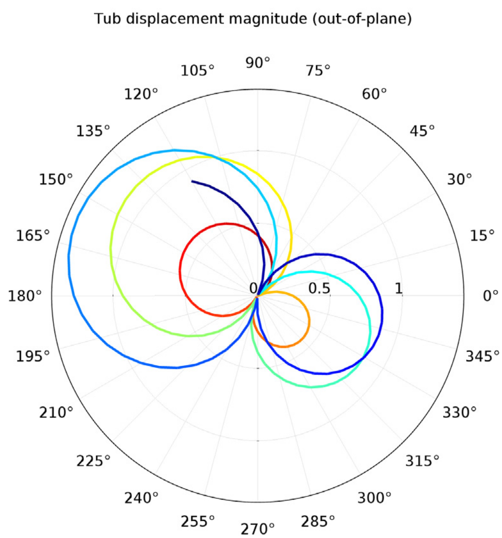


Figure 5: Out-of-plane (y) displacement magnitude of the tub with the position of unbalanced clothes.

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

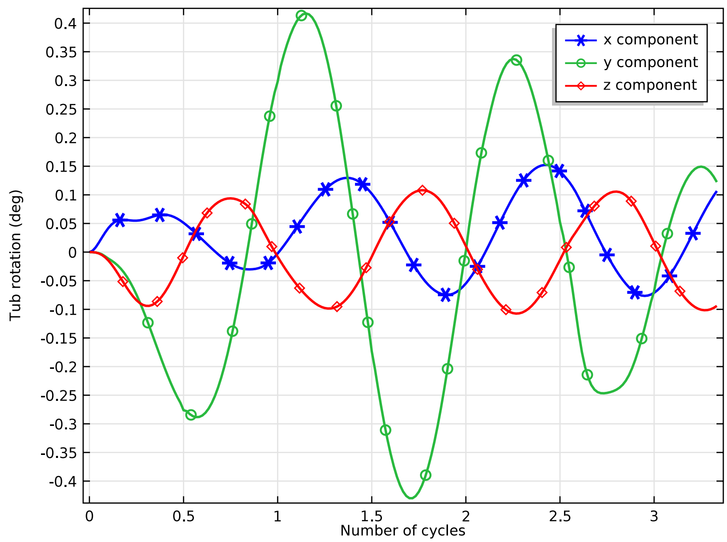


Figure 6: Tub rotation about all three axes.

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

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

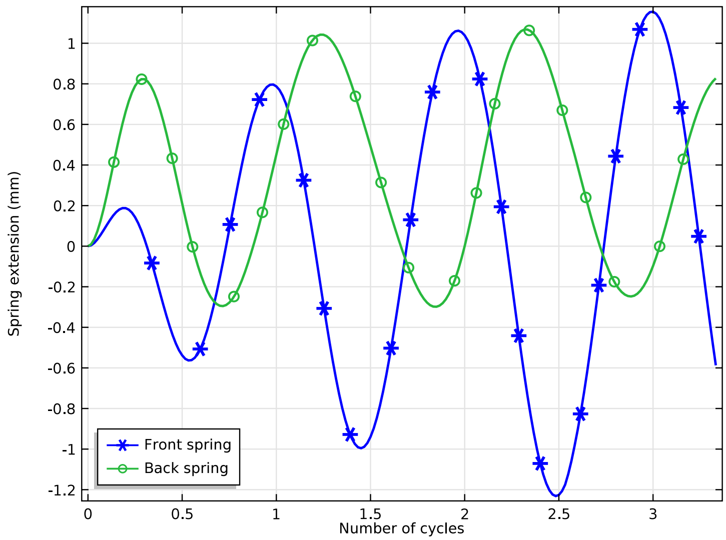


Figure 7: Extension of stabilizing springs.

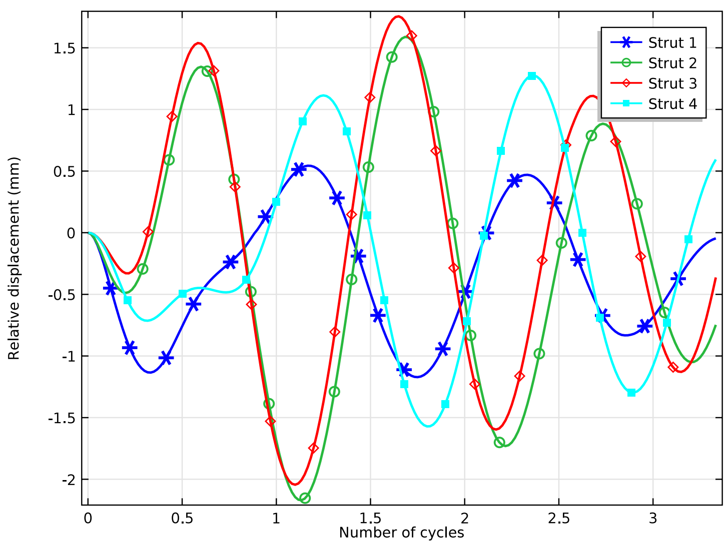


Figure 8: Relative displacement between piston and cylinder of different struts.

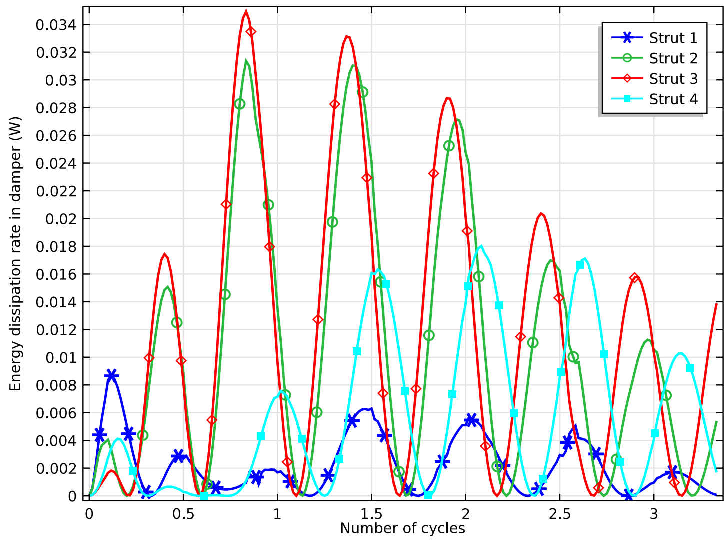


Figure 9: Energy dissipation rate in different struts.

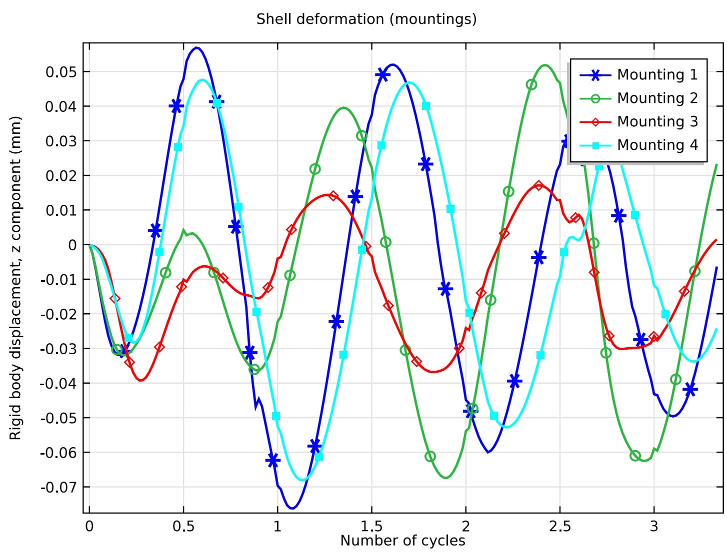


Figure 10: Vertical deformation of the housing at the mounting locations.

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

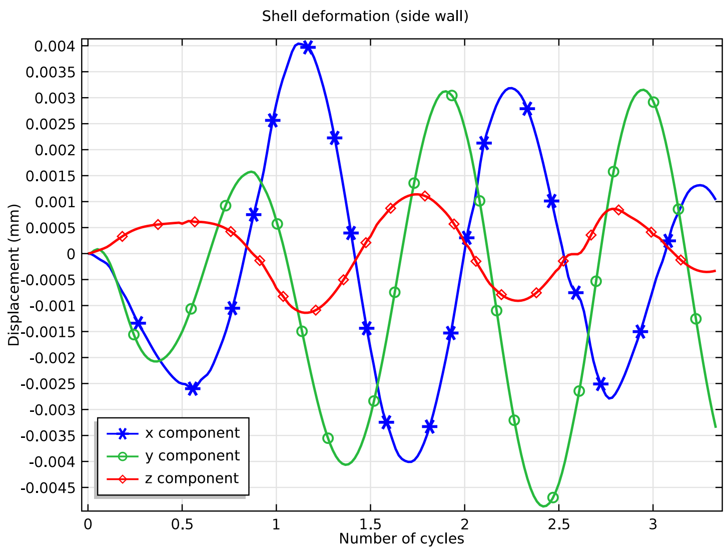


Figure 11: Deformation of the housing in different directions at a point on the side wall.

Figure 12 shows the normal acceleration of the housing at a point in the middle of the right side wall. Normal acceleration is a measure of noise emitted by the side walls.

The frequency spectrum of this acceleration can be seen in Figure 13. The frequency spectrum shows that the frequency content of side wall vibration is mainly in the range 0-30 Hz. This is interesting, since the excitation frequency, corresponding to the rotational speed of the drum or unbalanced mass is smaller, approximately 1.67 Hz.

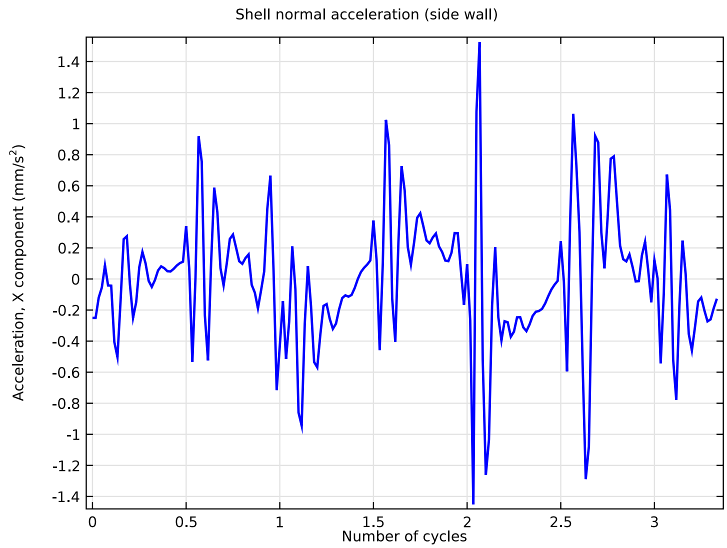


Figure 12: Normal acceleration of the housing, at a point on the side wall.

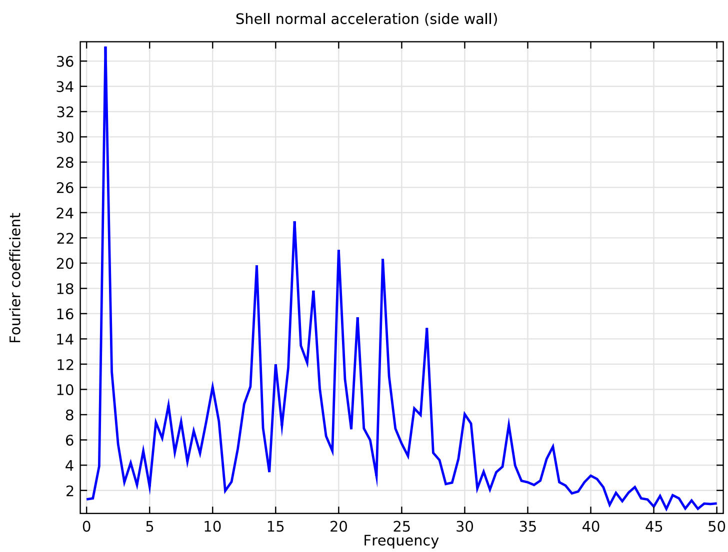


Figure 13: Frequency spectrum of normal acceleration of the housing.

Notes About the COMSOL Implementation

- Use **Mass and Moment of Inertia** subnode of the **Rigid Domain** 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 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.

Model Library path: `Multibody_Dynamics_Module/Machinery_and_Robotics/washing_machine_vibrations`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 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 **Preset Studies for Selected Physics Interfaces>Eigenfrequency**.

8 Click **Done**.

DEFINITIONS

Start by importing the model parameters and geometry.

Parameters

1 On the **Model** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the model's Model Library 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 On the **Model** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Model Library 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 On the **Model** 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.
4 On the **3D view** toolbar, click **Hide Geometry Objects**.
5 In the **Settings** window for Hide Geometry Objects, locate the **Selection** section.
6 From the **Geometric entity level** list, choose **Boundary**.
7 On the object **fin**, select Boundaries 1, 2, 5, 6, 10, 12, and 14 only.

Define some selections to be used later.

Explicit 1

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

Explicit 2

1 On 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 1.2

1 Right-click **Component 1 (comp1)>Definitions>Explicit 2** 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 2.2

1 Right-click **Component 1 (comp1)>Definitions>Strut axis 1.2** 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 3.2

1 Right-click **Component 1 (comp1)>Definitions>Strut axis 2.2** 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)

Rigid Domain 1

1 On the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.

2 In the **Settings** window for Rigid Domain, 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 **Rigid Domain** section. From the ρ list, choose **User defined**.

Mass and Moment of Inertia 1

1 On 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.

Applied Force 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** right-click **Clothes** 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

Rigid Domain 2

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

   Select the **Force initial values** in **Consistent initialization**, 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, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Drum** node, then 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, expand the **Initial Values 1** node, then click **Center of Rotation: Point 1**.
2 Select Points 141 and 151 only.

Rigid Domain 3

1 On the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.
2 In the **Settings** window for Rigid Domain, type `Tub` in the **Label** text field.
3 Select Domains 5, 15, and 16 only.

Rigid Domain 4

1 On the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.

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

3 Select Domain 14 only.

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

Mass and Moment of Inertia 1

1 On 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 **I** text field, type `Im`.

Rigid Domain 5

1 On the **Physics** toolbar, click **Domains** and choose **Rigid Domain**.

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

3 Select Domain 11 only.

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

TABLE 2: RIGID DOMAINS

Name	**Selection**
Piston 2	17
Piston 3	18
Piston 4	12
Cylinder 1	9
Cylinder 2	19
Cylinder 3	20
Cylinder 4	10
Mounting 1	6
Mounting 2	21
Mounting 3	22
Mounting 4	7
Base	1-4

Right-click **Component 1 (comp1)>Multibody Dynamics (mbd)>Base** 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	Name	Value	Unit	Property group
Density	rho	`rho0`	kg/m³	Basic

4 On the **Model** toolbar, click **More Windows** and choose **Add Material**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Aluminum**.

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

MATERIALS

Aluminum (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat2)**.

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

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

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

5 On the **Model** toolbar, click **Add Material** to close the **Add Material** window.

SHELL (SHELL)

On the **Physics** toolbar, click **Multibody Dynamics (mbd)** and choose **Shell (shell)**.

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**.

4 Locate the **Thickness** section. In the *d* text field, type `0.001[m]`.

Use linear elements for shell to reduce the computation time.

5 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

6 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 **Linear Elastic Material 1** node. This will reduce the analysis time.

SHELL [HOUSING] (SHELL)

On the **Physics** toolbar, click **Shell (shell)** and choose **Shell [Housing] (shell)**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell [Housing] (shell)** click **Linear Elastic Material 1**.

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

3 Select the **Force linear strains** check box.

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

Attachment 1

1 On 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.

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

TABLE 3: SHELL ATTACHMENTS

Name	**Selection (edge)**
Back Spring	47, 48, 51, and 52
Mounting 1	37, 38, 41, and 42
Mounting 2	53, 54, 57, and 58
Mounting 3	55, 56, 59, and 60
Mounting 4	39, 40, 43, and 44
Base 1	29, 30, 33, and 34

TABLE 3: SHELL ATTACHMENTS

Name	Selection (edge)
Base 2	61, 62, 65, and 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)

Spring-Damper 1

1 On 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 **Component 1 (comp1)>Multibody Dynamics (mbd)>Housing-tub (front)** node, then click **Destination Point 1**.

2 Select Points 246 and 248 only.

Housing-tub (front) 1

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

Hinge Joint 1

1 On 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

7 Locate the **Joint Forces and Moments** section. From the list, choose **Do not compute**.

Center of Joint: Point 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Tub-drum** node, then click **Center of Joint: Point 1**.

2 Select Points 141 and 151 only.

Prescribed Motion 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)** right-click **Tub-drum** 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.

Fixed Joint 1

1 On 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**.

5 Locate the **Joint Forces and Moments** section. From the list, choose **Do not compute**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Motor-tub** node, then click **Center of Joint: Boundary 1**.
2 Select Boundaries 168 and 171 only.

Motor-tub 1

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

Prismatic Joint 1

1 On 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**.
6 Locate the **Joint Forces and Moments** section. From the list, choose **Do not compute**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Cylinder 1-piston 1** node, then click **Center of Joint: Boundary 1**.
2 Select Boundary 144 only.

Joint Axis 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Cylinder 1-piston 1** 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**.

Spring and Damper 1

1 On 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`.

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

TABLE 4: PRISMATIC JOINTS

Name	Source	Destination	Center of Joint selection (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

Fixed Joint 3

1 On 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**.

7 Locate the **Joint Forces and Moments** section. From the list, choose **Do not compute**.

Center of Joint: Boundary 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Multibody Dynamics (mbd)>Tub-piston 1** node, then click **Center of Joint: Boundary 1**.

2 Select Boundary 149 only.

Joint Axis 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Tub-piston 1** 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, under **Component 1 (comp1)>Multibody Dynamics (mbd)>Tub-piston 1** 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`.

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

TABLE 5: FIXED JOINTS (WITH ELASTICITY)

Name	Source	Destination	Center of Joint selection (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, under **Component 1 (comp1)>Multibody Dynamics (mbd)** 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, under **Component 1 (comp1)>Multibody Dynamics (mbd)** 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, under **Component 1 (comp1)>Multibody Dynamics (mbd)** 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, under **Component 1 (comp1)>Multibody Dynamics (mbd)** 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, under **Component 1 (comp1)>Multibody Dynamics (mbd)** 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.

Define the connection between mountings-housing and housing-base.

Cylinder 4-mounting 4.1

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

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

TABLE 6: FIXED JOINTS (RIGID)

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 **Mesh Settings** section.

3 From the **Element size** list, choose **Fine**.

4 Click the **Build All** button.

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 On the **Model** toolbar, click **Compute**.

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

RESULTS

3D Plot Group 1

1 On the **Model** 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.
3 Locate the **Data** section. From the **Data set** list, choose **None**.

Mode Shape

1 Right-click **Results>3D Plot Group 1** and choose **Surface**.
2 In the **Model Builder** window, under **Results>Mode Shape** right-click **Surface 1** and choose **Deformation**.
3 Right-click **Mode Shape** and choose **Surface**.
4 In the **Settings** window for Surface, locate the **Expression** section.
5 In the **Expression** text field, type `shell.disp`.
6 Click to expand the **Title** section. From the **Title type** list, choose **None**.
7 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.
8 Select the **Reverse color table** check box.
9 Right-click **Results>Mode Shape>Surface 2** and choose **Deformation**.
10 In the **Settings** window for Deformation, locate the **Expression** section.
11 In the **x component** text field, type `u2`.
12 In the **y component** text field, type `v2`.
13 In the **z component** text field, type `w2`.
14 In the **Model Builder** window, click **Mode Shape**.
15 In the **Settings** window for 3D Plot Group, locate the **Data** section.
16 From the **Data set** list, choose **Study 1/Solution 1**.
17 From the **Eigenfrequency** list, choose **2.6521+0.65943i**.
18 On the **3D plot group** toolbar, click **Plot**.
19 Click the **Zoom Extents** button on the **Graphics** toolbar.

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

20 From the **Eigenfrequency** list, choose **3.6723+0.67709i**.
21 On the **3D plot group** toolbar, click **Plot**.
22 Click the **Zoom Extents** button on the **Graphics** toolbar.

Before performing a transient analysis, define additional variables to use them in post-processing.

DEFINITIONS

Variables 1a

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]	rad	Number of cycles

ROOT

On the **Model** toolbar, click **More Windows** and choose **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.
2 Find the **Studies** subsection. In the **Select study** tree, select **Preset Studies>Time Dependent**.
3 Click **Add Study** in the window toolbar.
4 On the **Model** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
2 In the **Settings** window for Time Dependent, locate the **Study Settings** section.
3 In the **Times** text field, type `range(0,0.01,2)`.
4 Select the **Relative tolerance** check box.
5 In the associated text field, type `0.001`.
6 In the **Model Builder** window, click **Study 2**.

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

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

9 On the **Study** toolbar, click **Show Default Solver**.

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

Solution 2

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

2 In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

3 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.

4 Right-click **Study 2>Solver Configurations>Solution 2>Time-Dependent Solver 1** and choose **Fully Coupled**.

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

6 Locate the **Method and Termination** section. From the **Jacobian update** list, choose **On every iteration**.

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

8 On the **Study** toolbar, click **Compute**.

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

RESULTS

Mode Shape 1

1 In the **Model Builder** window, under **Results** 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 **Data set** list, choose **Study 2/Solution 2**.

4 From the **Time (s)** list, choose **1.0000**.

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

Displacement

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.
4 In the associated text field, type `1`.
5 In the **Model Builder** window, expand the **Results>Displacement>Surface 2** node, then click **Deformation 1**.
6 In the **Settings** window for Deformation, locate the **Scale** section.
7 Select the **Scale factor** check box.
8 In the associated text field, type `1`.
9 On the **3D plot group** toolbar, click **Plot**.
10 Click the **Zoom Extents** button on the **Graphics** toolbar.

To get the polar plots of the tub displacement as shown in Figure 4 and Figure 5, follow the instructions below.

Polar Plot Group 3

1 On the **Model** 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 **Data set** list, choose **Study 2/Solution 2**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
5 In the **Title** text area, type `Tub displacement magnitude (in-plane)`.

Tub displacement magnitude (in-plane)

1 On the **Polar plot group** toolbar, click **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 **Model>Component 1>Definitions>Variables>uin_tub - Tub displacement magnitude (in-plane)**.
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 **Model>Component 1>Definitions>Variables>th_drum - Drum rotation**.
5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** text field, type `2`.

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

7 Right-click **Results>Tub displacement magnitude (in-plane)>Global 1** and choose **Color Expression**.

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

9 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

10 On the **Polar plot group** toolbar, click **Plot**.

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

Tub displacement magnitude (in-plane) 1

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.

3 Locate the **Title** section. In the **Title** text area, type `Tub displacement magnitude (out-of-plane)`.

Tub displacement magnitude (out-of-plane)

1 In the **Model Builder** window, expand the **Results> 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 **Model>Component 1> Definitions>Variables>uout_tub - Tub displacement magnitude (out-of-plane)**.

3 On the **Polar plot group** toolbar, click **Plot**.

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

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

1D Plot Group 5

1 On the **Model** 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 **Data set** list, choose **Study 2/Solution 2**.

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

Tub rotation

1 On the **1D plot group** toolbar, click **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 **Model>Component 1> Multibody Dynamics>Rigid domains>Tub>Rigid body rotation (Spatial)> 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 **Model>Component 1>Multibody Dynamics>Rigid domains>Tub> Rigid body rotation (Spatial)>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 **Model>Component 1>Multibody Dynamics>Rigid domains>Tub> Rigid body rotation (Spatial)>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 **Model>Component 1>Definitions>Variables> n_cycle - Number of cycles**.
7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** text field, type `2`.
8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
9 In the **Number** text field, type `20`.
10 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
x component
y component
z component

12 On the **1D plot group** toolbar, click **Plot**.

13 Click the **Zoom Extents** button on the **Graphics** toolbar.

Tub rotation 1

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 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Stabilizing spring extension

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

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

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

Legends
Front spring
Back spring

5 On the **1D plot group** toolbar, click **Plot**.

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

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

Stabilizing spring extension 1

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**.

Strut displacement

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

3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Multibody Dynamics>Prismatic joints>Cylinder 2-piston 2>mbd.prj2.u - Relative displacement**.

4 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Multibody Dynamics>Prismatic joints>Cylinder 3-piston 3>mbd.prj3.u - Relative displacement**.

5 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Multibody Dynamics>Prismatic joints>Cylinder 4-piston 4>mbd.prj4.u - Relative displacement**.

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

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

7 On the **1D plot group** toolbar, click **Plot**.

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

Strut displacement 1

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.

Energy dissipation rate

1 In the **Model Builder** window, expand the **Results>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 On the **1D plot group** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.

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

Energy dissipation rate 1

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 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Shell deformation (mountings)`.

Shell deformation (mountings)

1 In the **Model Builder** window, expand the **Results>Shell deformation (mountings)** 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 **Model>Component 1>Shell [Housing]>Attachments>Mounting 1>Rigid body displacement>shell.att3.w - Rigid body displacement, z component**.
3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Shell [Housing]>Attachments>Mounting 2>Rigid body displacement>shell.att4.w - Rigid body displacement, z component**.
4 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Shell [Housing]>Attachments>Mounting 3>Rigid body displacement>shell.att5.w - Rigid body displacement, z component**.

5 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Shell [Housing]>Attachments>Mounting 4> Rigid body displacement>shell.att6.w - Rigid body displacement, z component**.

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

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

7 On the **1D plot group** toolbar, click **Plot**.

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

Data Sets

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

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

3 From the **Data set** list, choose **Study 2/Solution 2**.

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 Select the **Snap to closest boundary** check box.

8 Click the **Plot** button.

1D Plot Group 10

1 On 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 **Data set** list, choose **Cut Point 3D 1**.

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

5 In the **Title** text area, type `Shell deformation (side wall)`.

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

7 In the associated text field, type `Number of cycles`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Displacement (mm)`.

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

Shell deformation (side wall)

1 On the **1D plot group** toolbar, click **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 **Model>Component 1>Definitions>Variables> n_cycle - Number of cycles**.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** text field, type 2.

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

7 In the **Number** text field, type 20.

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

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

10 In the table, enter the following settings:

Legends
x component

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

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

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

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

Legends
y component

15 Right-click **Results>Shell deformation (side wall)>Point Graph 2** and choose **Duplicate**.

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

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

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

Legends
z component

19 On the **1D plot group** toolbar, click **Plot**.

20 Click the **Zoom Extents** button on the **Graphics** toolbar.

To plot the shell normal acceleration with time and it's frequency spectrum as shown in Figure 12 and Figure 13 respectively, follow the steps below.

1D Plot Group 11

1 On the **Model** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

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

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

5 In the **Title** text area, type `Shell normal acceleration (side wall)`.

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

7 In the associated text field, type `Number of cycles`.

Shell normal acceleration (side wall): time

1 On the **1D plot group** toolbar, click **Point Graph**.

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

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

4 In the **Unit** field, type `mm/s^2`.

5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Definitions>Variables> n_cycle - Number of cycles**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** text field, type `2`.

7 On the **1D plot group** toolbar, click **Plot**.

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

Shell normal acceleration (side wall): time 1

1 In the **Model Builder** window, right-click **Shell normal acceleration (side wall): time** and choose **Duplicate**.

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

3 Locate the **Title** section. In the **Title** text area, type `Shell normal acceleration (side wall)`.

4 Locate the **Plot Settings** section. Clear the **x-axis label** check box.

Shell normal acceleration (side wall): frequency

1 In the **Model Builder** window, expand the **Results>Shell normal acceleration (side wall): frequency** node, then click **Point Graph 1**.
2 In the **Settings** window for Point Graph, locate the **x-Axis Data** section.
3 From the **Parameter** list, choose **Frequency spectrum**.
4 On the **1D plot group** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the **Graphics** toolbar.