Capacitive Micromachined Ultrasonic Transducer with Lumped Model

Introduction

A capacitive micromachined ultrasonic transducer (CMUT) converts ultrasound to an electrical signal for high-resolution imaging applications. As a mechanical system, a CMUT can be modeled as a spring-mass system with one degree of freedom. Such model can be useful when analyzing an array of CMUTs where FEM modeling would be impractical. This tutorial demonstrates how a lumped model of a MEMS transducer can be derived from its FEM model using the Lumped Mechanical System interface and the Parameter Estimation study. The Lumped Mechanical System interface is available the Multibody Dynamics Module. The Parameter Estimation study is available in the Optimization Module.

Model Definition

This tutorial makes use of an existing FEM model and adds a lumped model defined by the Lumped Mechanical System interface.

FEM Model of the CMUT

This tutorial starts by opening the Application Library model Capacitive Micromachined Ultrasonic Transducer. You can refer to the accompanying documentation for discussions on the device geometry and operation. A Stationary study is added to compute `keff`, an estimate for the effective spring constant of the CMUT. In this test, uniform pressure is applied to the top surface of the CMUT and membrane displacement is measured. The total force is `p_max*l^2`, with `p_max` the applied pressure and `l^2` the area of the CMUT. Varying `p_max` and then plotting the total force against displacement gives the value for `keff`. The CMUT has distributed mass and stiffness so `keff` is only an estimate. The second study is a Frequency Domain, Prestressed study for the frequency response around 7.5 MHz, its natural frequency. The result is used as the reference data for subsequent studies.

LMS Model of the CMUT

The Lumped Mechanical System interface is used to define the lumped model with three the parameters: the damping coefficient `c`, the effective spring constant `keff`, and the effective mass `meff`. The value for `meff` is $2.837 \cdot 10^{-11}$ kg, as calculated based on the equation

$$m_{\mathrm{eff}} = \frac{k_{\mathrm{eff}}}{(2\pi f_0)^2}$$

Using the lumped model, a Frequency Domain study is performed for the frequency response around 7.5 MHz.

The Parameter Estimation Study

The Parameter Estimation study is used to derive the accurate values of the model parameters. In the Parameter Estimation study, a model expression is computed from the Frequency Domain study of the circuit model defined by the three parameters, which are varied to fit the experimental data (from FEM). To fit the model expression to the experimental data, Parameter Estimation minimizes an objective function defined as the error between model expression and experimental data. The output of the Parameter Estimation (PE) is the set of circuit parameters that best approximates the frequency response from FEM.

Results and Discussion

Figure 1 shows the total force (integrated over the surface of CMUT) versus maximum displacement (measured at the center). The slope of this plot gives `keff`, which is approximately 63,000 N/m.

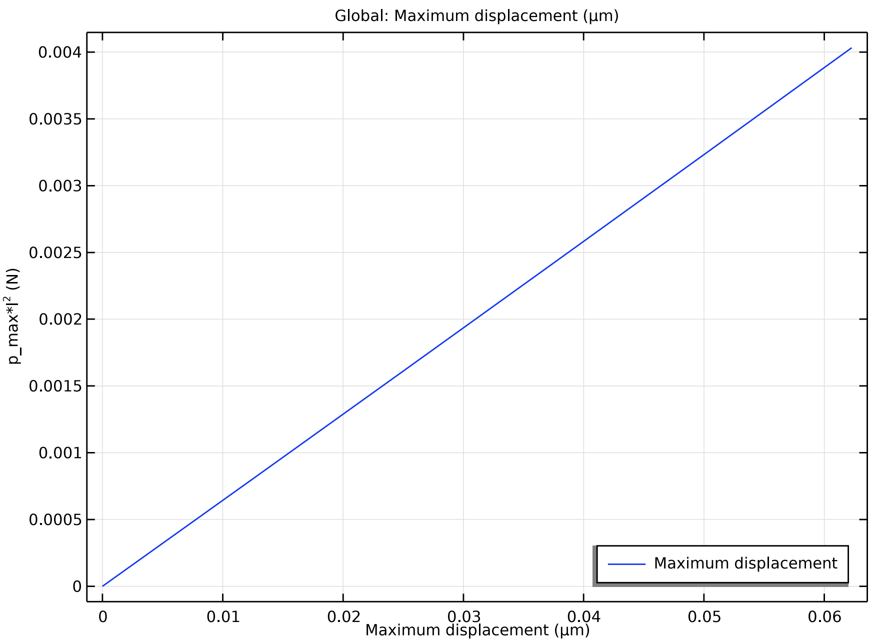


Figure 1: The plot total force versus z-displacement from the FEM simulation. The slope of the plot is the effective spring constant `keff` *in the lumped model.*

Figure 2 shows the z-displacement as a function of frequency from the Frequency Domain, Prestressed study frequency range of 7.2 to 7.8 MHz. The maximum z-displacement at the center of the device and is 30 nm at 7.5 MHz.

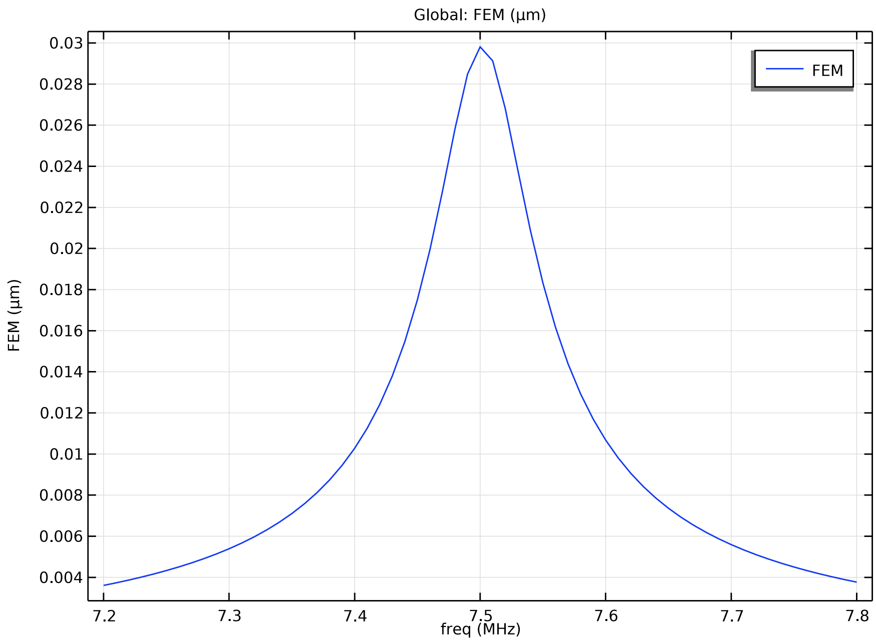


Figure 2: Plot of z-displacement versus frequency from FEM simulation.

From the previous results, the initial values of the lumped model parameters are summarized in Table 1.

TABLE 1: INITIAL VALUES OF LUMPED MODEL PARAMETERS.

Parameter	**Value**
`c`	1E-5 N·s/m
`keff`	63E3 N/m
`meff`	2.837 E-11 kg

Figure 3 compares the frequency response curves of the FEM and lumped models computed with initial values for `c`, `keff`, and `meff`.

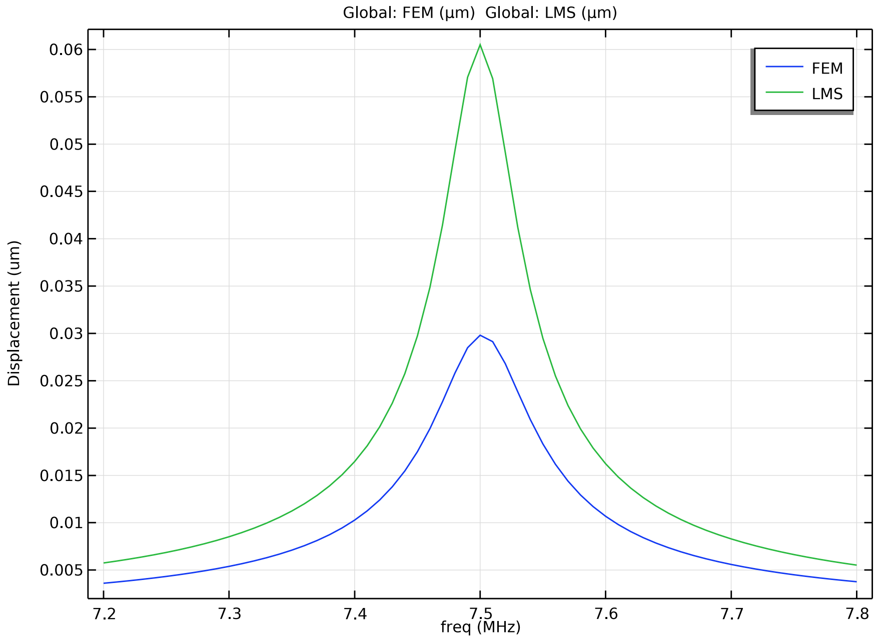


Figure 3: Plot of z-displacement versus frequency from FEM simulation and lumped model using the initial values in Table 1.

Figure 4 shows the output of the Parameter Estimation study comparison between experimental data and model expression. The fitted parameters are listed in Table 2.

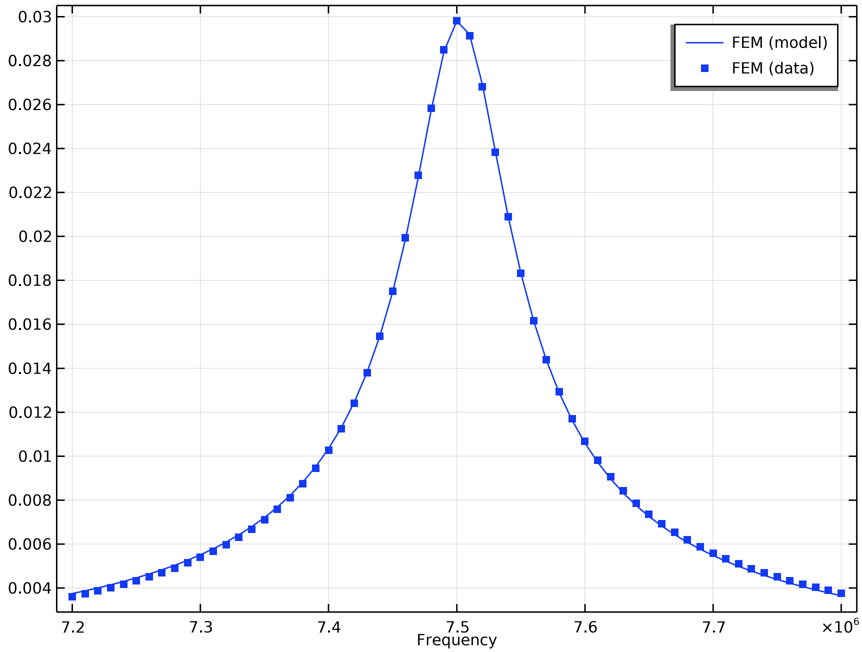


Figure 4: Plot of FEM result and lumped model using the final parameters from the Parameter Estimation study.

TABLE 2: VALUES OF LUMPED PARAMETERS COMPUTED FROM PARAMETER ESTIMATION STUDY.

Parameter	**Values**
`c`	1.986E-5 N·s/m
`keff`	98.677E3 N/m
`meff`	4.441E-11 kg

Reference

1. C. Chou, P. Chen, H. Wu, T. Hsu, and M. Li, "Piston-Shaped CMOS-MEMS CMUT Front-End Featuring Force-Displacement Transduction Enhancement," *Proceedings of the 21st International Conference on Solid-State Sensors, Actuators and Microsystems (Transducers)*, pp. 26–29, 2021.

Application Library path: `MEMS_Module/Sensors/capacitive_micromachined_ultrasonic_transducer_lumped_model`

Modeling Instructions

Start by opening the FEM model.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **MEMS Module>Sensors>capacitive_micromachined_ultrasonic_transducer** in the tree.

3 Click **Open**.

Before setting up the **Stationary** study to measure the effective spring constant, add a second **Boundary Load** feature.

COMPONENT 1 (COMP1)

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

SOLID MECHANICS (SOLID)

Boundary Load 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node.

2 Right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

-p_max	z

5 Select Boundary 22 only.

Set up a **Stationary** study to measure the effective spring constant. Use only the second **Boundary Load** by disabling the first **Boundary Load**.

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>Stationary**.

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

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

STUDY 3

Stationary - Force vs. Displacement

1 In the **Settings** window for **Stationary**, type `Stationary - Force vs. Displacement` in the **Label** text field.

2 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Boundary Load 1**.

4 Click **Disable**.

5 Clear the **Modify model configuration for study step** check box.

6 In the table, clear the **Solve for** check boxes for **Electrostatics (es)**, **Electrical Circuit (cir)**, and **Moving mesh (Component 1)**.

7 In the table, clear the **Solve for** check box for **Electromechanical Forces 1 (eme1)**.

8 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_max (Maximum pressure)	range(0,2e5,1e6)	Pa

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

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

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

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

From the results of Study 3, plot the force versus displacement.

RESULTS

1D Plot Group 5

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

Global 1

Right-click **1D Plot Group 5** and choose **Global**.

Effective Spring Constant

1 In the **Settings** window for **1D Plot Group**, type `Effective Spring Constant` in the **Label** text field.

2 Locate the **Data** section. From the **Dataset** list, choose **Study 3/Solution 4 (sol4)**.

3 Locate the **Plot Settings** section. Select the **Flip the x- and y-axes** check box.

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

Global 1

1 In the **Model Builder** window, 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
abs(minop1(w))	µm	

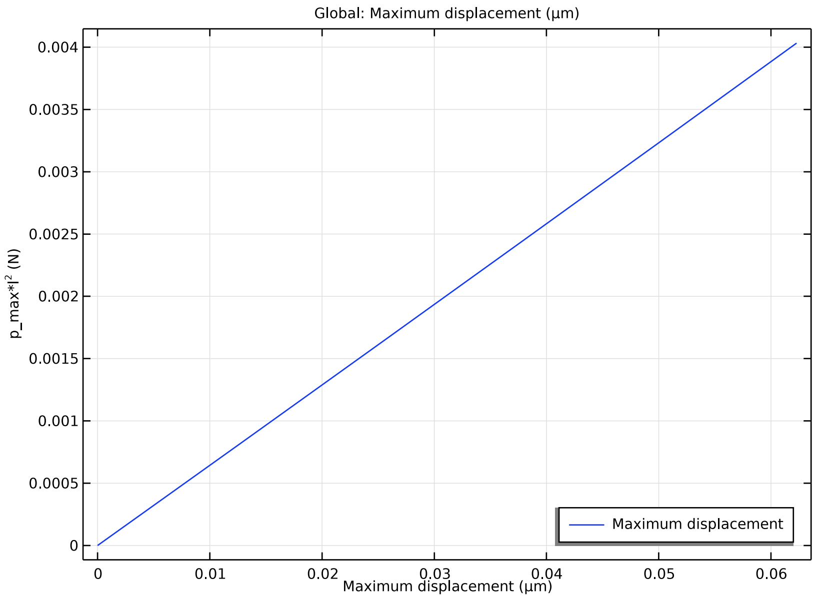
4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `p_max*l^2`.

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

Expression	Unit	Description
abs(minop1(w))	µm	Maximum displacement

7 In the **Effective Spring Constant** toolbar, click **Plot**.



From this plot, the effective spring constant is approximately 63,000 N/m. This value will be used in the lumped model.

For the next **Frequency Domain, Prestressed** study, add the **Damping** feature under **Linear Elastic Material**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

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

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

3 From the **Damping type** list, choose **Isotropic loss factor**.

4 From the η_s list, choose **User defined**. In the associated text field, type `1e-2`.

Set up a **Frequency Domain, Prestressed** study. Use only the first **Boundary Load** feature by disabling the second **Boundary Load**.

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 **Preset Studies for Selected Physics Interfaces>Solid Mechanics>Frequency Domain, Prestressed**.

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

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

STUDY 4

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** check box.

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Boundary Load 2**.

4 Right-click and choose **Disable**.

Frequency Domain Perturbation, FEM

1 In the **Model Builder** window, under **Study 4** click **Step 2: Frequency-Domain Perturbation**.

2 In the **Settings** window for **Frequency-Domain Perturbation**, type `Frequency Domain Perturbation, FEM` in the **Label** text field.

3 Locate the **Study Settings** section. From the **Frequency unit** list, choose **MHz**.

4 In the **Frequencies** text field, type `range(7.2,0.01,7.8)`.

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
p_max (Maximum pressure)	0.01[MPa]	Pa

8 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

9 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Boundary Load 2**.

10 Right-click and choose **Disable**.

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

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

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

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

From the results of Study 4, plot the frequency response of the FEM model.

RESULTS

Frequency Response

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4/Solution 5 (sol5)**.

Global 1

1 Right-click **Frequency Response** and choose **Global**.

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

3 In the table, enter the following settings:

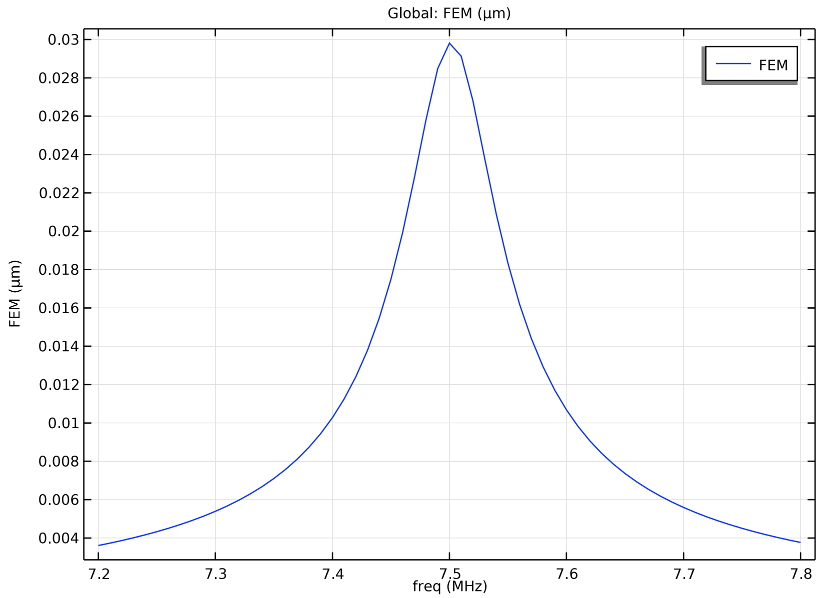
Expression	Unit	Description
minop1(w)	µm	FEM

4 Locate the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

5 Locate the **y-Axis Data** section. From the **Expression evaluated for** list, choose **RMS for total solution**.

6 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

7 In the **Frequency Response** toolbar, click **Plot**.



Define and enter the values for the following parameters. These are the initial values for the lumped model.

GLOBAL DEFINITIONS

Parameters 2

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
keff	63000[N/m]	63000 N/m	Effective spring constant
meff	keff/(2*pi* 7.5[MHz])^2	2.837E-11 kg	Effective mass
c	1e-5[N*s/m]	1E-5 N·s/m	Damping coefficient
fapp	p_max*l^2	0.0040322 N	Applied force

Add the **Lumped Mechanical System** interface and set up the lumped model.

ADD PHYSICS

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

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

3 In the tree, select **Structural Mechanics>Lumped Mechanical System (lms)**.

4 Click **Add to Component 1** in the window toolbar.

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

LUMPED MECHANICAL SYSTEM (LMS)

Spring 1 (K1)

1 Right-click **Component 1 (comp1)>Lumped Mechanical System (lms)** and choose **Spring**.

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

3 In the table, enter the following settings:

Label	Node names
p1	1
p2	0

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

Damper 1 (C1)

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

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

3 In the table, enter the following settings:

Label	Node names
p1	1
p2	0

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

Mass 1 (M1)

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

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

3 In the table, enter the following settings:

Label	Node names
p1	1
p2	2

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

Force Node 1 (frc1)

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

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

3 In the f_{p10} text field, type `fapp`.

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

Label	Node name
p1	2

For the next **Frequency Domain**, change the values of some LMS parameter.

GLOBAL DEFINITIONS

Parameters 2

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 2**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
fapp	0.01[MPa]*l^2	4.0323E-5 N	Applied force

Set up a **Frequency Domain** study for the LMS model.

ADD STUDY

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

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

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

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

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

STUDY 5

Frequency Domain, LMS

1 In the **Settings** window for **Frequency Domain**, type `Frequency Domain, LMS` in the **Label** text field.

2 Locate the **Study Settings** section. From the **Frequency unit** list, choose **MHz**.

3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electrostatics (es)**.

4 Locate the **Study Settings** section. In the **Frequencies** text field, type `ramnge(7.2,0.01,7.8)`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Solid Mechanics (solid)**, **Electrical Circuit (cir)**, and **Moving mesh (Component 1)**.

6 In the table, clear the **Solve for** check box for **Electromechanical Forces 1 (eme1)**.

7 Locate the **Study Settings** section. In the **Frequencies** text field, type `range(7.2,0.01,7.8)`.

8 In the **Model Builder** window, click **Study 5**.

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

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

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

From the results of Study 5, plot the frequency response of the LMS model.

RESULTS

Frequency Response

1 In the **Model Builder** window, under **Results** click **Frequency Response**.

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

3 Select the **x-axis label** check box.

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

Global 2 - LMS

1 Right-click **Frequency Response** and choose **Global**.

2 In the **Settings** window for **Global**, type `Global 2 - LMS` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 5/Solution 7 (sol7)**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Lumped Mechanical System>Two port components>M1>lms.M1_uRMS - Displacement, RMS (M1) - m**.

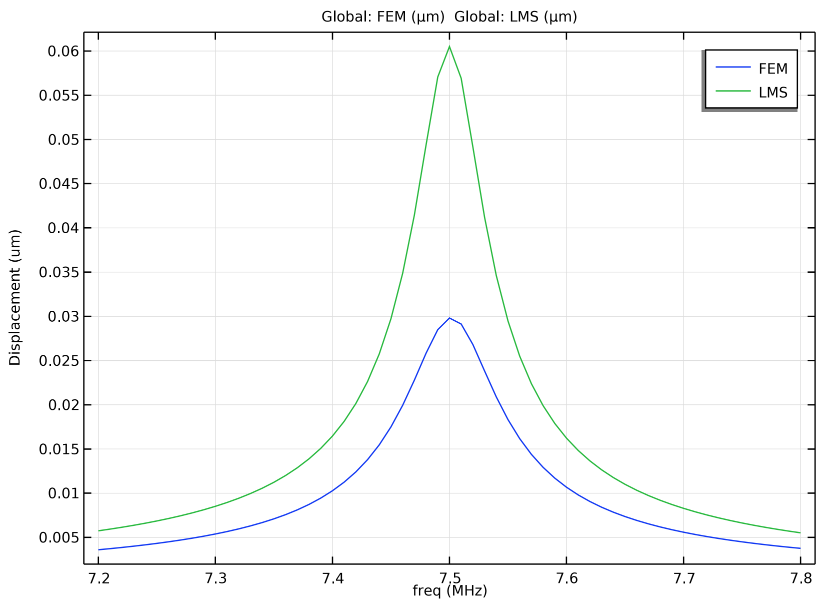
5 In the **Frequency Response** toolbar, click **Plot**.

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

Expression	Unit	Description
`lms.M1_uRMS`	µm	LMS

7 Locate the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

8 In the **Frequency Response** toolbar, click **Plot**.



Copy the result of Study 4 to a table for use as reference data in the **Parameter Estimation** study.

Global 1

In the **Model Builder** window, right-click **Global 1** and choose **Copy Plot Data to Table**.

FEM Reference Data

1 In the **Model Builder** window, under **Results>Tables** click **Table 1**.

2 In the **Settings** window for **Table**, type `FEM Reference Data` in the **Label** text field.

Set up a **Parameter Estimation** study based on the previous **Frequency Domain** study for the LMS model.

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 **Empty Study**.

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

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

STUDY 5

Step 1: Frequency Domain, LMS

In the **Model Builder** window, under **Study 5** right-click **Step 1: Frequency Domain, LMS** and choose **Copy**.

STUDY 6

In the **Model Builder** window, right-click **Study 6** and choose **Paste Frequency Domain**.

Parameter Estimation

1 In the **Study** toolbar, click **Optimization** and choose **Parameter Estimation**.

2 In the **Settings** window for **Parameter Estimation**, locate the **Experimental Data** section.

3 From the **Data source** list, choose **Result table**.

4 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
freq (MHz)	Frequency	Frequency unit=MHz

5 From the **Frequency unit** list, choose **MHz**.

6 In the table, click to select the cell at row number 2 and column number 2.

7 In the **Model expression** text field, type `comp1.lms.M1_uRMS`.

8 In the **Unit** text field, type `um`.

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

10 In the **Scale value** text field, type `1e-9`.

Select the model parameters to be included in the study. Specify their initial values, scaling, and the lower and upper bounds. For this study, a default plot will be generated automatically comparing the FEM reference data and the lumped model using the final values of the lumped parameters.

11 Locate the **Estimated Parameters** section. Click **Add**.

12 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
c (Damping coefficient)	1e-5[N*s/m]	3e-5[N*s/m]	1e-6[N*s/m]	3e-5[N*s/m]

13 Click **Add**.

14 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
meff (Effective mass)	2.882e-11[kg]	5e-11[kg]	1e-11[kg]	5e-11[kg]

15 Click **Add**.

16 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
keff (Effective spring constant)	63000[N/m]	1e5[N/m]	1e4[N/m]	1e5[N/m]

17 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **SNOPT**.

18 Find the **Solver settings** subsection. From the **Least-squares time/parameter method** list, choose **Use only least-squares data points**.

Because in the **Frequency Domain** study the variables are complex, the option for Split complex variables in real and imaginary parts must be enabled.

Solution 8 (sol8)

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

2 In the **Model Builder** window, expand the **Solution 8 (sol8)** node, then click **Compile Equations: Frequency Domain, LMS**.

3 In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.

4 Select the **Split complex variables in real and imaginary parts** check box.

5 Click **Compute**.

RESULTS

Parameter estimation