Created in COMSOL Multiphysics 5.6

Mechanical Multiport System: Elastic Wave Propagation in a Small Aluminum Plate

Introduction

In this tutorial, the vibrational behavior of a small aluminum plate with four waveguide structures is analyzed. This is an example of a structural component located in any device where elastic waves are propagating, like a smart speaker, an electric motor, or a MEMS device. The plate can be thought of as a mechanical multiport system. The model uses the port boundary condition at the inlet/outlet of the waveguide structures. The port conditions consistently capture and treat the different propagating elastic modes, like longitudinal, transverse, and torsional waves. The transmission and reflection of the various modes is characterized through the scattering matrix of the system, which is automatically computed. This allows the vibrational behavior of the component to be characterized in great detail, for example, for subsequent use in a system simulation.

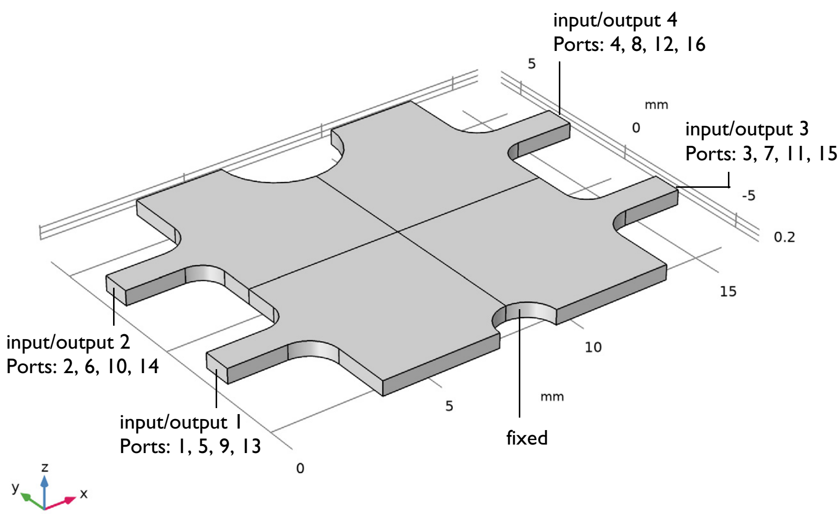


Figure 1: Model geometry layout, location of the inlet/outlets of the four waveguides, and the port numbers associated with each inlet/outlet.

Model Definition

The model consists of a small 16 mm by 12 mm by 0.5 mm aluminum piece that has four waveguide structures with an inlet/outlet (connection to other components), see Figure 1. The plate is fixed on one surface. The structure is meant to represent a small subcomponent in a device or system where vibrations are generated, for example, by a transducer. In order to control and understand transmission, such as feedback phenomena, it is necessary to understand how elastic waves (vibrations) propagate through the structure when excited. A compact lumped representation is given by the scattering or S-matrix of the system. The scattering matrix relates the incident and reflected mode amplitudes at all the inlet/outlets. Component S_{ij}, represents the amplitude of the outgoing mode at port i when the system is excited at port j. The the system is excited at 10 kHz.

In this model each inlet/outlet has four **Port** features added. Each port describes one specific propagating mode, either the longitudinal, the torsional, the transverse 1, or the transverse 2 mode. The modes are depicted in Figure 2. The scattering matrix elements are automatically computed when solving a model with a port feature; all the elements can be computed when the **Port Sweep** functionality is used. Using a combination of ports (representing all propagating modes) at an elastic waveguide outlet will also act as a perfect nonreflecting condition that is superior to a perfectly matched layer (PML) configuration.

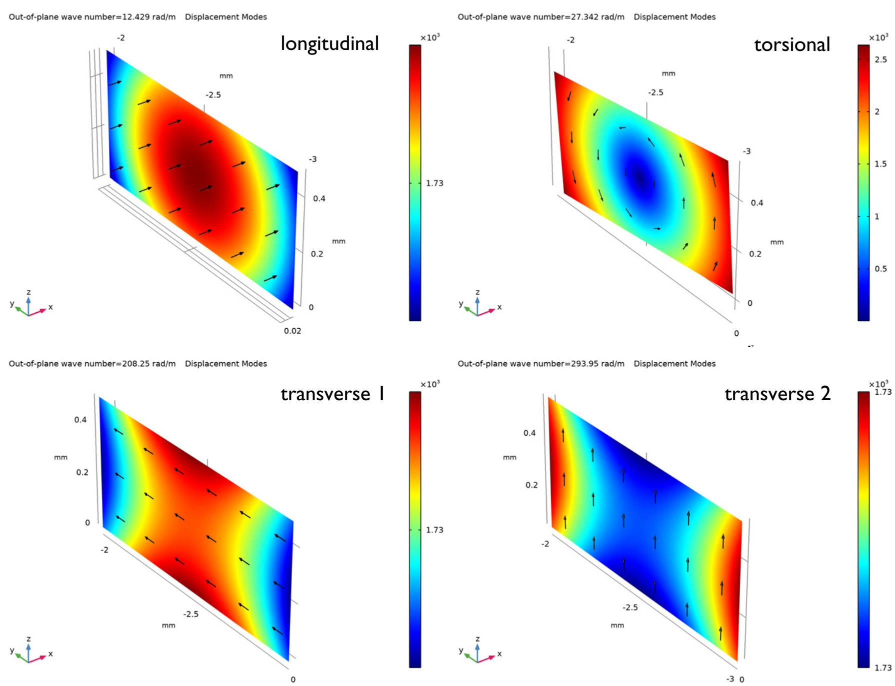


Figure 2: The four propagating modes captured by the ports located at the four inlet/outlets of the mechanical system.

Results and Discussion

The displacement of the multiport aluminum structure, when the four different modes are excited at inlet/outlet 1, is depicted in Figure 3, Figure 4, Figure 5, and Figure 6.

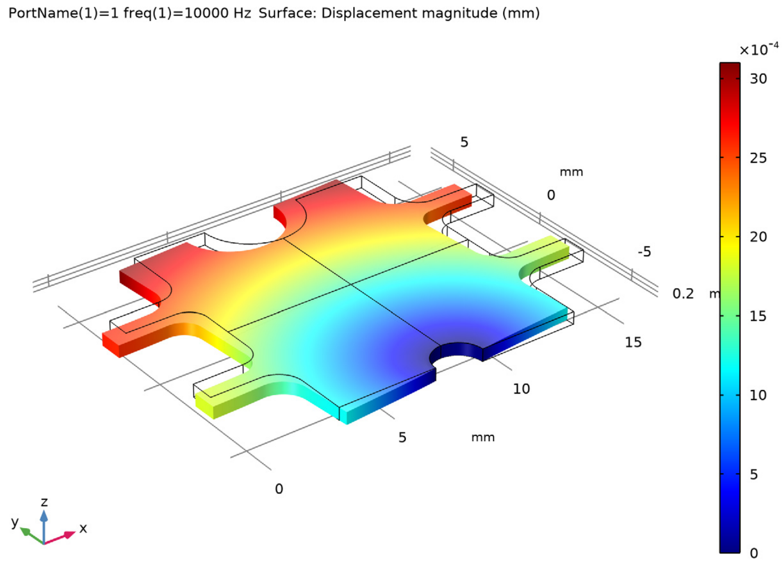


Figure 3: Displacement when the longitudinal mode is excited at the inlet/outlet 1.

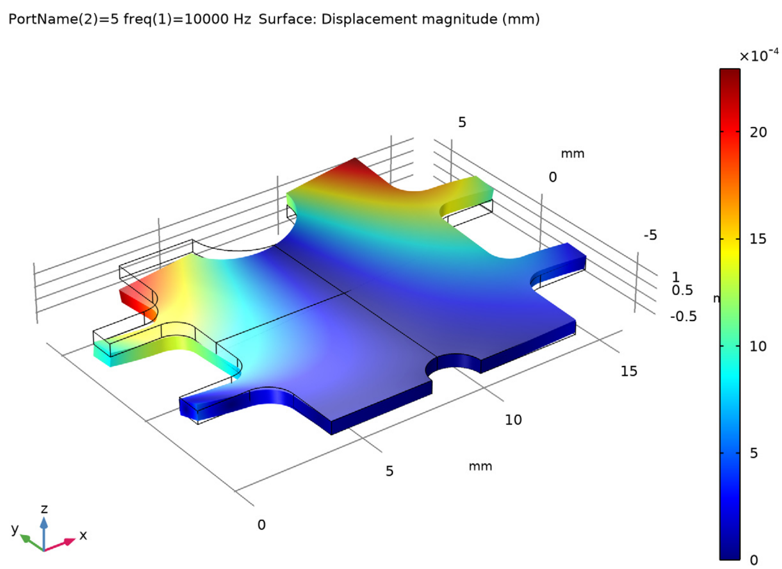


Figure 4: Displacement when the torsional mode is excited at the inlet/outlet 1.

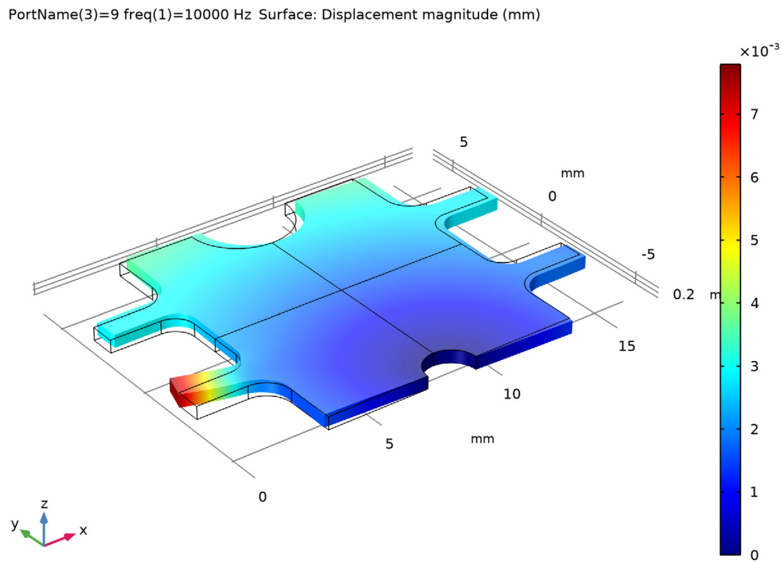


Figure 5: Displacement when the traverse 1 mode is excited at the inlet/outlet 1.

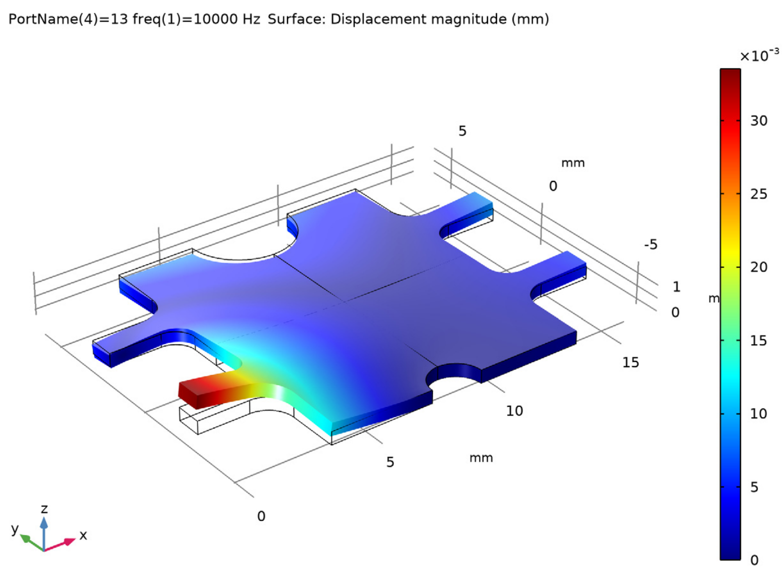


Figure 6: Displacement when the traverse 2 mode is excited at the inlet/outlet 1.

The power transmitted or reflected by the excitation of a given port can be assessed by plotting the absolute value of the scattering parameters squared $|S_{ij}|^2$. Note that the mode shapes are always scaled to unit power in the mechanical port formulation. For the acoustic ports both amplitude and power scaling options exist.

The power of the various outgoing modes at all inlet/outlets, when the system is excited with a specific port, is depicted in Figure 7. The graphs show how energy is transmitted and reflected in the system and how energy is converted from one propagating mode to another.

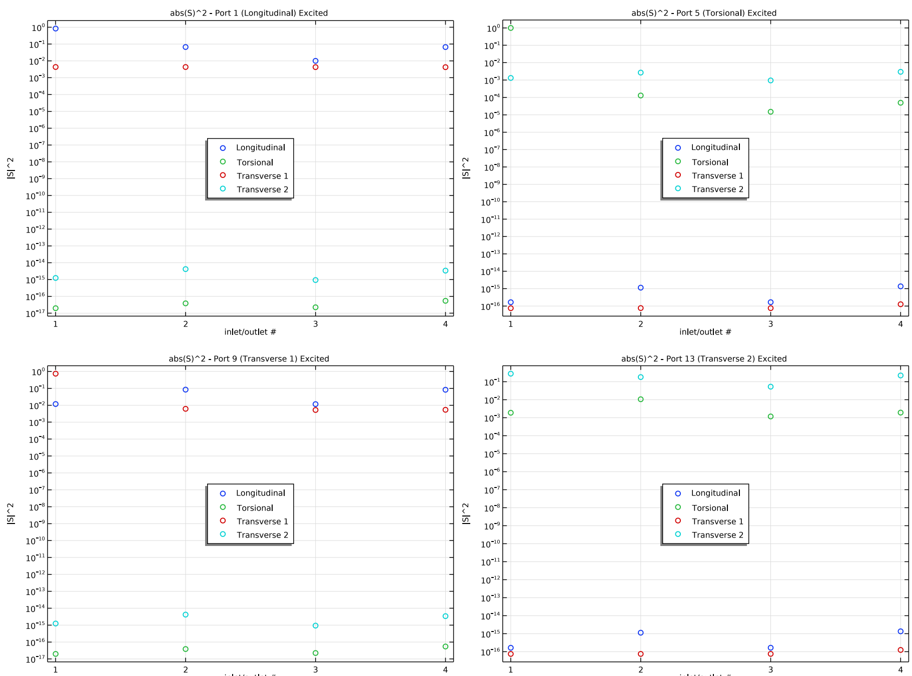


Figure 7: Power of the various outgoing modes at all inlet/outlets when the system is excited with a specific port.

Application Library path: `Acoustics_Module/Elastic_Waves/mechanical_multiport_system`

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 **Acoustics>Elastic Waves>Solid Mechanics (Elastic Waves)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Empty Study**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

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

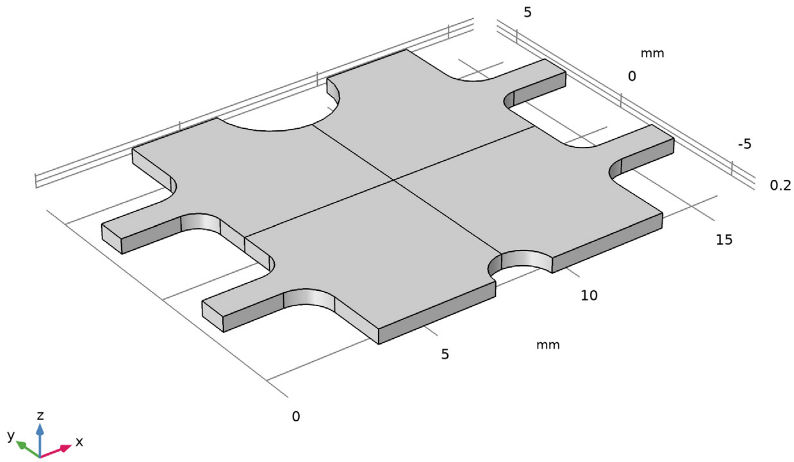
2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

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

5 Click **Import**.

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



Create selections for the inlet/outlet boundary of the four waveguide structures. On each waveguide inlet/outlet boundary several **Port** features will be added, each capturing one specific propagating mode.

DEFINITIONS

input/output 1

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `input/output 1` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 1 only.

input/output 2

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `input/output 2` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 6 only.

input/output 3

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `input/output 3` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 47 only.

input/output 4

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `input/output 4` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 48 only.

MATERIALS

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-in>Aluminum 6063-T83**.
3 Click **Add to Component** in the window toolbar.
4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.
2 Select Boundaries 23 and 25 only.

Now, proceed and add four port conditions per input/output boundary. First add the ports that represent the longitudinal modes, group them, and then duplicate them. Then add the ports that represent the torsional and the two translational modes.

Port 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2 In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **input/output 1**.

4. Locate the **Incident Mode Settings** section. From the **Incident wave excitation at this port** list, choose **Off**.

   Note that per default the first port added has the excitation turned on. We will turn it off here; you can then select any port as source and run the model. Here we will use a port sweep to excite the desired ports. When a port sweep is performed, all ports are (automatically) one by one used as the excitation source. If all ports are excited, the full scattering matrix is computed.

Port 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **input/output 2**.

Port 3

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **input/output 3**.

Port 4

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **input/output 4**.

Port 1, Port 2, Port 3, Port 4

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)**, Ctrl-click to select **Port 1**, **Port 2**, **Port 3**, and **Port 4**.
2. Right-click and choose **Group**.

Longitudinal Mode

In the **Settings** window for **Group**, type `Longitudinal Mode` in the **Label** text field.

Torsional Mode

1. Right-click **Longitudinal Mode** and choose **Duplicate**.
2. In the **Settings** window for **Group**, type `Torsional Mode` in the **Label** text field.

Translational Mode 1

1. Right-click **Torsional Mode** and choose **Duplicate**.
2. In the **Settings** window for **Group**, type `Translational Mode 1` in the **Label** text field.

Translational Mode 2

1 Right-click **Translational Mode 1** and choose **Duplicate**.

2 In the **Settings** window for **Group**, type `Translational Mode 2` in the **Label** text field.

Torsional Mode

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

Translational Mode 1

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

MESH 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)> Translational Mode 2** node, then click **Component 1 (comp1)>Mesh 1**.

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

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `1`.

5 In the **Minimum element size** text field, type `0.05`.

6 In the **Curvature factor** text field, type `0.3`.

7 In the **Resolution of narrow regions** text field, type `5`.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Delete**.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 4 and 9 only.

3 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Copy Face 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 Select Boundaries 4 and 9 only.

3 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.

4 Select the **Activate Selection** toggle button.

5 Select Boundaries 27 and 31 only.

6 Click **Build Selected**.

Swept 1

In the **Mesh** toolbar, click **Swept**.

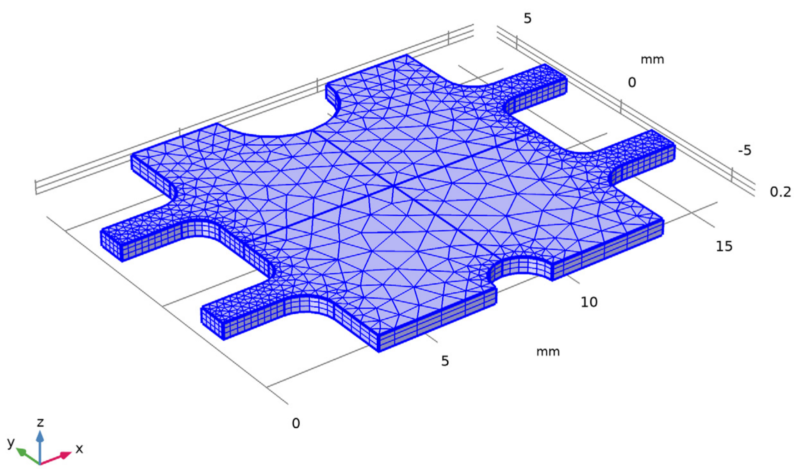
Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 3.

4 Click **Build All**.



Before solving the full model, perform a single **Boundary Mode Analysis** study to compute the propagating modes and identify their mode wave numbers. Note that the values for this specific setup are already found in the **Parameters** list.

STUDY 1 - MODES

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

2 In the **Settings** window for **Study**, type `Study 1 - Modes` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Boundary Mode Analysis

1 In the **Study** toolbar, click **Study Steps** and choose **Other>Boundary Mode Analysis**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Mode analysis frequency** text field, type `f0`.

4 Select the **Desired number of modes** check box.

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

6 Select the **Search for modes around** check box.

7 In the associated text field, type `0.8*kp`.

8 From the **Mode search method around shift** list, choose **Larger real part**.

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

RESULTS

Displacement Modes

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

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

3 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **input/output 1**.

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

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

Right-click **Displacement Modes** and choose **Surface**.

Deformation 1

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

Arrow Surface 1

1 In the **Model Builder** window, right-click **Displacement Modes** and choose **Arrow Surface**.

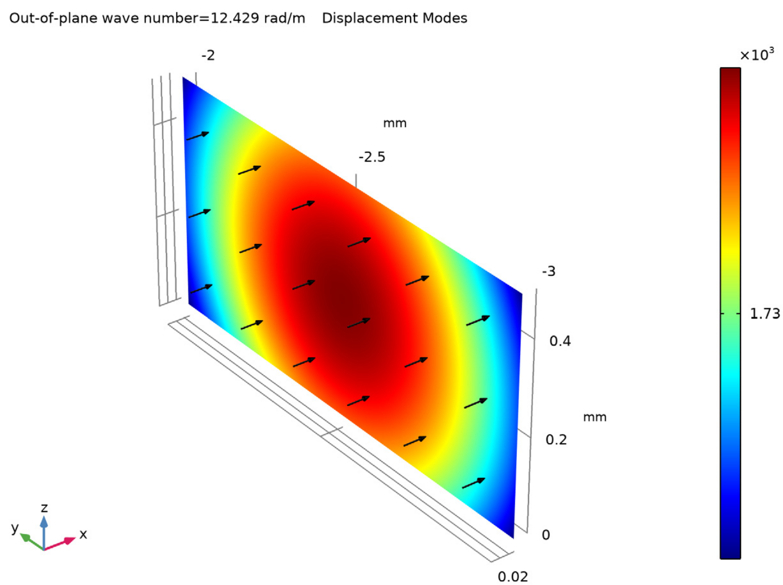
2 In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.

3 In the **Number of arrows** text field, type 20.

4 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

5 From the **Color** list, choose **Black**.

6 In the **Displacement Modes** toolbar, click **Plot**.



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

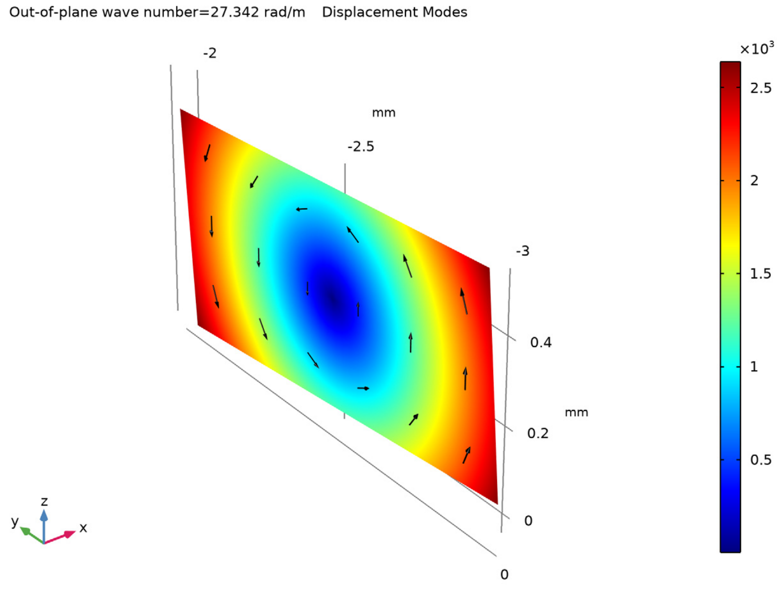
Displacement Modes

1 In the **Model Builder** window, click **Displacement Modes**.

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

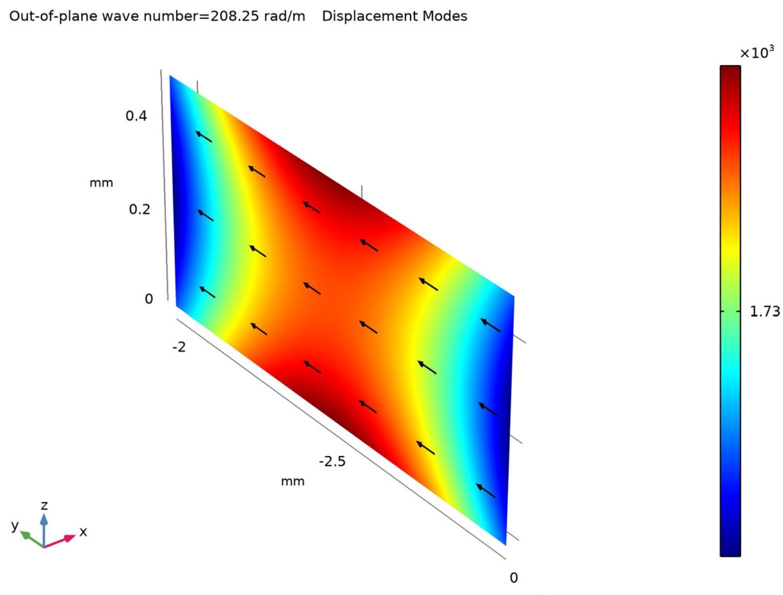
3 From the **Out-of-plane wave number (rad/m)** list, choose **27.342**.

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



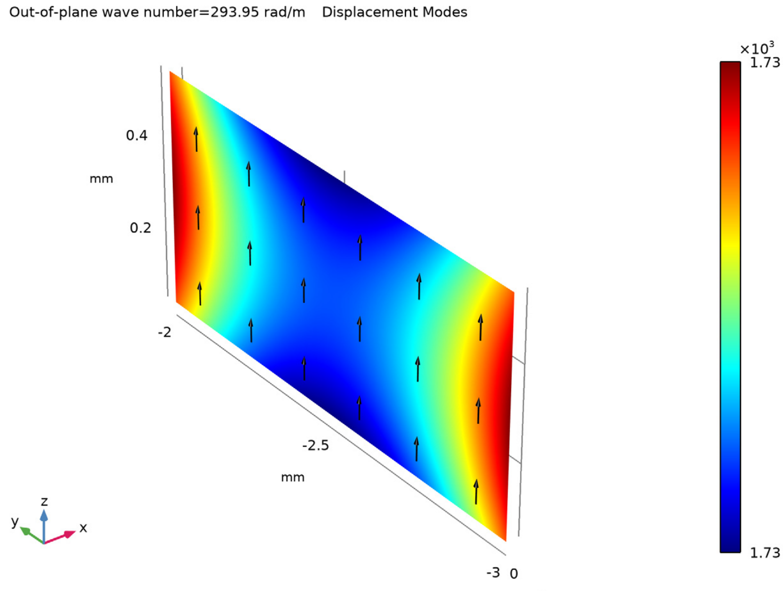
5 From the **Out-of-plane wave number (rad/m)** list, choose **208.25**.

6 In the **Displacement Modes** toolbar, click **Plot**.



7 From the **Out-of-plane wave number (rad/m)** list, choose **293.95**.

8 In the **Displacement Modes** toolbar, click **Plot**.



Proceed to do the full system analysis. This requires setting up a study with one **Boundary Mode Analysis** per port feature. In each step, it should be ensured that the correct mode is computed (longitudinal, torsional, transverse). The search uses the values of the mode wave numbers computed in the previous study (also found in the **Parameters** list).

For simplicity, one inlet/outlet boundary will be selected as the source of excitations (here the inlet/outlet 1). The desired modes will be activated one at a time using the port sweep functionality. It is activated in the **Port Sweep Settings** section and ports 1, 5, 9, and 13 are used as sources.

ROOT

In the **Home** toolbar, click **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 **Empty Study**.
3. Click **Add Study** in the window toolbar.
4. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 - FREQUENCY DOMAIN (PORT SWEEP)

1. In the **Settings** window for **Study**, type `Study 2 - Frequency Domain (Port Sweep)` in the **Label** text field.
2. Locate the **Study Settings** section. Clear the **Generate default plots** check box.
3. Clear the **Generate convergence plots** check box.

Boundary Mode Analysis

1. In the **Study** toolbar, click **Study Steps** and choose **Other>Boundary Mode Analysis**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.
3. In the **Mode analysis frequency** text field, type `f0`.
4. Select the **Desired number of modes** check box.
5. Select the **Search for modes around** check box.
6. In the associated text field, type `0.9*k_lo`.
7. From the **Mode search method around shift** list, choose **Larger real part**.

Step 2: Boundary Mode Analysis 1

1. Right-click **Study 2 - Frequency Domain (Port Sweep)>Step 1: Boundary Mode Analysis** and choose **Duplicate**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.
3. In the **Port name** text field, type `2`.

Step 3: Boundary Mode Analysis 2

1. Right-click **Step 2: Boundary Mode Analysis 1** and choose **Duplicate**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.
3. In the **Port name** text field, type `3`.

Step 4: Boundary Mode Analysis 3

1. Right-click **Step 3: Boundary Mode Analysis 2** and choose **Duplicate**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.
3. In the **Port name** text field, type `4`.

Step 5: Boundary Mode Analysis 4

1. Right-click **Step 4: Boundary Mode Analysis 3** and choose **Duplicate**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.
3. In the **Port name** text field, type `5`.
4. In the **Search for modes around** text field, type `0.9*k_to`.

Step 6: Boundary Mode Analysis 5

1 Right-click **Step 5: Boundary Mode Analysis 4** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `6`.

Step 7: Boundary Mode Analysis 6

1 Right-click **Step 6: Boundary Mode Analysis 5** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `7`.

Step 8: Boundary Mode Analysis 7

1 Right-click **Step 7: Boundary Mode Analysis 6** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `8`.

Step 9: Boundary Mode Analysis 8

1 Right-click **Step 8: Boundary Mode Analysis 7** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `9`.

4 In the **Search for modes around** text field, type `0.9*k_tr1`.

Step 10: Boundary Mode Analysis 9

1 Right-click **Step 9: Boundary Mode Analysis 8** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `10`.

Step 11: Boundary Mode Analysis 10

1 Right-click **Step 10: Boundary Mode Analysis 9** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `11`.

Step 12: Boundary Mode Analysis 11

1 Right-click **Step 11: Boundary Mode Analysis 10** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `12`.

Step 13: Boundary Mode Analysis 12

1 Right-click **Step 12: Boundary Mode Analysis 11** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `13`.

4 In the **Search for modes around** text field, type `0.9*k_tr2`.

Step 14: Boundary Mode Analysis 13

1 Right-click **Step 13: Boundary Mode Analysis 12** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `14`.

Step 15: Boundary Mode Analysis 14

1 Right-click **Step 14: Boundary Mode Analysis 13** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `15`.

Step 16: Boundary Mode Analysis 15

1 Right-click **Step 15: Boundary Mode Analysis 14** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Port name** text field, type `16`.

Frequency Domain

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain**.

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

3 In the **Frequencies** text field, type `f0`.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, click to collapse the **Port Sweep Settings** section.

3 Click to expand the **Port Sweep Settings** section. Select the **Activate port sweep** check box.

Remember to add the `PortName` parameter to the **Parameters** list.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`PortName`	`1`	1	`Port name parameter`

STUDY 2 - FREQUENCY DOMAIN (PORT SWEEP)

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
PortName (Port name parameter)	`1, 5, 9, 13`	

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

RESULTS

Displacement

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

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 - Frequency Domain (Port Sweep)/Parametric Solutions 1 (sol19)**.

Surface 1

Right-click **Displacement** and choose **Surface**.

Deformation 1

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

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

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

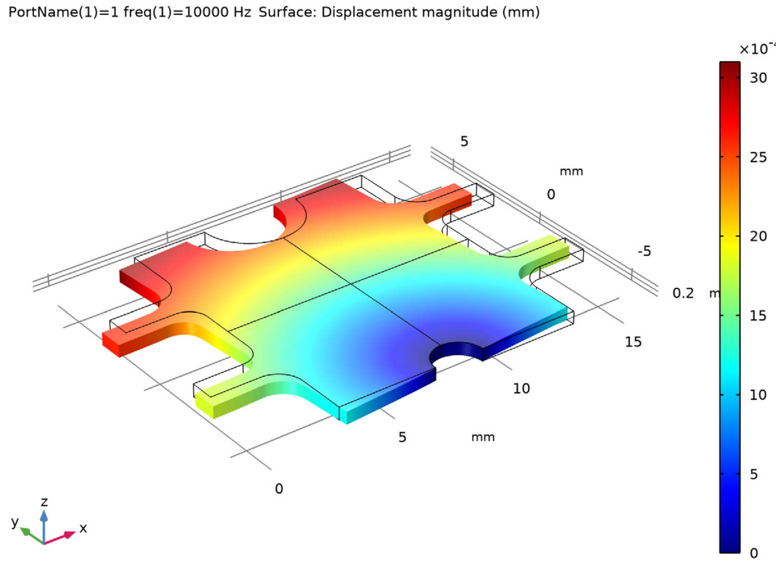
Displacement

1 In the **Model Builder** window, click **Displacement**.

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

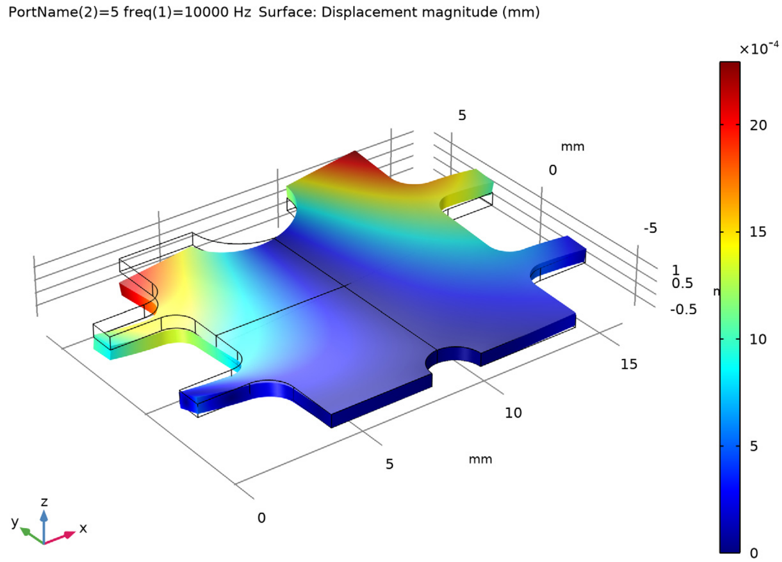
3 From the **Parameter value (PortName)** list, choose **1**.

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



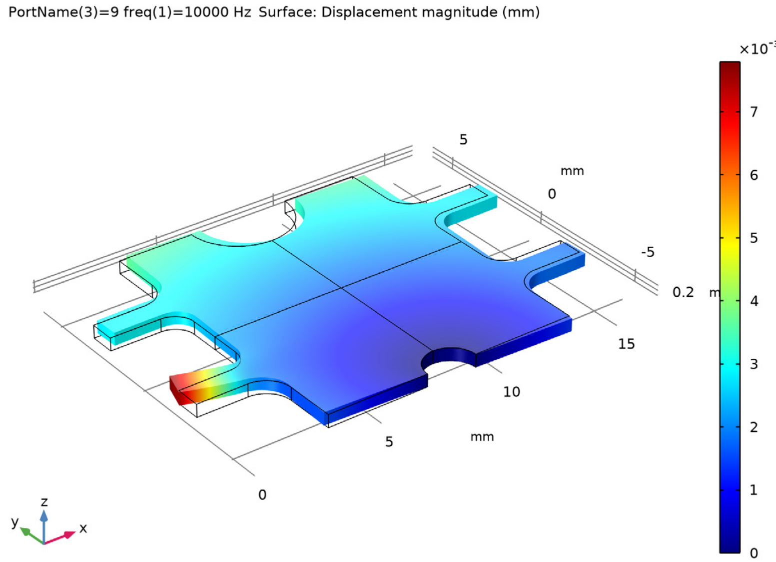
5 From the **Parameter value (PortName)** list, choose **5**.

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



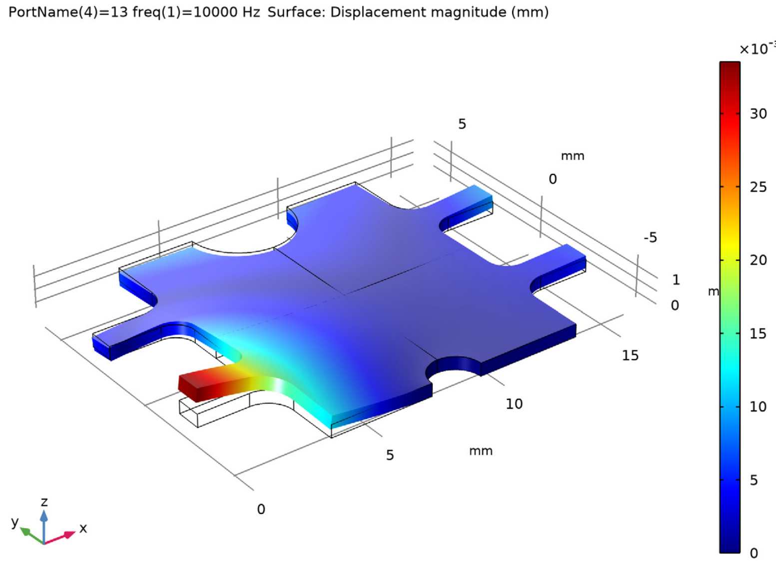
7 From the **Parameter value (PortName)** list, choose **9**.

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



9 From the **Parameter value (PortName)** list, choose **13**.

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



Included in the solution, when using the port feature, is the automatic computation of the scattering parameter S_ij. This represents the amplitude of the outgoing wave at port i when excited at port j. The variable S11 is, for example, called `solid.Smatrix11`, if the port number has two digits the format is, for example, `solid.Smatrix1_12`.

Now, set up 4 plots that show the outgoing energy of the various modes (proportional to abs(S_ij)^2) at the four inlet/outlets. Each plot corresponds to exciting a specific mode at the inlet/outlet number 1. Setting up the plots is a bit time consuming but can be simplified using **Duplicate** as well as copy/paste of expressions. The instructions are omitted here as they are long. For details, open the model and inspect the plots.