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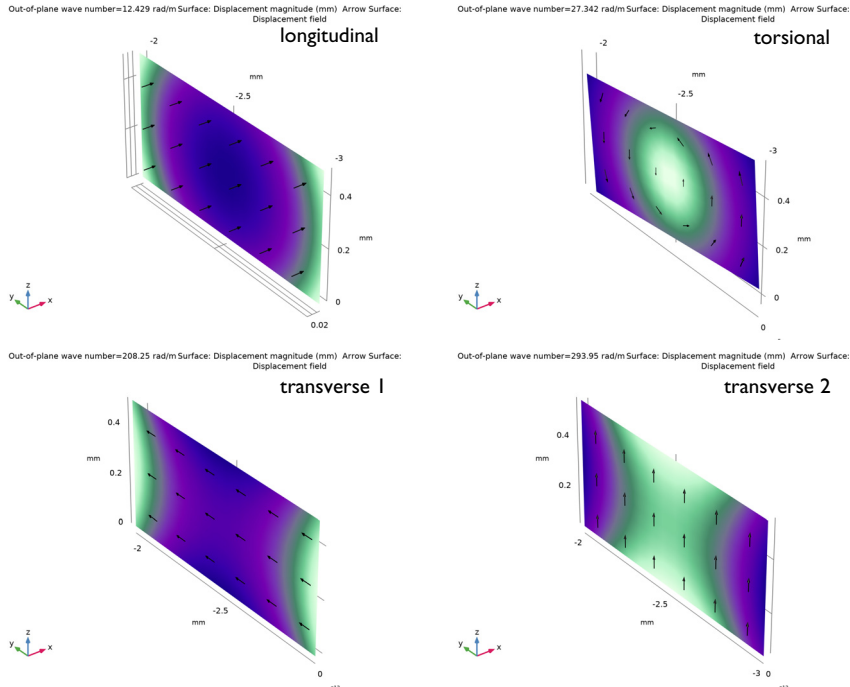
Figure 2: The four propagating modes captured by the ports located at the four inlet/outlets of the mechanical system.

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.

PortName(1)=1 freq(1)=10000 Hz Volume: Displacement magnitude (mm)
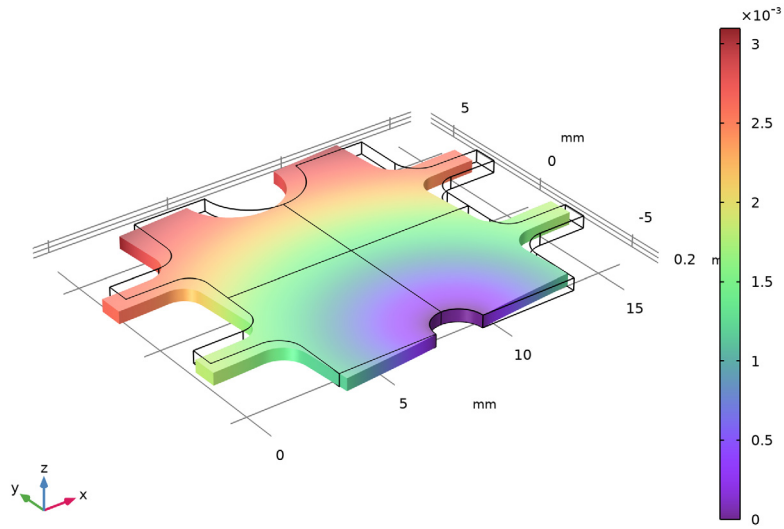
Figure 3: Displacement when the longitudinal mode is excited at the inlet/outlet 1.

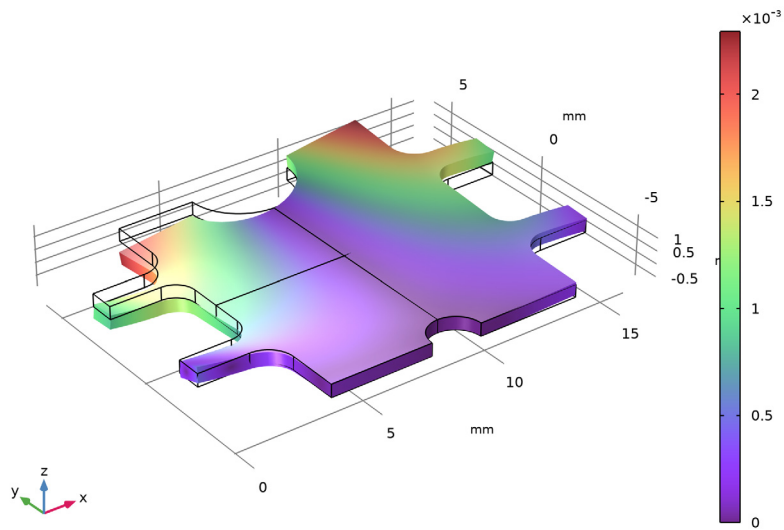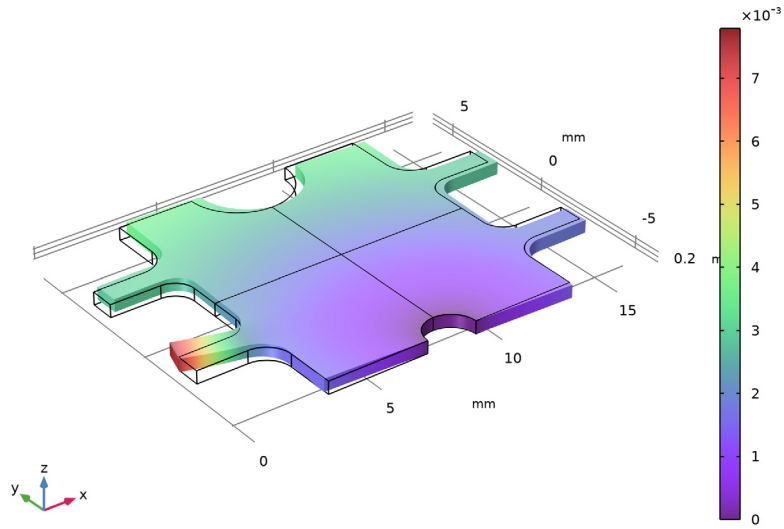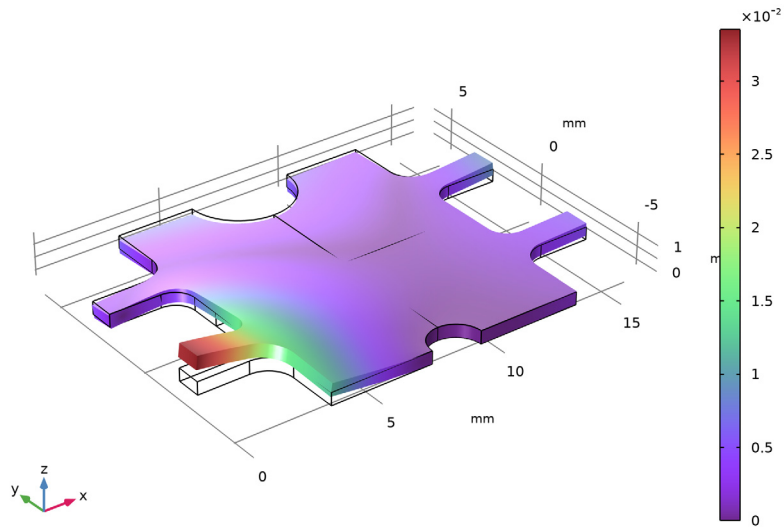PortName(2)=5 freq(1)=10000 Hz Volume: Displacement magnitude (mm)

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

PortName(3)=9 freq(1)=10000 Hz Volume: Displacement magnitude (mm)

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

PortName(4)=13 freq(1)=10000 Hz Volume: Displacement magnitude (mm)

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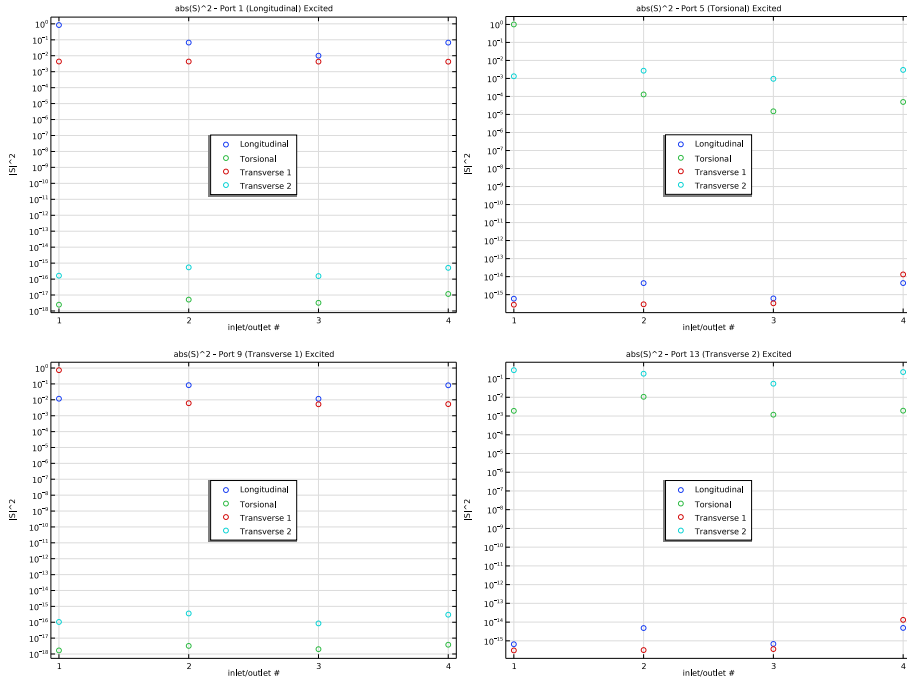
Figure 7: Power of the various outgoing modes at all inlet/outlets when the system is excited with a specific port.

Application Library path: Acoustics_Module/Elastic_Waves/
mechanical_multiport_system

Modeling Instructions

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

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) (solid).

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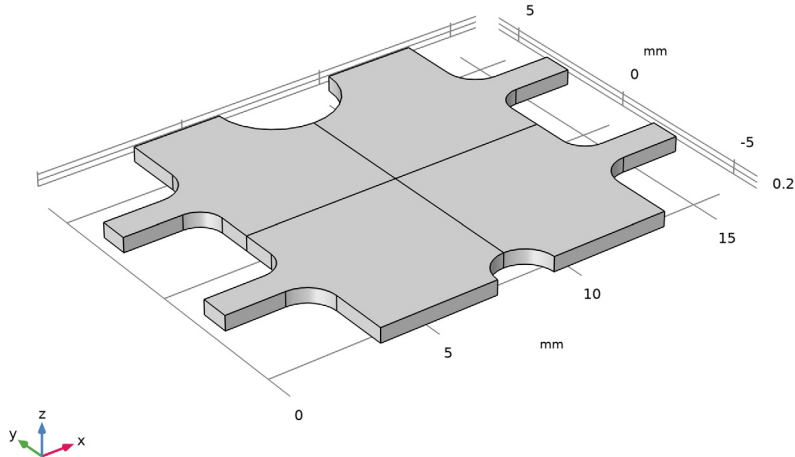
Create selections for the inlet/outlet boundary of the four waveguide structures. On each waveguide inlet/outlet boundary several **Port** features will be added, each capturing one specific propagating mode.

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.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Aluminum 6063-T83**.

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

5 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 by default the first port added has the excitation turned on. Here, turn it off; you can then select any port as source and run the model and 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

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

2 Right-click **Longitudinal Mode** and choose **Duplicate**.

Torsional Mode

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

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

3 Right-click **Torsional Mode** and choose **Duplicate**.

Translational Mode 1

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

2 In the **Settings** window for **Group**, type Translational Mode 1 in the **Label** text field.

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

Translational Mode 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Translational Mode 1.1**.

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 **Sequence Type** section.

3 From the 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 △ **More Generators** 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 Click to select the ▭ **Activate Selection** toggle button.

5 Select Boundaries 27 and 31 only.

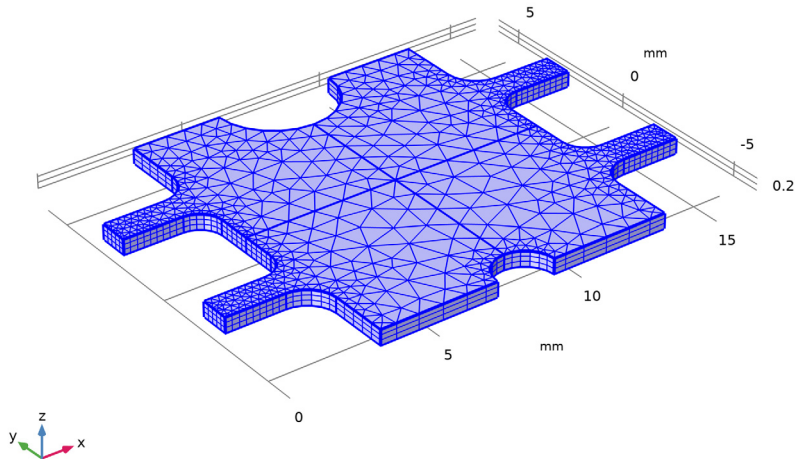**6** Click ▦ **Build Selected**.

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.

Step 1: 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. In the associated text field, type 4.

5 Select the **Search for modes around shift** check box. In the associated text field, type 0.8*kp.

6 From the **Search method around shift** list, choose **Larger real part**.

7 In the **Study** toolbar, click ▬ **Compute**.

1 In the **Home** toolbar, click ▣ **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1 - Modes/Solution 1 (sol1)>Solid Mechanics>Mode Shape (solid)**.

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

5 In the **Home** toolbar, click ▣ **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Arrow Surface 1

1 Right-click **Mode Shape (solid)** 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**.

Selection 1

**1** Right-click **Arrow Surface 1** and choose **Selection**.

2 Select Boundary 1 only.

Mode Shape (solid)

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

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

3 Clear the **Plot dataset edges** check box.

4 In the **Mode Shape (solid)** toolbar, click ▣ **Plot**.

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

Out-of-plane wave number=12.429 rad/m Surface: Displacement magnitude (mm) Arrow Surface:
Displacement field
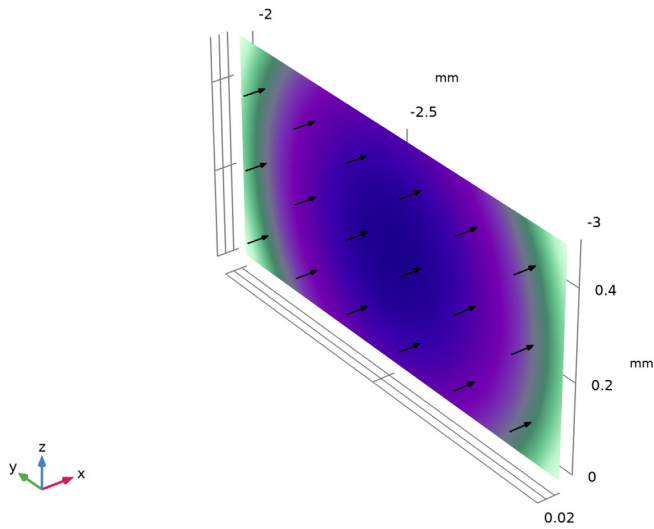
6 Locate the **Data** section. From the **Out-of-plane wave number (rad/m)** list, choose **27.342**.

7 In the **Mode Shape (solid)** toolbar, click 🔘 **Plot**.

Out-of-plane wave number=27.342 rad/m Surface: Displacement magnitude (mm) Arrow Surface: Displacement field

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

9 In the **Mode Shape (solid)** toolbar, click 🔲 **Plot**.

Out-of-plane wave number=208.25 rad/m Surface: Displacement magnitude (mm) Arrow Surface: Displacement field

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

II In the **Mode Shape (solid)** toolbar, click **Plot**.

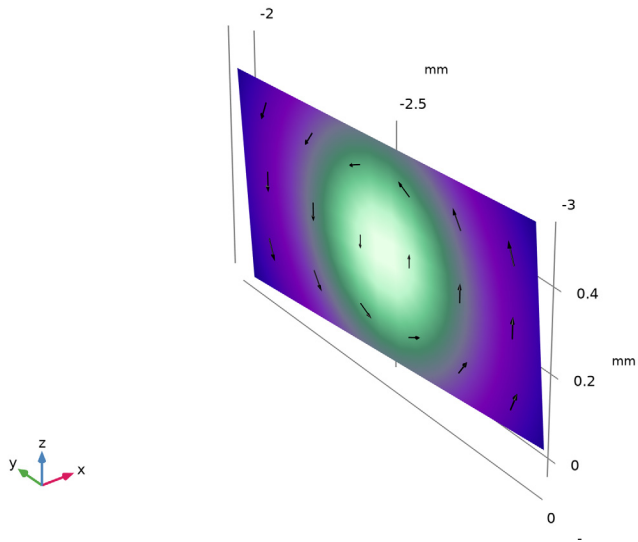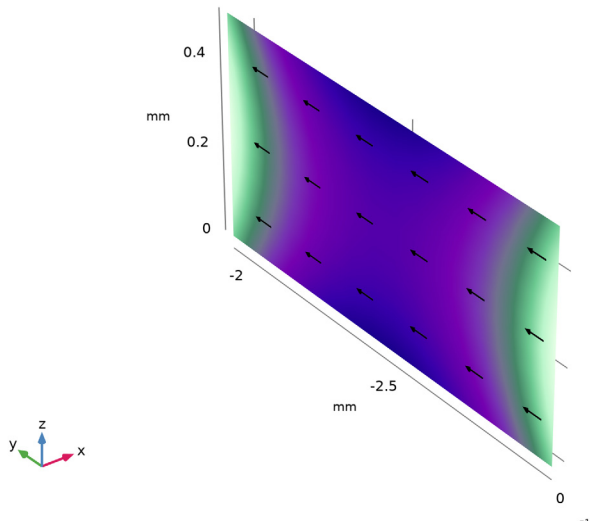Out-of-plane wave number=293.95 rad/m Surface: Displacement magnitude (mm) Arrow Surface:
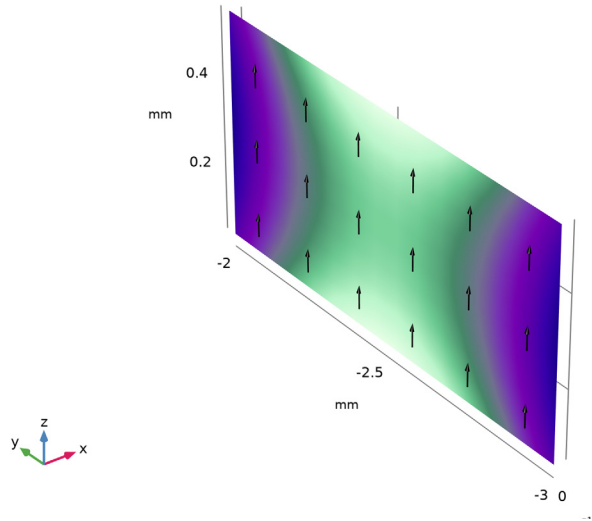Displacement field

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.

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 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.

Step 1: 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 shift** check box. In the associated text field, type 0.9*k_lo.

6 From the **Search method around shift** list, choose **Larger real part**.

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

Step 2: Boundary Mode Analysis 1

1 In the **Model Builder** window, click **Step 2: Boundary Mode Analysis 1**.

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

3 In the **Port name** text field, type 2.

4 Right-click **Step 2: Boundary Mode Analysis 1** and choose **Duplicate**.

Step 3: Boundary Mode Analysis 2

1 In the **Model Builder** window, click **Step 3: Boundary Mode Analysis 2**.

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

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

4 Right-click **Step 3: Boundary Mode Analysis 2** and choose **Duplicate**.

Step 4: Boundary Mode Analysis 3

1 In the **Model Builder** window, click **Step 4: Boundary Mode Analysis 3**.

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

3 In the **Port name** text field, type 4.

4 Right-click **Step 4: Boundary Mode Analysis 3** and choose **Duplicate**.

Step 5: Boundary Mode Analysis 4

1 In the **Model Builder** window, click **Step 5: Boundary Mode Analysis 4**.

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 shift** text field, type 0.9*k_to.

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

Step 6: Boundary Mode Analysis 5

1 In the **Model Builder** window, click **Step 6: Boundary Mode Analysis 5**.

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

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

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

Step 7: Boundary Mode Analysis 6

1 In the **Model Builder** window, click **Step 7: Boundary Mode Analysis 6**.

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

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

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

Step 8: Boundary Mode Analysis 7

1 In the **Model Builder** window, click **Step 8: Boundary Mode Analysis 7**.

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

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

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

Step 9: Boundary Mode Analysis 8

1 In the **Model Builder** window, click **Step 9: Boundary Mode Analysis 8**.

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 shift** text field, type 0.9*k_tr1.

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

Step 10: Boundary Mode Analysis 9

1 In the **Model Builder** window, click **Step 10: Boundary Mode Analysis 9**.

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

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

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

Step 11: Boundary Mode Analysis 10

1 In the **Model Builder** window, click **Step 11: Boundary Mode Analysis 10**.

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

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

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

Step 12: Boundary Mode Analysis 11

1 In the **Model Builder** window, click **Step 12: Boundary Mode Analysis 11**.

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

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

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

Step 13: Boundary Mode Analysis 12

1 In the **Model Builder** window, click **Step 13: Boundary Mode Analysis 12**.

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 shift** text field, type 0.9*k_tr2.

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

Step 14: Boundary Mode Analysis 13

1 In the **Model Builder** window, click **Step 14: Boundary Mode Analysis 13**.

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

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

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

Step 15: Boundary Mode Analysis 14

1 In the **Model Builder** window, click **Step 15: Boundary Mode Analysis 14**.

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

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

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

Step 16: Boundary Mode Analysis 15

1 In the **Model Builder** window, click **Step 16: Boundary Mode Analysis 15**.

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

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

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. From the list, choose
Activate port sweep.

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**.

1 In the **Home** toolbar, click ■ **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 2 - Frequency Domain (Port Sweep)/ Parametric Solutions 1 (sol19)>Solid Mechanics>Displacement (solid)**.

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

5 In the **Home** toolbar, click ■ **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Displacement (solid)

1 Click the ↔ **Zoom Extents** button in the **Graphics** toolbar.

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

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

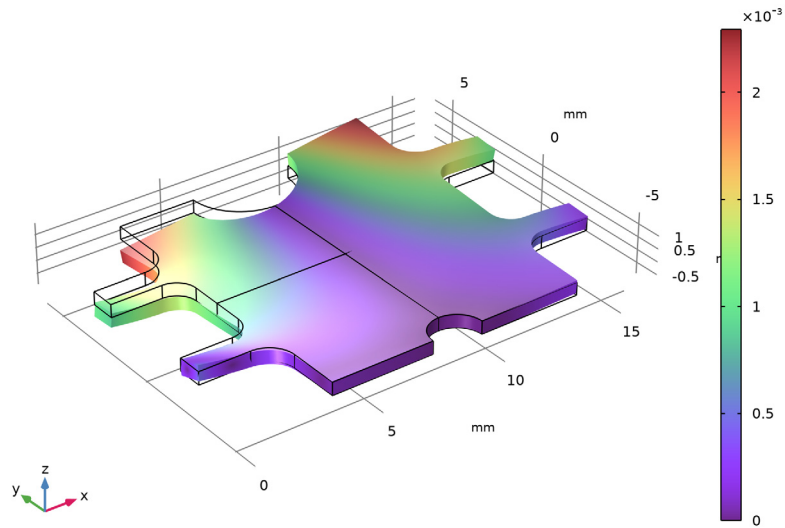4 In the **Displacement (solid)** toolbar, click ▶ **Plot**.

PortName(1)=1 freq(1)=10000 Hz Volume: Displacement magnitude (mm)
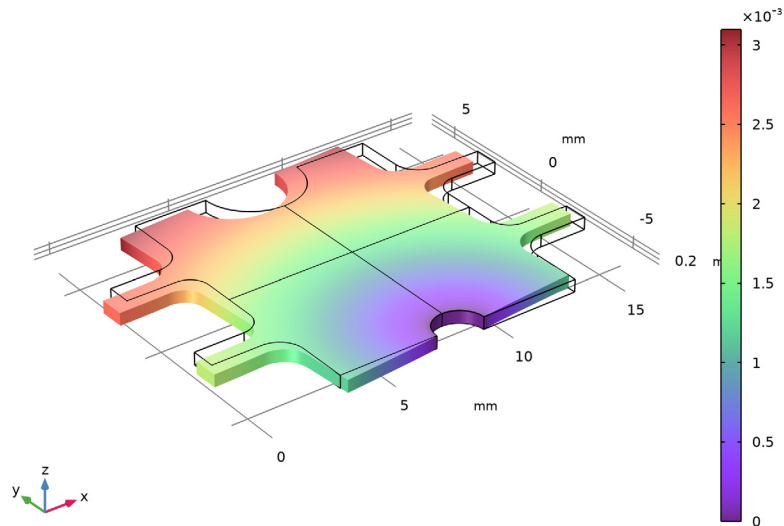
5 From the **Parameter value (PortName)** list, choose **5**.

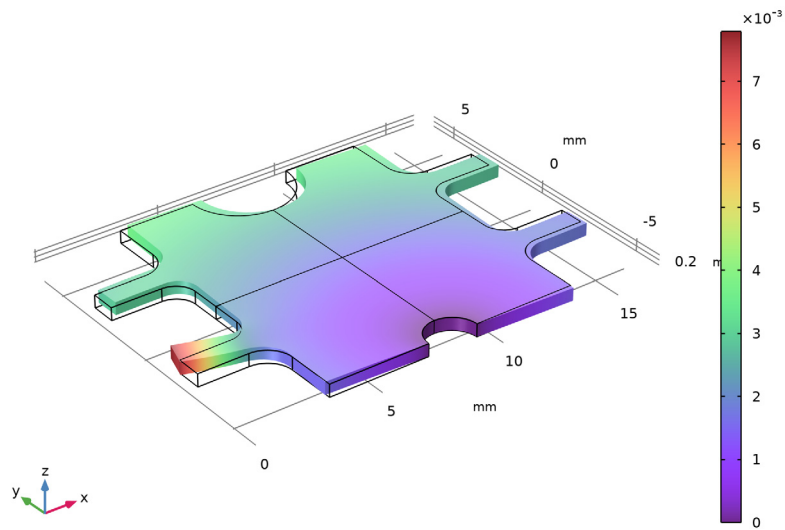**6** In the **Displacement (solid)** toolbar, click ▶ **Plot**.

PortName(2)=5 freq(1)=10000 Hz Volume: Displacement magnitude (mm)

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

8 In the **Displacement (solid)** toolbar, click ▶ **Plot**.
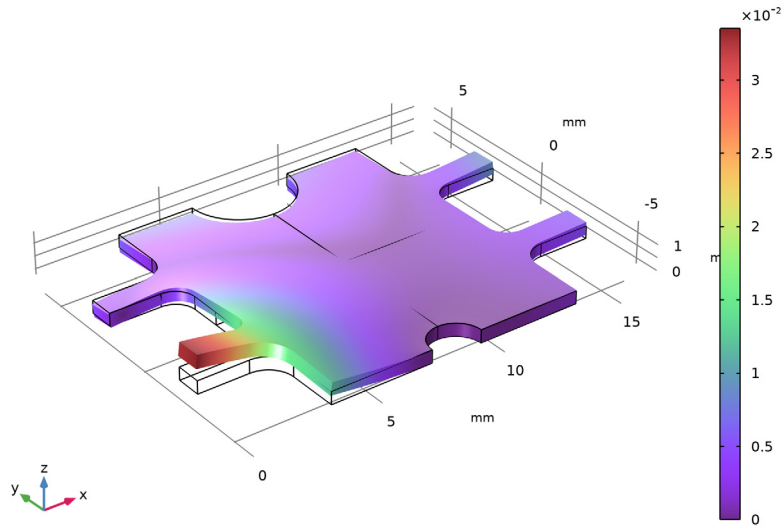
PortName(3)=9 freq(1)=10000 Hz Volume: Displacement magnitude (mm)

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

10 In the **Displacement (solid)** toolbar, click ◉ **Plot**.

PortName(4)=13 freq(1)=10000 Hz Volume: Displacement magnitude (mm)

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.

abs(S)^2 - Port 1 (Longitudinal) Excited