Model created in COMSOL Multiphysics 6.4

Isospectral Drums

Introduction

This example examines an interesting question posed by Mark Kac in 1966 (Ref. 1): "Can one hear the shape of a drum?"

Striking a drum excites a spectrum of vibration modes that together make up the instrument's characteristic sound or acoustic signal. These vibration modes correspond to the eigenmodes, or eigenfunctions, of the drum's membrane. Thus you can study this problem by solving eigenvalue problems for stretched membranes.

If you can find two differently shaped membranes that have identical eigenvalues — in other words, they are *isospectral* — then it is not possible to hear the shape of a specific drum.

In 1992, Gordon, Webb, and Wolpert (Ref. 2) showed that there are indeed sets of different planar shapes (nonisometric shapes) that are isospectral.

Work by Driscoll (Ref. 3) contains the following example of two planar shapes that sound the same.

Model Definition

The model shows the eigensolutions (the eigenvalues and eigenmodes) in two isospectral domains. Both cases use the solution to the same eigenvalue PDE:

$$\Delta u = \lambda u$$

The membranes are fixed at the boundaries; that is, a homogeneous Dirichlet boundary condition applies for all boundaries.

Results and Discussion

The sets of eigenvalues show that the domains are isospectral. Figure 1 shows two eigenmodes, one for each membrane shape, with the same eigenvalue. Note that eigenfunction normalization and sign are arbitrary and can vary from case to case.

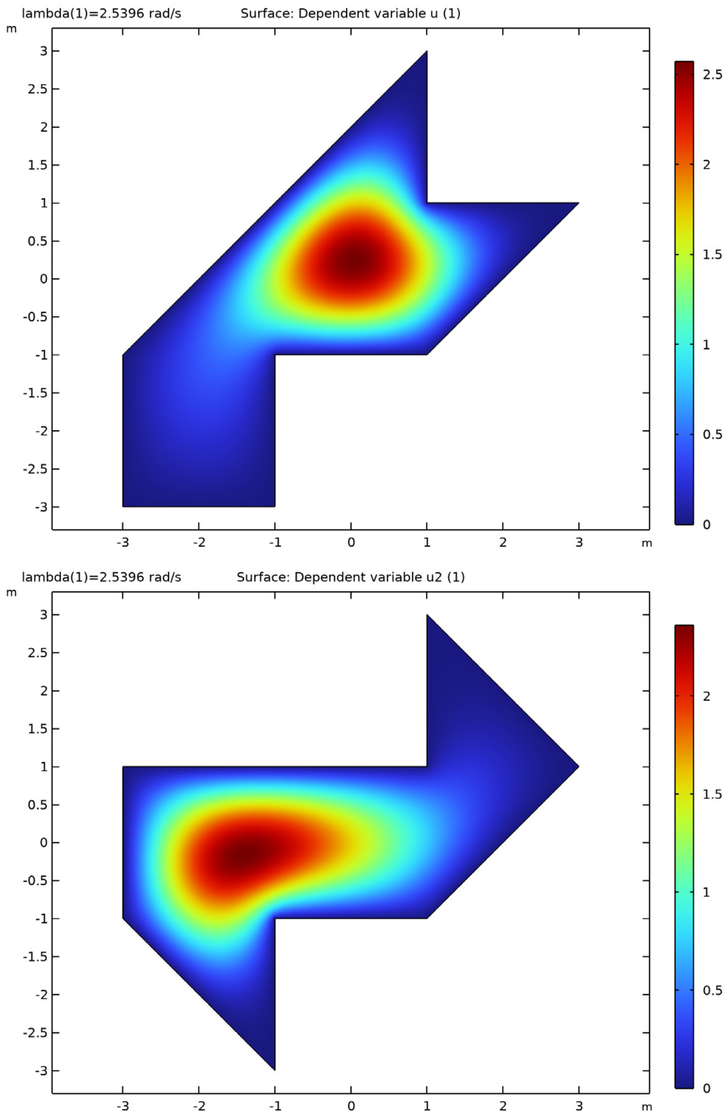


Figure 1: Eigenfunctions for the two different membranes with the same eigenvalue.

For each membrane shape, a straightforward postprocessing step shows that the eigenmodes found are orthogonal.

Notes About the COMSOL Implementation

In a single MPH-file, build two COMSOL Multiphysics models that solve the eigenvalue PDE on two different 2D domains, and compare the sets of eigenvalues.

The model shows how to use the `with` operator to access different eigenmodes during postprocessing.

References

1. M. Kac, "Can One Hear the Shape of a Drum?," *American Math. Mon.*, vol. 73 Part II, pp. 1–23, 1966.

2. C. Gordon, D. Webb, and S. Wolpert, "Isospectral Plane Domains and Surfaces via Riemannian Orbifolds," *Invent. Math.*, vol. 110, pp. 1–22, 1992.

3. T. Driscoll, "Eigenmodes of Isospectral Drums," *Technical Report-Center for Theory and Simulation in Science and Engineering*, Cornell University, Ithaca, N.Y., CTC95TR209, May 1995.

Application Library path: `COMSOL_Multiphysics/Equation_Based/isospectral_drums`

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 **2D**.

2 In the **Select Physics** tree, select **Mathematics** > **PDE Interfaces** > **Coefficient Form PDE (c)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Eigenvalue**.

6 Click **Done**.

GEOMETRY 1

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

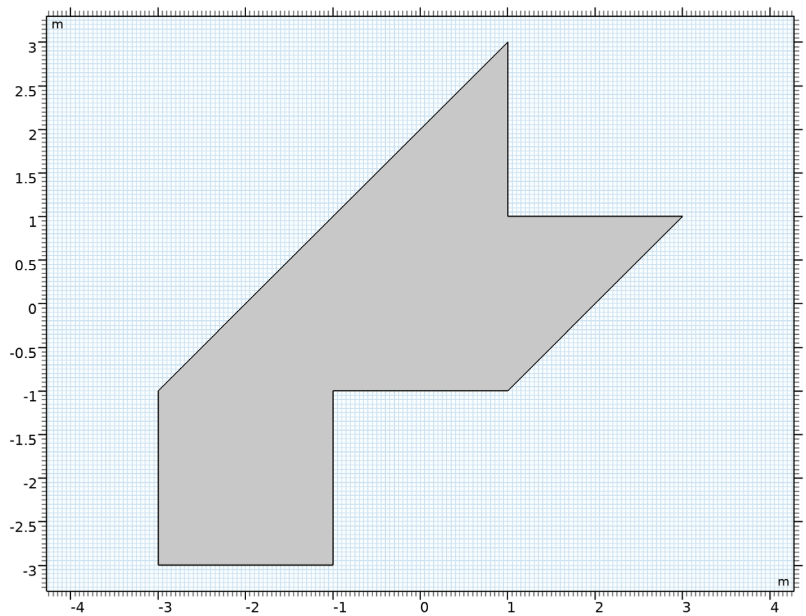
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)
-3	-3
-3	-1
1	3
1	1
3	1
1	-1
-1	-1
-1	-3

4 Click **Build All Objects**.

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



COEFFICIENT FORM PDE (C)

Use the default values of the coefficients of PDE to model this problem.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Applying Dirichlet boundary conditions corresponds to clamping the drum's membrane at the edges.

MESH 1

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

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

3 From the **Element size** list, choose **Extra fine**.

By using a finer mesh, you reduce the apparent discrepancies between the matching eigenvalues caused by numerical errors.

4 Click **Build All**.

STUDY 1

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

The default settings for an Eigenvalue study step gives the six lowest eigenvalues.

RESULTS

Coefficient Form PDE

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

The default plot shows the eigenfunctions for the lowest eigenmode; compare with the upper plot in Figure 1.

ROOT

Now add a second model for the alternative membrane shape.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **2D**.

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 **Mathematics** > **PDE Interfaces** > **Coefficient Form PDE (c)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study 1**.

5 Click the **Add to Component 2** button in the window toolbar.

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

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Coefficient Form PDE (c)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Eigenvalue**.

5 Click the **Add Study** button in the window toolbar.

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

GEOMETRY 2

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

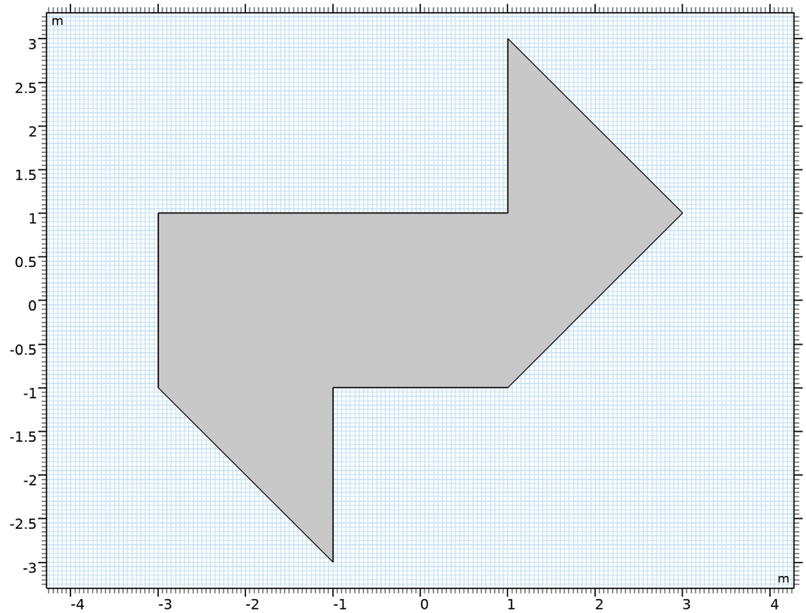
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)
-3	1
1	1
1	3
3	1
1	-1
-1	-1
-1	-3
-3	-1

4 Click **Build All Objects**.

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



COEFFICIENT FORM PDE 2 (C2)

Use the same domain settings and boundary conditions as for the first model.

Dirichlet Boundary Condition 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.
2. In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.

MESH 2

1. In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. From the **Element size** list, choose **Extra fine**.
4. Click **Build All**.

STUDY 2

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

RESULTS

Coefficient Form PDE 2

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

The new default plot should now look like the lower plot in Figure 1.

Use the `with` operator to access different eigenmodes in order to show that they are orthogonal:

Surface Integration 1

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Integration** > **Surface Integration**.

2 Select Domain 1 only.

3 In the **Settings** window for **Surface Integration**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`with(1,u)*with(2,u)`	`m^2`	

5 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The value of integral, which is very small (of the order of 10^{-16}), appears in the **Table** window. Ideally, the result should be zero when the eigenmodes are orthogonal, but using a numerical method you can expect a small nonzero number.

RESULTS

Finally, do the same for the second model.

Surface Integration 2

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration** > **Surface Integration**.

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

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

4 Select Domain 1 only.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`with(1,u2)*with(2,u2)`	`m^2`	`Dependent variable u`

6 Click **Evaluate**.