Maximizing the Eigenfrequency of a Shell

Introduction

Shape optimization can be used to alter the geometry of an existing product to improve its performance. You can do that using the **Deformed Geometry** interface, but you have to decide which shape deformations to allow. It is important to impose some restriction to preserve the mesh quality during the optimization. One approach is to use a polynomial to introduce some regularization (in combination with a maximum displacement parameter), so that the slope of the shape variation is bounded. This type of regularized shape optimization can be set up using equation based modeling, but it is also built into the **Free Shape Shell** feature. This feature differs from the **Free Shape Boundary** feature in that it can be used on boundaries that are not adjacent to meshed domains.

Model Definition

Shape optimization is often subject to constraints on the geometry deformation, and this model shows how the **Free Shape Shell** feature can be combined with the **Symmetry/Roller** feature to restrict edges to move along an imaginary boundary defined by a normal vector. The sides as well as the loaded edges are fixed using the **Fixed Edge** feature. The eigenmodes are not symmetric, so the entire shell has to be modeled in every optimization iteration, but it is possible to enforce a symmetric design using the **Mirror Symmetry** feature.The initial geometry of the shell is shown in Figure 1.

An **Eigenvalue** solver is required for computing the eigenfrequency of the shell, while the dependent variables associated with the **Free Shape Shell** features requires a **Stationary** solver. Gradient based optimization over multiple study steps is unsupported, but it s possible to combine the two solver types using a single **Stationary Then Eigenfrequency** study step.

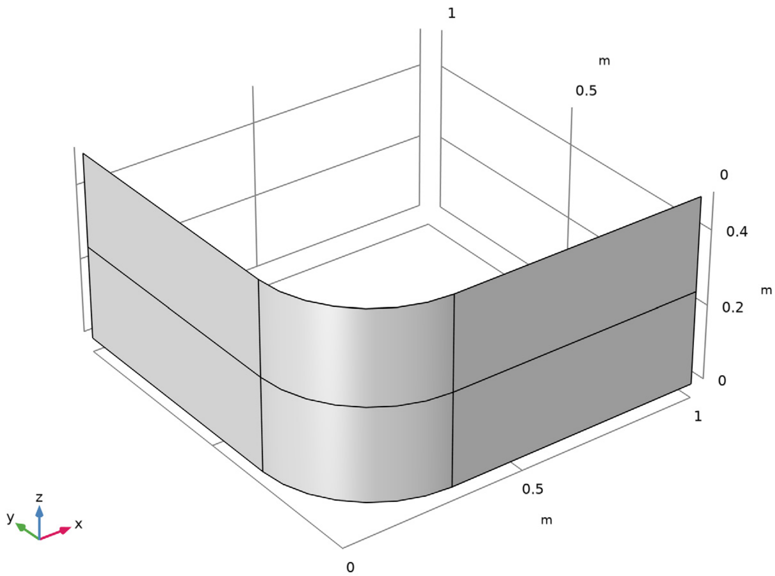

Figure 1: The initial geometry is shown. The displacement and rotation is fixed at the leftmost edge of the shell. The shape deformation of this edge is restricted to the xz-plane.

The shell is made of steel and the objective is to maximize the lowest eigenfrequency by deforming the mesh used for the discretization of the shell.

Results and Discussion

The optimal design is intuitive in the sense that it deforms the shell, so that material is moved away from the midplane, increasing the stiffness of the shell; see Figure 2.

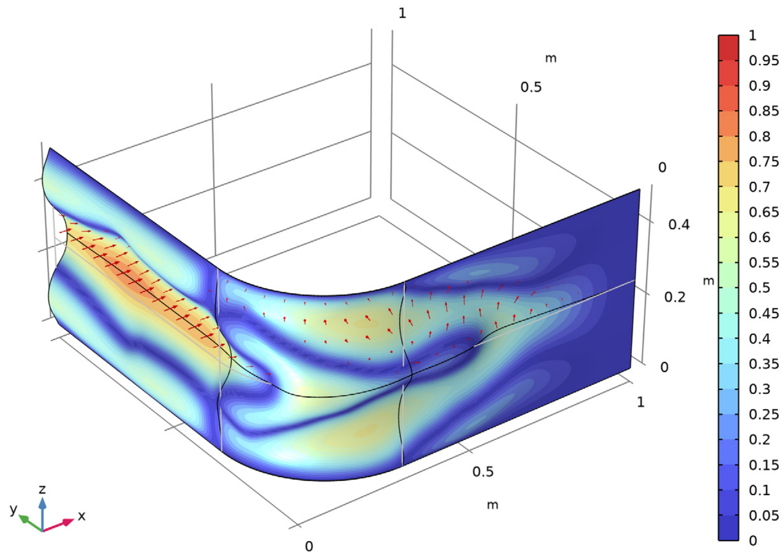

Figure 2: The default shape optimization plot shows the edges of the old geometry in gray together with a surface plot of the relative normal boundary displacement in colors. The actual displacement is shown with red arrows.

By deforming the shell the optimization is able to increase the lowest eigenfrequency by a factor of four. The area (and thus the mass) of the shell is increased by the optimization, but the tip of the shell remains relatively flat to avoid introducing more mass in this area.

Notes About the COMSOL Implementation

This model combines the **Shape Optimization** and **Shell** interfaces. The **Shell** interface computes fold lines automatically, but the details of this computation is affected by the presence of **Deformed Geometry** and/or **Shape Optimization**. Therefore it is good practice to perform a verification analysis in a new component, and this model includes such a verification analysis, which results in a somewhat lower eigenfrequency. The eigenmodes can also appear different, but this is primarily due to the fact that the optimization produces degenerate eigenmodes, so one can only expect that the eigenmodes (for the low frequencies) span the same space.

Application Library path: `Optimization_Module/Shape_Optimization/shell_eigenfrequency_shape_optimization`

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 **Structural Mechanics>Shell (shell)**.
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 In the table, enter the following settings:

Name	Expression	Value	Description
Lmax	5[cm]	0.05 m	Maximum displacement

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Square 1 (sq1)

In the **Work Plane** toolbar, click **Square**.

Work Plane 1 (wp1)>Fillet 1 (fil1)

1 In the **Work Plane** toolbar, click **Fillet**.
2 On the object **sq1**, select Point 1 only.
3 In the **Settings** window for **Fillet**, locate the **Radius** section.
4 In the **Radius** text field, type `0.3`.

Work Plane 1 (wp1)>Convert to Curve 1 (ccur1)

1 In the **Work Plane** toolbar, click **Conversions** and choose **Convert to Curve**.
2 Select the object **fil1** only.

Edges to Delete

1 In the **Work Plane** toolbar, click **Selections** and choose **Box Selection**.
2 In the **Settings** window for **Box Selection**, type `Edges to Delete` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Box Limits** section. In the **xw minimum** text field, type `0.9`.
5 In the **yw minimum** text field, type `0.9`.

Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 Right-click **Plane Geometry** and choose **Delete Entities**.
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
3 From the **Selection** list, choose **Edges to Delete**.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (m)
0.5

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
5 From the **Show in physics** list, choose **Boundary selection**.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `0.25`.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

3 From the **Objects to partition** list, choose **Extrude 1**.

4 From the **Partition with** list, choose **Work plane**.

5 In the **Geometry** toolbar, click **Build All**.

Exterior Edges

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Exterior Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Locate the **Output Entities** section. From the **Geometric entity level** list, choose **Adjacent edges**.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog box, select **Extrude 1** in the **Input selections** list.

7 Click **OK**.

Fixed Edge

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Edge**.

4 In the **Label** text field, type `Fixed Edge`.

5 Locate the **Box Limits** section. In the **y minimum** text field, type `0.99`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Internal Symmetry

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Internal Symmetry` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Edge**.
4 Locate the **Box Limits** section. In the **z minimum** text field, type `0.24`.
5 In the **z maximum** text field, type `0.26`.
6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.
7 In the **Geometry** toolbar, click **Build All**.

The model geometry is now complete.

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>Structural steel**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Extra fine**.
4 Click **Build All**.

SHELL (SHELL)

We expect the **Free Shape Shell** feature to preserve continuity of the normal, so that correct fold line constraints are applied.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.
2 In the **Settings** window for **Shell**, click to expand the **Fold-Line Settings** section.
3 In the α text field, type `0`.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.
2. In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Fixed Edge**.

COMPONENT 1 (COMP1)

Define the shape optimization problem using the **Free Shape Shell**, **Symmetry/Roller**, and **Fixed Edge** features.

Free Shape Shell 1

1. In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization, Shell**.
2. In the **Settings** window for **Free Shape Shell**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.
4. Locate the **Control Variable Settings** section. In the text field, type `Lmax`.
5. Locate the **Filtering** section. From the R_{min} list, choose **Small**.

Fixed Edge 1

1. In the **Shape Optimization** toolbar, click **Fixed Edge**.
2. In the **Settings** window for **Fixed Edge**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Exterior Edges**.

Symmetry/Roller 1

1. In the **Shape Optimization** toolbar, click **Symmetry/Roller**.
2. In the **Settings** window for **Symmetry/Roller**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Edge**.
4. From the **Selection** list, choose **Fixed Edge**.
5. Locate the **Prescribed Normal Vector** section. Specify the **n** vector as

0	X
1	Y
0	Z

Preserving the symmetry of the design requires the use of the **Symmetry/Roller** and **Mirror Symmetry** features, because the eigenmodes are not symmetric, so the entire shell has to be modeled in every optimization iteration.

Symmetry/Roller 2

1. In the **Shape Optimization** toolbar, click **Symmetry/Roller**.

2 In the **Settings** window for **Symmetry/Roller**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Internal Symmetry**.

Mirror Symmetry 1

1 In the **Shape Optimization** toolbar, click **Mirror Symmetry**.

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

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

4 Select Boundaries 1, 2, and 5 only.

STUDY 1

Step 1: Stationary Then Eigenfrequency

In the **Study** toolbar, click **Study Steps** and choose **Eigenfrequency>Stationary Then Eigenfrequency**.

Shape Optimization

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

It is often difficult to converge shape- and topology optimization problems on the tolerance, so limit the number of GCMMA iterations to reduce the computational time.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type `20`.

4 In the **Move limits** text field, type `0.2`.

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

Expression	Description
freq	Frequency

6 From the **Type** list, choose **Maximization**.

7 From the **Solution** list, choose **Minimum of objectives**.

8 From the **Objective scaling** list, choose **Initial solution based**.

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

10 In the **Settings** window for **Study**, type `Maximize Lowest Eigenfrequency` in the **Label** text field.

11 In the **Study** toolbar, click **Show Default Plots**.

RESULTS

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

4 From the **Plane** list, choose **XY-planes**.

5 In the **Z-coordinate** text field, type `0.25`.

Shape Optimization

1 In the **Model Builder** window, click **Shape Optimization**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

MAXIMIZE LOWEST EIGENFREQUENCY

1 In the **Model Builder** window, under **Maximize Lowest Eigenfrequency** click **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Shape Optimization**.

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

RESULTS

1 In the **Model Builder** window, under **Results** click **Shape Optimization**.

2 In the **Shape Optimization** toolbar, click **Plot**.

The **Shell** interface treats fold lines differently with shape optimization, so it is good practice to perform a verification in a new component.

Filter 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **Filter**.

3 In the **Settings** window for **Filter**, locate the **Expression** section.

4 In the **Expression** text field, type `1`.

5 Click **Plot**.

6 Right-click **Filter 1** and choose **Create Mesh in New Component**.

MESH 2

Import 1

1. In the **Settings** window for **Import**, locate the **Import** section.
2. From the **Boundary partitioning** list, choose **Minimal**.
3. Click **Import**.
4. Click **Build All**.

Paste the **Shell** interface and material from the first component.

SHELL (SHELL)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Shell (shell)** and choose **Copy**.

COMPONENT 2 (COMP2)

In the **Model Builder** window, right-click **Component 2 (comp2)** and choose **Paste Shell**.

SHELL (SHELL2)

In the **Messages from Paste** dialog box, click **OK**.

Fixed Constraint 1

1. In the **Model Builder** window, expand the **Shell (shell2)** node, then click **Fixed Constraint 1**.
2. In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.
3. From the **Selection** list, choose **Fixed Edge**.

MATERIALS

Structural steel (mat1)

In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Structural steel (mat1)** and choose **Copy**.

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Paste Material**.

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>Eigenfrequency**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Shell (shell)**.
5 Click **Add Study** in the window toolbar.
6 In the **Model Builder** window, click the root node.
7 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Disable the irrelevant physics interfaces in the study steps.

MAXIMIZE LOWEST EIGENFREQUENCY

Step 1: Stationary Then Eigenfrequency

1 In the **Settings** window for **Stationary Then Eigenfrequency**, locate the **Physics and Variables Selection** section.
2 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Shell (shell)	√	Automatic (Stationary then eigenfrequency)
Shell (shell2)		Automatic (Stationary)
Shape Optimization (Component 1)	√	Automatic

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Shell (shell)		Automatic (Stationary)
Shell (shell2)	√	Automatic (Eigenfrequency)
Shape Optimization (Component 1)		Automatic

4 In the **Model Builder** window, click **Study 2**.
5 In the **Settings** window for **Study**, type `Verification` in the **Label** text field.
6 In the **Home** toolbar, click **Compute**.

Delete irrelevant result features.

RESULTS

Eigenfrequencies (Verification), Participation Factors (Verification), Shape Optimization 1

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Shape Optimization 1**, **Eigenfrequencies (Verification)**, and **Participation Factors (Verification)**.
2. Right-click and choose **Delete**.

   Note that the optimization causes the first two eigenfrequencies to become degenerate, so the first two eigenmodes in the verification should span the same space as in the optimization, but the eigenmodes themselves can be different.