Created in COMSOL Multiphysics 5.6

Optimizing a Flywheel Profile

Introduction

Flywheels are designed to store kinetic energy by rotating a mass at high speed. The inertial forces load the flywheel causing structural stress which can destroy the flywheel for high rotational frequencies. This model starts out by computing the moment of inertia, the mass, as well as the stress distribution in a simple disk-shaped flywheel. It then uses shape optimization to increase the moment of inertia without increasing the mass or maximum von Mises stress.

Model Definition

The setup of the Solid Mechanics interface is straightforward, and so is the computation of the total mass and moment of inertia via the Mass Properties feature. The main difficulty with the modeling setup thus relates to the use of gradient based optimization in combination with the maximum von mises stress as a constraint.

THE OPTIMIZATION PROBLEM

The maximum von Mises stress will be approximated with a p-norm ($p = 10$):

$$(\sigma_{\mathrm{max}} = \mathrm{max}(\sigma)) \approx \left(\int_{\Omega} \sigma^p d\Omega\right)^{1/p} . \qquad (1)$$

This is a constraint aggregation strategy that performs well for moderate values of p. Too small values cause a bad approximation of the maximum operator, while too large values cause numerical problems for the optimization solver.

The shape optimization problem is setup using the **Free Shape Domain**, **Free Shape Boundary**, and **Free Shape Symmetry** features, such that the inner and outer radii of the flywheel is fixed, that is, it is the profile of the flywheel that is optimized. The **Free Shape Boundary** feature is set up with a filter radius only one 3rd of the maximum displacement. This means that the slope of the flywheel profile can be as large as 3, which would normally lead to solver errors, but in this model the initial geometry is well suited for a mapped mesh, which is less prone to inverted elements.

Results and Discussion

Figure 1 shows the optimized and initial flywheel profiles.

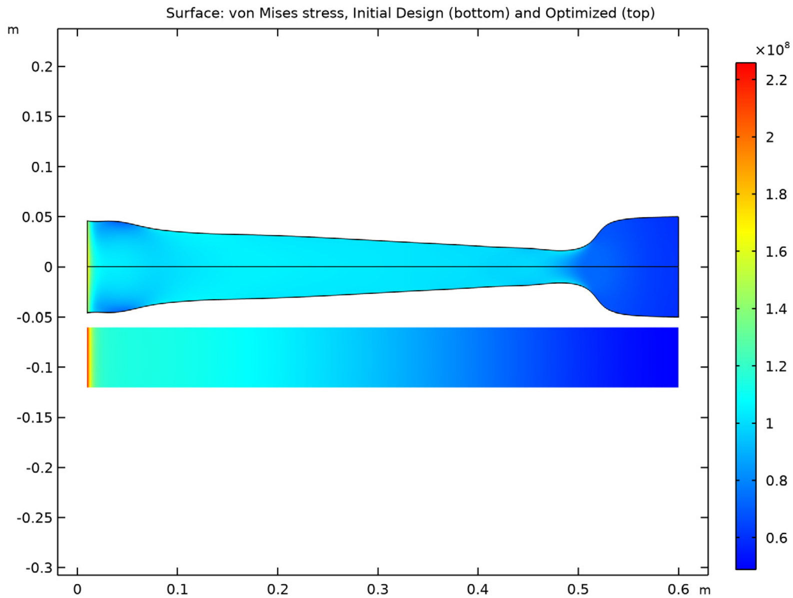

Figure 1: The von Mises stress for the initial and optimized flywheel design.

The moment of inertia has increased by 9%, while the maximum von Mises stress has been decreased by 36%.

Application Library path: `Optimization_Module/Shape_Optimization/flywheel_profile`

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 Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Type in the parameters for the initial geometry. Note that we exploit symmetry by only modeling half of the flywheel.

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
r0	1[cm]	0.01 m	Inner flywheel radius
r1	60[cm]	0.6 m	Outer flywheel radius
H0	3[cm]	0.03 m	Initial flywheel thickness
omega	2*pi*50[rad/s]	314.16 rad/s	Angular velocity
pExp	10	10	P-norm exponent

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `r1-r0`.
4 In the **Height** text field, type `H0`.
5 Locate the **Position** section. In the **r** text field, type `r0`.

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.

SOLID MECHANICS (SOLID)

Rotating Frame 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Volume Forces>Rotating Frame**.

2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

3 In the Ω text field, type `omega`.

4 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

Symmetry Plane 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.

2 Select Boundary 2 only.

MESH 1

Mapped 1

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

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 **Extremely fine**.

DEFINITIONS

The **Mass Properties** feature allows computation of the total mass as well as the moment of inertia, but it is not visible by default.

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, select **Physics>Advanced Physics Options** in the tree.

3 In the tree, select the check box for the node **General>Variable Utilities**.

4 Click **OK**.

Mass Properties 1 (mass1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variable Utilities>Mass Properties**.

2 In the **Settings** window for **Mass Properties**, locate the **Density** section.

3 From the **Density source** list, choose **From physics interface**.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

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

INITIAL DESIGN

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

2 In the **Settings** window for **Study**, type `Initial Design` in the **Label** text field.

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

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

RESULTS

Initial Design

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Initial Design` in the **Label** text field.

Initial Constraints

1 Right-click **Initial Design** and choose **Global Evaluation**.

Compute the initial P-norm von Mises stress and mass.

2 In the **Settings** window for **Global Evaluation**, type `Initial Constraints` in the **Label** text field.

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

Expression	Unit	Description
`aveop1(solid.mises^pExp)^(1/pExp)`	`N/m^2`	`Average 1`

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Mass Properties 1>mass1.mass - Mass - kg**.

5 In the **Initial Design** toolbar, click **Evaluate**.

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
sigma0	96.2[MPa]	9.62E7 Pa	Initial p-norm stress
M0	266[kg]	266 kg	Initial mass

DEFINITIONS

Setup the shape optimization problem and use the p-norm of the von Mises stress as a constraint.

Free Shape Domain 1

1. In the **Definitions** toolbar, click **Optimization** and choose **Free Shape Domain**.
2. In the **Settings** window for **Free Shape Domain**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **All domains**.

Symmetry/Roller 1

1. In the **Definitions** toolbar, click **Optimization** and choose **Symmetry/Roller**.
2. Select Boundaries 1 and 4 only.

Free Shape Boundary 1

1. In the **Definitions** toolbar, click **Optimization** and choose **Free Shape Boundary**.
2. Select Boundary 3 only.
3. In the **Settings** window for **Free Shape Boundary**, locate the **Control Variable Settings** section.
4. In the d_{max} text field, type `2*H0/3`.
5. Locate the **Filtering** section. In the R_{min} text field, type `2*H0/3`.

Variables 1

1. In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`stress_Pnorm`	`aveop1((solid.mises/sigma0)^pExp)^(1/pExp)`		`Scaled P-norm of von Mises stress`

This P-norm is an approximate max operator, which is compatible with gradient based optimization (a real max operator is not). The p-norm is expected to result in unstable behavior for large p.

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.

SHAPE OPTIMIZATION

1. In the **Model Builder** window, click **Study 2**.
2. In the **Settings** window for **Study**, type `Shape Optimization` in the **Label** text field.
3. In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Arrow Line 1

1. In the **Model Builder** window, expand the **Shape Optimization** node, then click **Arrow Line 1**.
2. In the **Settings** window for **Arrow Line**, locate the **Arrow Positioning** section.
3. From the **Placement** list, choose **Mesh nodes**.

SHAPE OPTIMIZATION

Shape Optimization

1. In the **Model Builder** window, right-click **Shape Optimization** and choose **Optimization>Shape Optimization**.
2. In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.
3. From the **Method** list, choose **IPOPT**.

4 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>Mass Properties 1> Moment of inertia, principal values>comp1.mass1.Ip1 - First moment of inertia principal value - kg·m²**.

5 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)>Definitions>Mass Properties 1>comp1.mass1.mass - Mass - kg**.

6 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>comp1.stress_Pnorm - Scaled P-norm of von Mises stress**.

Rescale the mass and moment of inertia. Constrain the mass and stress from above, while maximizing the moment of inertia.

7 Locate the **Objective Function** section. From the **Objective scaling** list, choose **Initial solution based**.

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

9 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound
comp1.mass1.mass/M0		1
comp1.stress_Pnorm		1

10 Locate the **Output While Solving** section. Select the **Plot** check box.

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

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

The optimization moves mass toward the axis to reduce stress, but this reduces the moment of inertia, so material is also moved to the perimeter of the flywheel. Modify the stress plot to include the initial stress distribution.

RESULTS

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

2 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

3 In row **Point 2**, set **R** to 1.

4 In row **Point 2**, set **Z** to 0.

5 Click to expand the **Advanced** section. Select the **Define variables** check box.

Mirror 2D 2

1 Right-click **Mirror 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Mirror 2D**, locate the **Data** section.

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

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

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

3 From the **Dataset** list, choose **Mirror 2D 2**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Surface: von Mises stress, Initial Design (bottom) and Optimized (top)`.

Surface 2

1 In the **Model Builder** window, expand the **Stress (solid)** node.

2 Right-click **Results>Stress (solid)>Surface 1** and choose **Duplicate**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Deformation** and choose **Delete**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **x component** text field, type `0`.

4 In the **y component** text field, type `if(mir1side,1,-1)*-3*H0`.

5 Locate the **Scale** section. Select the **Scale factor** check box.

6 In the associated text field, type `1`.

7 In the **Stress (solid)** toolbar, click **Plot**.

It is clear that the optimization has decreased the maximum stress in the flywheel.