



Model created in COMSOL Multiphysics 6.4

# 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 ( $p = 10$ ):

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

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 set up 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.

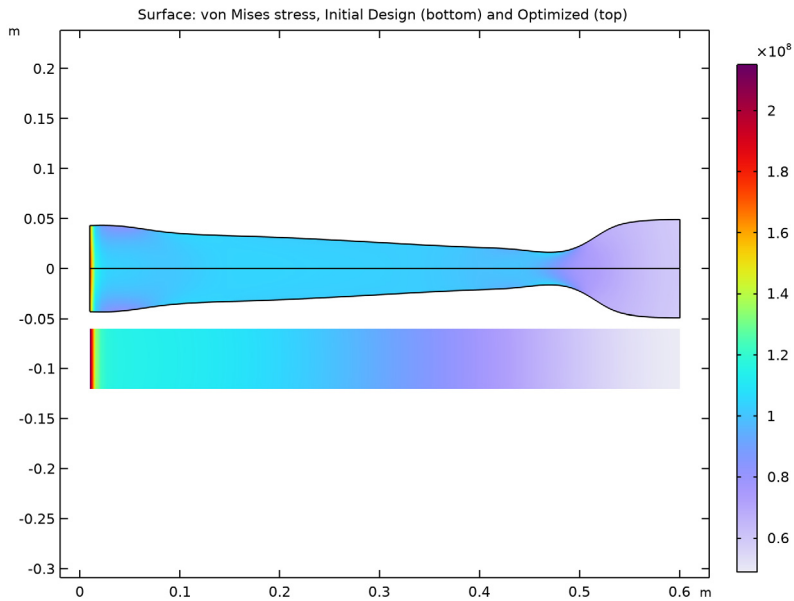


Figure 1: The von Mises stress for the initial and optimized flywheel design.

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 **Materials** 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 the **Add to Component** button in the window toolbar.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

## SOLID MECHANICS (SOLID)

### *Rotating Frame I*


- 1 In the **Physics** toolbar, click  **Domains** and choose **Rotating Frame**.
- 2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.
- 3 In the  $\Omega$  text field, type omega.

### *Symmetry Plane I*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry Plane**.
- 2 Select Boundary 2 only.

## MESH I

### *Mapped I*

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.

### *Mass Properties I (mass I)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Physics 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**.


### *P-norm I (pnorm I)*

- 1 In the **Definitions** toolbar, click  **Physics Utilities** and choose **P-norm**.
- 2 In the **Settings** window for **P-norm**, locate the **Geometric Entity Selection** section.

- 3 From the **Selection** list, choose **All domains**.
- 4 Click **Replace Expression** in the upper-right corner of the **P-norm** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Stress > comp1.solid.mises - von Mises stress - N/m<sup>2</sup>**.
- 5 Locate the **P-norm** section. From the *p* list, choose **User defined**.
- 6 In the **Norm** text field, type *pExp*.
- 7 Locate the **Quadrature Settings** section. Select the **Compute integral in revolved geometry** checkbox.


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*.

## STUDY 1: INITIAL DESIGN

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, type Study 1: Initial Design in the **Label** text field.
- 3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.
- 4 In the **Study** 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.
- 2 In the **Settings** window for **Global Evaluation**, type Initial Constraints in the **Label** text field.
- 3 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1) > Definitions > pnorm1 - P-norm - N/m<sup>2</sup>**.
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
pnorm1	MPa	p-norm

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


## COMPONENT 1 (COMP1)

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


### *Free Shape Domain 1*

In the **Physics** toolbar, click  **Optimization** and choose **Shape Optimization**.

### *Symmetry/Roller 1*

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

### *Free Shape Boundary 1*



- 1 In the **Shape Optimization** toolbar, click  **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 `a` text field, type  $2*H0/3$ .
- 5 Locate the **Filtering** section. From the  $R_{min}$  list, choose **Small**.

## DEFINITIONS

### *P-norm 1 (pnorm1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Definitions** click **P-norm 1 (pnorm1)**.
- 2 In the **Settings** window for **P-norm**, locate the **P-norm** section.
- 3 In the `a` text field, type `solid.mises/sigma0`.

## 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 the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

## STUDY 1: INITIAL DESIGN

### *Step 1: Stationary*

- 1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 2 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Deformed Geometry**.

## STUDY 2: SHAPE OPTIMIZATION

- 1 In the **Model Builder** window, click **Study 2**.
- 2 In the **Settings** window for **Study**, type Study 2: 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 vertices**.

## STUDY 2: SHAPE OPTIMIZATION


### *Shape Optimization*

- 1 In the **Study** toolbar, click  **Optimization** and choose **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·m<sup>2</sup>.**

- 5 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1) > Definitions > Free Shape Domain 1 > comp1.fsd1.relVolume - Material volume divided by geometry volume - I**.
- 6 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1) > Definitions > comp1.pnorm1 - P-norm - I**.  
Scale the moment of inertia with the initial value. Constrain the volume and stress from above, while maximizing the moment of inertia.
- 7 Locate the **Objective Function** section. Find the **Objective settings** subsection. 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.fsd1.relVolume		1
comp1.pnorm1		1

- 10 Click to expand the **Output** section. Select the **Plot** checkbox.
- 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.

### *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 **Study 2: Shape Optimization/Solution 2 (sol2)**.

#### *Stress (solid)*

- 1** In the **Model Builder** window, under **Results** 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, under **Results > Stress (solid)** 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  $\text{if}(\text{mir1side}, 1, -1) * -3 * H0$ .
- 5** Locate the **Scale** section.
- 6** Select the **Scale factor** checkbox. 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.