Created in COMSOL Multiphysics 6.1

Stresses in a Pulley

Introduction

This example contains a study of the stress distribution in a driving pulley. The analysis shows the stresses as functions of the pulley's angular velocity.

Model Definition

Figure 1 shows the pulley under study (to the right) and the external forces applied due to the driving belt.

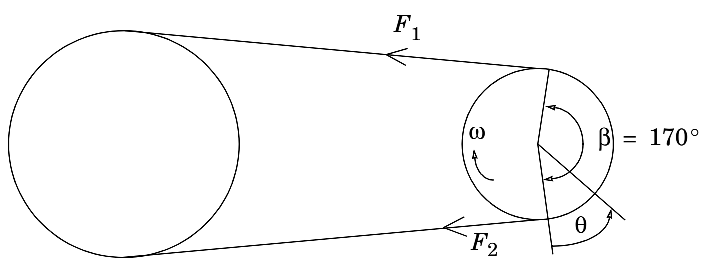

Figure 1: Pulley and driving belt with the external forces F_1 and F_2.

Here, F_1 and F_2 are the loads in the load side and in the slack side of the belt, respectively. The relationship between these forces is given by the capstan equation (or Eytelwein's formula, as it is referred to in the German literature):

$$\frac{F_1}{F_2} = e^{\mu\beta}$$

where μ is the coefficient of friction and β is the contact angle between the belt and the pulley. This equation is valid if a condition of impending slippage between the belt and the pulley prevails.

It is also necessary to state that the peripheral force (the force that transmits the power) is

$$F_u = F_1 - F_2 = \frac{M}{R}$$

Where R is the outer radius of the pulley and M is the applied torque. It is then possible to define F_2 as

$$F_2 = \frac{F_u}{(e^{\mu\beta} - 1)}$$

Now that you know the force in the slack side of the belt, you can derive the loads on the boundary of the pulley. In mechanics and the theory for flexible wires the following equilibrium equations appear:

$$F_n = \frac{S}{R}$$

$$F_t = \mu F_n$$

where F_n is the normal component (directed inward) and F_t the tangential component (directed against the rotation) of the external forces, S is the tension force, and R is the radius.

If you apply these equations, the loads on the boundary of the pulley become

$$F_n(\theta) = \frac{F_2}{R} e^{\mu\theta}$$

and

$$F_t(\theta) = \mu F_n(\theta)$$

where θ is the angle for which the forces are calculated (see Figure 1).

Due to the rotation of the pulley, inertia loads are generated. These loads can be calculated as

$$F_r = r\omega^2\rho$$

where r is the radius, ω is the rotation speed, and ρ is the density.

The pulley is fixed at its inner diameter and the inertia loads are active in the entire geometry.

A parametric analysis shows how the rotational speed affects the stress distribution in the pulley. Because the power at the pulley shaft remains constant, the torque (defined as the ratio of the power by the rotational speed) decreases with increased speed. This means that with increased rotational speed, the inertial load increases while the driving-belt force decreases.

Results and Discussion

The following plots show the von Mises stress distribution inside the pulley for different rotational speeds in rpm (revolutions per minute).

As is evident from the plots, the stress distribution changes as the rotational speed increases.

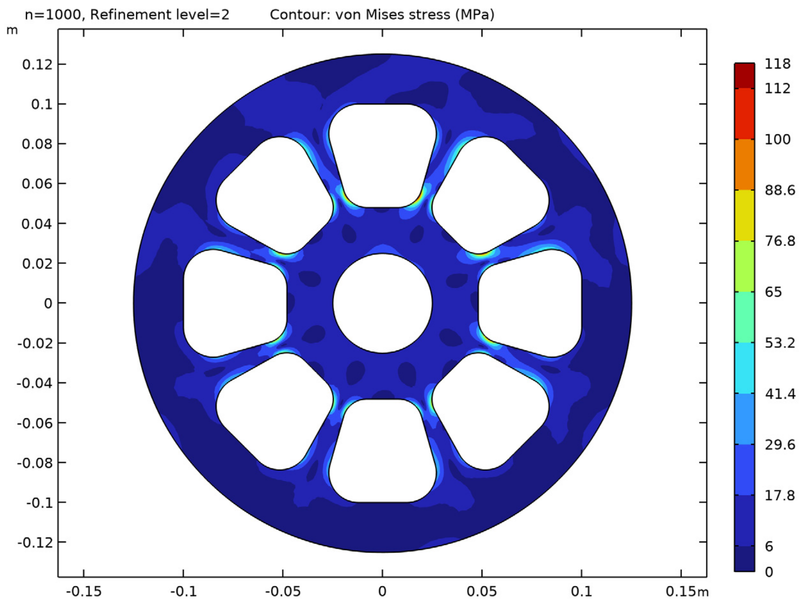

Figure 2: von Mises stress distribution at n = 1000 rpm.

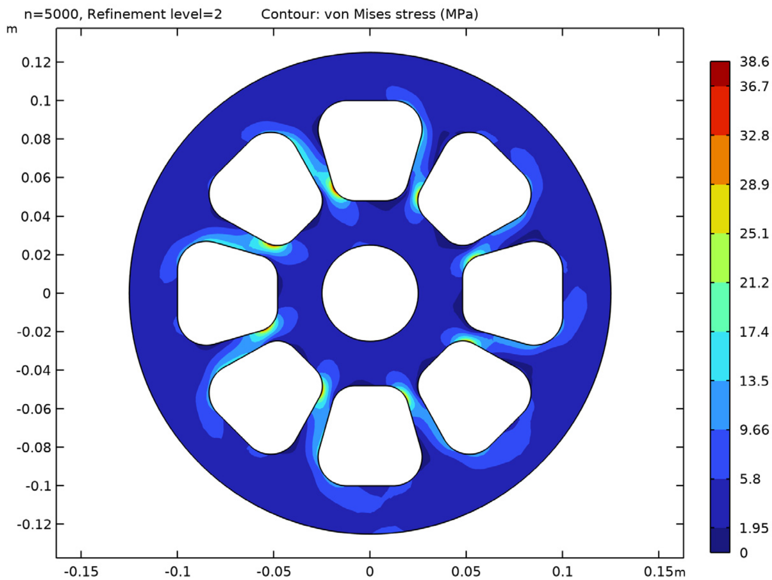

Figure 3: von Mises stress distribution at n = 5000 rpm.

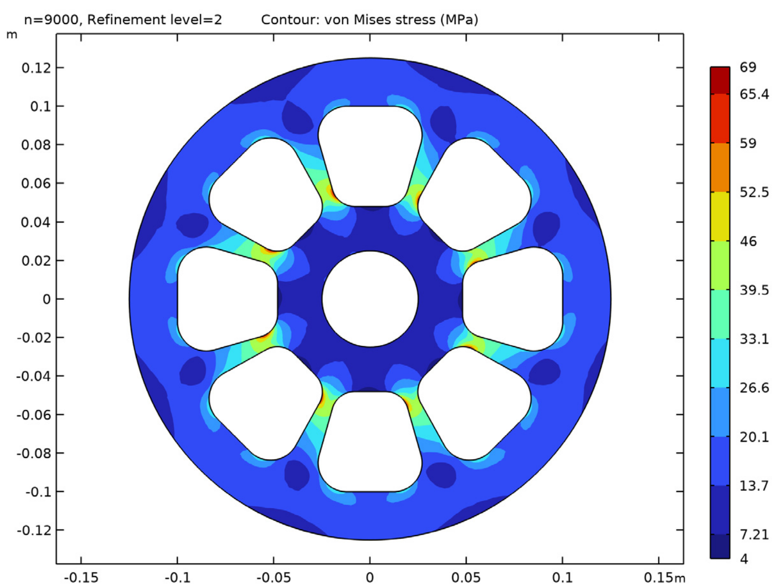

Figure 4: von Mises stress distribution at n = 9000 rpm.

At the point (0.019, 0.054), the von Mises stress is maximal for the first rotational speed (n = 1000 rpm). The plot in Figure 5 shows how the rotational speed affects the von Mises stress at this specific point. First the stress decreases, but then the effect of the inertial loads becomes dominating and the stress begins to increase.

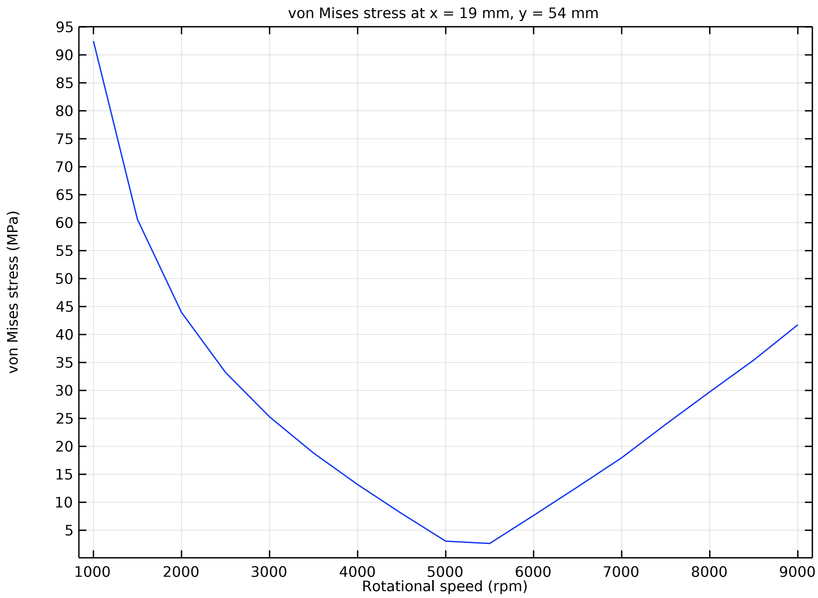

Figure 5: von Mises stress as function of rotational speed at point (0.019, 0.054).

Figure 6 shows the von Mises stresses in as part of the pulley. The stress field was evaluated using the accurate derivative recovery method, which makes it smoother.

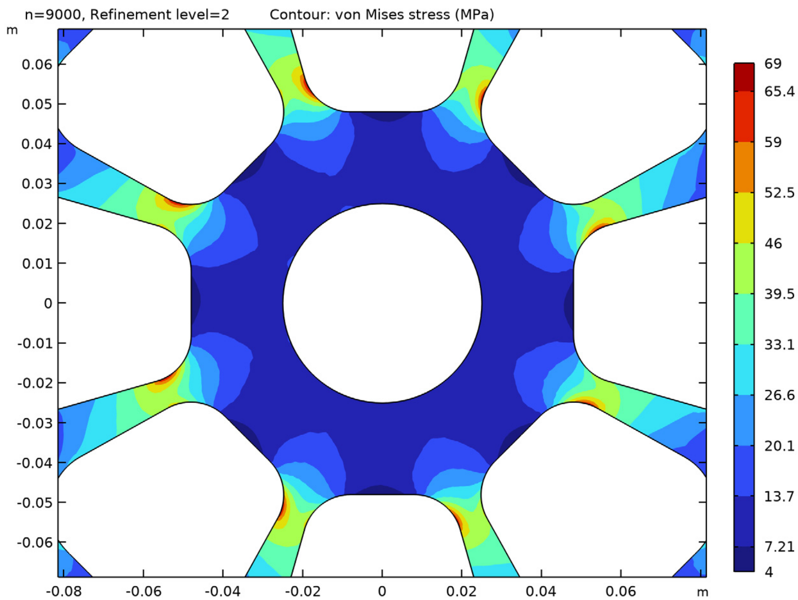

Figure 6: von Mises stress in a part of the pulley evaluated using accurate derivative recovery.

In this example, dynamic effects have been ignored. In reality, it is possible that vibrations in the pulley could occur if the rotational speed coincides with natural frequencies of the pulley.

Notes About the COMSOL Implementation

When solving, adaptive mesh refinement helps to compute accurate stresses, as the stress concentration is not known in advance.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/stresses_in_pulley`

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

GEOMETRY 1

Import the model geometry.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 From the **Source** list, choose **COMSOL Multiphysics file**.
4 Click **Browse**.
5 Browse to the model's Application Libraries folder and double-click the file `stresses_in_pulley.mphbin`.
6 Click **Import**.

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 `stresses_in_pulley_parameters.txt`.

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Global Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.
3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `stresses_in_pulley_variables.txt`.

MATERIALS

Define aluminum as the pulley material.

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**.

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

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Cylindrical System 2 (sys2)

In the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

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**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `d`.

Body Load 1

1 In the **Physics** toolbar, click **Domains** and choose **Body Load**.

2 Select Domain 1 only.

3 In the **Settings** window for **Body Load**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{V}$ vector as

`r*omega^2*solid.rho`	r
`0`	phi

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 57, 58, 62, and 63 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

Boundary Load 1

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

2 Select Boundaries 67 and 68 only.

3 In the **Settings** window for **Boundary Load**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

-Fn	r
Ft	phi

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 **Fine**.

4 Click **Build All**.

The mesh should contain about 2600 elements.

STUDY 1

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
n (Revolutions per minute)	range(1e3,5e2,9e3)

Since the stress concentrations depend on the rpm, mesh adaptation must be run separately for each case. Therefore, disable the parametric solver.

5 Click to expand the **Advanced Settings** section. From the **Use parametric solver** list, choose **Off**.

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Adaptation and Error Estimates** section.

3 From the **Adaptation and error estimates** list, choose **Adaptation and error estimates**.

4 From the **Error estimate** list, choose **Functional**.

5 From the **Functional type** list, choose **Manual**.

6 In the **Functional** text field, type `comp1.solid.Ws_tot`.

7 Find the **Mesh adaptation** subsection. From the **Adaptation method** list, choose **Rebuild mesh**.

Change the **Adaptive Mesh Refinement** solver settings to use an error estimate which is more sensitive to stress concentrations. The total elastic energy has this property, since it is in fact quadratic in the local stress. As the critical stress concentration regions are small, select **Mesh initialization** as refinement method, since it can refine more aggressively in small areas.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Adaptive Mesh Refinement**.

4 In the **Settings** window for **Adaptive Mesh Refinement**, locate the **General** section.

5 Find the **Mesh adaptation** subsection. In the **Element count growth factor** text field, type `1.3`.

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

RESULTS

von Mises

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `von Mises` in the **Label** text field.

Use a filled contour plot to clearly see where different stress levels occur. For that purpose, add a contour plot and define its settings to display the von Mises stress variation.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

Contour 1

1 Right-click **von Mises** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics> Stress>solid.mises - von Mises stress - N/m²**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 Locate the **Levels** section. In the **Total levels** text field, type `10`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

6 In the **von Mises** toolbar, click **Plot**.

You can also experiment with different quality settings.

7 Click to expand the **Quality** section. From the **Resolution** list, choose **Fine**.

8 From the **Smoothing** list, choose **Everywhere**.

9 From the **Recover** list, choose **Within domains**.

The last setting switches on the accurate derivative recovery method, which makes the stress field smoother.

10 In the **von Mises** toolbar, click **Plot**.

To visualize the results for different rotational speeds, use the solution for the desired rpm value by selecting it from the **Parameter value** list in the **Settings** window for the **2D Plot Group**.

von Mises

1 In the **Model Builder** window, click **von Mises**.

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

3 From the **Parameter value (n)** list, choose **1000**.

4 In the **von Mises** toolbar, click **Plot**.

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

6 From the **Parameter value (n)** list, choose **5000**.

7 In the **von Mises** toolbar, click **Plot**.

8 From the **Parameter value (n)** list, choose **9000**.

9 In the **von Mises** toolbar, click **Plot**.

To get a line plot of the von Mises stress at a specific point as a function of the rotational speed, make use of the feature **Cut Point 2D** in a **1D Plot Group**.

Cut Point 2D 1

1 In the **Results** toolbar, click **Cut Point 2D**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 Locate the **Point Data** section. In the **X** text field, type `0.019`.

5 In the **Y** text field, type `0.054`.

von Mises vs. Rotational Speed

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `von Mises vs. Rotational Speed` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 2D 1**.

4 From the **Parameter selection (Refinement level)** list, choose **Last**.

Point Graph 1

1 Right-click **von Mises vs. Rotational Speed** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>solid.mises - von Mises stress - N/m²**.

3 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **n**.

von Mises vs. Rotational Speed

1 In the **Model Builder** window, click **von Mises vs. Rotational Speed**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box. In the associated text field, type `Rotational speed (rpm)`.

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

5 In the **Title** text area, type `von Mises stress at x = 19 mm, y = 54 mm`.

Notice that by selecting different solution-set entries in the **Dataset** list in the **Settings** window for the 1D and 2D plot groups, you can visualize results at different mesh refinements.

6 In the **von Mises vs. Rotational Speed** toolbar, click **Plot**.

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

von Mises

Finally, zoom in on the center of the pulley to get a close-up view of the stress distribution. In particular, notice that the accurate derivative recovery method gives a smoother and more accurate solution.

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