Model created in COMSOL Multiphysics 6.4

Fatigue Failure of an Eyeglass Frame

Introduction

In the search for weight reduction, the cross section of an eyeglass frame is continuously reduced. The thin section over the nose transfers the entire load between the two halves. This example predicts the fatigue life using the combined Basquin and Coffin–Manson model when eyeglasses are subjected to bending.

Model Definition

The frame of the eyeglasses is made of the MONEL alloy 400. The eyeglass lenses are made of a CR-39 material that is lighter in weight than glass. The risk of fatigue in the lenses is avoided by using a coating that holds together any shards in case of fracture. Therefore only the fatigue life of the frame is predicted.

The frame of the eyeglasses is very thin, 1 mm, and has a shape according to Figure 1.

Figure 1: Shape of the eyeglasses.

Young's modulus of the MONEL alloy 400 is taken from the COMSOL Material Library and the Poisson's ratio is 0.32. For the CR-39 material, the Young's modulus is 2.1 GPa and the Poisson's ratio is 0.4.

Fatigue data for the MONEL alloy 400 has been obtained in rotating bending tests, and fitted to the combined Basquin and Coffin–Manson relation according to where the strain amplitude, ε_a, is expressed as

$$\varepsilon_a = \frac{\sigma_f'}{E} \cdot (2N)^{-b} + \varepsilon_f' \cdot (2N)^{-c}$$

where ε_a is the strain amplitude, E is the Young's modulus, N represents the number of cycles to failure, and σ_f', ε_f', b, and c are fatigue material parameters.

Bending of the eyeglasses is simulated by fixing one side of the eyeglasses and applying an alternating vertical 4 N force on the other side of the eyeglasses.

Results and Discussion

The stress distribution in the eyeglasses at the peak load is shown in Figure 2.

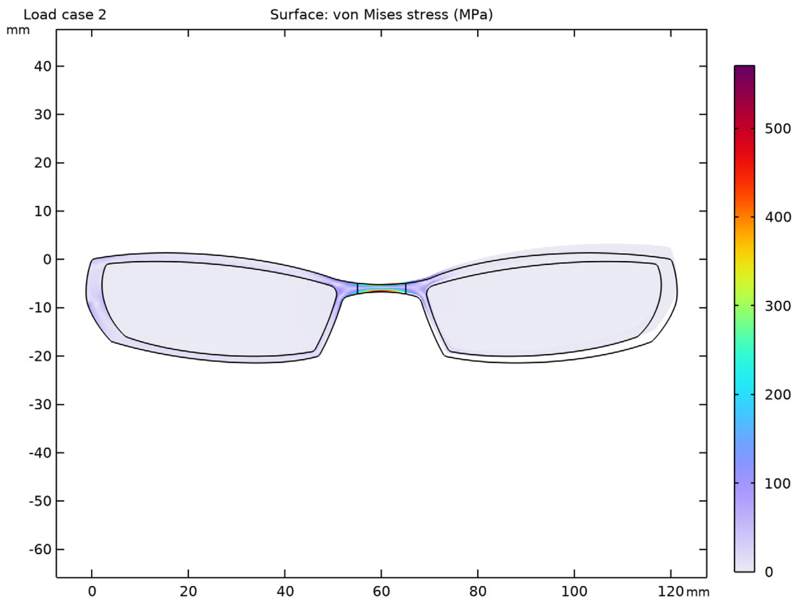


Figure 2: Equivalent stress in eyeglasses.

Bending of the eyeglasses causes high stresses on both sides of the thin central part. Since the equivalent stress does not discriminate between tension and compression the same stress levels are encountered on both sides. The resulting stress contours are similar if the bending is reversed when the right part of the glasses is pulled down instead.

Since the fatigue data is obtained in a rotating bending test, the stresses and strains alternate between tension and compression during one test cycle. This is exactly the same structural behavior as in one load cycle of the eyeglasses and therefore the fatigue curve is directly applicable to the resulting principal strains. The highest and smallest principal strains at peak loads are shown in Figure 3.

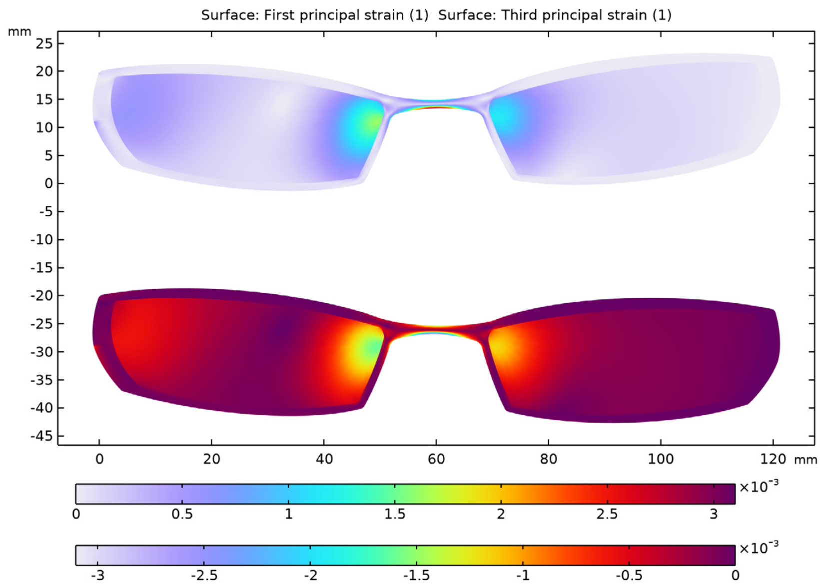


Figure 3: Principal strains in the eyeglasses. Above: first principal strain when pulling upward. Below: third principal strain when pulling downward.

When pulling up, the lower side of the thin central section experiences peak tensile strains. The peak compressive strains are experienced in the same point when pulling the eyeglasses down. On the upper side of the thin central section over the nose, an opposite situation is encountered, with peak compression when pulling up and peak tension when pulling down. The difference in strain during both load events controls the fatigue life that is predicted in Figure 4. The estimated life is approximately 100,000 cycles.

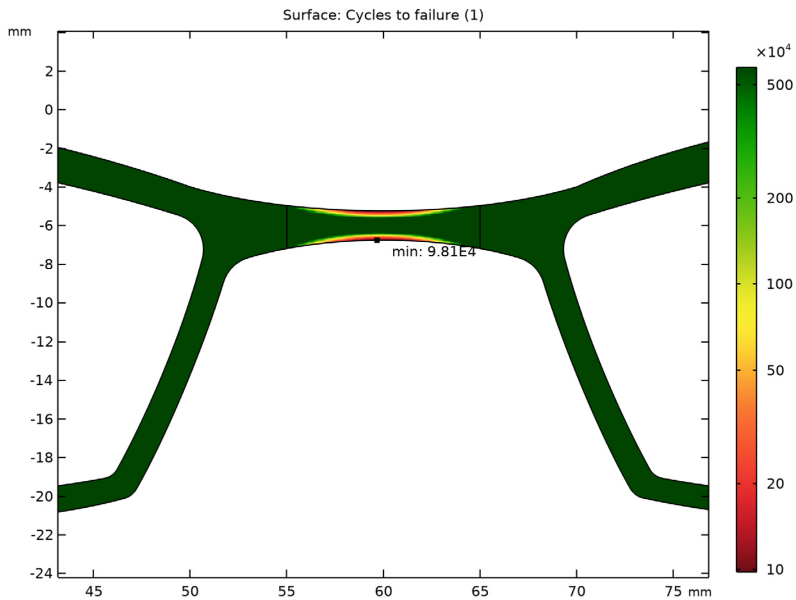


Figure 4: Fatigue life computed by the combined Basquin and Coffin–Manson relation.

Application Library path: `Fatigue_Module/Strain_Life/eyeglass_frame_fatigue`

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

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Import 1 (imp1)

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

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

3 Click **Browse**.

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

5 Click **null**.

The thin central part of the eyeglasses transfers the entire load between the two halves. Make a rectangle in the center to create a domain for fine structured mesh.

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 `10`.

4 In the **Height** text field, type `4`.

5 Locate the **Position** section. In the **x** text field, type `55`.

6 In the **y** text field, type `-8`.

Intersection 1 (int1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Intersection**, locate the **Intersection** section.

4 Select the **Keep input objects** checkbox.

Delete Entities 1 (del1)

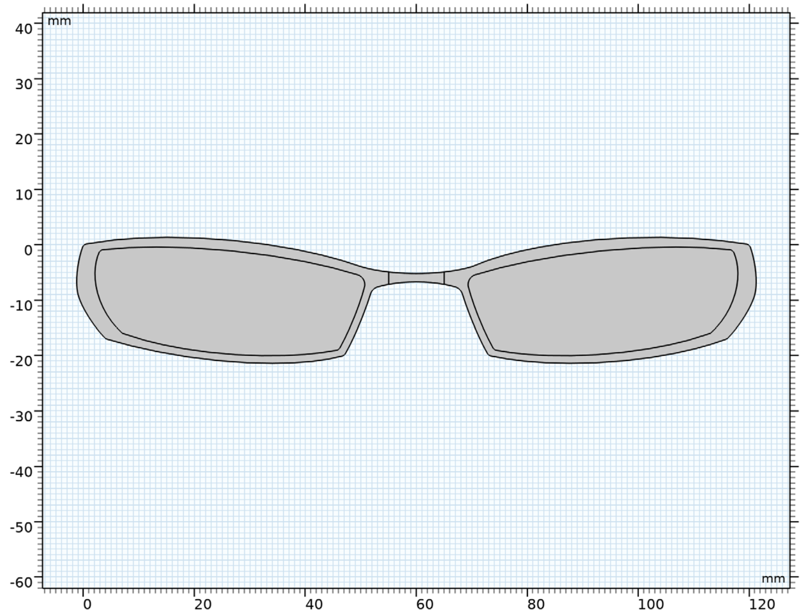
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **r1**, select Domain 1 only.

5 Click **Build All Objects**.



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 **Search** text field, type `monel 400`.

4 Click **Search**.

5 In the tree, select **Material Library** > **Nickel Alloys** > **Monel 400** > **Monel 400 [solid]** > **Monel 400 [solid,annealed]**.

6 Select Domains 1–3 only.

7 Click the **Add to Component** button in the window toolbar.

8 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Monel 400 [solid,annealed] (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.32	kg/m³	Basic

CR-39

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `CR-39` in the **Label** text field.

3 Select Domains 4 and 5 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	2.1[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.4	1	Young's modulus and Poisson's ratio
Density	rho	1.3[g/cm^3]	kg/m³	Basic

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 **Thickness** section.

3 In the d text field, type `0.001`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 3 only.

Boundary Load 1

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

2 Select Boundary 42 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

0	x
1	y

6 In the **Physics** toolbar, click **Load Group** and choose **New Load Group**.

MESH 1

Mapped 1

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

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

4 Select Domain 2 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `15`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 22 only.

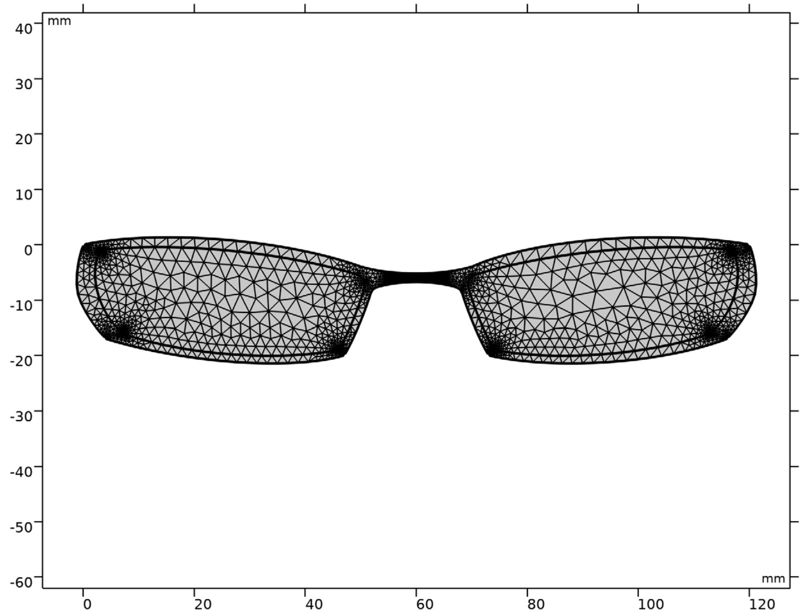
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `20`.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Define load cases** checkbox.

4 Click **Add** four times.

5 In the table, enter the following settings:

Load case	lg1	Weight
Load case 1	√	0
Load case 2	√	4
Load case 3	√	-4
Load case 4	√	0

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

RESULTS

Stress (solid)

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

2 From the **Load case** list, choose **Load case 2**.

Surface 1

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

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

3 From the **Unit** list, choose **MPa**.

Deformation

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

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

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

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

Stress (solid)

The equivalent stress does not discriminate between tension and compression. Evaluate how strains change at peak loads.

Principal strain (solid)

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Principal strain (solid)` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

4 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

Surface 1

1 In the **Model Builder** window, expand the **Principal strain (solid)** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Load case** list, choose **Load case 2**.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.ep1`.

Display both the largest principal strain and the smallest principal strain in the same figure.

Deformation

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

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

3 In the **Y-component** text field, type `v+0.020`.

Surface 2

1 In the **Model Builder** window, under **Results** > **Principal strain (solid)** right-click **Surface 1** and choose **Duplicate**.

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

3 From the **Load case** list, choose **Load case 3**.

4 Locate the **Expression** section. In the **Expression** text field, type `solid.ep3`.

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 **Y-component** text field, type `v-0.020`.

4 In the **Principal strain (solid)** toolbar, click **Plot**.

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 **Structural Mechanics** > **Fatigue (ftg)**.

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

5 Click the **Add to Component 1** button in the window toolbar.

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

FATIGUE (FTG)

Strain-Life 1

1 In the **Physics** toolbar, click **Domains** and choose **Strain-Life**.

2 Select Domains 1–3 only.

3 In the **Settings** window for **Strain-Life**, locate the **Fatigue Model Selection** section.

4 From the **Criterion** list, choose **Combined Basquin and Coffin–Manson**.

5 Locate the **Solution Field** section. From the **Physics interface** list, choose **Solid Mechanics (solid)**.

6 Locate the **Evaluation Settings** section. In the N_{cut} text field, type `5.75e6`.

MATERIALS

Monel 400 [solid,annealed] (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Monel 400 [solid,annealed] (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Fatigue strength coefficient	sigmaf_Basquin	`970[MPa]`	Pa	Basquin
Fatigue strength exponent	b_Basquin	`-0.077`	1	Basquin
Fatigue ductility coefficient	epsilonf_CM	`0.738`	1	Coffin-Manson
Fatigue ductility exponent	c_CM	`-0.54`	1	Coffin-Manson

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 **Solid Mechanics (solid)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Fatigue**.

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

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

STUDY 2

Step 1: Fatigue

1 In the **Settings** window for **Fatigue**, locate the **Values of Dependent Variables** section.

2 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

3 From the **Method** list, choose **Solution**.

4 From the **Study** list, choose **Study 1, Stationary**.

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

The plot generated by default shows cycles to failure. Zoom in to reproduce Figure 4.