Model created in COMSOL Multiphysics 6.4

Cycle Counting in Fatigue Analysis — Benchmark

Introduction

In this benchmark model for the Rainflow counting algorithm, values computed by the Fatigue Module in COMSOL are compared with the ASTM standard E1049-85 (Ref. 1). An extension to the benchmark compares the Palmgren–Miner cumulative damage model to analytical expressions.

Model Definition

A flat test specimen is subjected to a repeated load cycle. The material has elastic properties defined by Young's modulus E = 69 GPa and Poisson's ratio ν = 0.34. The thickness of the specimen is 6.25 mm. The remaining dimensions are given in Figure 1.

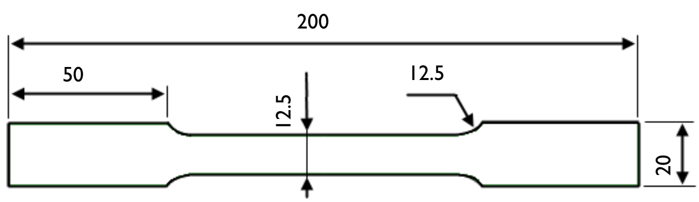

Figure 1: Flat test specimen. Dimensions are given in mm.

The Wöhler diagram (S-N curve) is given by the expression

$$\sigma_a = 94e6(R/(-0.36))^{1,15}N^{-0.119}$$

where σ_a is the stress amplitude, R is the R-value and N is the number of cycles to failure for a constant stress cycle defined by σ_a and R. The relation is valid for $-2.5 \le R \le -0.2$, while the parameter $N \ge 10^8$ is seen as infinite life and it should not be taken into account in damage calculations.

An ASTM cycle (Ref. 1) is evaluated in the example. The load history of the cycle is presented in Table 1.

TABLE 1: FATIGUE CYCLE.

Step	Load units
1	-2
2	1
3	-3
4	5
5	-1

TABLE 1: FATIGUE CYCLE.

Step	Load units
6	3
7	-4
8	4
9	2

The unit load can be chosen arbitrarily and is here selected so that one unit load corresponds to 10 MPa stress in the central cross section.

The ASTM example is further extended to examine the fatigue damage caused by 100000 blocks of the cycle.

In the test specimen, the stress state in the central part away from the fillets does not vary with the position. Therefore, any point in the central thin cross section can be chosen for evaluation.

Results and Discussion

Because of the symmetry, only a quarter of the test specimen is modeled. The 2D Solid Mechanics interface is used with a plane stress assumption. Figure 2 shows the axial stress caused by a unit load. Although a quarter of the specimen was modeled, the results for the whole specimen can be examined using datasets of the type **Mirror 2D**.

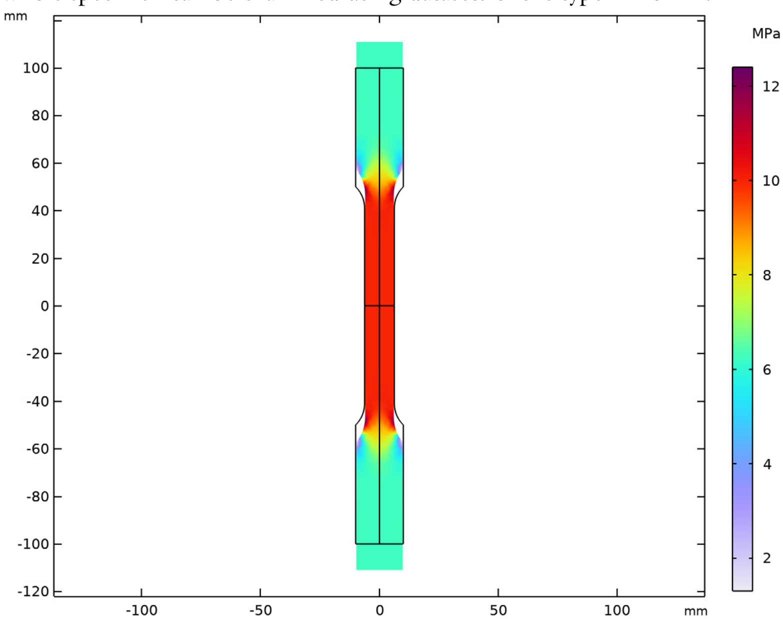

Figure 2: Axial stress in the specimen caused by a unit load.

The **Cumulative Damage** feature generates results of the cycle counting algorithm as well as a damage calculation. A point in the thin central cross section is evaluated. The applied load cycle, see Table 1, is quantified with the Rainflow Counting algorithm and shown in Figure 3. The ASTM results (Ref. 1) are shown in Table 2

TABLE 2: ASTM RAINFLOW COUNTING RESULTS.

σ_a (MPa)	σ_m (MPa)	n
20	-10	0.5
15	-5	0.5
40	0	0.5
45	5	0.5
40	10	0.5
30	10	0.5
1	10	1.0

here, σ_m is the cycle mean stress amplitude and n is number of cycles.

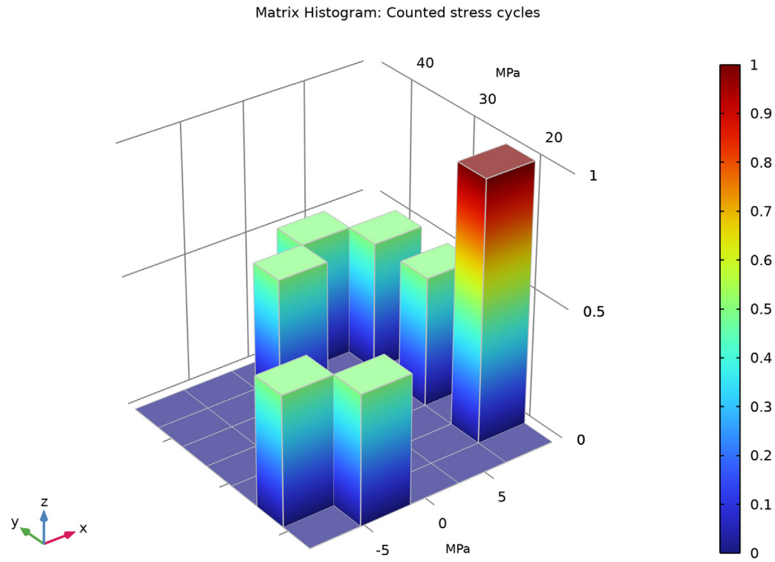

Figure 3: Load cycle quantified with the Rainflow Counting option.

The Rainflow Counting results by ASTM and COMSOL are in perfect agreement.

The evaluation of the cumulative damage is now compared against analytical expressions. In the Palmgren–Miner model, the damage is calculated for each stress bin. The number of cycles to failure for a constant cycle is taken from the S-N curve which is evaluated at the center of the bin.

With the chosen bin discretization the damage is evaluated at following bin stress centers:

$$\sigma_a^b = 17.1\text{ MPa}, 21.4\text{ MPa}, 25.7\text{ MPa}, 30.0\text{ MPa}, 34.3\text{ MPa}, 38.6\text{ MPa}, 42.9\text{ MPa}$$

and

$$\sigma_m^b = -8.0\text{ MPa}, -4.0\text{ MPa}, 0.0\text{ MPa}, 4.0\text{ MPa}, 8.0\text{ MPa}$$

The key values for the damaging bins are presented in Table 3. The superscript 'b' denotes that the variable is evaluated at the bin center.

TABLE 3: DAMAGING BIN DATA.

σ_a (MPa)	σ_m (MPa)	$\sigma_a{}^b$ (MPa)	$\sigma_m{}^b$ (MPa)	R^b	n^b	N^b
20.0	-10.0	21.4	-8.0	-2.19	0.500	Inf
15.0	-5.0	17.1	-4.0	-1.61	0.500	Inf
40.0	0.0	38.6	0.0	-1.00	0.500	3.44e7
45.0	5.0	42.9	4.0	-0.829	0.500	2.31e6
40.0	10.0	38.6	8.0	-0.656	0.500	5.84e5
30.0	10.0	30.0	8.0	-0.579	0.500	1.45e6
20.0	10.0	21.4	8.0	-0.456	1.00	2.47e6

The fatigue usage factor f_{us} following the Palmgren–Miner linear damage rule is calculated using the expression

$$f_{\mathrm{us}} = m \sum_{i=1}^{p} n_i^b / N_i^b \tag{1}$$

where m is the number or repeatable block and p is the number of bins.

Following Equation 1 the fatigue usage factor of the ASTM cycles is $f_{\mathrm{us}}^{\mathrm{ASTM}} = 0.184$. The COMSOL result is $f_{\mathrm{us}} = 0.182$. The small discrepancy between the results is attributed to the evaluation of the S-N curve.

The relative damage contribution from each bin is shown in Figure 4.

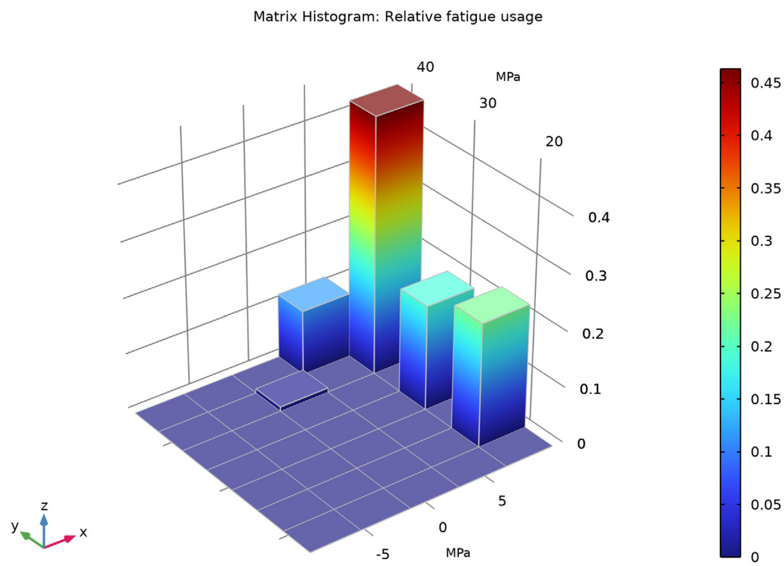

Figure 4: Contribution of each stress bin to the fatigue usage.

The Cumulative Damage model is based on the Palmgren–Miner linear damage rule. It is a discrete model in the sense that the calculations are based on stress bins which hold all stress cycles within a certain stress amplitude and mean stress range.

In this example, the stress amplitude range in a bin is 20 MPa/5=4 MPa and the mean stress range in a bin is 30 MPa/7 MPa = 4.3 MPa. By changing the bin discretization, the graph showing counted stress cycles in Figure 3, appears with a different resolution.

As for the results of the relative damage in Figure 4, a change in discretization can affect the results significantly. The damage is evaluated based on the bin stresses and not true cycle stresses. Consider the cycle defined by σ_a = 45.0 MPa and σ_m = 5.0 MPa. It is evaluated in a bin defined by σ_a^b = 42.9 MPa and σ_m^b = 4.0 MPa. Since bin stresses are lower than true stresses in that cycle, they predict less damage and thus give a nonconservative contribution to the fatigue usage factor.

Notes About the COMSOL Implementation

In the Cumulative Damaged feature, you can use three different types of S-N curves — **S-N curve with R-value dependence**, **S-N curve with mean stress dependence**, or **S-N curve for**

amplitude stress. In this model, the S-N curve is defined using the R-value definition. The arguments to this S-N curve are specified by first giving the R-value followed by the number of cycles, as it is done in the Analytic function in the Arguments field.

Reference

1. ASTM International, Standard Practices for Cycle Counting in Fatigue Analysis, Designation: E1049-85 (Reapproved 2011).

Application Library path: `Fatigue_Module/Verification_Examples/cycle_counting_benchmark`

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**.

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 `100`.

Rectangle 2 (r2)

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 `10-6.25`.

4 In the **Height** text field, type `50-sqrt(12.5^2-8.75^2)`.

5 Locate the **Position** section. In the **x** text field, type `6.25`.

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Position** section.

3 In the **x** text field, type `6.25+12.5`.

4 In the **y** text field, type `50-sqrt(12.5^2-8.75^2)`.

5 Locate the **Size and Shape** section. In the **Radius** text field, type `12.5`.

Difference 1 (dif1)

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

2 From the bigger rectangle subtract the smaller rectangle and the circle.

GLOBAL DEFINITIONS

Specify the load cycle.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Interpolation**.

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

3 In the table, enter the following settings:

t	f(t)
1	-2
2	1
3	-3
4	5
5	-1
6	3
7	-4

t	f(t)
8	4
9	-2

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 `0.00625`.

Symmetry 1

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

2 Select Boundaries 1 and 2 only.

Boundary Load 1

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

2 Select Boundary 3 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 $\mathbf{F}_{\mathrm{tot}}$ vector as

0	x
F*int1(case)	y

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
F	10*6.25*12.5/2	390.63	Load unit
case	1	1	Load case

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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
Young's modulus	E	69e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.34	1	Young's modulus and Poisson's ratio
Density	rho	2700	kg/m³	Basic

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 **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list
case (Load case)	`range(1,1,9)`

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Apply**.

Stress (solid)

Mirror solution of a quarter of a specimen and create results for a full specimen.

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 **Y** to `100`.

Mirror 2D 2

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

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

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

4 Locate the **Axis Data** section. In row **Point 1**, set **x** to `-6.25`.

5 In row **Point 2**, set **x** to `6.25` and **y** to `0`.

Display stress state in the whole specimen.

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 From the **Parameter value (case)** list, choose **2**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **Second Piola–Kirchhoff stress (material and geometry frames) - N/m²** > **solid.SGpYY - Second Piola–Kirchhoff stress, YY-component**.

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

Load Cycle Response

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

2 In the **Settings** window for **1D Plot Group**, type `Load Cycle Response` in the **Label** text field.

Point Graph 1

1 Right-click **Load Cycle Response** and choose **Point Graph**.

2 Select Point 3 only.

3 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** > **Second Piola–Kirchhoff stress (material and geometry frames) - N/m²** > **solid.SGpYY - Second Piola–Kirchhoff stress, YY-component**.

4 In the **Load Cycle Response** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Specify the load cycle.

Analytic 1 (an1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global** > **Analytic**.

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

3 In the **Arguments** text field, type `R, N`.

4 In the **Expression** text field, type `(94e6*(R/-0.36)^1.15)*N^-0.119`.

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)

Cumulative Damage 1

1 In the **Physics** toolbar, click **Points** and choose **Cumulative Damage**.

Any point in the thin section away from the notch will give the same fatigue response.

2 Select Point 3 only.

3 In the **Settings** window for **Cumulative Damage**, locate the **Solution Field** section.

4 From the **Physics interface** list, choose **Solid Mechanics (solid)**.

5 Locate the **Cycle Counting Parameters** section. Find the **Discretization** subsection. In the N_{m} text field, type `5`.

6 In the N_{r} text field, type `7`.

7 Locate the **Damage Model Parameters** section. From the $f_{\mathrm{SN}}(R,N)$ list, choose **an1**.

8 In the m text field, type `100000`.

Set cutoff which can be seen as a limit for infinite life that does not contribute to the damage.

9 Find the **Evaluation settings** subsection. In the N_{cut} text field, type `1e8`.

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**.

RESULTS

Matrix Histogram 1

1 In the **Model Builder** window, expand the **Stress Cycle Distribution (ftg)** node, then click **Matrix Histogram 1**.

2 In the **Settings** window for **Matrix Histogram**, locate the **Axes** section.

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

4 In the **Stress Cycle Distribution (ftg)** toolbar, click **Plot**.

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

Matrix Histogram 1

1 In the **Model Builder** window, expand the **Fatigue Usage Distribution (ftg)** node, then click **Matrix Histogram 1**.

2 In the **Settings** window for **Matrix Histogram**, locate the **Axes** section.

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

4 In the **Fatigue Usage Distribution (ftg)** toolbar, click **Plot**.

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