Frame with Cutout

Introduction

A thin-walled frame member with a central cutout is subjected to a random load scenario. Although the stresses are expected to be far below the yield level of the material, a concern arises whether or not the component will fail due to fatigue.

This example demonstrates an approach to damage quantification of a long load history. The Rainflow counting algorithm is used to define the load scenario and Palmgren-Miner linear damage model quantifies the damage.

Model Definition

The load carrying beam is shown in Figure 1. It has a length of 1.1 m, a thin-walled square cross section having the dimension 160 mm x160 mm and a thickness of 6 mm. The cutout is centrally placed on one of the faces and is 100 mm long, 80 mm wide and has a fillet with radius 10 mm in each corner.

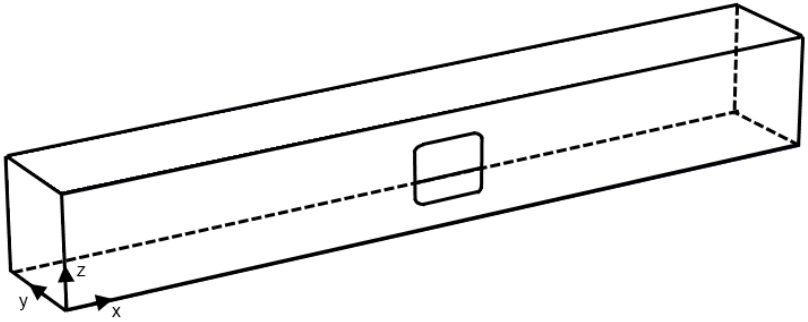

Figure 1: Geometry of the frame member.

The loading consists of bending moments in both directions and a twisting moment. All three loads can vary independently. The information about the load is obtained using three strain gauges which are glied on the bottom and the back sides of the frame. The back side is the one opposite of the face with the hole. The location and position of the strain gauges are shown in Figure 2. Strain gauges 1 and 2 are oriented at a 45° angle to the global coordinate system while the 3rd gauge is aligned with the length of the face (the x-axis). Both are located 850 mm from the edge along the length and in the middle along the width of the faces.

©2013 COMSOL

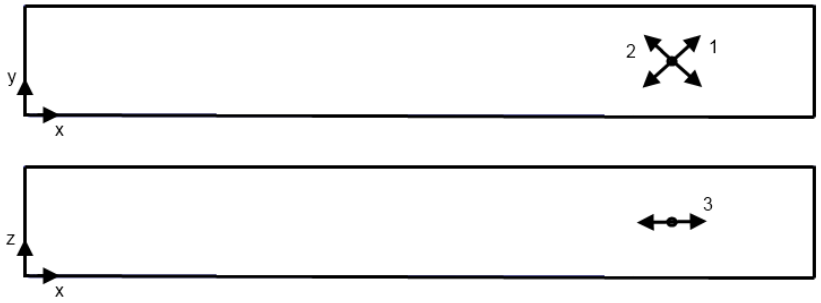

Figure 2: Location and orientation of the strain gauges.

The frame is made out of steel with Young's modulus, Poisson's ratio and density given by E = 200 GPa, ν = 0.33 and ρ = 7800 kg/m^3.

The fatigue behavior follows material library data for iron alloy 4340 for variant defined by phase **UTS 200 Ksi - 293K** and variation **unnotched**.

The left end of the structure, x = 0, is clamped. One twisting moment, along x, and two bending moments, along y and z, are applied on the right end at x = 1.1 m. The lifetime for which the frame is designed is 10000 longer than the what has been recorded be the strain gauges. The response to one loading cycle block in all three gauges is captured in Figure 3. From the history it is clear that the loading event is non-proportional. However stress concentrations will arise around the cutout where the stress state is uniaxial. Therefore used models, Rainflow Counting and Palmgren-Miner, can be seen as appropriate for fatigue evaluation.

©2013 COMSOL

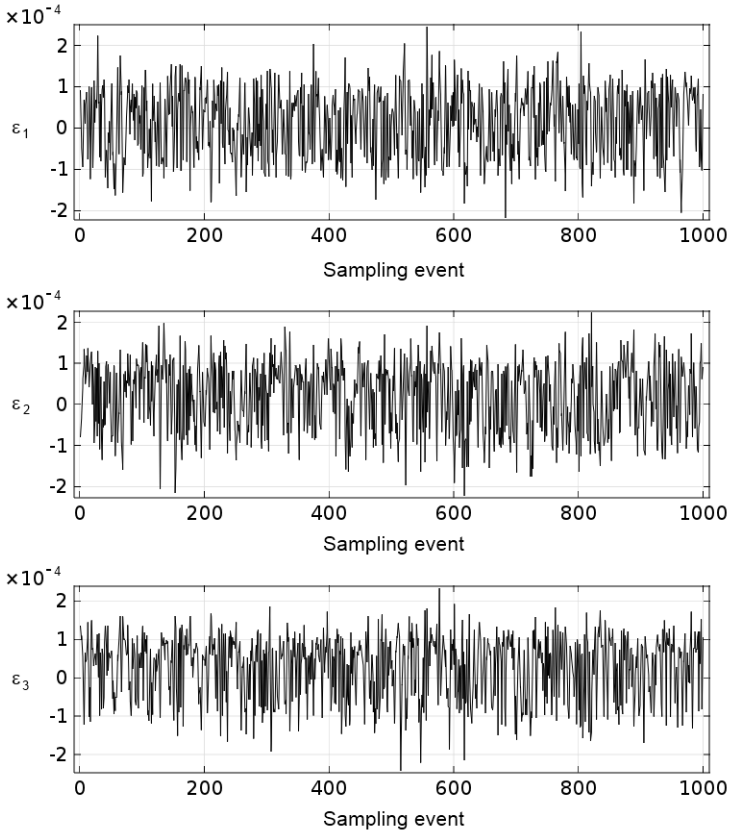

Figure 3: Strain history in tree gauges. The subscript indicates the gauge number.

In order to apply the strain history to the frame, the strain history needs to be transferred to loads which can be applied on the boundary. The transformation can be done in two ways. In the first one a relation between a unit moment and a response in each strain gauge must be obtained using COMSOL. By repeating it for each unit moment and inverting the relation a transformation matrix relating strains to moments will be obtained.

An alternative way is to use an approximative analytical relation based on beam theory with a thin-walled assumption, Hooke's law, and rotation of stresses in a plane. The result is given below without a detailed derivation.

©2013 COMSOL

$$\begin{aligned}\varepsilon_1 &= B_1(A_1C_1 + A_2D_1)M_Y + 2B_2(A_1C_3 + A_2D_3)M_X\\ \varepsilon_2 &= B_1(A_1D_1 + A_2C_1)M_Y + 2B_2(A_1D_3 + A_2C_3)M_X\\ \varepsilon_3 &= A_1B_1M_Z\end{aligned} \quad (1)$$

where $A_1 = 1/E$, $A_2 = -\nu/E$, $B_1 = 3(b+t)/(4tb^3)$, $B_2 = 1/(2tb^2)$, $C_1 = \cos 45^\circ \cos 45^\circ$, $C_2 = \sin 45^\circ \sin 45^\circ$, $C_3 = \cos 45^\circ \sin 45^\circ$, $D_1 = \cos 135^\circ \cos 135^\circ$, $D_2 = \sin 135^\circ \sin 135^\circ$ and $D_3 = \cos 135^\circ \sin 135^\circ$. The variables $t = 6$ mm and $b = 154$ mm define the thickness and the side (using midsurfaces) of the cross sect ion.

Based on the geometrical and material constants the Equation 1gives the following relation

$$\begin{bmatrix}\varepsilon_1\\ \varepsilon_2\\ \varepsilon_3\end{bmatrix} = \begin{bmatrix}2.34\text{e-}8 & -9.18\text{e-}9 & 0\\ -2.34\text{e-}8 & -9.18\text{e-}9 & 0\\ 0 & 0 & -2.73\text{e-}8\end{bmatrix}\begin{bmatrix}M_X\\ M_Y\\ M_Z\end{bmatrix}. \quad (2)$$

Results and Discussion

According to FE analysis the applied moments and gauge strains are related by

$$\begin{bmatrix}\varepsilon_1\\ \varepsilon_2\\ \varepsilon_3\end{bmatrix} = \begin{bmatrix}2.62\text{e-}8 & -9.45\text{e-}9 & 0\\ -2.62\text{e-}8 & -9.55\text{e-}9 & 0\\ 0 & 0 & -2.86\text{e-}8\end{bmatrix}\begin{bmatrix}M_X\\ M_Y\\ M_Z\end{bmatrix}. \quad (3)$$

The coefficients differ somewhat when compared to the analytical results, Equation 2. It is reasonable to assume that the final fatigue prediction will also differ, depending on used transformation matrix. Since Equation 2 is based on approximations further results are based on the FE-relation, Equation 3.

©2013 COMSOL

Stresses on the inner side of the shell are more critical from the fatigue point of view. The variation of the fatigue usage factor along the fillets is shown in Figure 4. It reaches about 0.11 and thus the examined frame should not fail in fatigue.

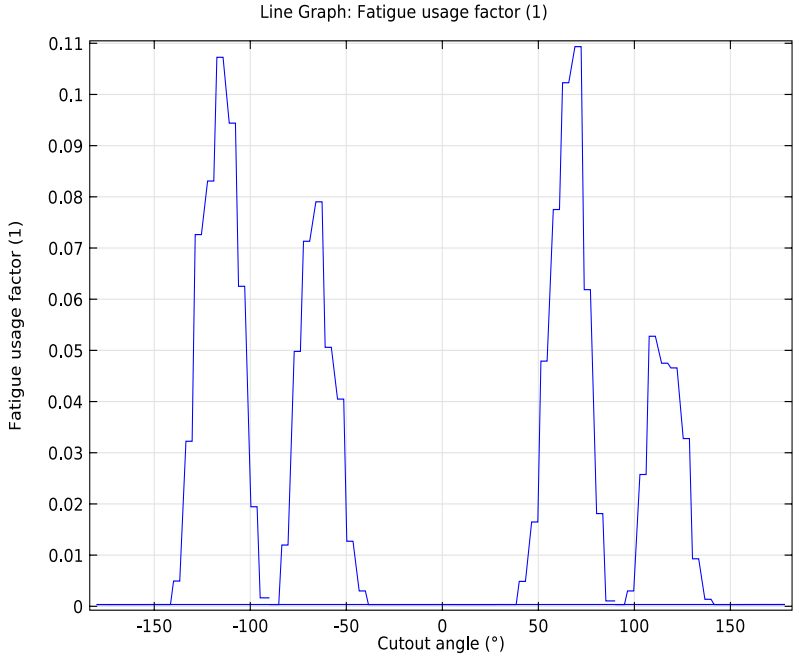

Figure 4: Fatigue usage factor along the cutout fillets. The angle is measured in the xz-plane with the angle starting from the x-axis.

In the most loaded point in the structure the stress history seems to be fairly symmetric around a zero mid stress. The mid stress is found in the range from -250 MPa to 250 MPa and the amplitude extend almost up to 600 MPa. The load distribution in the critical point is captured and shown in the Rainflow histogram in Figure 5.

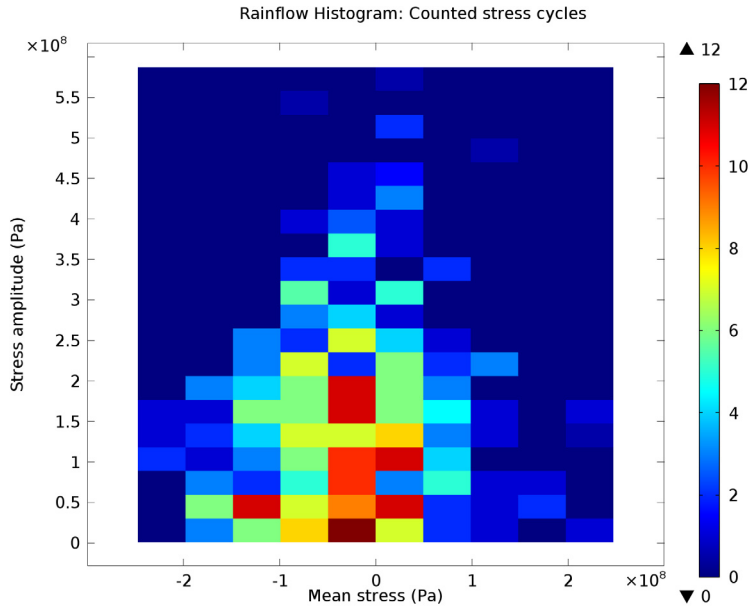

Figure 5: Cycle counting, following the rainflow theory, in the point of the highest fatigue usage factor.

In Figure 6 the relative contribution to the damage is shown for the same location. The dark blue area indicates stress cycles which are non-damaging. This means that they are below the endurance limit in the Wöhler curve, also called the S-N curve. The damaging stress cycles are found for high stress amplitudes. Since the Palmgren-Miner rule scales damage linearly with the number of cycles, when the number of cycles increases with factor $1/0.11$ the fatigue usage factor will exceed 1 an thus a failure will occur. In practice, the linearity assumption of the Palmgren-Miner rule can be questioned, so a proper safety factor should be applied.

An important information when evaluating Figure 5 and Figure 6 is that 32% of the fatigue damage comes from one single event in the load history. This indicates that the load history recorded is to short to make good predictions. Either new, longer, measurements should be made, or a high safety factor should be used in combination with some statistical considerations.

©2013 COMSOL

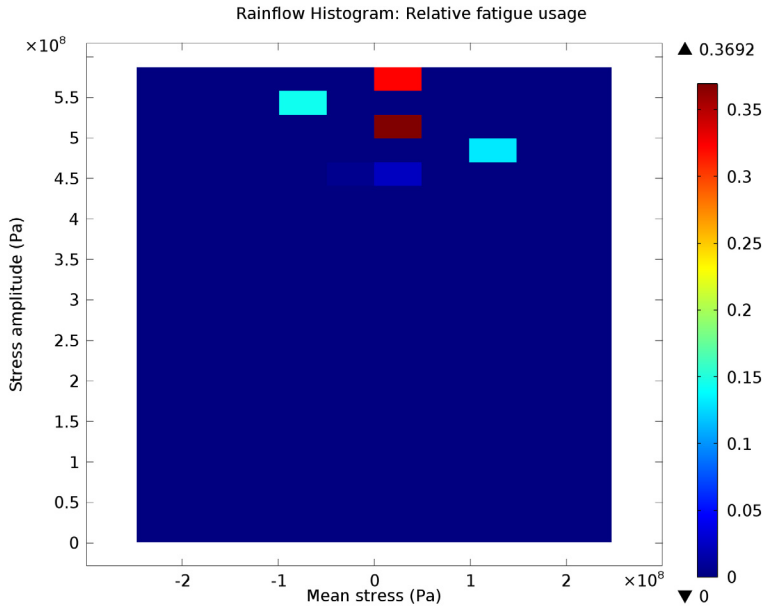

Figure 6: Relative fatigue usage, following the Palmgren-Miner damage rule, in the point of the highest usage factor.

Notes About the COMSOL Implementation

In COMSOL, several functions with the same argument list can be put in one file. This is demonstrated in the example with three load functions define the moment prescribed on one end of the frame. When an interpolation function is read from a file it treats the first columns as arguments and the following one as function response. As an example, in the interpolation function defining the moment around the y-axis the number of arguments is assigned 1 and the function position is assigned 2. Thus the first column is the argument and the third column is the function value (1+2).

COMSOL provides several options for specification of the S-N curve. **Interpolation** function with options data type **Grid**, interpolation **Linear** and extrapolation **Constant** is recommended. Those options are optimal for search of the life once R-value, and the amplitude stress are known. The input for S-N curve defined in the example is shown in Figure 7.

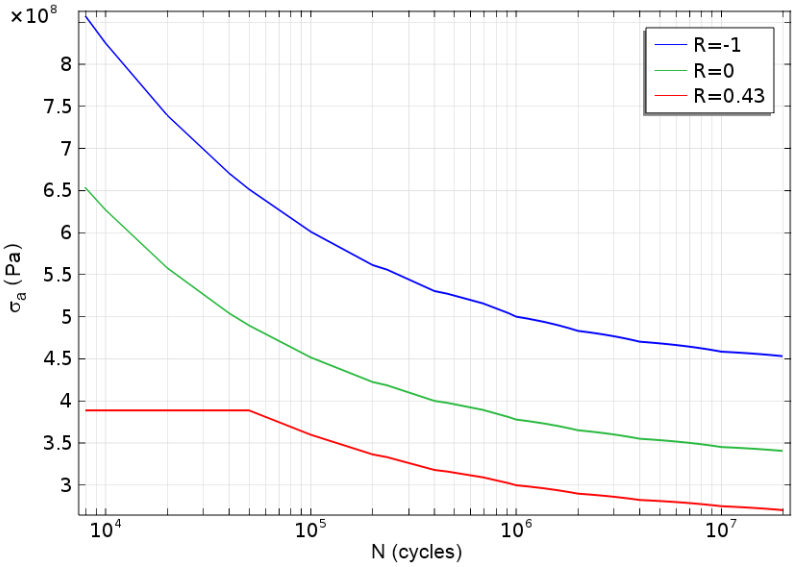

Figure 7: Wöhler curve of the material.

The fatigue material data in the material library gives the maximum fatigue stress, σ_{max}, as the function of the number of cycles and R-value. In order to transfer it to the stress amplitude, σ_a ,which is required by the **Cumulative Damage** fatigue feature, just apply following transformation

$$\sigma_a = \frac{\sigma_{max}(1-R)}{2}. \quad (4)$$

Model Library path: `Fatigue_Module/Damage/frame_with_cutout`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Shell (shell)**.

 ©2013 COMSOL

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

GEOMETRY 1

Block 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type `1.1`.

4 In the **Depth** edit field, type `0.154`.

5 In the **Height** edit field, type `0.154`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Work Plane 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Work Plane**.

2 In the **Work Plane** settings window, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **blk1**, select Boundary 3 only.

Rectangle 1

1 In the **Model Builder** window, under **Model 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Position** section.

3 From the **Base** list, choose **Center**.

4 Locate the **Size** section. In the **Width** edit field, type `0.08`.

5 In the **Height** edit field, type `0.1`.

Fillet 1

1 Right-click **Plane Geometry** and choose **Fillet**.

2 On the object **r1**, select Points 1–4 only.

3 In the **Fillet** settings window, locate the **Radius** section.

4 In the **Radius** edit field, type `0.01`.

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Form Union**.

2 In the **Finalize** settings window, locate the **Finalize** section.

3 In the **Relative repair tolerance** edit field, type `1e-6`.

Create points for strain evaluation where strain gauges are placed.

Point 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **More Primitives>Point**.

2 In the **Point** settings window, locate the **Point** section.

3 In the **x** edit field, type `0.3`.

4 In the **z** edit field, type `-0.077`.

Point 2

1 Right-click **Geometry 1** and choose **More Primitives>Point**.

2 In the **Point** settings window, locate the **Point** section.

3 In the **x** edit field, type `0.300`.

4 In the **y** edit field, type `0.077`.

5 Click the **Build All** button.

DEFINITIONS

Create a new coordinate system that is aligned with the strain gauge directions on the bottom side of the frame.

Base Vector System 2

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Coordinate Systems>Base Vector System**.

2 In the **Base Vector System** settings window, locate the **Settings** section.

3 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	y
x1	`cos(pi/4)`	`sin(pi/4)`
x2	`-sin(pi/4)`	`cos(pi/4)`

4 Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

5 Click the **Wireframe Rendering** button on the Graphics toolbar.

SHELL

1 In the **Model Builder** window, under **Model 1** click **Shell**.

2 Select Boundaries 2–5 only.

 ©2013 COMSOL

3 In the **Shell** settings window, locate the **Thickness** section.

4 In the d edit field, type `0.006[m]`.

Linear Elastic Material 2

1 Right-click **Model 1>Shell** and choose **Linear Elastic Material**.

2 Select Boundary 3 only.

3 In the **Linear Elastic Material** settings window, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Base Vector System 2**.

Prescribed Displacement/Rotation 1 1

1 In the **Model Builder** window, right-click **Shell** and choose **Prescribed Displacement/Rotation**.

2 Select Edges 1, 2, 4, and 6 only.

3 In the **Prescribed Displacement/Rotation** settings window, locate the **Prescribed Displacement** section.

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

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

6 Select the **Prescribed in z direction** check box.

Apply one twisting and two bending unit moments and differentiate them using load cases.

Rigid Connector 1

1 Right-click **Shell** and choose **Rigid Connector**.

2 Select Edges 17–20 only.

Applied Moment 1

1 Right-click **Model 1>Shell>Rigid Connector 1** and choose **Applied Moment**.

2 In the **Applied Moment** settings window, locate the **Applied Moment** section.

3 In the **M** table, enter the following settings:

M	x
0	y
0	z

4 Right-click **Model 1>Shell>Rigid Connector 1>Applied Moment 1** and choose **Rename**.

5 Go to the **Rename Applied Moment** dialog box and type `Twisting Moment (x)` in the **New name** edit field.

6 Click **OK**.

Applied Moment 2

1 In the **Model Builder** window, under **Model 1>Shell** right-click **Rigid Connector 1** and choose **Applied Moment**.

2 In the **Applied Moment** settings window, locate the **Applied Moment** section.

3 In the **M** table, enter the following settings:

0	x
M	y
0	z

4 Right-click **Model 1>Shell>Rigid Connector 1>Applied Moment 2** and choose **Rename**.

5 Go to the **Rename Applied Moment** dialog box and type `Bending Moment (y)` in the **New name** edit field.

6 Click **OK**.

Applied Moment 3

1 In the **Model Builder** window, under **Model 1>Shell** right-click **Rigid Connector 1** and choose **Applied Moment**.

2 In the **Applied Moment** settings window, locate the **Applied Moment** section.

3 In the **M** table, enter the following settings:

0	x
0	y
M	z

4 Right-click **Model 1>Shell>Rigid Connector 1>Applied Moment 3** and choose **Rename**.

5 Go to the **Rename Applied Moment** dialog box and type `Bending Moment (z)` in the **New name** edit field.

6 Click **OK**.

Bending Moment (y)

In the **Model Builder** window, under **Model 1>Shell>Rigid Connector 1** right-click **Twisting Moment (x)** and choose **Load Group>New Load Group**.

©2013 COMSOL

Bending Moment (z)

1 In the **Model Builder** window, under **Model 1>Shell>Rigid Connector 1** right-click **Bending Moment (y)** and choose **Load Group>New Load Group**.

2 Right-click **Bending Moment (z)** and choose **Load Group>New Load Group**.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Load Group 1**.

2 In the **Load Group** settings window, locate the **Group Identifier** section.

3 In the **Identifier** edit field, type `lgX`.

4 In the **Model Builder** window, under **Global Definitions** click **Load Group 2**.

5 In the **Load Group** settings window, locate the **Group Identifier** section.

6 In the **Identifier** edit field, type `lgY`.

7 In the **Model Builder** window, under **Global Definitions** click **Load Group 3**.

8 In the **Load Group** settings window, locate the **Group Identifier** section.

9 In the **Identifier** edit field, type `lgZ`.

10 In the **Model Builder** window, expand the **Global Definitions** node.

Parameters

1 Right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression
M	1 [N*m]

MATERIALS

Material 1

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

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 2–5 only.

5 In the **Material** settings window, locate the **Material Contents** section.

6 In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	200e9
Poisson's ratio	nu	0.33
Density	rho	7800

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extremely fine**.

4 Click the **Build All** button.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

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

3 Select the **Continuation** check box.

4 Select the **Define load cases** check box.

5 Click **Add** three times.

6 In the table, enter the following settings:

Load case	lgX	lgY	lgZ
Load case X	√	(As is)	(As is)
Load case Y	(As is)	√	(As is)
Load case Z	(As is)	(As is)	√

7 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

8 Right-click **Study 1** and choose **Plot**.

RESULTS

Derived Values

Create a table showing relation between gauge strain and each unit moment.

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Point Evaluation**.

©2013 COMSOL

2 Select Point 13 only.

3 In the **Point Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Shell>Strain>Strain tensor (local)>Strain tensor (local), xx component (shell.elxx)**.

4 Locate the **Expression** section. Select the **Description** check box.

5 In the associated edit field, type `e1`.

6 Find the **Parameters** subsection. In the table, enter the following settings:

Name	Value	Description
shell.z	1	Height of evaluation in shell, [-1,1]

7 Click the **Evaluate** button.

8 Right-click **Results>Derived Values>Point Evaluation 1** and choose **Duplicate**.

9 In the **Point Evaluation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Shell>Strain>Strain tensor (local)>Strain tensor (local), yy component (shell.elyy)**.

10 Locate the **Expression** section. In the **Description** edit field, type `e2`.

11 Click the **Evaluate** button.

12 Right-click **Point Evaluation 1** and choose **Duplicate**.

13 Select Point 14 only.

14 In the **Point Evaluation** settings window, locate the **Expression** section.

15 In the **Description** edit field, type `e3`.

16 Click the **Evaluate** button.

GLOBAL DEFINITIONS

Load S-N curve.

Interpolation 1

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Functions>Interpolation**.

2 In the **Interpolation** settings window, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click the **Browse** button.

5 Browse to the model's Model Library folder and double-click the file `frame_with_cutout_SN_curve.txt`.

6 Click the **Import** button.

Load twist and bending moments.

Interpolation 2

1 Right-click **Global Definitions** and choose **Functions>Interpolation**.

2 In the **Interpolation** settings window, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click the **Browse** button.

5 Browse to the model's Model Library folder and double-click the file `frame_with_cutout_load.txt`.

6 In the **Number of arguments** edit field, type 1.

7 Click the **Import** button.

When an interpolation function is read from a file it treats the first columns as arguments and the following one as function response. For example in the interpolation function defining the moment around the y-axis the number of arguments is assigned 1 and the function position is assigned 2. Thus the first column is the argument and the third column is the function value (1+2).

8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
fHisX	1
fHisY	2
fHisZ	3

MODEL 1

In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 In the **Add physics** tree, select **Structural Mechanics>Fatigue (ftg)**.

3 Click **Next**.

4 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	Solve for
Shell (shell)	×

 ©2013 COMSOL

5 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

6 Click **Finish**.

FATIGUE

Cumulative Damage 1

1 In the **Model Builder** window, under **Model 1** right-click **Fatigue** and choose **Edges>Cumulative Damage**.

2 Select all edges around the cutout.

3 In the **Cumulative Damage** settings window, locate the **Analysis** section.

4 From the **Type** list, choose **Generalized loads**.

5 Locate the **Solution Field** section. From the **Physics** list, choose **Shell**.

6 Locate the **Cycle Counting Parameters** section. Find the **Discretization** subsection. In the N_m edit field, type `10`.

7 In the N_r edit field, type `20`.

8 Locate the **Damage Model Parameters** section. In the m edit field, type `10000`.

9 From the $\sigma_a(R,N)$ list, choose **Interpolation 1**.

10 Locate the **Generalized Load Definition** section. In the s_f edit field, type `1000`.

11 Click **Add** three times.

12 Click **Add**.

13 Click **Add**.

14 In the table, enter the following settings:

Generalized load history
fHisX
fHisY
fHisZ

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Stationary** settings window, click to expand the **Values of Dependent Variables** section.

3 Select the **Values of variables not solved for** check box.

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

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

6 From the **Load case** list, choose **All**.

7 Click to expand the **Study Extensions** section. Clear the **Define load cases** check box.

8 Click to collapse the **Study Extensions** section. In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

Change the x-axis display to be an expression of the corner angles.

RESULTS

Fatigue Usage Factor (ftg)

1 In the **Model Builder** window, expand the **Results>Fatigue Usage Factor (ftg)** node, then click **Line Graph 1**.

2 In the **Line Graph** settings window, locate the **x-Axis Data** section.

Define rounding position using an angle.

3 From the **Parameter** list, choose **Expression**.

4 In the **Line Graph** settings window, locate the **x-Axis Data** section.

5 In the **Expression** edit field, type `atan2(z-0.03,x-0.04)*(dom==15)+atan2(z-0.03,x+0.04)*(dom==11)+atan2(z+0.03,x+0.04)*(dom==10)+atan2(z+0.03,x-0.04)*(dom==14)`.

6 From the **Unit** list, choose **×**.

7 Select the **Description** check box.

8 In the associated edit field, type `Cutout angle`.

9 Click the **Plot** button.

10 In the **Model Builder** window, right-click **Fatigue Usage Factor (ftg)** and choose **Rename**.

11 Go to the **Rename 1D Plot Group** dialog box and type `Fatigue Usage Factor Top (ftg)` in the **New name** edit field.

12 Click **OK**.

Stress Cycle Distribution (ftg)

1 In the **Model Builder** window, under **Results** right-click **Stress Cycle Distribution (ftg)** and choose **Rename**.

2 Go to the **Rename 2D Plot Group** dialog box and type `Stress Cycle Distribution Top (ftg)` in the **New name** edit field.

 ©2013 COMSOL

3 Click **OK**.

Fatigue Usage Distribution (ftg)

1 In the **Model Builder** window, under **Results** right-click **Fatigue Usage Distribution (ftg)** and choose **Rename**.

2 Go to the **Rename 2D Plot Group** dialog box and type `Fatigue Usage Distribution Top (ftg)` in the **New name** edit field.

3 Click **OK**.

Examine fatigue at the bottom side of the shell.

SHELL

1 In the **Shell** settings window, locate the **Height of Evaluation in Shell, [-1,1]** section.

2 In the *z* edit field, type `-1`.

MODEL 1

In the **Model Builder** window, right-click **Model 1** and choose **Add Physics**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 In the **Add physics** tree, select **Structural Mechanics>Fatigue (ftg)**.

3 Click **Next**.

4 Find the **Selected physics** subsection. In the table, enter the following settings:

Physics	Solve for
Shell (shell)	×
Fatigue (ftg)	×

5 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

6 Click **Finish**.

FATIGUE 2

Cumulative Damage 1

1 In the **Model Builder** window, under **Model 1** right-click **Fatigue 2** and choose **Edges>Cumulative Damage**.

2 Select all edges around the cutout.

3 In the **Cumulative Damage** settings window, locate the **Analysis** section.

4 From the **Type** list, choose **Generalized loads**.

5 Locate the **Solution Field** section. From the **Physics** list, choose **Shell**.

6 Locate the **Cycle Counting Parameters** section. Find the **Discretization** subsection. In the N_m edit field, type `10`.

7 In the N_r edit field, type `20`.

8 Locate the **Damage Model Parameters** section. In the m edit field, type `10000`.

9 From the $\sigma_a(R,N)$ list, choose **Interpolation 1**.

10 Locate the **Generalized Load Definition** section. In the s_f edit field, type `1000`.

11 Click **Add** three times.

12 In the table, enter the following settings:

Generalized load history
fHisX
fHisY
fHisZ

STUDY 3

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.

2 In the **Stationary** settings window, click to expand the **Values of Dependent Variables** section.

3 Select the **Values of variables not solved for** check box.

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

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

6 From the **Load case** list, choose **All**.

7 In the **Model Builder** window, right-click **Study 3** and choose **Compute**.

RESULTS

Fatigue Usage Factor (ftg2)

1 In the **Model Builder** window, expand the **Results>Fatigue Usage Factor (ftg2)** node, then click **Line Graph 1**.

2 In the **Line Graph** settings window, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

 ©2013 COMSOL

4 In the **Line Graph** settings window, locate the **x-Axis Data** section.

5 In the **Expression** edit field, type `atan2(z-0.03,x-0.04)*(dom==15)+atan2(z-0.03,x+0.04)*(dom==11)+atan2(z+0.03,x+0.04)*(dom==10)+atan2(z+0.03,x-0.04)*(dom==14)`.

6 From the **Unit** list, choose **×**.

7 Select the **Description** check box.

8 In the associated edit field, type `Cutout angle`.

9 Right-click **Results>Fatigue Usage Factor (ftg2)>Line Graph 1** and choose **Rename**.

10 Go to the **Rename Line Graph** dialog box and type `Fatigue Usage Factor Bottom (ftg2)` in the **New name** edit field.

11 Click **OK**.

Stress Cycle Distribution (ftg2)

1 In the **Model Builder** window, under **Results** right-click **Stress Cycle Distribution (ftg2)** and choose **Rename**.

2 Go to the **Rename 2D Plot Group** dialog box and type `Stress Cycle Distribution Bottom (ftg2)` in the **New name** edit field.

3 Click **OK**.

Fatigue Usage Distribution (ftg2)

1 In the **Model Builder** window, under **Results** right-click **Fatigue Usage Distribution (ftg2)** and choose **Rename**.

2 Go to the **Rename 2D Plot Group** dialog box and type `Fatigue Usage Distribution Bottom (ftg2)` in the **New name** edit field.

3 Click **OK**.

©2013 COMSOL