COMSOL MULTIPHYSICS®

Shaft with Fillet

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

COMSOL

Shaft with Fillet

Introduction

This model is of benchmark type, based on the example found in section 5.4.3 of Ref. 1, and shows how to perform a high-cycle fatigue analysis for nonproportional loading using critical planes.

Model Definition

The geometry is a circular shaft with two different diameters, 10 mm and 20 mm. At the transition, between the two parts, there is a fillet with a radius of 3 mm.

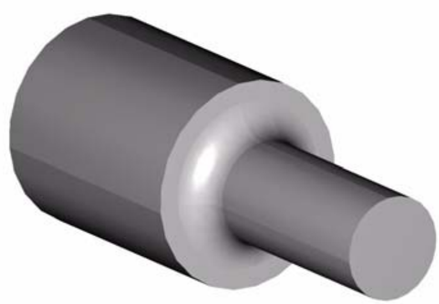

Figure 1: Model geometry

Two time-dependent loads are applied to the small end of the shaft: a transverse force, causing bending, and a twisting moment. As Figure 2 shows, the force varies between 0 and 2.95 kN and the torque between –30.3 Nm and +30.3 Nm.

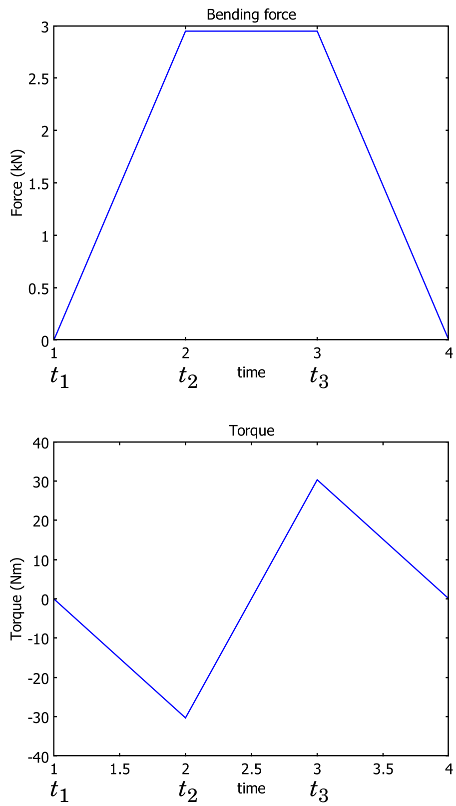

Figure 2: Variation of the bending force and twisting moment during one load cycle.

Compute the fatigue usage factor by analyzing the total stress distribution at times t_1, t_2 and t_3.

MATERIAL PROPERTIES

- Elastic data: Isotropic with $E = 100$ GPa, $\nu = 0$
- Fatigue data: The fatigue limit is known for two cases with pure axial loading. For pure tension it is 560 MPa ($\sigma_{max} = 1120$ MPa, $\sigma_{min} = 0$ MPa), and for fully reversed loading it is 700 MPa ($\sigma_{max} = 700$ MPa, $\sigma_{min} = -700$ MPa). This gives the

Findley parameters $f = 440$ MPa and $k = 0.23$ if Equation 13-16 on page 415 in the *Structural Mechanics Module User's Guide* is applied.

CONSTRAINTS

The thick end of the bar is fixed.

LOADS

- Bending load: The transverse force is evenly distributed as a shear traction over the small end.
- Torsional load: A shear stress in the circumferential direction is applied to the bar end. It is given as a linear variation with the radius in correspondence with analytical solutions for torsion.

Results and Discussion

Figure 3 shows the von Mises stresses for the bending load cases. The location for the maximum effective stress is at the surface of the fillet, at a radius slightly larger than the minimum radius of the shaft. The maximum effective stress can also be found at a diametrically opposite location. Here, the bending load gives rise to a compression stress of equal magnitude as the tension stress at the other side. Due to numerical reasons, the maximum stresses do not occur exactly in the *xz*-plane.

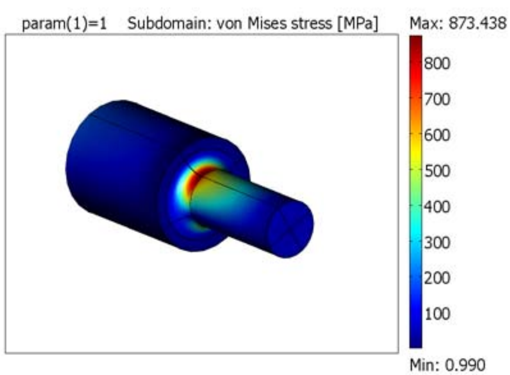

Figure 3: von Mises stress distribution from the bending load case.

Figure 4 shows the effective stress distribution for the torsional load. You can find the maximum value along the surface of the fillet, where the shear stress in the circumferential direction is at a maximum.

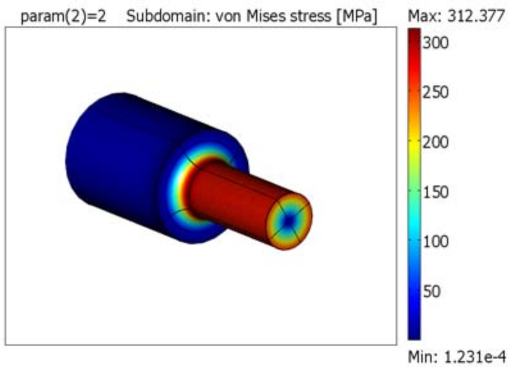

Figure 4: von Mises stress distribution for the torsional load case.

In Figure 5 you can see that the highest value of the utilization factor, 0.962, is found where the positive bending stress is combined with the shear stress from torsion. Note that on the diametrically opposite side of the shaft, with the maximum compressive stress due to the bending, the utilization factor is only slightly increased compared to the surrounding areas.

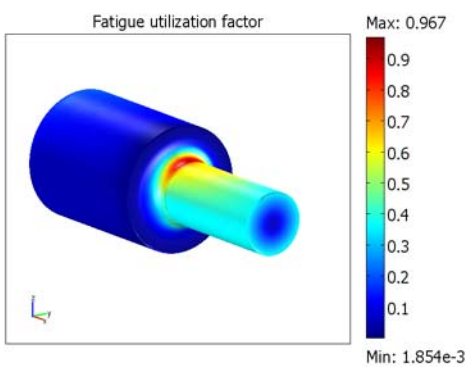

Figure 5: Fatigue utilization factor.

The history of each of the global stress components in the most critical point is shown in Figure 6. Because the peak stresses occur at a small distance up along the fillet and because, for numerical reasons, they are not found exactly in the plane $y = 0$, all stress components are nonzero.

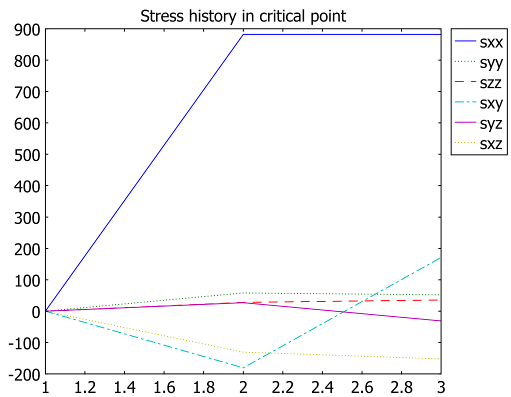

Figure 6: Histories for global stress components.

For the example in Ref. 1 the computed Findley parameter is 433 MPa, which gives a utilization factor of 0.984. This is slightly higher than the utilization factor computed by this model, which is 0.962. The difference is mainly due to the simplified approach of the referenced example, where only the normal stress component from the bending load and the shear stress component from the torsional load are considered. In reality, there are additional stress components, especially in the bending case. Also, the peak stress concentration does not occur at exactly the same location in the two load cases.

Modeling in COMSOL Multiphysics

Start by using the parametric solver to solve the static problem for the load cases of the maximum bending force and the maximum twisting moment, respectively. These are the two basic load cases that you can combine and use for the fatigue analysis.

Carry out the fatigue analysis in MATLAB by doing the following steps:

- Extract the stress distribution (stress tensor) for both basic load cases from the FEM structure of the model.

- Set up the matrix containing the combinations of basic load cases which you can use for the fatigue analysis. The rows of this matrix correspond to the time instances you are analyzing, and the columns correspond to the basic load cases.

  In this case the matrix is a 3x2 matrix. If you look at Figure 2 you can see that at time t_1 both loads are zero, thus the first row has only zeros. At time t_2 the bending force is at maximum, while the torque at minimum. You obtain the combined stresses by subtracting the solution of the twisting moment from that of the bending force. The second row of the matrix is thus $1\ -1$. Both loads are at maximum at the third time, t_3, which means that you can add the basic load cases and the third row of the matrix contains two ones.
- Run the fatigue analysis function `hcfmultiax`.

Reference

1. D, F. Socie and G.B. Marquis, *Multiaxial Fatigue*, SAE, 1999.

Model Library path: `Structural_Mechanics_Module/Fatigue/shaft_with_fillet`

Note: This model requires MATLAB to run the fatigue script `shaft_with_fillet_fatigue.m`

Modeling Using the Graphical User Interface

This section describes how to solve the two basic load cases using COMSOL Multiphysics.

MODEL NAVIGATOR

1. On the **New** page, select **3D** from the **Space dimension** list.
2. From the **Application Modes** list, select **Structural Mechanics Module>Solid, Stress-Strain>Static analysis**.
3. Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu select **Constants** and enter the following constants (the descriptions are optional); then click **OK**.

NAME	EXPRESSION	DESCRIPTION
M	30.3[N*m]	Torque amplitude
R	5[mm]	Radius where torque and force are applied
k	M/(R^4*pi/2)	Constant used in distributed moment calculation
param	1	Parameter used to control the load cases
F	2.95[kN]	Maximum bending force
A	pi*R^2	Area of surface where force is applied

2 Choose **Model Settings** from the **Physics** menu to open the **Model Settings** dialog box.

3 Select **MPa** from the **Base unit system** list to use mm as length scale and MPa as stress unit. Click **OK**.

GEOMETRY MODELING

Create the geometry by drawing a 2D plane and rotate it.

1 Select **Work-Plane Settings** from the **Draw** menu to open the **Work-Plane Settings** dialog box.

2 Click **OK** to create a 2D work plane in the *xy*-plane which is the default settings.

3 Select **Axes/Grid Settings** from the **Options** menu and give axis and grid settings according to the following table. On the **Grid** page, clear the **Auto** check box to enter the grid spacing. When done, click **OK**.

AXIS		GRID	
x min	-5	x spacing	1
x max	60	Extra x	
y min	-15	y spacing	1
y max	15	Extra y	

4 Click the **Line** button on the Draw toolbar. Click the left mouse button at (0, 0), then move the mouse to (0, 10) and click the left mouse button again.

5 Move the mouse to (25, 10) and click the left mouse button again.

6 Move the mouse to (25, 8) and click the left mouse button.

7 Click the **2nd Degree Bézier Curve** button on the Draw toolbar.

8 Move the mouse to (25, 5) and click the left mouse button.

9 Move the mouse to (28, 5) and click the left mouse button.

10 Click the **Line** button on the Draw toolbar.

11 Move the mouse to (49, 5) and click the left mouse button.

12 Move the mouse to (49, 0) and click the left mouse button.

13 Click the right mouse button to form a 2D solid.

14 Select **Revolve** from the **Draw** menu to open the **Revolve** dialog box.

15 Select **Axis direction through: Second point**.

16 Enter **x** `1` **y** `0` as the coordinates of the **Second point**.

17 Click **OK** to close the **Revolve** dialog box and create the shaft.

PHYSICS SETTINGS

Boundary Conditions

1 Select **Boundary Settings** from the **Physics** menu.

2 Specify boundary settings according to the following table; when done, click **OK**.

SETTINGS	BOUNDARIES 1, 3, 5, 7	BOUNDARIES 21–24
Page	Constraint	Load
Constraint condition	Fixed	
F_x		`0`
F_y		`-z*k*(param>1.5)`
F_z		`y*k*(param>1.5)-F/A*(param<1.5)`

Subdomain Settings

1 Select **Subdomain Settings** from the **Physics** menu.

2 Specify subdomain settings according to the following table; when done, click **OK**.

SETTINGS	SUBDOMAIN 1
Page	Material
Material model	Isotropic
E	`100[GPa]`
ν	`0`

MESH GENERATION

1 Select **Free Mesh Parameters** from the **Mesh** menu to open the **Free Mesh Parameters** dialog box.

2 On the **Subdomain** page, type `1.2` in the **Element growth rate** edit field.

3 On the **Boundary** page, select Boundaries 11, 12, 14, and 16.

4 Enter `1` in the **Maximum element size** edit field. Click **OK**.

COMPUTING THE SOLUTION

1 Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2 Select **Parametric** in the **Solver** list.

3 Specify `param` in the **Parameter name** edit field.

4 Specify `1,2` in the **Parameter values** edit field.

5 Select **Conjugate gradients** from the **Linear system solver** list. Click **OK**.

6 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 Select **1** from the **Parameter value** list on the **General** page to plot the von Mises stress distribution from the bending.

3 Select **2** from the **Parameter value** list to plot the von Mises stress distribution from torsion.

Fatigue Analysis

This section describes how to solve the fatigue problem.

You have already solved the static bending and torsion load cases.

- Select **File>Export>FEM Structure as 'fem'** from the **File** menu to export the FEM structure containing the static load cases to the command line.
- Run the script `shaft_with_fillet_fatigue.m` by typing `shaft_with_fillet_fatigue` and pressing Return.

The script `shaft_with_fillet_fatigue.m` is shown below.

```
% Compute all components of the stress tensor
[sxx, syy, szz, sxy, syz, sxz] = posteval(fem,'sx_smsld','sy_smsld',...
                                          'sz_smsld',...
                                          'sxy_smsld','syz_smsld',...
                                          'sxz_smsld',...
                                      'edim',2,'solnum','all','cont','on');
stress = zeros(6,size(sxx.d,2),size(sxx.d,1));
stress(1,:,:)=sxx.d';
```

```
stress(2,:,:)=syy.d';
stress(3,:,:)=szz.d';
stress(4,:,:)=sxy.d';
stress(5,:,:)=syz.d';
stress(6,:,:)=sxz.d';

% Fatigue data for the material
params.f = 440;
params.k = 0.23;

% Combine the basic loadcases to form the fatigue loadcycle
lccomb = [0 0; 1 1; 1 -1];

% Resolution when searching for the critical plane
angle_step = 6; % degrees

% Compute the fatigue damage
[resval, sigma_max, delta_tau, worst_ind, s_history]= hcfmultiax(stress, ...
               'lccomb', lccomb, 'anglestep', angle_step, 'params', params);

% Copy a postdata struct from the stress evaluation and assign the damage data
% to the value field (d)
findley_res = sxx;
findley_res.d(1,:) = resval;
figure(1);

% Plot the fatigue damage
postdataplot(findley_res,...
             'scenelight','on',...
             'campos',[244,-161,77],...
             'camtarget',[24.5,0,0],...
             'camva',9.5,...
             'title','Fatigue utilization factor');

% Plot the maximum normal stress in the critical plane
sigma_max_res = sxx;
sigma_max_res.d(1,:) = sigma_max;
figure(2);
postdataplot(sigma_max_res,...
             'scenelight','on',...
             'campos',[244,-161,77],...
             'camtarget',[24.5,0,0],...
             'camva',9.5,...
             'title','Maximum normal stress');

% Plot the shear stress in the critical plane
delta_tau_res = sxx;
delta_tau_res.d(1,:) = delta_tau;
figure(3);
postdataplot(delta_tau_res,...
             'scenelight','on',...
             'campos',[244,-161,77],...
             'camtarget',[24.5,0,0],...
             'camva',9.5,...
             'title','Shear stress amplitude');
```

```
% Plot the stress history for all stress components at the point with
% maximum fatigue damage
x = 1:(size(s_history,2));
figure(4);
if isscript
  plot(x,s_history,'linestyle','cycle');
else
  plot(x,s_history);
end
legend('sxx','syy','szz','sxy','syz','sxz')
title('Stress history in critical point')
```