Model created in COMSOL Multiphysics 6.4

Rolling Contact Fatigue in a Linear Guide

Introduction

A linear guide has been loaded above the manufacturer specification limit. A concern arises whether the contact loads will introduce fatigue spalling. In a system analysis, the entire guide has been analyzed and the mostly damaging contact load has been identified to occurs on the rail raceway.

In this analysis, the rolling contact motion of the mostly loaded rolling element is simulated, and fatigue is evaluated using the Dang Van model.

Model Definition

The linear guide consists of a rail, a carriage, two recirculation parts on both ends of the carriage, and multiple sets of ball chains that transfer the load between the rail and the carriage. The carriage contains recirculation channels that leads the rolling element back to the raceway. The structural loads are transferred through rail, rolling elements and carriage. The role of the recirculation parts is to force the rolling elements into a recirculating pattern and to keep out the contamination. A schematic description of a linear guide is shown in Figure 1, where both recirculation parts have been removed in order to show the rolling elements.

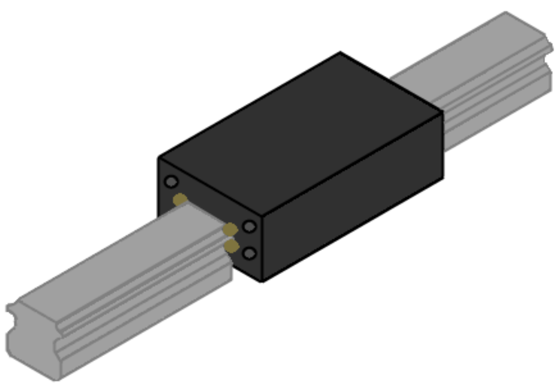


Figure 1: Schematic representation of a linear guide.

The cross section of the rail is shown in Figure 2.

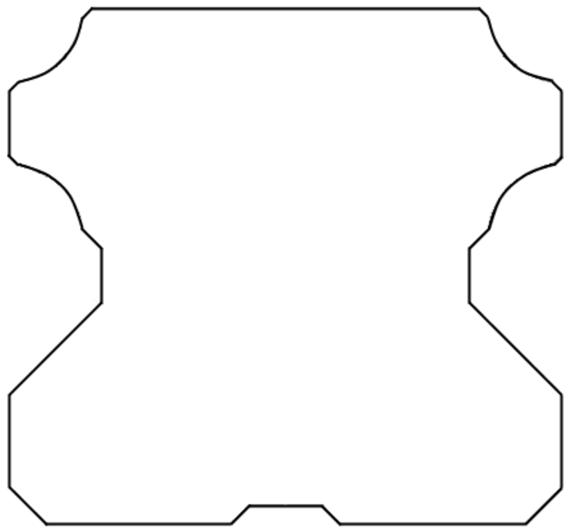


Figure 2: Rail cross section.

The rail has four raceway grooves where vertical and horizontal loads, as well as the twisting moments, are transferred. The ball chains consist of many rolling elements and it is not uncommon that in a loaded condition more than 30 rolling elements simultaneously transfer load from a carriage to the raceway.

An earlier performed system analysis has concluded that the most loaded rolling element transfers 13.75 N at a 45° angle.

The elastic properties of the rail are defined with Young's modulus and Poisson's ratio being E = 200 GPa and ν = 0.30, respectively.

NUMERICAL MODEL

The contact load from the rolling element is modeled as a contact pressure ellipse. Since the groove has the radius of 2 mm and the rolling element has a radius of 1.8 mm, a total load of 13.75 N according to the Hertzian theory results in a contact ellipse characterized by

- maximum contact pressure, p_{max} = 1.14 GPa
- semi-major axis, a = 161 µm
- semi-minor axis, b = 36 µm.

In a contact analysis the element size is of great importance. The elements must be small enough to correctly resolve the contact pressure on the surface. However as we deal with rolling contact, the contact pressure is moving along the surface and thus the entire area of the traveling contact must consist of small elements. Moreover, the highest effective and shear stresses in a contact analysis are often found on the subsurface level, close to the surface. Therefore, a fine mesh is required also through the depth of the model.

In order to reduce the model size, only a 3.6 mm long slice of the rail is modeled. This length corresponds to one rolling element diameter. The fine elements of the contact zone are located along a length of 500 µm, which is about 7 times larger than the contact length in the rolling direction.

The moving contact load is prescribed along a stretch of 400 µm using 50 load steps. In each step the center of the load the moves 8 µm. Since the ellipse axis along the traveling direction is 36 µm, the analysis requires nine steps in order for the load to cross over a size of the contact area.

In a model of a long rail, the fatigue results of the cross section of the rail will be constant along the rail length. With the use of a truncated contact path, the results along the cross section will differ depending on the position along the length. The used dimensions of the rail length and the contact path are sufficient to avoid the edge effects when evaluating the fatigue results at the center of the fine contact zone.

Results and Discussion

The equivalent stress resulting from the rolling element contact is shown in Figure 3. The highest stress component, normal to the surface, is prescribed via the contact pressure from a rolling element, see Figure 4. The maximum equivalent stress is however twice as high on the subsurface level than it is at the surface, see Figure 5.

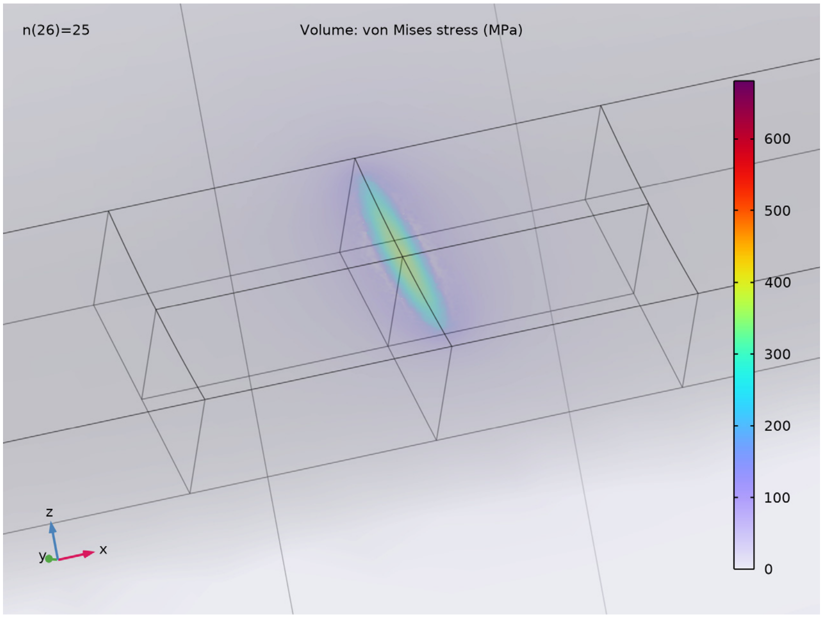


Figure 3: Equivalent stress.

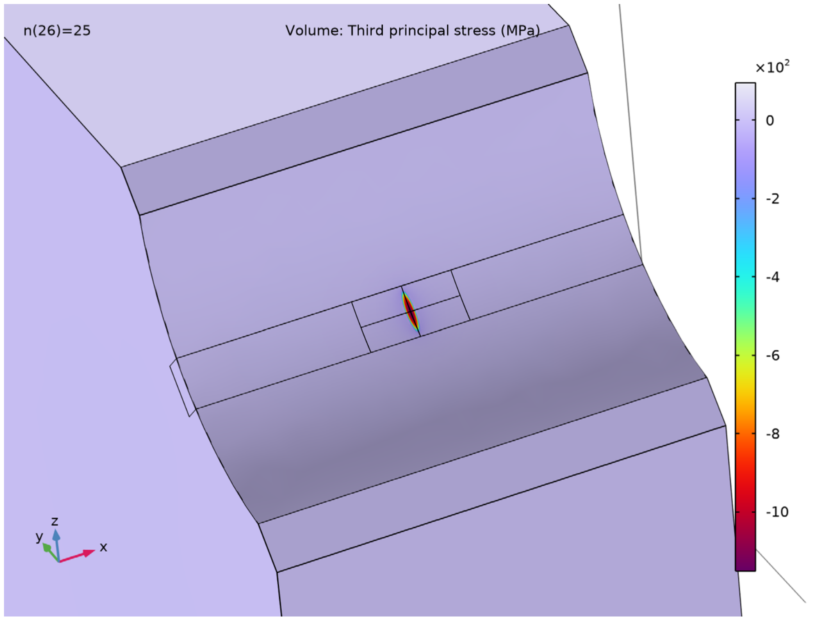


Figure 4: Contact stress.

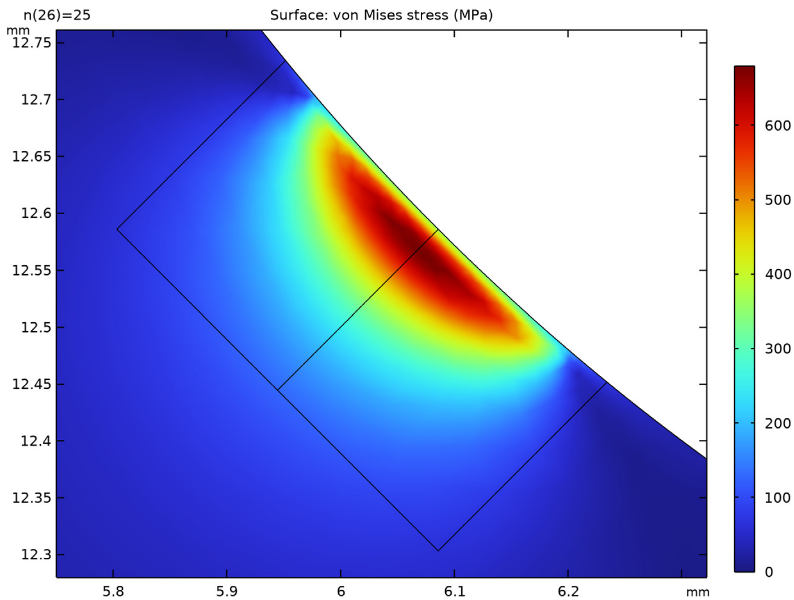


Figure 5: Equivalent stress through the depth.

The development of the profile of the equivalent stress through the depth, as the load passes, is shown in Figure 6. Similarly, the development of the profile of the shear stress through the depth, as the load passes, is shown in Figure 7. From these figures, it is clear that the locations of the highest effective and the highest shear stresses do not coincide. The highest equivalent stress is located 23.5 μm below the surface, while the highest shear stress is located 16.6 μm below the surface. This indicates that the loading is nonproportional. The two locations are examined further in Figure 8 and Figure 9 where the stress history is shown as the contact load travels along the surface.

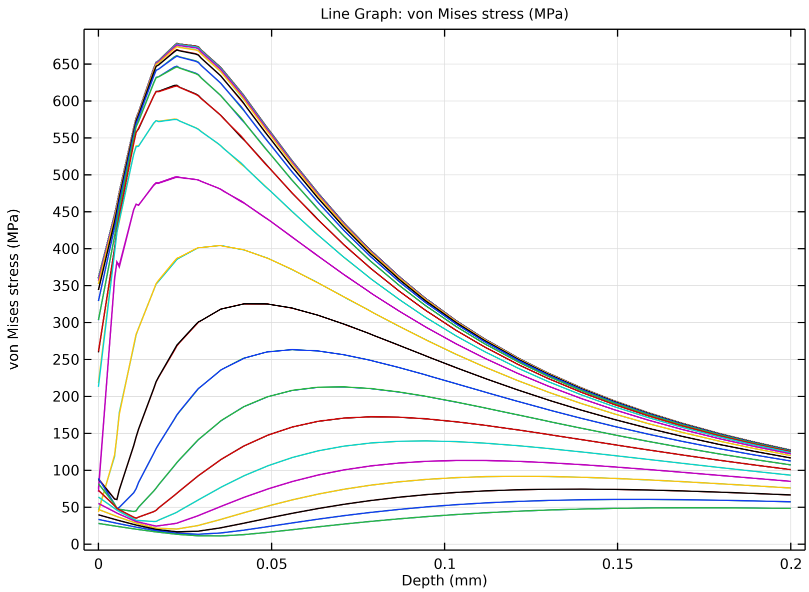


Figure 6: The development of the equivalent stress profile through the depth as the load passes.

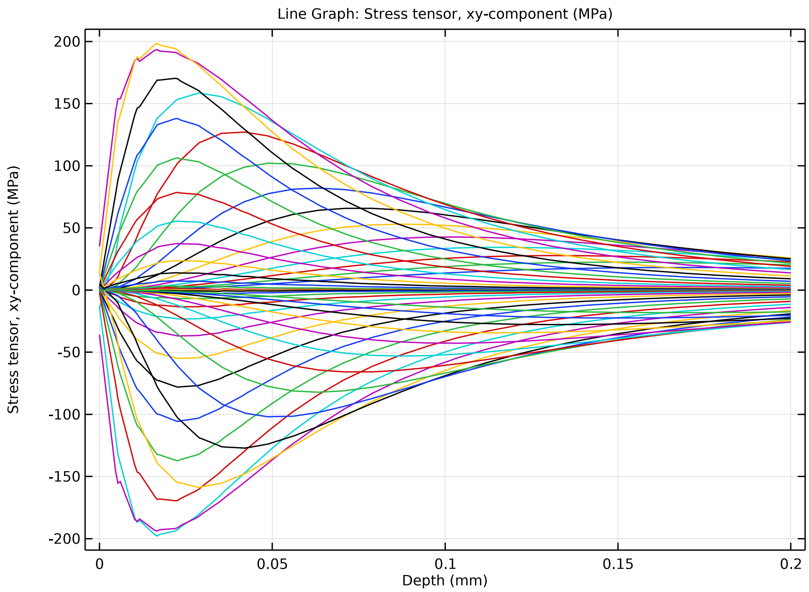


Figure 7: The development of the shear stress profile through the depth as the load passes.

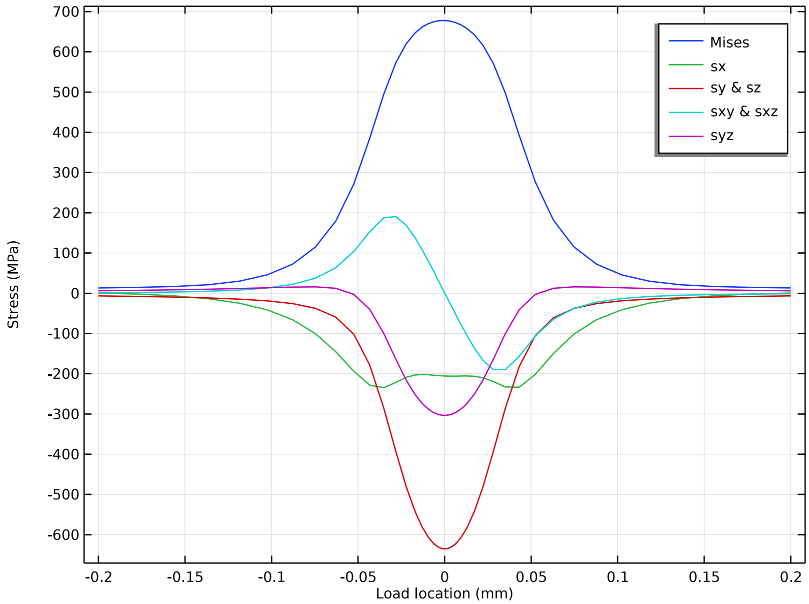


Figure 8: The equivalent stress history 23.5 μm below the surface.

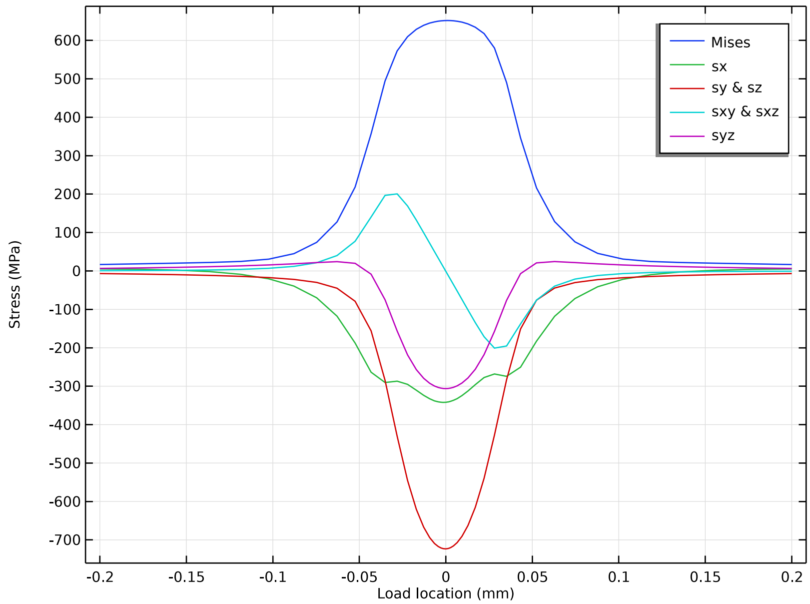


Figure 9: The shear stress history 16.6 μm below the surface.

The results of the fatigue analysis is shown in Figure 10. Since the fatigue usage factor is close to 1.0, fatigue failure through spalling of the raceway can be expected.

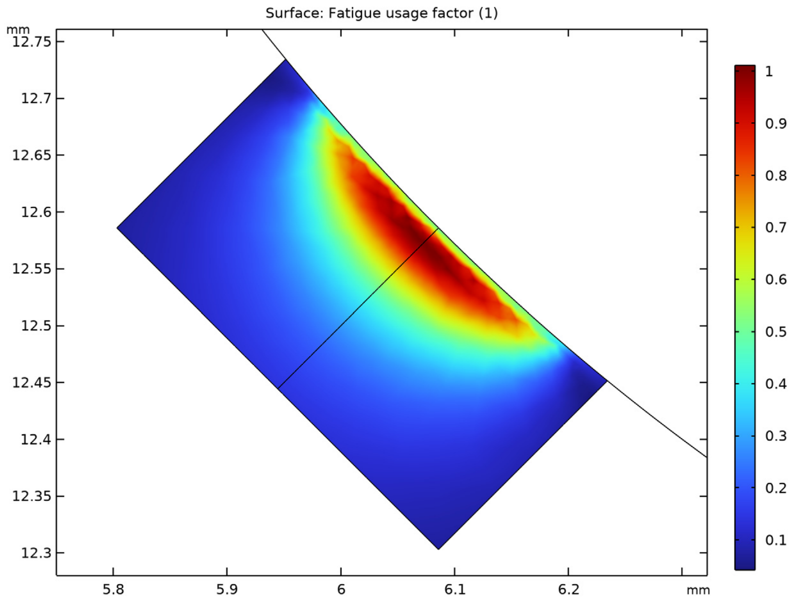


Figure 10: The fatigue usage factor as predicted by the Dang Van model.

Application Library path: `Fatigue_Module/Stress_Based/linear_guide`

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 **3D**.

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

Next load the model geometry. First, however, change the length unit to millimeters.

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 `linear_guide_geometry.mphbin`.

5 Click **null**.

Create selections to apply features more easily.

DEFINITIONS

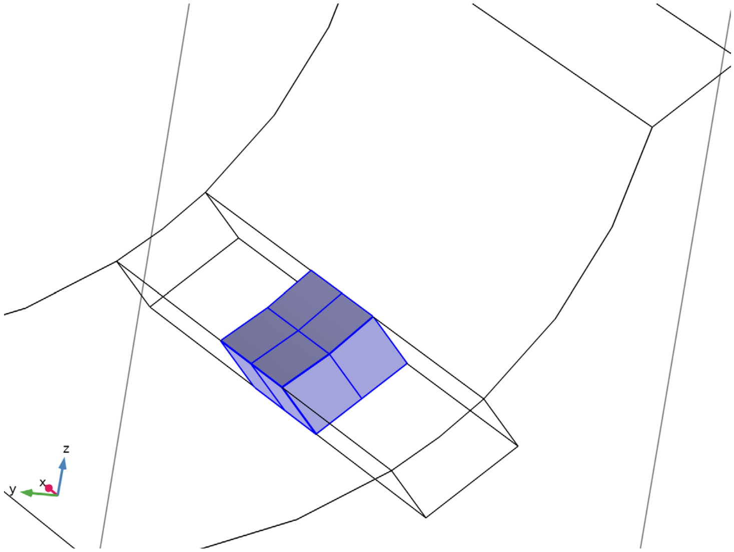
Contact Volume

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Contact Volume` in the **Label** text field.

3 Click the **Wireframe Rendering** button in the **Graphics** toolbar. Rotate the geometry and zoom in on the contact region.

4 Select Domains 3–6 only.



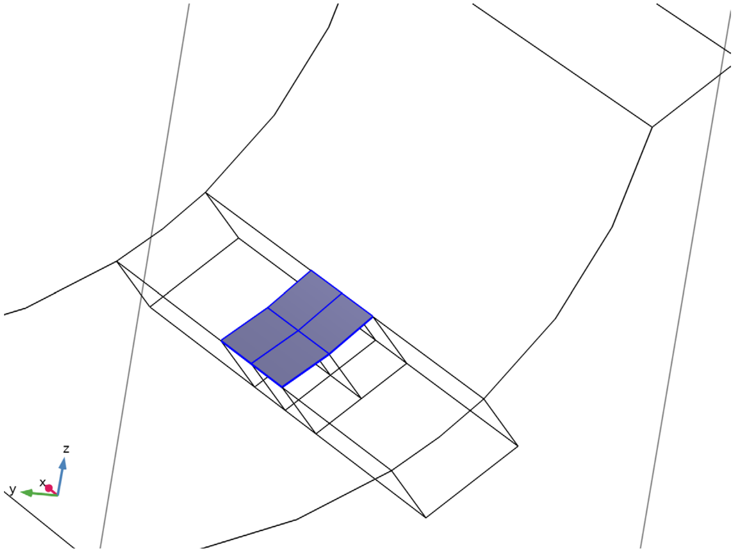
Contact Area

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Contact Area` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 30, 32, 39, and 41 only.



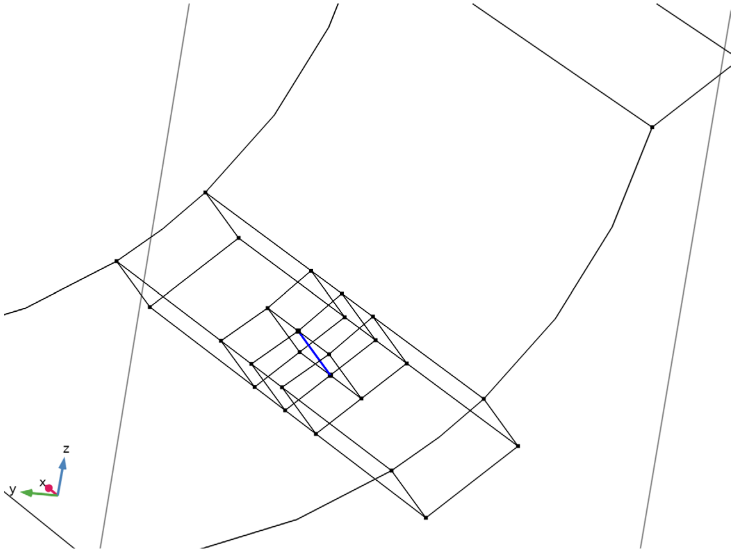
Contact Depth Line

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Contact Depth Line` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 59 only.



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

Load model parameters.

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 `linear_guide_parameters.txt`.

DEFINITIONS

Load model variables.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **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 `linear_guide_variables.txt`.

Create the load function.

Analytic 1 (an1)

1 In the **Definitions** toolbar, click **f(x)** **Analytic**.

2 In the **Settings** window for **Analytic**, type `cPos` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `alpha*(x-nStep/2)^3`.

4 Locate the **Units** section. In the **Function** text field, type `m`.

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	200e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7800	kg/m³	Basic

SOLID MECHANICS (SOLID)

Boundary Load 1

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

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Contact Area**.

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the *p* text field, type `pMax*pMag`.

Fixed Constraint 1

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

2 Select Boundary 6 only.

Roller 1

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

2 Select Boundaries 1, 2, 13, 48, and 49 only.

Create a mesh with fine elements in the vicinity of the contact area.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Contact Area**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

4 Click to expand the **Element Size Parameters** section. Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section.

6 Select the **Maximum element size** checkbox. In the associated text field, type `0.01`.

Free Tetrahedral 1

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

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

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

4 From the **Selection** list, choose **Contact Volume**.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

Distribution 1

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

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Contact Depth Line**.

4 Locate the **Distribution** section. From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 25.

6 In the **Element ratio** text field, type 2.

7 Select the **Reverse direction** checkbox.

Free Tetrahedral 2

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size 1

1 Right-click **Free Tetrahedral 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Minimum element size** checkbox. In the associated text field, type 0.1.

6 Click **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 **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
n (Analysis step)	range(0,1,nStep)	

Use an iterative solver that is efficient for large well-conditioned models, such as a block.

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.

4 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Suggested Iterative Solver (solid)** and choose **Enable**.

5 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**.

Study1/Mirror 3D

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **More 3D Datasets** > **Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, type `Study1/Mirror 3D` in the **Label** text field.

4 Locate the **Plane Data** section. From the **Plane** list, choose **XZ-planes**.

Surface: Equivalent Stress

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Surface: Equivalent Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study1/Mirror 3D**.

4 From the **Parameter value (n)** list, choose **25**.

5 Locate the **Plot Settings** section. From the **View** list, choose **New view** to create a dedicated view for this plot.

6 In the **Surface: Equivalent Stress** toolbar, click **Plot**.

Volume 1

In the **Model Builder** window, expand the **Surface: Equivalent Stress** node.

Deformation

1 In the **Model Builder** window, expand the **Volume 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 Click the **Transparency** button in the **Graphics** toolbar and zoom manually to reproduce Figure 3.

5 Click the **Transparency** button in the **Graphics** toolbar again to restore the opaque mode.

Surface: Equivalent Stress

Duplicate the plot group to generate Figure 4.

Surface: Contact Stress

1 In the **Model Builder** window, right-click **Surface: Equivalent Stress** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Surface: Contact Stress` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

4 In the **Surface: Contact Stress** toolbar, click **Plot**.

Volume 1

1 In the **Model Builder** window, expand the **Surface: Contact Stress** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **Principal stresses** > **solid.sp3Gp - Third principal stress - N/m²**.

3 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

4 Zoom out a little and compare with Figure 4.

Examine stresses at the subsurface level; see Figure 5.

Study1/Cut Plane: Through Thickness

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `Study1/Cut Plane: Through Thickness` in the **Label** text field.

Subsurface: Equivalent Stress

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Subsurface: Equivalent Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (n)** list, choose **25**.

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 In the **Subsurface: Equivalent Stress** toolbar, click **Plot**.

Surface 1

1 Right-click **Subsurface: Equivalent Stress** and choose **Surface**.

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** > **solid.misesGp - von Mises stress - N/m²**.

Evaluate stresses along a subsurface central line; see Figure 6 and Figure 7.

Through Thickness: Equivalent Stress

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

2 In the **Settings** window for **1D Plot Group**, type `Through Thickness: Equivalent Stress` in the **Label** text field.

Line Graph 1

1 Right-click **Through Thickness: Equivalent Stress** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 From the **Selection** list, choose **Contact Depth Line**.

4 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.misesGp - von Mises stress - N/m²**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Reversed arc length**.

Through Thickness: Equivalent Stress

1 In the **Model Builder** window, click **Through Thickness: Equivalent Stress**.

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

3 Select the **x-axis label** checkbox. In the associated text field, type `Depth (mm)`.

Through Thickness: Shear Stress

1 Right-click **Through Thickness: Equivalent Stress** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Through Thickness: Shear Stress` in the **Label** text field.

Line Graph 1

1. In the **Model Builder** window, expand the **Through Thickness: Shear Stress** node, then click **Line Graph 1**.
2. In the **Settings** window for **Line 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** > **Stress tensor (spatial frame) - N/m²** > **solid.sGpxy - Stress tensor, xy-component**.

The highest equivalent stress and the highest shear stress are found at different subsurface levels. Create new datasets in order to evaluate the stress history at these levels and reproduce Figure 8 and Figure 9.

Cut Point 3D: Max Mises

1. In the **Results** toolbar, click **Cut Point 3D**.
2. In the **Settings** window for **Cut Point 3D**, type `Cut Point 3D: Max Mises` in the **Label** text field.
3. Locate the **Point Data** section. In the **X** text field, type `0`.
4. In the **Y** text field, type `7.5-2.0235*cos(45[deg])`.
5. In the **Z** text field, type `14-2.0235*sin(45[deg])`.

Cut Point 3D: Max Shear

1. Right-click **Cut Point 3D: Max Mises** and choose **Duplicate**.
2. In the **Settings** window for **Cut Point 3D**, type `Cut Point 3D: Max Shear` in the **Label** text field.
3. Locate the **Point Data** section. In the **Y** text field, type `7.5-2.0166*cos(45[deg])`.
4. In the **Z** text field, type `14-2.0166*sin(45[deg])`.

Point of Max Mises: Stress Components

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Point of Max Mises: Stress Components` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: Max Mises**.

Point Graph 1

1. Right-click **Point of Max Mises: Stress Components** 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.misesGp - von Mises stress - N/m²**.

3 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `cPos(n)`.

5 Click to expand the **Legends** section. Select the **Show legends** checkbox.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Mises

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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** > **Stress tensor (spatial frame) - N/m²** > **solid.sGpxx - Stress tensor, xx-component**.

3 Locate the **Legends** section. In the table, enter the following settings:

Legends
sx

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

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** > **Stress tensor (spatial frame) - N/m²** > **solid.sGpyy - Stress tensor, yy-component**.

3 Locate the **Legends** section. In the table, enter the following settings:

Legends
sy & sz

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

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** > **Stress tensor (spatial frame) - N/m²** > **solid.sGpxy - Stress tensor, xy-component**.

3 Locate the **Legends** section. In the table, enter the following settings:

Legends
sxy & sxz

Point Graph 5

1 Right-click **Point Graph 4** and choose **Duplicate**.

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** > **Stress tensor (spatial frame) - N/m²** > **solid.sGpyz - Stress tensor, yz-component**.

3 Locate the **Legends** section. In the table, enter the following settings:

Legends
syz

4 In the **Point of Max Mises: Stress Components** toolbar, click **Plot**.

Point of Max Mises: Stress Components

1 In the **Model Builder** window, click **Point of Max Mises: Stress Components**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Load location (mm)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Stress (MPa)`.

7 In the **Point of Max Mises: Stress Components** toolbar, click **Plot**.

Point of Max Shear: Stress Components

1 Right-click **Point of Max Mises: Stress Components** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Point of Max Shear: Stress Components` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: Max Shear**.

4 In the **Point of Max Shear: Stress Components** toolbar, click **Plot**.

Now, perform the fatigue analysis.

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)

Stress-Based 1

1 In the **Physics** toolbar, click **Domains** and choose **Stress-Based**.

2 In the **Settings** window for **Stress-Based**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Contact Volume**.

4 Locate the **Fatigue Model Selection** section. From the **Criterion** list, choose **Dang Van**.

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

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Material 1 (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
Hydrostatic stress sensitivity coefficient	a_DangVan	0.23	1	Dang Van
Limit factor	b_DangVan	248[MPa]	Pa	Dang Van

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

Study2/Mirror 3D

1 In the **Model Builder** window, right-click **Study1/Mirror 3D** and choose **Duplicate**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Label** text field, type `Study2/Mirror 3D`.

Fatigue Usage Factor (ftg)

1 In the **Model Builder** window, under **Results** click **Fatigue Usage Factor (ftg)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study2/Mirror 3D**.

Evaluate fatigue on the subsurface level to generate Figure 10.

Study2/Cut Plane: Through Thickness

1 In the **Model Builder** window, right-click **Study1/Cut Plane: Through Thickness** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Label** text field, type `Study2/Cut Plane: Through Thickness`.

Subsurface: Fatigue Usage Factor

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Subsurface: Fatigue Usage Factor` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study2/ Cut Plane: Through Thickness**.
4 Locate the **Plot Settings** section. From the **View** list, choose **View 2D 5**.

Surface 1

1 Right-click **Subsurface: Fatigue Usage Factor** and choose **Surface**.
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)** > **Fatigue** > **ftg.fus - Fatigue usage factor - 1**.
3 In the **Subsurface: Fatigue Usage Factor** toolbar, click **Plot**.