

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 v = 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, $p_{\text{max}} = 1.14$ GPa
- semi-major axis, $a = 161 \,\mu\text{m}$
- semi-minor axis, $b = 36 \,\mu\text{m}$.

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 μ m, 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 μ m using 50 load steps. In each step the center of the load the moves 8 μ m. Since the ellipse axis along the traveling direction is 36 μ m, 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 μ m below the surface, while the highest shear stress is located 16.6 μ m below the surface. This indicates that the loading is non-proportional. 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 μ m 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

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Structural Mechanics>Solid Mechanics (solid).
- 3 Click Add.
- 4 Click \bigcirc Study.

- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click 🗹 Done.

GEOMETRY I

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

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Import I (imp1)

- I In the Home toolbar, click 🖽 Import.
- 2 In the Settings window for Import, locate the Import section.
- 3 Click **Browse**.
- **4** Browse to the model's Application Libraries folder and double-click the file linear_guide_geometry.mphbin.
- 5 Click া Import.

Create selections to apply features more easily.

DEFINITIONS

Contact Volume

- I In the **Definitions** toolbar, click **herefore 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

- I 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

- I 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Click **b** 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 I

- I In the Model Builder window, under Component I (compl) right-click Definitions and choose Variables.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** Click **b** 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 I (an I)

- I In the **Definitions** toolbar, click $\begin{bmatrix} f \\ Q \end{bmatrix}$ 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 I (mat1)

- I In the Model Builder window, under Component I (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	I	Young's modulus and Poisson's ratio
Density	rho	7800	kg/m³	Basic

SOLID MECHANICS (SOLID)

Boundary Load 1

- I In the Model Builder window, under Component I (comp1) right-click Solid Mechanics (solid) 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 I

- I In the Physics toolbar, click 🔚 Boundaries and choose Fixed Constraint.
- **2** Select Boundary 6 only.

Roller I

- I 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 I

Free Triangular 1

- I In the Mesh toolbar, click \bigwedge Boundary 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 I

- I Right-click Free Triangular I 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 check box. In the associated text field, type 0.01.

Free Tetrahedral I

- I In the Mesh toolbar, click \land 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

- I Right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 From the Predefined list, choose Extra fine.

Distribution I

- I In the Model Builder window, right-click Free Tetrahedral I 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** check box.

Free Tetrahedral 2

In the Mesh toolbar, click \land Free Tetrahedral.

Size 1

- I 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 check box. In the associated text field, type 0.1.

6 Click 📗 Build All.

STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study Extensions section.
- 3 Select the Auxiliary sweep check box.
- 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 (soll)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Stationary Solver I node.
- 4 Right-click Study I>Solver Configurations>Solution I (solI)>Stationary Solver I> Suggested Iterative Solver (solid) and choose Enable.
- **5** In the **Study** toolbar, click **= Compute**.

RESULTS

Study I/Mirror 3D

- I 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

- I 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 Study I/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

- I In the Model Builder window, expand the Surface: Equivalent Stress node, then click Volume I.
- 2 In the Settings window for Volume, locate the Expression section.
- 3 From the Unit list, choose MPa.

Deformation

- I In the Model Builder window, expand the Volume I node, then click Deformation.
- 2 In the Settings window for Deformation, locate the Scale section.
- 3 Select the Scale factor check box. 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

I 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 **O** Plot.

Volume 1

- I In the Model Builder window, expand the Surface: Contact Stress node, then click Volume I.
- 2 In the Settings window for Volume, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (comp1)>Solid Mechanics> Stress>Principal stresses>solid.sp3 - 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.

Study I / Cut Plane: Through Thickness

- I 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

- I 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 **OM** Plot.

Surface 1

- I 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 I (comp1)>Solid Mechanics> Stress>solid.mises - von Mises stress - N/m².
- 3 Locate the Expression section. From the Unit list, choose MPa.

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

Through Thickness: Equivalent Stress

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID Plot Group, type Through Thickness: Equivalent Stress in the Label text field.

Line Graph 1

- I 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 I (compl)>Solid Mechanics>Stress>solid.mises von Mises stress N/m².
- 5 Locate the y-Axis Data section. From the Unit list, choose MPa.
- 6 Locate the x-Axis Data section. From the Parameter list, choose Reversed arc length.

Through Thickness: Equivalent Stress

- I In the Model Builder window, click Through Thickness: Equivalent Stress.
- 2 In the Settings window for ID Plot Group, locate the Plot Settings section.
- 3 Select the x-axis label check box. In the associated text field, type Depth (mm).

Through Thickness: Shear Stress

- I Right-click Through Thickness: Equivalent Stress and choose Duplicate.
- 2 In the Settings window for ID Plot Group, type Through Thickness: Shear Stress in the Label text field.

Line Graph 1

- I 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 I (compl)>
 Solid Mechanics>Stress tensor (spatial frame) N/m²>solid.sxy 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

I 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

- I 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

- I In the **Results** toolbar, click \sim **ID** Plot Group.
- 2 In the Settings window for ID 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

- I 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 I (compl)> Solid Mechanics>Stress>solid.mises von Mises stress N/m².
- 3 Locate the y-Axis Data section. From the Unit list, choose MPa.
- 4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 5 In the Expression text field, type cPos(n).
- 6 Click to expand the Legends section. Select the Show legends check box.
- 7 From the Legends list, choose Manual.
- 8 In the table, enter the following settings:

Legends

Mises

Point Graph 2

I Right-click Point Graph I 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 I (compl)> Solid Mechanics>Stress>Stress tensor (spatial frame) N/m²>solid.sxx Stress tensor, xx-component.
- 3 Locate the Legends section. In the table, enter the following settings:

Legends

SX

Point Graph 3

- I 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 I (compl)> Solid Mechanics>Stress>Stress tensor (spatial frame) N/m²>solid.syy Stress tensor, yy-component.
- 3 Locate the Legends section. In the table, enter the following settings:

Legends

sy & sz

Point Graph 4

- I 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 I (compl)> Solid Mechanics>Stress>Stress tensor (spatial frame) N/m²>solid.sxy Stress tensor, xy-component.
- **3** Locate the **Legends** section. In the table, enter the following settings:

Legends

sxy & sxz

Point Graph 5

- I 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 I (compl)> Solid Mechanics>Stress>Stress tensor (spatial frame) N/m²>solid.syz 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 **O Plot**.

Point of Max Mises: Stress Components

- I In the Model Builder window, click Point of Max Mises: Stress Components.
- 2 In the Settings window for ID 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 check box. In the associated text field, type Load location (mm).
- 6 Select the y-axis label check box. 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

- I Right-click Point of Max Mises: Stress Components and choose Duplicate.
- 2 In the Settings window for ID 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 **I** Plot.

Now, perform the fatigue analysis.

ADD PHYSICS

- I 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 check box for Study 1.
- 5 Click Add to Component I in the window toolbar.
- 6 In the Home toolbar, click 🖄 Add Physics to close the Add Physics window.

FATIGUE (FTG)

Stress-Based I

- I Right-click Component I (comp1)>Fatigue (ftg) and choose the domain evaluation 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 I (mat1)

- I In the Model Builder window, under Component I (compl)>Materials click Material I (matl).
- 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

- I In the Home toolbar, click $\stackrel{\sim}{\xrightarrow{}}$ 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** check box for **Solid Mechanics (solid)**.
- 4 Find the Studies subsection. In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Fatigue.
- 5 Click Add Study in the window toolbar.
- 6 In the Home toolbar, click $\stackrel{\sim}{\longrightarrow}$ Add Study to close the Add Study window.

STUDY 2

Step 1: Fatigue

- I 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 I, Stationary.
- **5** In the **Home** toolbar, click **= Compute**.

RESULTS

Study2/Mirror 3D

- I In the Model Builder window, right-click Study I/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)

- I 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

- I In the Model Builder window, right-click Study I/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

- I In the Home toolbar, click 🚛 Add Plot Group and choose 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

- I Right-click Subsurface: Fatigue Usage Factor and choose Surface.
- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>Fatigue>ftg.fus Fatigue usage factor.
- 3 In the Subsurface: Fatigue Usage Factor toolbar, click 💽 Plot.