Creep Analysis of a Turbine Stator Blade

Introduction

The conditions within gas turbines are extreme. The pressure can be as high as 40 bars, and the temperature far above 1000 K. Any new component must therefore be carefully designed to be able to withstand thermal stress and vibrations due to the rotating machinery and aerodynamic loads exerted by the fluid rushing through the turbine. If a component fails, the high rotational speeds can result in a complete collapse of the whole turbine.

This example performs an analysis of creep deformation caused by thermal stress during operation conditions. The model is an extension of the *Thermal Stress Analysis of a Turbine Stator Blade* example from the Structural Mechanics Module Application Library. The setup of the stationary thermal stress problem is discussed thereafter. The focus of this example is on the extension of the original model to include creep deformation in a time-dependent analysis.

Note: This application requires the CFD Module or the Heat Transfer Module to solve the stationary problem.

Model Definition

The model geometry is shown in Figure 1 The stator blade profile is a modified version of a design shown in Ref. 1. The geometry also includes some generic mounting details as well as a generic internal cooling duct.

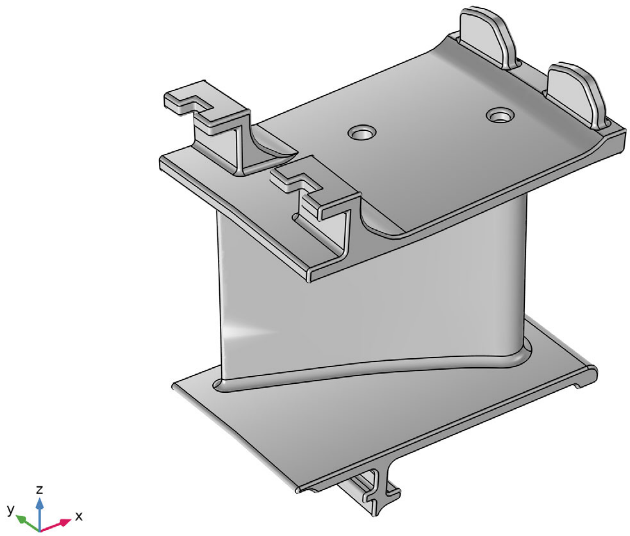

Figure 1: A stator blade with mounting details.

The blade and the mounting details are made of a directionally solidified (DS) GTD111 nickel-based alloy. The basic properties of this alloy are available in the COMSOL Material Library, including for example a temperature-dependent Young's modulus. In addition to the data included in the Material Library, the linear elastic model set up requires a reference temperature of 310 K and a Poisson's ratio equal to 0.33, a value comparable to other stainless steels.

Creep deformation is included in the analysis by adding a **Creep** node to the linear elastic material model. The creep properties of (DS) GTD111 are highly influenced by both temperature and stress. Furthermore, the directional solidification of the alloy leads to anisotropic behavior. While this anisotropy could be included by using the Hill's orthotropic equivalent stress available under the **Creep** node, it is however assumed that creep deformation is isotropic, given that the grain directions in the component are unknown.

The example considers secondary creep only, which can be described by a thermally activated Norton creep model. The rate of the creep strain ε_{cr} is then given by

$$\dot{\varepsilon}_{cr} = A\left(\frac{\sigma_{mises}}{\sigma_{ref}}\right)^{n}\exp\left[-\frac{Q}{R}\left(\frac{1}{T}-\frac{1}{T_{ref}}\right)\right]\frac{3}{2}\frac{\mathrm{dev}(\sigma)}{\sigma_{mises}} \qquad (1)$$

where A is the creep rate coefficient, n is the stress exponent, and σ_{ref} is a reference stress level. The temperature dependence is controlled by an Arrhenius function where Q is the activation energy, R is the gas constant, T is the current absolute temperature of the material, and T_{ref} is the reference temperature for the activation energy.

Creep properties of (DS) GTD111 are difficult to determine, the data to predict the secondary creep using Equation 1 is summarized in Table 1.

TABLE 1: CREEP PROPERTIES USED FOR (DS) GTD111.

Property	**Symbol**	**Value**	**Unit**
Creep rate	A	2.5e-23	1/h
Stress exponent	n	6.75	1
Reference stress	σ_{ref}	1	MPa
Activation energy	Q	350	kJ/mol
Reference temperature	T_{ref}	900	K

The creep deformation of the component is studied during a period of 400 h of constant and stable operation. During this period, the temperature conditions are assumed constant, and the temperature is therefore not solved in the time-dependent study step. The temperature and initial conditions for the displacements and stress are solved in the preceding stationary study step set up, described in the reference model. That model also includes the definition of all relevant boundary conditions and loads.

Results and Discussion

The accumulated amount of creep strain after 400 h of isothermal operation is shown in Figure 2. It is observed that the maximum creep strain reaches a value of approximately 1%. Creep rupture of (DS) GTD111 has been reported to occur at around 10% creep strain, so the stator blade still has plenty of safety margin after 400 h of stable operation. There is no risk that cracks start to form, neither that the tertiary creep regime is entered.

An interesting observation, when comparing the distribution of the equivalent creep strain in Figure 2 to the stress state at the beginning of the time-dependent step shown in Figure 3; is that the maximum creep strain does not coincide with the peak stress. This is caused by the temperature distribution, where areas around the cooling ducts' inlets and outlets have a lower temperature than other parts of the stator blade.

The stress relaxation caused by creep is clearly visible when comparing Figure 3 to the stress distribution after 400 h of operation. The stress level decreases by several hundreds of MPa after this time lapse.

The variation of the maximum creep strain in the component is shown in Figure 5. It is concluded that most of the creep deformation occurs during the initial operational hours, and that stress relaxation also leads to a reduced creep rate. Note that Figure 5 shows the maximum creep strain in the entire component, and that the actual material point where maximum creep occurs may change between time steps.

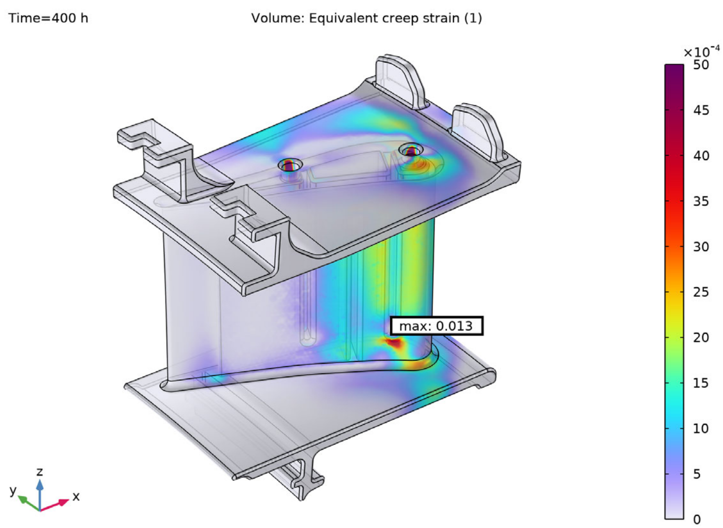

Figure 2: Distribution of the equivalent creep strain after 400 h of stable operation.

.

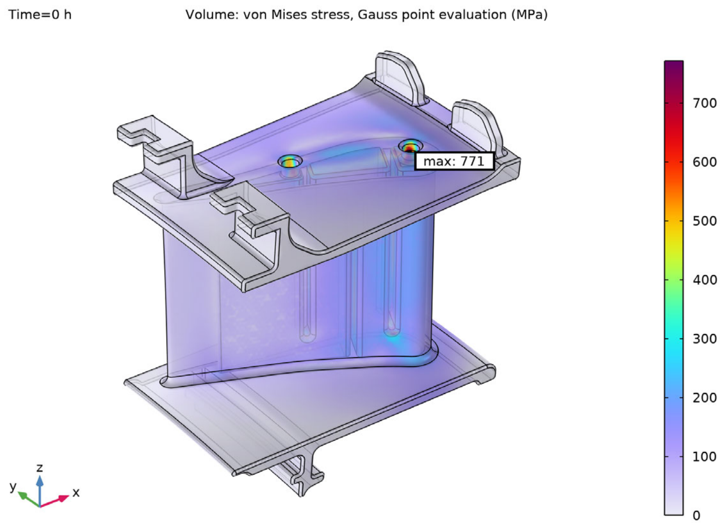

Figure 3: Distribution of von Mises stress at the start of operation.

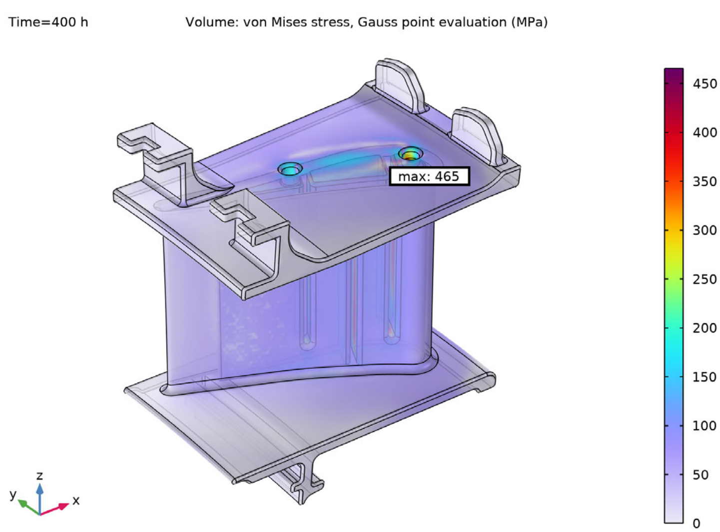

Figure 4: Distribution of von Mises stress after 400 h of stable operation.

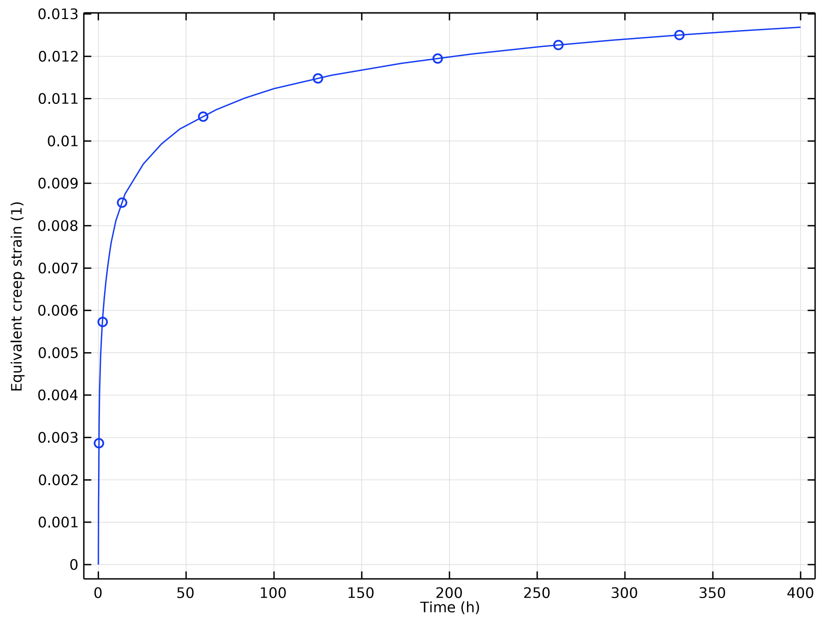

Figure 5: Maximum creep strain in the stator blade versus time.

Reference

1. NASA, "Power Turbine", Glenn Research Center, www.grc.nasa.gov/WWW/K-12/airplane/powturb.html.

Application Library path: `Nonlinear_Structural_Materials_Module/Creep/turbine_stator_creep`

Modeling Instructions

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **Structural Mechanics Module>Thermal-Structure Interaction>turbine_stator** in the tree.

3 Click **Open**.

COMPONENT 1 (COMP1)

Add a creep model to the physics and creep properties to the material.

1 In the **Model Builder** window, expand the **Component 1 (comp1)** node.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

The local computations at Gauss points during assembly are expensive for creep models. Using a reduced integration scheme will reduce the overall simulation time.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Quadrature Settings** section.

3 Select the **Reduced integration** check box.

Creep 1

1 In the **Physics** toolbar, click **Attributes** and choose **Creep**.

2 In the **Settings** window for **Creep**, locate the **Creep Model** section.

3 Find the **Thermal effects** subsection. From the $g(T)$ list, choose **Arrhenius**.

4 In the T_{ref} text field, type `T_work`.

5 In the Q text field, type `350[kJ/mol]`.

MATERIALS

GTD111 DS [solid,longitudinal] (mat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **GTD111 DS [solid,longitudinal] (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
Creep rate coefficient	A_nor	`2.5e-23[1/h]`	1/s	Norton
Reference stress	sigRef_nor	`1[MPa]`	N/m²	Norton
Stress exponent	n_nor	`6.75`	1	Norton

Monitor the maximum equivalent creep strain in the component using a **Domain Probe**.

DEFINITIONS

Domain Probe 1 (dom1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Probes>Domain Probe**.

3 In the **Settings** window for **Domain Probe**, locate the **Probe Type** section.

4 From the **Type** list, choose **Maximum**.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.eceGp`.

6 Select the **Description** check box.

STUDY 1

Step 2: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node.

2 Right-click **Study 1** and choose **Study Steps>Time Dependent>Time Dependent**.

3 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

4 From the **Time unit** list, choose **h**.

5 In the **Output times** text field, type `0 10 100 400`.

The temperature is assumed constant during operations.

6 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Heat Transfer in Solids (ht)**.

Reset the solver to its default, and modify the generated solver sequence to improve the performance of the model.

Solution 1 (sol1)

1 In the **Model Builder** window, right-click **Solver Configurations** and choose **Reset Solver to Default**.

2 Expand the **Solution 1 (sol1)** node.

Heat Transfer in Solids and **Solid Mechanics** are in this model one-way coupled. This allows the physics to be solved in series, which is more efficient than the suggested default solver.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 From the **Termination technique** list, choose **Iterations**.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node, then click **Temperature**.

7 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

8 From the **Termination technique** list, choose **Tolerance**.

9 In the **Tolerance factor** text field, type `1`.

10 In the **Model Builder** window, click **Solid Mechanics**.

11 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

12 From the **Termination technique** list, choose **Tolerance**.

13 In the **Tolerance factor** text field, type `1`.

When working with inelastic material models such as creep, it is preferable to store data at steps taken by the solver to avoid interpolation errors during result evaluation.

14 In the **Model Builder** window, click **Time-Dependent Solver 1**.

15 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

16 From the **Steps taken by solver** list, choose **Strict**.

17 Select the **Initial step** check box. In the associated text field, type `0.1`.

Step 2: Time Dependent

In the **Model Builder** window, under **Study 1** right-click **Step 2: Time Dependent** and choose **Get Initial Value for Step** to customize the probe plot before solving.

RESULTS

Equivalent Creep Strain vs Time

1 In the **Settings** window for **1D Plot Group**, type `Equivalent Creep Strain vs Time` in the **Label** text field.

2 Locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Equivalent creep strain (1)`.

4 Locate the **Legend** section. Clear the **Show legends** check box.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Equivalent Creep Strain vs Time** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

STUDY 1

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

RESULTS

Stress (solid)

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

Equivalent Creep Strain

1 Right-click **Stress (solid)** and choose **Duplicate**.

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

Volume 1

1 In the **Model Builder** window, expand the **Equivalent Creep Strain** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.eceGp`.

4 In the **Equivalent Creep Strain** toolbar, click **Plot**.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `0.005`.

7 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Nonlinear**.

8 In the **Color calibration parameter** text field, type `-.7`.

Marker 1

1 In the **Model Builder** window, expand the **Volume 1** node, then click **Marker 1**.

2 In the **Settings** window for **Marker**, locate the **Text Format** section.

3 In the **Display precision** text field, type `2`.

4 In the **Equivalent Creep Strain** toolbar, click **Plot**.