

# Thermal Stress 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.

The most extreme conditions are found in the high pressure section, downstream of the combustion chamber where hot combustion gas flows through a cascade of rotors and stators. To prevent melting, relatively "cold" air is taken from bleeding vanes located in the high-pressure compressor casing; this cold air is then led past the combustion chamber into the turbine casing as coolant. Directly behind the combustion chamber, both internal cooling within ducts, and film cooling over the blade side surfaces are applied. Besides, a surface treatment (coating) is often added to the blades to protect them from hot corrosion. Further downstream, where the temperature is somewhat lower, it may suffice with internal cooling. For more details on gas turbines, see Ref. 1.

Since the physics within a gas turbine is very complex, simplified approaches are often used at the initial stages of the development of new components. In this tutorial, the thermal stresses in a stator blade with internal cooling are analyzed (coating and film cooling are not considered in this example).

**Note:** This application requires the Structural Mechanics Module and the CFD Module or Heat Transfer Module. It also uses the Material Library.

# Model Definition

The model geometry is shown in Figure 1. The stator blade profile is a modified version of a design shown in Ref. 2. The geometry includes some generic mounting details as well as a generic internal cooling duct.



Figure 1: A stator blade with mounting details.

Use the Thermal Stress interface from the Structural Mechanics Module to set up the model. The blade and the mounting details are assumed to be made of the directionally solidified (DS) GTD111 nickel-based alloy with high tensile strength (Ref. 1). This material is available in the COMSOL Material Library. In addition to the data covered by the Material Library, the linear elastic model requires a reference temperature that is set to 310 K and a Poisson's Ratio that is set to 0.33, a number comparable to that for other stainless steels.

Figure 2 shows the cooling duct. The duct geometry is simplified and does not include details such as the ribs typical for cooling ducts (Ref. 3). Instead of simulating the complicated flow in the duct, an average Nusselt number correlation from Ref. 3 is used to calculate a heat transfer coefficient. Assume the cooling fluid to be air at 30 bar and 650 K.



Figure 2: The internal cooling duct.

The heat flux on the stator blade surfaces is calculated using the heat transfer coefficient. The pressure and suction sides are approximated as two flat plates using the local heat transfer coefficient for external forced convection. The combustion gases are approximated as air at 30 bar and 1150 K. The corresponding speed of sound is approximately 650 m/s.

Ref. 4 contains a Mach number plot of stators without film cooling. A typical Mach number is 0.45 on the pressure side (the concave side) and 0.7 on the suction side (the convex side). This corresponds to approximately 300 m/s on the pressure side and 450 m/ s on the suction side.

The platform walls adjacent to the stator blades are treated in the same way as the stator itself but with the free stream velocity set to 350 m/s.

The stator blade exchanges heat with the cooling air through the boundaries highlighted in Figure 3. It is assumed that the turbine has a local working temperature of 900 K, and that the heat transfer coefficient to the stator is 25 W/( $m^2 \cdot K$ ).



Figure 3: Boundaries through which heat is exchanged with the cooling air.

The attachment of the stator element to a ring support is simulated via roller and spring foundation boundary conditions on a few boundaries. All other boundaries are free to deform as a result of thermal expansion.

# Results and Discussion

Figure 4 shows a temperature surface plot. The internal cooling creates significant temperature gradients within the blade. However, the trailing edge reaches a temperature close to that of the combustion gases, which indicates that the cooling might be insufficient. The side walls also become very hot, and some additional cooling can be beneficial.



Figure 4: Surface temperature plot.

Figure 5 shows a surface plot of the von Mises stress. The maximum stress does not exceed the yield stress of the material (Ref. 5), which indicates that no static failure occurs. However, the component may still be at risk for thermal fatigue, so no definite assessment can be made without conducting a more advanced analysis that includes, for example, creep and other transient effects.

Volume: von Mises stress, Gauss point evaluation (MPa)



Figure 5: Surface plot of the von Mises stress.

Surface: Displacement magnitude (mm)



Figure 6: Surface plot of the displacement.

# Notes About the COMSOL Implementation

The default suggested solver for a thermal stress analysis assumes that the displacement field and the temperature field are two-way coupled. However, in this example there is only a one-way coupling such that the temperature field is independent of the deformations. In this case, a more efficient solver can be set up where the physics are solved in series. To modify the default solver, set **Termination technique** to **Iterations** in the **Segregated** solver node and make sure that **Iterations** are set to 1. Moreover, set **Termination technique** to **Tolerance** in each **Segregated step** node. With these modifications, the temperature field is first solved to convergence, before the displacements are solved for.

# References

1. M.P. Boyce, *Gas Turbine Engineering Handbook*, 2nd ed., Gulf Professional Publishing, 2001.

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

3. J. Bredberg, "Turbulence Modelling for Internal Cooling of Gas-Turbine Blades", Thesis for the degree of doctor of philosophy, *Chalmers University of Technology*, 2002.

4. P. Dahlander, "Source Term Model Approaches to Film Cooling Simulations", Thesis for the degree of doctor of philosophy, *Chalmers University of Technology*, 2001.

5. http://www.cnalloys.co.uk/stainless-jethete-m152

Application Library path: Heat\_Transfer\_Module/Thermal\_Stress/ turbine\_stator **Note:** Instructions below require to select entities corresponding to a particular numbers list. For example:

Select Boundaries 113 and 139 only.

In most cases the easiest way to select them is to click the **Paste Selection** button and paste the numbers as they are printed in the document (for example paste "113 and 139" for the example above).

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>Thermal-Structure Interaction> Thermal Stress, Solid.
- 3 Click Add.
- 4 Click  $\bigcirc$  Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click **M** Done.

# 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** In the table, enter the following settings:

Name	Expression	Value	Description
Pr_cool	0.72	0.72	Cooling Prandtl number
U_suction_side	450[m/s]	450 m/s	Gas velocity on stator suction side
U_pressure_side	300[m/s]	300 m/s	Gas velocity on stator pressure side

Name	Expression	Value	Description
U_platform	350[m/s]	350 m/s	Gas velocity along platform walls
T_gas	1150[K]	1150 K	Gas temperature
p_high	30[bar]	3E6 Pa	High pressure level
mu_cool	3.1e-5[Pa*s]	3.1E-5 Pa∙s	Viscosity of the cooling air
Cp_cool	770[J/kg/K]	770 J/(kg·K)	Heat capacity of the cooling air
T_cool	650[K]	650 K	Cooling air temperature
H_cool	0.01[m]	0.01 m	Characteristic length scale of cooling channels
T_work	900[K]	900 K	Working temperature
Nu_cool	400	400	Average Nusselt number in cooling channel

## GEOMETRY I

Import I (impl)

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 turbine\_stator.mphbin.
- 5 Click ा Import.

To see the interior:

6 Click the Transparency button in the Graphics toolbar.

The imported geometry should look as shown in the Figure 1.

- 7 In the Home toolbar, click 📗 Build All.
- 8 Click the  $\leftarrow$  Zoom Extents button in the Graphics toolbar.

Define a number of selections to simplify the model setup. First define the internal cooling duct boundaries.

#### DEFINITIONS

Cooling Duct

- I In the Definitions toolbar, click 🐚 Explicit.
- 2 In the Settings window for Explicit, type Cooling Duct in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- 4 Click **Paste Selection**.
- **5** In the **Paste Selection** dialog box, type **59-64**, **66**, **74-114** in the **Selection** text field. If you are reading an electronic version of this document, you can copy the geometric entity numbers from the text.
- 6 Click OK.
- 7 Click the Transparency button in the Graphics toolbar.
- 8 Click the 🔁 Wireframe Rendering button in the Graphics toolbar.

The selection is shown in Figure 2.

Proceed to select the boundaries through which the heat exchange with the rest of the turbine occurs (Figure 3).

Exchange Boundaries

- I In the **Definitions** toolbar, click **here Explicit**.
- 2 In the Settings window for Explicit, type Exchange Boundaries in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- 4 Click Paste Selection.
- **5** In the **Paste Selection** dialog box, type 1-4, 9, 11, 14, 16, 17, 21-31, 36-39, 42-58, 65, 67-72, 115-136 in the **Selection** text field.
- 6 Click OK.
- 7 Click the Wireframe Rendering button in the Graphics toolbar.

## ADD MATERIAL

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Material Library>Nickel Alloys>GTD111 DS>GTD111 DS [solid]> GTD111 DS [solid,longitudinal].
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

#### MATERIALS

## GTD111 DS [solid,longitudinal] (mat1)

I In the Settings window for Material, locate the Material Contents section.

**2** In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.33	1	Young's modulus and Poisson's ratio
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	k(T)	W/(m·K)	Basic

3 In the Model Builder window, expand the GTD111 DS [solid,longitudinal] (mat1) node.

Piecewise (E\_solid\_longitudinal\_l)

- I In the Model Builder window, expand the Component I (comp1)>Materials> GTD111 DS [solid,longitudinal] (mat1)>Young's modulus and Poisson's ratio (Enu) node, then click Piecewise (E\_solid\_longitudinal\_1).
- 2 In the Settings window for Piecewise, locate the Definition section.
- **3** From the **Extrapolation** list, choose **Nearest function**.

## MULTIPHYSICS

#### Thermal Expansion 1 (tel)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Thermal Expansion I (tel).
- 2 In the Settings window for Thermal Expansion, locate the Model Input section.
- 3 Click **Go to Source** for Volume reference temperature.

#### **GLOBAL DEFINITIONS**

#### Default Model Inputs

- I In the Model Builder window, under Global Definitions click Default Model Inputs.
- 2 In the Settings window for Default Model Inputs, locate the Browse Model Inputs section.
- **3** Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type **310**[K].

#### HEAT TRANSFER IN SOLIDS (HT)

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Heat Transfer in Solids (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *T* text field, type T\_gas.

## Heat Flux 1

- I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.
- 2 In the Settings window for Heat Flux, locate the Boundary Selection section.
- 3 From the Selection list, choose Exchange Boundaries.
- 4 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux.
- **5** In the h text field, type 25.
- **6** In the  $T_{\text{ext}}$  text field, type T\_work.

## Heat Flux 2

- I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.
- 2 In the Settings window for Heat Flux, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Cooling Duct**.
- 4 Locate the Heat Flux section. From the Flux type list, choose Convective heat flux.
- **5** In the *h* text field, type Nu\_cool\*mu\_cool\*Cp\_cool/2/Pr\_cool/H\_cool.
- **6** In the  $T_{\text{ext}}$  text field, type T\_cool.

#### Heat Flux 3

- I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.
- **2** Select Boundary 41 only.
- 3 In the Settings window for Heat Flux, locate the Heat Flux section.
- 4 From the Flux type list, choose Convective heat flux.
- 5 From the Heat transfer coefficient list, choose External forced convection.
- 6 From the list, choose Plate, local transfer coefficient.
- 7 In the  $x_{\rm pl}$  text field, type 0.1675-x.
- 8 In the U text field, type U\_suction\_side.
- **9** In the  $p_A$  text field, type p\_high.
- **IO** In the  $T_{\text{ext}}$  text field, type T\_gas.

#### Heat Flux 4

- I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.
- **2** Select Boundary 40 only.
- 3 In the Settings window for Heat Flux, locate the Heat Flux section.
- 4 From the Flux type list, choose Convective heat flux.
- 5 From the Heat transfer coefficient list, choose External forced convection.
- 6 From the list, choose Plate, local transfer coefficient.
- 7 In the  $x_{\rm pl}$  text field, type 0.1675-x.
- 8 In the U text field, type U\_pressure\_side.
- **9** In the  $p_A$  text field, type p\_high.
- IO In the  $T_{\rm ext}$  text field, type T\_gas.

#### Heat Flux 5

- I In the Physics toolbar, click 🔚 Boundaries and choose Heat Flux.
- 2 Select Boundaries 10, 15, and 32–35 only.
- 3 In the Settings window for Heat Flux, locate the Heat Flux section.
- 4 From the Flux type list, choose Convective heat flux.
- 5 From the Heat transfer coefficient list, choose External forced convection.
- 6 From the list, choose Plate, local transfer coefficient.
- 7 In the  $x_{\rm pl}$  text field, type 0.19-x.
- 8 In the *U* text field, type U\_platform.
- **9** In the  $p_A$  text field, type p\_high.
- **IO** In the  $T_{\text{ext}}$  text field, type T\_gas.

## SOLID MECHANICS (SOLID)

In the Model Builder window, under Component I (compl) click Solid Mechanics (solid).

Spring Foundation 1

- I In the Physics toolbar, click 🔚 Boundaries and choose Spring Foundation.
- **2** Select Boundaries 42, 43, 56, 129, and 131 only.
- 3 In the Settings window for Spring Foundation, locate the Spring section.
- 4 From the list, choose Diagonal.

**5** In the  $\mathbf{k}_{A}$  table, enter the following settings:

1e9	0	0
0	0	0
0	0	0

# Spring Foundation 2

- I In the Physics toolbar, click 📄 Boundaries and choose Spring Foundation.
- 2 Select Boundaries 27, 29, 53, and 70 only.
- 3 In the Settings window for Spring Foundation, locate the Spring section.
- 4 From the list, choose **Diagonal**.
- **5** In the  $\mathbf{k}_{A}$  table, enter the following settings:

0	0	0
0	0	0
0	0	1e9

# Spring Foundation 3

- I In the Physics toolbar, click 📄 Boundaries and choose Spring Foundation.
- **2** Select Boundaries 6 and 12 only.
- 3 In the Settings window for Spring Foundation, locate the Spring section.
- 4 From the list, choose **Diagonal**.
- **5** In the  $\mathbf{k}_{A}$  table, enter the following settings:

0	0	0
0	1e9	0
0	0	0

# MESH I

Size 1

In the Model Builder window, under Component I (comp1) right-click Mesh I and choose Size.

# Size

- I In the Settings window for Size, locate the Element Size section.
- 2 Click the **Custom** button.

- **3** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.025.
- 4 In the Minimum element size text field, type 0.0025.
- 5 In the Maximum element growth rate text field, type 2.
- 6 In the Curvature factor text field, type 0.75.
- 7 In the Resolution of narrow regions text field, type 0.5.

## Size I

- I In the Model Builder window, click Size I.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Boundary**.
- 4 From the Selection list, choose Cooling Duct.

Also add boundaries 40 and 41 to the selection.

- 5 Locate the Element Size section. Click the Custom button.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.0035.
- 8 Select the Maximum element growth rate check box.
- 9 In the associated text field, type 1.1.

#### Free Tetrahedral I

- I In the Mesh toolbar, click 🧄 Free Tetrahedral.
- 2 In the Settings window for Free Tetrahedral, click 📗 Build All.

#### STUDY I

**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.

#### Solution 1 (soll)

- I In the Study toolbar, click **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, then click Segregated I.
- 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 I>Solver Configurations>
  Solution I (soll)>Stationary Solver I>Segregated I 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, under Study I>Solver Configurations>Solution I (soll)> Stationary Solver I>Segregated I click Solid Mechanics.
- II In the Settings window for Segregated Step, locate the Method and Termination section.

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

**I3** In the **Tolerance factor** text field, type 1.

**I4** In the **Study** toolbar, click **= Compute**.

#### RESULTS

#### Stress (solid)

The first default plot shows the von Mises stress. Disable the deformation and create a max/min marker to identify the critical point in the stator.

#### Volume 1

- I In the Model Builder window, expand the Stress (solid) node, then click Volume I.
- 2 In the Settings window for Volume, locate the Expression section.
- 3 In the Expression text field, type solid.misesGp.
- 4 From the Unit list, choose MPa.

#### Deformation

- I In the Model Builder window, expand the Volume I node.
- 2 Right-click **Deformation** and choose **Disable**.

#### Marker I

- I In the Model Builder window, right-click Volume I and choose Marker.
- 2 In the Settings window for Marker, locate the Display section.
- 3 From the Display list, choose Max.
- 4 Locate the Text Format section. In the Display precision text field, type 4.
- 5 Locate the Coloring and Style section. From the Background color list, choose From theme.

6 Select the Show frame check box.

#### Transparency I

- I Right-click Volume I and choose Transparency.
- 2 In the Settings window for Transparency, locate the Transparency section.
- **3** In the **Transparency** text field, type **0.2**.
- 4 In the Fresnel transmittance text field, type 0.5.
- 5 In the Stress (solid) toolbar, click 💿 Plot.

The second default plot shows the temperature distribution Figure 4.

#### Temperature (ht)

- I In the Model Builder window, under Results click Temperature (ht).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- **3** Clear the **Plot dataset edges** check box.

#### Transparency I

- I In the Model Builder window, expand the Temperature (ht) node.
- 2 Right-click Surface and choose Transparency.
- 3 In the Settings window for Transparency, locate the Transparency section.
- 4 In the **Transparency** text field, type 0.2.
- 5 In the Fresnel transmittance text field, type 0.5.
- 6 In the Temperature (ht) toolbar, click **D** Plot.
- 7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Finally, plot the displacement (Figure 6).

#### Displacement

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, type Displacement in the Label text field.

# Surface 1

- I Right-click Displacement 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 (comp1)>Solid Mechanics> Displacement>solid.disp Displacement magnitude m.
- 3 Locate the Expression section. From the Unit list, choose mm.
- 4 Locate the Coloring and Style section. From the Color table list, choose SpectrumLight.

**5** In the **Displacement** toolbar, click **O** Plot.

6 Click the 🕂 Zoom Extents button in the Graphics toolbar.

20 | THERMAL STRESS ANALYSIS OF A TURBINE STATOR BLADE