CILAMCE 2017
Florianópolis - Santa Catarina - Brazil

STUDY OF THE HEAT TRANSFER PROCESSES IN GAS TURBINE BLADES VIA FINITE ELEMENT METHOD

Franciele Patricia da Silva Muchick

Sara Del Vecchio

fran_muchick@hotmail.com

sara.vecchio@ifsudestemg.edu.br

Federal Institute of Education, Science and Technology from South-East of Minas Gerais, Department of Education and Technology – Mechanics.

Bernardo Mascarenhas St., 1283, Fábrica, 36080-001, Juiz de Fora, Minas Gerais, Brazil

Flávia de Souza Bastos

flavia.bastos@ufjf.edu.br

Universidade Federal de Juiz de Fora, Department of Computational Mechanics.

José Lourenço Kelmer St., s/n, 36030-330, Juiz de Fora, Minas Gerais, Brazil.

Abstract. *Nowadays, the capability of making turbines that withstand higher temperatures is of extreme value for companies, which can make turbines with greater efficiency to distribute in the market. Using the Surrogate Loss System Method, an aerodynamic profile for the blade was calculated and generated according to the initial parameters selected in order to design an optimum blade for the turbine rotor, aiming the analysis of the existing methods for cooling it, in pursuance of extend the efficiency of the turbine as a whole. The heat transfer processes were simulated via finite element method. Considering the blades made of stainless steel AISI 310, the blade was submitted to a temperature of 1723.15K, which was the temperature of the inlet gases in the turbine module. Then a convection cooling was investigated into blade channels using the drained air of the last stage of the compressor, which was at 823.15K. Six different configurations were simulated increasing the number of cooling channels and redistributing them along the cross section of the blade, but for the purpose of this paper only three of them will be discussed, showing mostly the behavior of the heat transfer process in Abaqus.*

Keywords: *Gas turbine, Cooling channels, Convection, Conduction, Numerical simulation.*

INTRODUCTION

At the present time, there is not much research that take into consideration the heat transfer processes that occur in gas turbines, which ends up seriously harming the aeronautics industry, since it is not possible to increase the inlet temperatures in the turbine stage without damaging the structure or harming the operation and safety of the gas turbine (BOYCE, 2012).

This paper aims to study via a finite element method the heat transfer process throughout the blade when cooling channels are applied to it. In order to pursue the objective of this article, a blade was designed following the boundary conditions stipulated for the gas turbine in study.

By using the Surrogate Loss System method and all the initial parameters for the turbine, such as inlet temperature, linear and angular velocities on the turbine axis, power, inlet and outlet pressures and the mass flux of air, an aerodynamic profile was designed (MARCU, 2015). Then, this profile was submitted to simulation using the software Abaqus^{TM} 6.13. Six different cases has been studied in order to proof that by increasing the number of cooling channel lower temperatures are reached on the blades, which increases the efficiency of the turbine as a whole. In this work, only three cases will be presented: no cooling channels; one channel with the blade shape and the last with multiple circular channels.

In this article, the numerical method used by Abaqus to get the solutions will be on spot.

1 PREVIOUS STUDIES

One of the most important consideration while projecting a gas turbine is the necessity that some of his components have to not absorb too much heat during the operation, because that can be risk to the structure and safety. On the other hand, higher efficiencies depends on the inlet temperature, which is limited by the materials used to manufacture the turbine blades. In order to solve this problem, the application of a continuum cooling method allow operation temperatures higher than the melting temperature of the structural materials, with no harm to the parts.

Multiples methods of cooling can be used in order to raise the inlet temperatures on the turbine stage, which is the one responsible for increasing the efficiency of the turbo machine.

The cooling air is drained from the compressor and directed to the stator, rotor and other parts that needs cooling. The effect of the cooling fluid depends on the type of it, location and direction that the fluid is injected on the system.

Nowadays, to provide cooling some concepts have been used such as: Convection cooling, Impact cooling, Film cooling, Transpiration Cooling and Injection Cooling (Water) (SOARES, 2015).

In this paper, the Convection cooling will be on spot. This type of cooling is achieved making an air flux through the blade channels, removing heat of the internal walls. Usually, the air flux is radial and travels throughout the channels as can be seen on Fig. 1.

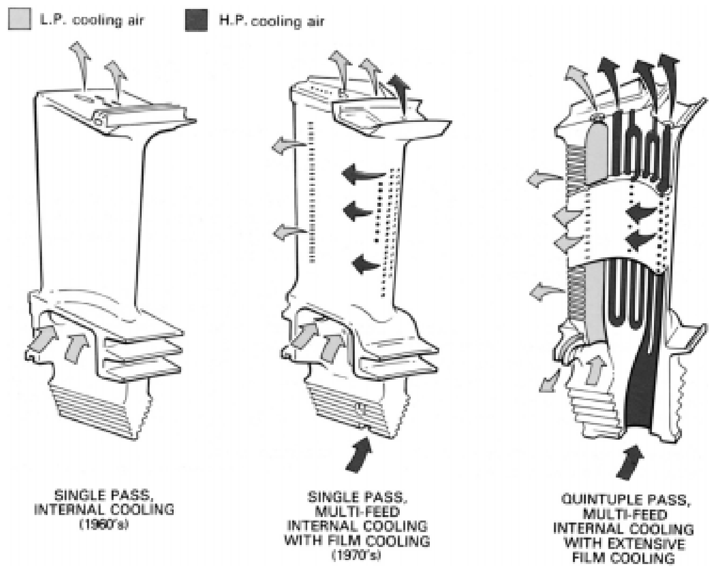

Figure 1. Evolution of the Convection cooling method.
Source: SOARES, 2015.

Many engineering phenomena can be described in analytical form using differential equations. Nevertheless, there is cases where the geometry of the problem became too complex, and those classical forms cannot be applied anymore. In those cases, in order to make it possible to solve, the Finite Element Method is used.

Based on the fact that by increasing the exhaustion temperature of the combustion chamber it is possible to increase expressively the turbine power, some researches have been done aiming to raise the efficiency of the turbo machines in study.

Findlay (2005), developed on his doctoral thesis, a 3D computational methodology to make conjugated aerodynamic and heat transfer analysis. He used tridimensional algorithms based on finite elements theory and the Reynolds equation corrected by Navier-Stokes equation.

In his study, he implemented reflective boundary conditions, which allowed a special simulation that produced more efficiently the development of the problem, using less iterations and accelerating the convergence process. By using his methodology, he provided a study that concerns the coupling of the conduction of heat in the channel and the heat transfer between the cooling fluid traveling inside the blades and the hot gases flux externally to it.

One of his conclusions were that, because the turbulence, higher temperatures, pressures and velocities that coexists in a gas turbine, the only possible way to understand and model a flow both thermodynamically and aerodynamically is by using computational methods.

Gas turbines operate with pressurized air (550°C – 700°C), which is redirect from the last stage of compressor to refrigerate the turbine blades. The pressure is higher once the air needs to go inside the blades and be release with the main gas stream. Besides this temperature can be considered high, the temperatures encountered on the combustion gases go beyond 1700°C.

Sadowski (2010) proved that in pursuance of improving the temperature resistance of the blades, a unique layer of ceramic coating can also be included. He simulated the blade including the turbulence model k-ε, a two-equation model that uses two additional differential equation in order to evaluate the turbulent viscosity that appear in the Boussinesq model for the Reynolds tension Eq. (1).

$$\underbrace{\overline{\tau}^{(t)}_{yx}}_{1} = \underbrace{-\mu^{(t)}}_{2} \underbrace{\frac{\partial \overline{v_x}}{\partial y}}_{3} \tag{1}$$

The term 1 represents the Reynolds tension in the plane xy, the term 2 the turbulent viscosity, and the term 3 the partial derivative of the x-direction smoothed speed in y-direction. The smoothed speed being an average of all the instant speed during a predefined time.

This model is high applicable in fluid dynamics to simulate fluid characteristics in turbulence conditions. After the simulation, Sadowski realized that blades that include only cooling channels reduce their temperatures in approximately 7.3%, which compared to 25% when both technologies are applied together. The main issue on applying then is the cost. Which stimulate this paper on work with common materials and no coating.

2 SIMULATION PROCEDURE

Within a collection of data about the commercial gas turbines, the parameters were chosen and the aerodynamic profile for the rotor blade was obtained applying the Surrogate Loss System Method (MARCU, 2015).

The parameters used for this blade can be seen in the table 1 bellow:

Table 1: Parameters of the turbine stage on a gas turbine.

Parameter	Value
Turbine Inlet Temperature (average)	1723.15K
Colling Fluid Temperature	823.15K
Axis Velocity	460 m/s
Angular Velocity	4000 rpm
Inlet Velocity of Combustion Gases	762 m/s
Turbine Power	180 MW
Inlet Pressure	180 kPa
Outlet Pressure	88.5 kPa
Mass flux of air	595 kg/s

This method work with an iteration process. The efficiency estimated is recalculated until the value of loss for the real cycle be reached. Numerically, the efficiency increase with the iterations, until the stipulated value.

The profile was then generated using MatLab® 2015.a, and can be seen on Fig. 2.

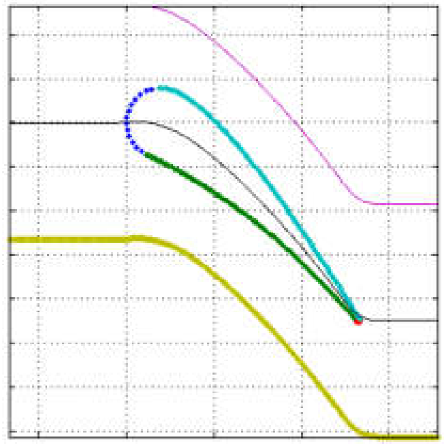

Figure 2. Aerodynamic profile for the rotor turbine blade.

After the creation of the aerodynamic profile, the file was imported to Abaqus CAE to be simulated as a 3D homogeneous extrusion solid.

The material chosen was Stainless Steel 310, an austenitic stainless steel medium carbon content, with 25% chrome and 20% nickel, which makes the blade more resistant to corrosion, oxidation and thermal fatigue.

As the study was mainly thermal, the conductivity was set to vary linearly with temperature, as shown on Eq. (2), according to Findlay (2005).

$$k = 6.811 + 0.020176 \times T \qquad (2)$$

Density (7900kg/m³) and specific heat (585.15J/kg.K) were kept constant during the simulation and the type of material was set isotropic. The same material was applied to the whole geometry of the blade.

The geometry was separated into two surfaces: the first one defining the external surface of the blade, where the combustion gases touch, and the second one limiting the cooling channels.

For this kind of analysis, where heat transfer by convection and diffusion is studied, Abaqus only allows his users to apply hexagonal elements. For this case, the element applied was DCC3D8, an 8-node convection/diffusion brick. After defining the mesh, boundary conditions were stated, considering no initial conditions and analyzing the steady state response.

The convection study needs an interaction called surface film condition to be applied to the cooling channels walls as well as to the external surface of the blade, that is the reason why this simulation could not be a 2D simulation, since Abaqus only allows the application of this interaction condition in 3D elements. The convective coefficient (h) and the bulk temperature (T_b) described it. Beyond the film condition, it was also applied boundary conditions simulating the external temperature and a surface thermal flux internally to the channels, simulating the cooling of the blade.

2.1 Study formalism

To model heat transfer with convection and diffusion in Abaqus is necessary to understand the following formulation. The elements that results for this formulation can be used in any heat transfer mesh.

When heat transfer elements were used, a nonsymmetrical Jacobian matrix is automatically invoked, and it is possible to obtain satisfactory results for either steady state or transient, which depends on the way that the formulation is handed.

The general thermal equilibrium equation in which a fluid is flowing with velocity v, is presented in Eq. (3):

$$\int \delta\theta \left[\rho c \left\{ \frac{\partial \theta}{\partial t} + v \frac{\partial \theta}{\partial x} \right\} - \frac{\partial}{\partial t}\left(k \frac{\partial \theta}{\partial x} \right) - q \right] dV + \int_{S_q} \delta\theta \left[nk \frac{\partial \theta}{\partial x} - q_s \right] dS = 0 \qquad (3)$$

In this equation, the temperature at a specific point and time is given by *θ(x,t)*, and an arbitrary field is represented by *δθ(x,t)*. The properties of the fluid are given as follow: *ρ(θ)* is the density, *c(θ)* is the specific heat, *k(θ)* is the conductivity and q is the heat added per unit of volume *(V)* from external sources, q_s is the heat flowing into the volume across the control surface on which temperature is not prescribed (S_q). The outward normal to the surface is represented by *n*, *x* is the spatial position and *t* is time.

Working with this equation as it is going to be used during the simulation some simplifications are made. The first one is that the heat transfer processes are simulated considering steady state which means that Eq. (4) and Eq. (5) can be applied. It is considered that the fluid temperature does not change during simulation.

$$\frac{\partial \theta}{\partial t} = 0 \qquad (4)$$

$$\frac{\partial}{\partial t}\left(k \frac{\partial \theta}{\partial x} \right) = 0 \qquad (5)$$

In the moment of simulation, the velocity of the fluid is not a input parameter for simulation, so it is stated as Eq. (6)

$$v = 0 \Rightarrow v \frac{\partial \theta}{\partial x} = 0 \qquad (6)$$

Therefore, the Eq. (3) is simplified and turns into Eq. (7)

$$\underbrace{\int \delta\theta [-q] dV}_{1} + \int_{S_q} \delta\theta \left[\underbrace{nk \frac{\partial \theta}{\partial x}}_{2} - \underbrace{q_s}_{3} \right] dS = 0 \qquad (7)$$

The term 1 in Eq. (7) represents the heat flux that is inserted on the cooling channels and tends to refrigerate the blade within the control volume. The term 2 represents the conduction contribution and the term 3 the convection contribution.

Inside the control surface, there is conduction between the isotherm (external surface of the blade) and the channel surfaces, where the film conditions, that are responsible to simulate the convection process, are applied.

2.2 Definition of convective coefficient

To use the interaction of the type Surface film condition two parameters are needed: the bulk temperature and the convective coefficient.

The bulk temperature is the temperature of the gases interfacing with the surface. The convective coefficient (h) is derived from dimensionless numbers such as Nusselt (Nu), Reynold (Re) and Prandtl (Pr).

The Nusselt number is showed on Eq. (8), where L is the characteristic length:

$$Nu = \frac{h \times L}{k_f} \tag{8}$$

Those parameters defines the temperature gradient at the surface, providing an approximation of the heat transfer via convection.

Here, it is necessary to consider the flow as laminar to simplify the analysis. The Eq. (9) shows how the Nusselt number can be related to the Reynolds and Prandtl numbers.

$$Nu = 0.664 \times \mathrm{Re}^{1/2} \times \mathrm{Pr}^{1/3} \tag{9}$$

The electric conductivity can be modeled as null, since the temperatures of the gases both internally and externally to the blade are high. Therefore, the photonic conductivity (k_f) and the thermal conductivity (k) are the same.

The Prandtl number can be defined as the ratio between the dynamic viscosity (μ) and the specific heat (c_p) and the thermal conductivity, as showed in Eq. (10):

$$\mathrm{Pr} = \frac{c_p \times \mu}{k} \tag{10}$$

The specific heat for the air, which is the fluid used since the mass flow of air is much larger than the mass flow of fuel, can be obtained from Eq. (11).

$$c_p = 1{,}05 - 0{,}365 \times \frac{T}{1000} + 0{,}85 \times \left(\frac{T}{1000}\right)^2 - 0{,}39\left(\frac{T}{1000}\right)^3 \tag{11}$$

The viscosity of the air depends on temperature, and is given by the Sutherland Equation, Eq. (12).

$$\mu = \frac{1{,}458 \times 10^{-6} \times T^{\frac{1}{2}}}{1 + \frac{110{,}4}{T}} \tag{12}$$

For this study, the following parameters were used for the film condition, as showed in table 2.

Table 2: Parameters for the film condition.

Parameters for Convection Simulation		
Parameters	**External Surface of the blade**	**Cooling Channels**
Bulk Temperature (T_b)	1723.15K	823.15K
Reynolds Number (Re)	1.5 x 10^5	1.15 x 10^5
Convective Coefficient (h)	3333.195W/m².K	2659.442 W/m²K

3 RESULTS

By applying the finite element method using the Abaqus software six different cases were studied, the first one using no cooling channels and the five others varying the shape and position of the cooling channels. For the purpose of this paper, only three of the results obtained is going to be shown.

In the first case, no cooling channel was applied. As boundary conditions the external temperature of the blade was set as 1723.15K, and the convective film was added to the external surface as well.

The simulation was set to steady state, in such a manner, after 1 second, the equilibrium was achieved, and the whole blade acquire the external temperature, as shown in Fig. 3.

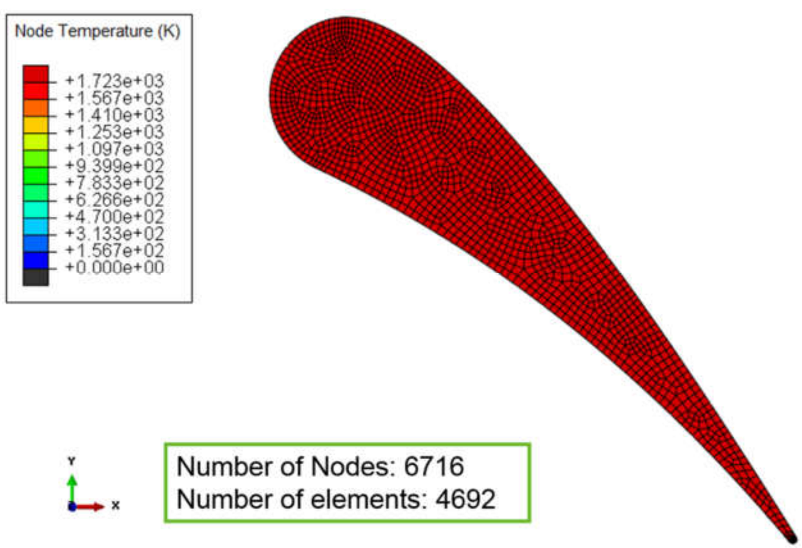

Figure 3. Result of the blade with no cooling channel.

It is possible to observe that the whole blade tends to achieve the equilibrium temperature, which is the combustion gases temperature, which is highly prejudicial to the structure of the blade itself.

In the second case, the cooling channel had the same shape as the blade, dislocated from the leading edge by 20mm. Beyond the boundary conditions already discussed, it was also applied a heat flux load of -2.39 MW/m² and a film coefficient internally to the channel, with a bulk temperature of 823.15K and a convective coefficient of 2659.442 W/m².K. The results obtained can be seen on Fig. 4.

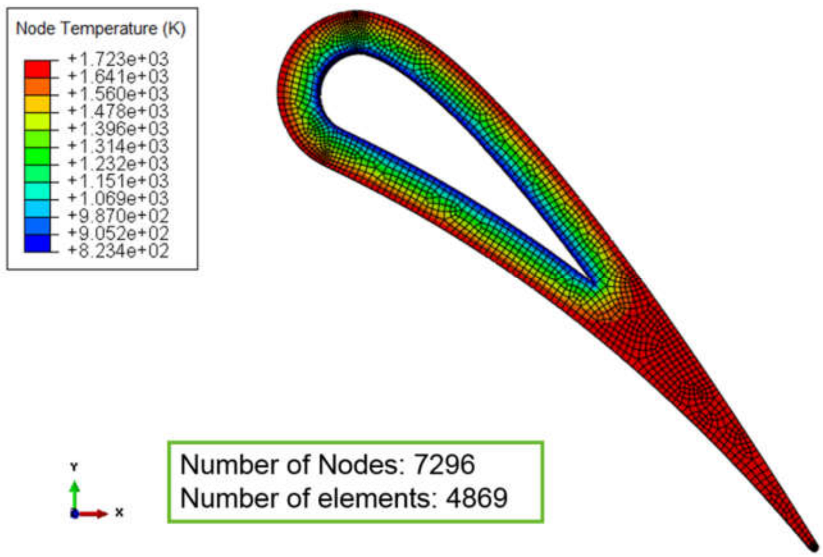

Figure 4. Result with a blade-shape cooling channel.

Due to the geometry of the channel, in his vicinity it was possible to observe an efficient cooling process. Nevertheless, points closer to the trailing edge were not influenced by the presence of the cooling channel, which can cause thermal tensions in those points.

By using the same boundary conditions as the second case, in a third case, the generation of the cooling channels was modified, since, now, there were ten channels, distributed along the transversal section of the blade. The results for this case can be seen on Fig. (5).

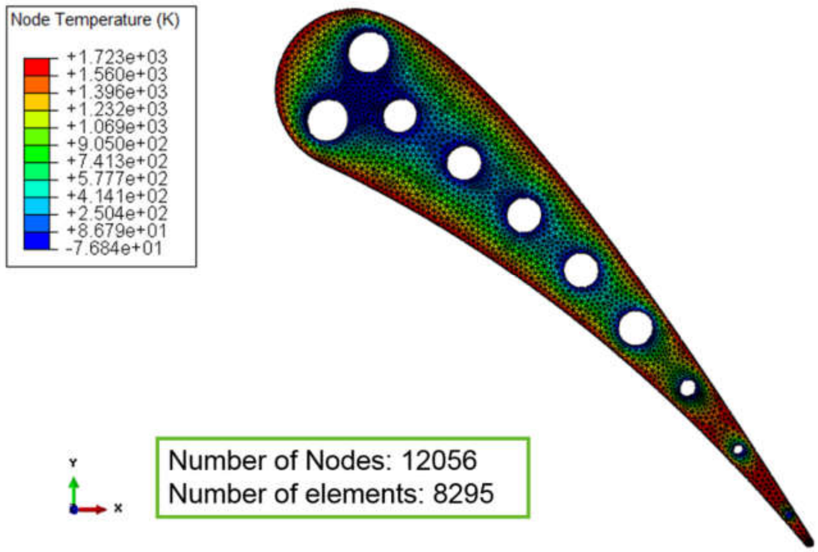

Figure 5. Result with multiple circular cooling channels.

In this last case, the blade was better cooled, which increased, in a sensible way, the turbine efficiency. For this configuration of channels, 80% of the surface area presented lower temperatures than the combustion gases one, presenting a cooling efficiency of 22%.

When it comes to discuss the mesh, it is important to observe the data in table 3.

Table 3. Mesh Characteristics.

Mesh Characteristics		
Cases Studied	**Number of Nodes**	**Number of Elements**
Case 1: No cooling channels	6716	4692
Case 2: One blade-shape cooling channel	7296	4869
Case 3: Multiple circular cooling channels	12056	8295

To be possible to do a mesh analysis it is important to remember that the element used was an 8-node hexagonal convection/diffusion brick. By keeping this in mind and analyzing the previous table, it is feasible to see that as cooling channels were applied to the instance, more nodes and, consequently, more elements were included in the analysis, which occur since an adaptive mesh was enforced, making the software to refine the mesh around the critical surfaces, which was, basically, all the channels. In this case, the adaptive mesh was used to make the mash smoother, since it was used curvature control. Indeed, as more channels were added, more elements were needed in the mesh.

CONCLUSION

The proposed model discriminated on Eq. (1) met the expectations of the study in such a way that the presumed results were achieved: as the number of channels were increased the temperatures in the blade were lower. The number of nodes and elements increased as well, modeling the geometry. In addition, the temperatures obtained in the vicinity of the channels were lower than the one in the isotherm around the blade.

Therefore, this paper achieved his purpose, which was the study via a finite element method of the heat transfer process throughout the blade when cooling channels are applied to it. More studies need to be done in this area of knowledge such as: a thermomechanical coupled study as well as a fluid-structure interaction study, which the authors highly recommend as following studies for this paper.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the support of FAPEMIG (Foundation for Research Support of Minas Gerais), through the provision of financial support to participate in the event.

The first author would also like to acknowledge CAPES (Coordination for the Improvement of Higher Education Personnel), for the scholarship and all support given during the Science without Borders program. In addition, the Professor Bogdan Marcu, whose class at USC, helped with the development of the blade profile. This period was primal to develop the knowledge needed to finish this study.

REFERENCES

BOYCE, M.P. *Gas turbine engineering handbook*. 4th edition. Elsevier: 2012.

FINDLAY, J. P. *3-D Conjugate Heat Transfer analysis of a cooled transonic turbine blade using non-reflecting boundary conditions*. McGill University. Montreal: 2005.

MARCU, B. *Turbine Design and Analysis*. Lectures. University of Southern California. California: 2015.

MANUAL, A., 2013. *Abaqus Documentation* Version 6.13.

SADOWSKI, T.; GOLEWSKI, P**.** *Multidisciplinary analysis of the operational temperature increase of turbine blades in combustion engines by application of the ceramic thermal barrier coatings (TBC)*. Computational Materials Science, p. 1326-1335. Elsevier: 2011.

SOARES, C. *Gas Turbines: A Handbook of Air, Land and Sea Applications.* 2nd edition. Elsevier: 2015.