24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-0291
STUDY OF THE HEAT PROCESSES IN GAS TURBINE BLADES VIA FINITE ELEMENT METHOD

Franciele Patricia da Silva Muchick
Sara Del Vecchio
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
fran_muchick@hotmail.com
sara.vecchio@ifsudestemg.edu.br

Flávia de Souza Bastos
Federal *University* of *Juiz de Fora, Department of Computational Mechanics.*José Lourenço Kelmer St., s/n, 36030-330, Juiz de Fora, Minas Gerais, Brazil.
flavia.bastos@ufjf.edu.br

Abstract. *Aerospace companies aim to produce turbines that can withstand higher temperatures, which improves exponentially the efficiency of them. Nevertheless, sometimes this is not possible, since the materials and the project itself cannot sustain those temperatures levels. In order to extend the efficiency of the turbine as a whole and according to the initial parameters selected, an optimum blade was designed using the Surrogate Loss System method, and then the heat transfer in the cross section of the blade was simulated via finite element method. Considering the blades made of stainless steel AISI 310, and submitting them to the temperature of the inlet gases in the turbine module, 1723.15K, a convection cooling was investigated, making the drained air of the last stage of the compressor to flow into the blade channels, 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, resulting that the most efficient blade was the one that had as many cooling channels as possible, maintaining the mechanical structure integrity.*

Keywords: *Gas Turbine, Cooling Channels, Convection, Conduction, Numerical Simulation.*

1. INTRODUCTION

Gas turbines are highly used as a propulsion method to airplanes. The aeronautics industry ends up seriously harming by the fact that not much research has been done in the context of heat transfer along the blades and structure of the gas turbines. The lack of research come to a halt that it is impossible to raise the inlet temperatures in the turbine stage, which could result in an increased thrust and, consequently, a major efficiency of the turbomachinery (Soares, 2015).

This paper intent to design, via Surrogate Loss System method (Marcu, 2015), a blade that support major inlet temperatures and follows all boundary conditions previously stipulated, such as inlet temperature, linear and angular velocities on the turbine axis, power, inlet and outlet pressures and the mass flux of air. In addition to that, a study of the heat transfer along the blades was performed, varying the number of cooling channels, the geometry as well as the position of them. Six cases were analyzed: no cooling channels; one circular channel close to the leading edge of the blade; one cooling channel with the blade profile; three cooling channels; five cooling channels and ten cooling channels, being the last three cases mentioned, with the channels distributed along the cross section of the blade.

In this article, the process of calculating and designing the blade, as well as the impact of using the cooling channels to improve the heat transfer process will be on spot.

2. EARLIER STUDIES

By the fact that increasing the temperature of the gases that entered the first stage of the turbine module it is possible to increase the efficiency of the whole turbomachinery, some researchers had been done studies in order to make the cooling processes, that occur inside the turbine blades, more efficient, in a way which will be possible to boost the turbine efficiency with no harm to its operation or security.

Some concepts have been used as far as cooling concerns, for instance: Convection Cooling; Impact Cooling; Film Cooling, Transpiration Cooling and Injection of water for cooling (Soares, 2015 and Boyce, 2012). In this paper, the Convection Cooling will be on spot. This type of cooling is achieved making an air flux to pass through the blade channels, removing heat of its internal walls. Usually, the air flux is radial and travels throughout the channels.

At the beginning of the 21st century, Kumar (2002), using a ceramic coating in the external surfaces of the blade, performed studies in spite of elevate the inlet temperatures to 1973K. In his paper, both convection and radiation heat transfer processes were analyzed, and he discovered that using the coating the radiation effects could be blocked, suppressing the metal temperatures e reducing the cooler heat loss.

Three years later, Findlay (2005), published his research in which he developed a 3D computational methodology, aiming to analyze the conjugated heat transfer in refrigerated turbine blades along with its aerodynamic. Even nowadays, the cooler used to refrigerate the blades is redirected from the last compressor stage, which makes the flow inside the cooling channels to be much disorganized and turbulent presenting higher velocity and temperature gradients. By knowing that and using his methodology, Findlay (2005) was able to study the heat transfer along the blade and in its cross section. After the cooling process, when the air collected the heat of the blade and now flows combined with the discharge flow, the blade present lower temperatures. The error in his simulation, when compared to real projects, was 10%, which was a great percentage since some simplifications were done in order to proceed with the study.

Sadowski (2010) considered the interaction fluid-structure in order to have a more realistic result. Although, the problem of simulating both together generated many complex equations, which made he decided to do it separately. For the heat transfer condition, he created a blade, introduced many parameters of the exhaust gases such as density, thermal capacity, viscosity, and then, calculated the problem using the turbulence model k-ε. His results proved that when the blade was used with no coating but with cooling channels, the results were less effective than when both characteristics were applied together. For a blade with ceramic coating and cooling channels, the temperature reduction was 25%.

3. CALCULATION AND DESING PROCEDURE

The parameters for the turbine was chosen using multiple data collected on commercial gas turbines. The list of those parameters can be seen on table 1.

Table 1. Parameters for the turbine stage.

Parameter	Value
Turbine Inlet Temperature (average)	1723.15K
Cooling 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

When designing a gas turbine blade profile, the main purpose is to maximize the outlet specific work, and to do that both the blade velocity and the axial velocity should be maximized as well. In order to do that, designers are limited by the formation of thermal shocks, that need to be controlled to minimum, and by tensions that are developed due to centrifugal forces on the material of the blade, in which friction are generated when the axial velocity is elevated.

Multiple rules should be followed to design a turbine blade. First of all, the power generated by the turbine and the velocity achieved should match the ones required on the system. The energy available, which is calculated through the difference in pressure and temperature between the inlet and outlet, should be distributed among all the stages. The blade angles and the flow angles should follow the ones in the velocity triangles, and finally, the narrow area of the profile should be able to permit the mass flow to pass.

The blade can be design by using multiple methods, which will depend on the company and designer, in this paper, it was chosen the Surrogate Loss System method (Marcu, 2015). The Surrogate Loss System method is a model that provide a tractable substitute to expensive physical simulations, being an effective solution to design complex systems such as gas turbine blades. This model provide an abstraction of the real system behavior, and using standard error measurements allow companies and designers to produce models that can be used with security in industry.

The chosen algorithm works recalculating the efficiency based on a supposition of the losses on the real cycle. The efficiency increases as the algorithm iterates until it reaches a value close enough to the one stipulated at the beginning.

By using the enthalpy-entropy graph, it was possible to get most of the quantities of interest: such as isentropic enthalpy drop (Δh_{is}), isentropic velocity (C_o) and work (w_{is}), and the kinetic residual energy (k_r).

With those results in hand coupled with the reaction (R_c), the total to total efficiency (η_{TT}), and the flow coefficient (ϕ) the tangential velocity on the middle radio (U) was calculated, as in Eq. 1.

$$U = \sqrt{2\Delta h_{is}} \times \sqrt{\frac{1}{\phi^2 - (R_c - 1)\frac{4}{\eta_{TT}}}} \tag{1}$$

With the tangential velocity, the axial velocity (C_a) was calculated, as well as the ideal (w) and real work (W). A verification was done: if the required power was achieved, the calculation can be resumed, otherwise, it was necessary to change the values for the reaction or the flow coefficient, until the required power been achieved.

By using the velocity triangles all the kinematic parameters can be calculated, as shown in Fig. 1.

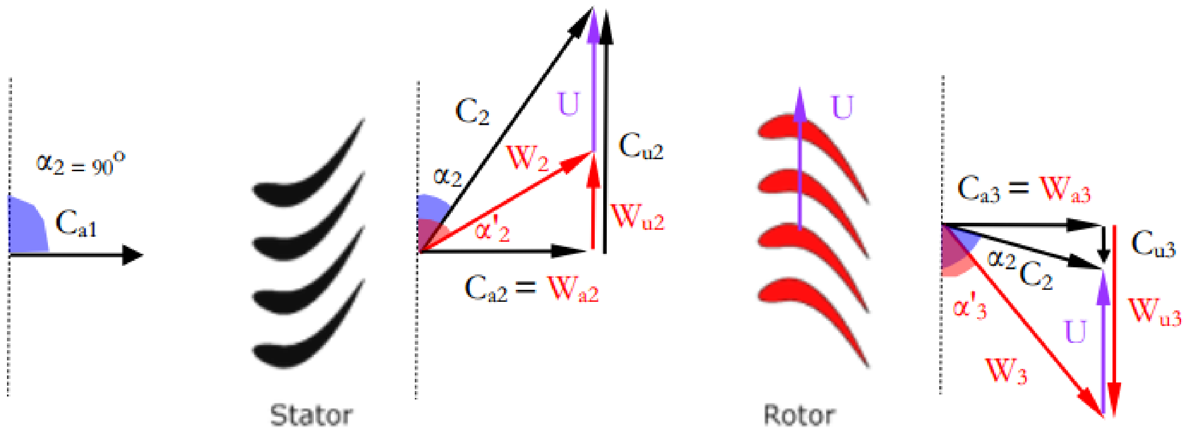

Figure 1. Velocity triangles for Stator and Rotor – Gas Turbines.
Source: Marcu, 2015.

To proceed with calculations it was necessary to estimate the thermodynamic parameters such as pressure, temperatures and density of the flow. The flow throughout the stator occurs at an enthalpy of constant stagnation and at the stagnation temperature, so that was conceivable to use the stagnation temperature ratios at the output of the stator. As the algorithm used was an iterative one, an initial value of temperature was used, and by doing iterations, the real one was discovered. With temperatures and the Mach number, the pressures was computed.

The thermodynamic and kinematic parameters allowed the designer to proceed and calculate the geometry of the profile, such as chord, angles and pitch, and then determine the number of blades that were required to generate the power needed.

The work fluid was considered to be air, since the mass flux of fuel was almost despicable when compared to the one of air.

In order to generate the blade, some specification, such as flow area and Mach number in the stator outlet, number of blades on the rotor, middle radium of the blade, clearance angle, outlet angle of the stator and axial chord of the blade was required. Using the mentioned values associated with the graphical interface created by Marcu (2015) on MatLab®, it was possible to generate the blade curves, which for this article can be seen on Fig. 2.

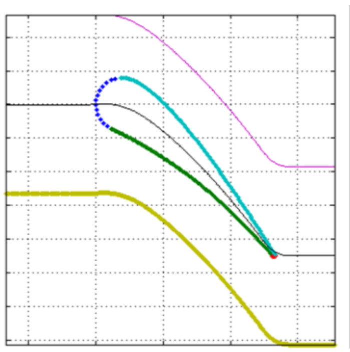

Figure 2. Blade profile generated on MatLab.

4. SIMULATION PROCEDURE

After the generation of the blade profile, the curves were used to generate the solid, and six configurations were designed, as shown in Fig. 3.

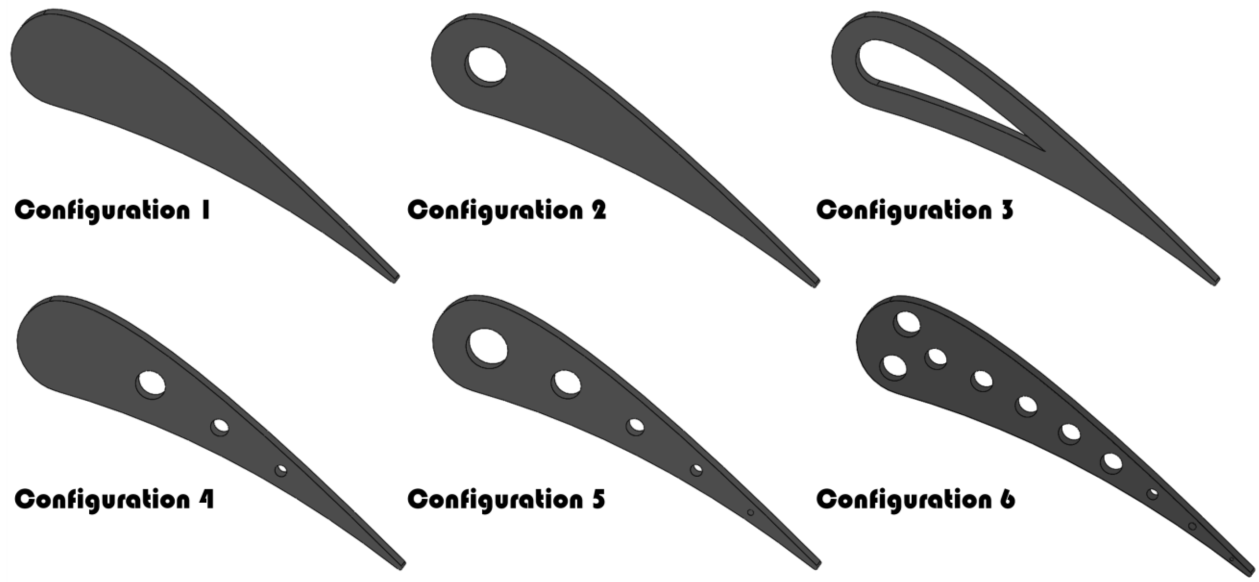

Figure 3. Configurations for the turbine blade.

To simulate the heat transfer on the blade, it was used the software Abaqus® CAE 6.13. The material chosen was Stainless Steel AISI 310, an austenitic stainless steel medium carbon content, with 25% chrome and 20% nickel, which is highly resistant to corrosion, oxidation, and thermal fatigue. A section with this material was applied to the whole geometry. Two surfaces were elaborated so that the boundary conditions could be applied to the model. One containing the external surface of the cross section, which received the combustion gases temperature and a convection film to simulate the external convection, and another one containing the cooling channels surfaces to apply the internal film convection and the cooling thermal flux.

The surface film condition is a requirement for convection studies. Because it can only be applied on 3D elements, the simulation was chosen to be 3D. The convective coefficient (h) and the bulk temperature (Tb) were the parameters needed to describe those conditions.

The mesh was then created using DCC3D8 elements, hexahedral 8-node bricks used only for convection and diffusion heat transfer analysis. The mesh refinement was set to be adaptive, which means that when the surface becomes thinner the mesh adapts to its surfaces automatically.

After the completion of all steps mentioned, the model was simulated and the results were as shown on the following chapter.

5. RESULTS

By applying the finite element method on Abaqus software, the cases were simulated, generating the results as follows.

In the first case, where no cooling channels were applied to the blade, only two boundary conditions were set to the model. One including the external temperature of the blade, considered to be the same as the turbine module inlet temperature, 1723.15K, and a convective film simulating the external convection, which coefficient was 3333.195W/m².K. The simulation was set to be steady, in such a manner that after one second the whole blade cross section was at equilibrium, as shown in Fig. 4.

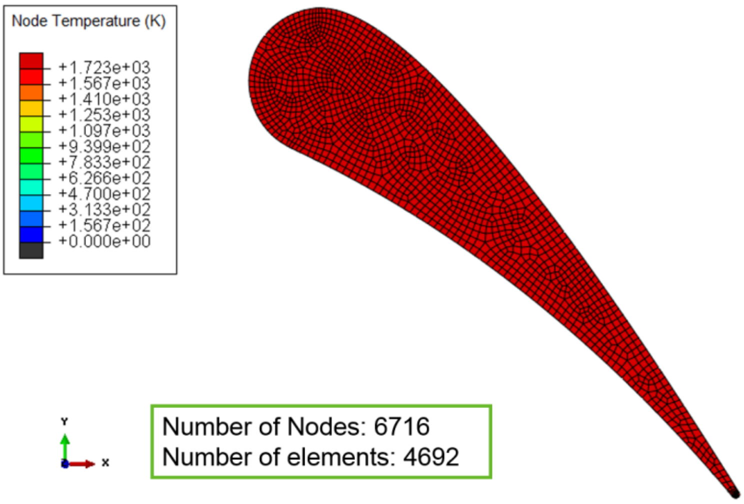

Figure 4. Result of the blade with no cooling channel.

After achieve equilibrium, the whole cross section was at the temperature of the inlet gases, which can highly harm the mechanical structure of the blade.

In the second case, it was created a cooling channel with the same shape of the blade with an offset of 20mm from the leading edge. In this case, beyond the boundary conditions already applied on the previous case, it was added two new boundary conditions, a cooling heat flux load of -2,39MW/m² and a film condition internally to the cooling channel, which parameters were a bulk temperature of 823.15K and a convective coefficient of 2659.442 W/m².K. In Fig. 5, the results of this case can be analyzed.

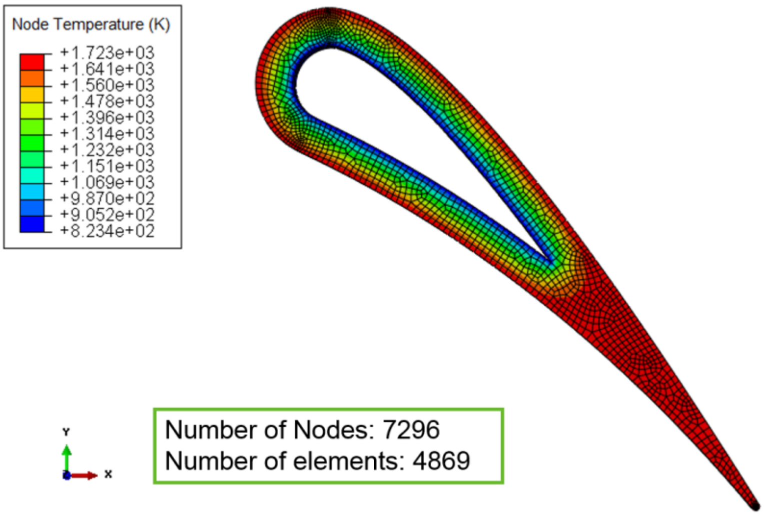

Figure 5. Result with a blade-shape cooling channel.

In the vicinity of the channels, it can be observed an efficient cooling process, whereas close to the leading edge there was no influence of the cooling channel, which can be a point of thermal tensions.

A third case was simulated, with the same conditions as the second one, differing only on the shape of the channel, and the results obtained were worse than the second one, as it is possible to see on Fig. 6.

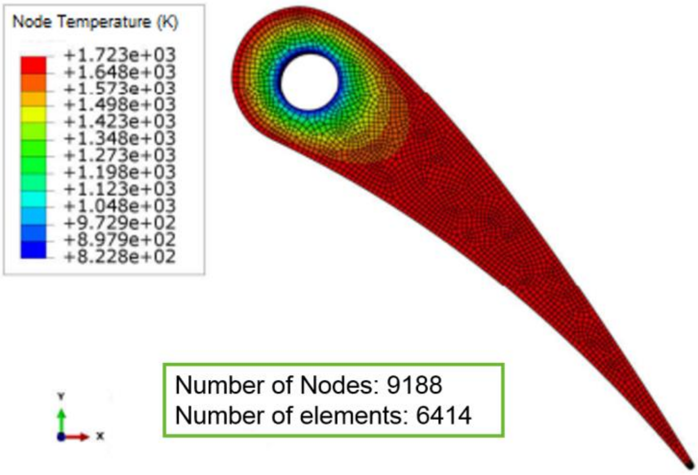

Figure 6. Result with a circular cooling channel.

It was decided to keep the circular format for the channels but instead of applying only one, to introduce multiple channels. In the first try of this new approach, it was employed three channels, centralized longitudinally on the cross section of the blade, measuring 25mm, 15mm and 10mm, as shown on Fig.7.

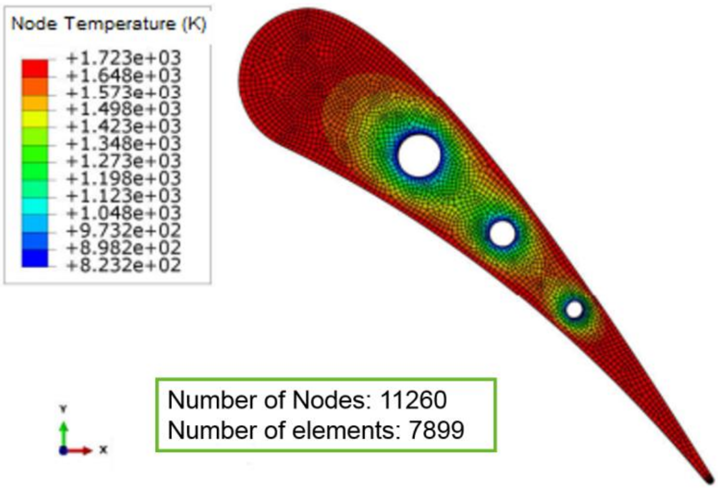

Figure 7. Result with three circular cooling channels.

For this case, the cooling occurred uniformly and expansively along the entire cross section, however the results showed that only three centralized channels are not sufficient to cool the whole blade, since the profile was cooled only in the region close to the channels, maintaining at high temperatures even in the most critical region, the leading edge.

A fifth case was then studied, now introducing cooling channels on the critical regions, the leading and trailing edges.

In this configuration, the channels had diameters of 35mm, 25mm, 15mm, 10mm and 5mm respectively. The result for this case is shown in Fig. 8.

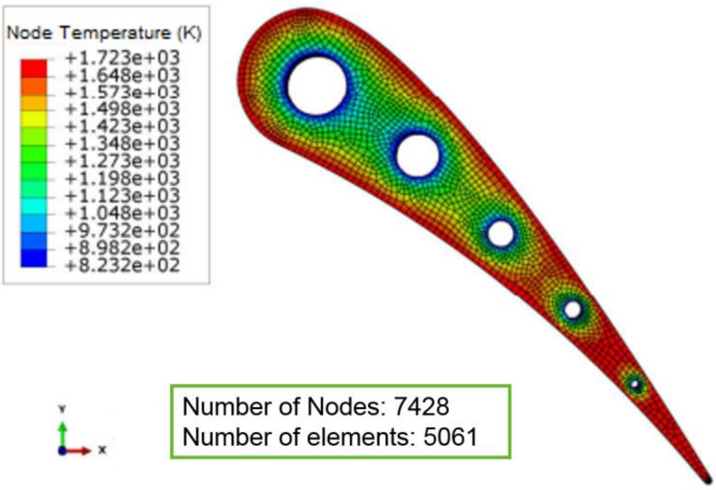

Figure 8. Result with five circular cooling channels.

For this distribution of the cooling channels, much of the cross section of the blade was at a temperature of 1348K, which makes the blade to work more safely.

A last case was simulated, now, including five more channels and distributing them all along the cross section, as shown in Fig. 9.

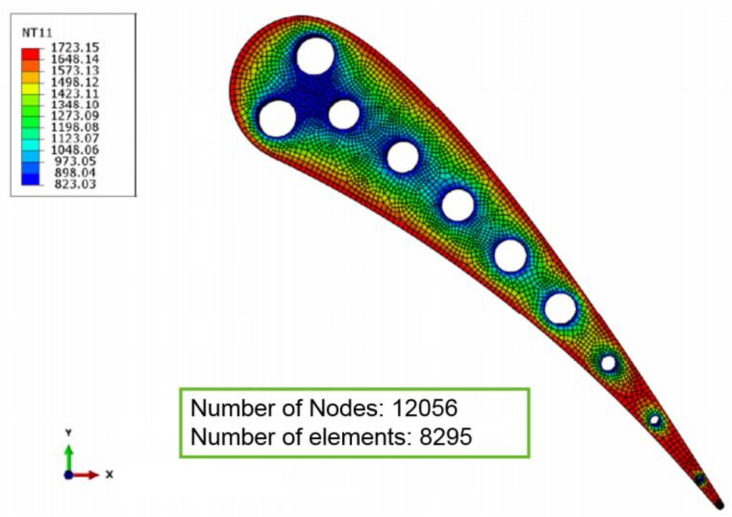

Figure 9. Result with ten circular cooling channels.

It is plausible that in this last configuration, the blade was better cooled, which will significantly increase the efficiency of the turbine. For this configuration, about 90% of the cross-sectional area of the blade exhibited temperatures lower or equal to 1350K, about 370K less than the inlet gas temperature in the turbine module, which represents a cooling efficiency of about 22%, for a reduction in the structural mass of about 13.7%.

Since blades are mainly produced by the die-casting process and then tubes for the passage of the cooling air are included on its structure, increasing the number of internal tubes for cooling contributes to the maintenance of the structural integrity of the blade and also the heat transfer process in it.

6. CONCLUSION

By using the Surrogate Loss System method and the finite element method, the proposed model was successfully generated and studied, within the pre-selected parameters.

It was simulated six configurations varying the number, position and size of the cooling channels and considering the convection process performed by the drained air flow of the last stage of compression at 823,15K, the method of cooling, it was possible to conclude that the higher the number of channels and the better distributed along the cross section, the better the results obtained with respect to the cooling of the blade.

For the configurations studied, the best result was the one obtained in the case where ten channels of different diameters distributed along the cross section of the blade were applied, in which there was a cooling efficiency of 22%, considering the higher and lower temperatures the blade reached post-simulation, in which the three processes of heat transfer occurred simultaneously, internal and external convection and conduction.

Even though, this paper achieved his main purpose, which was to study via finite element method the heat transfer process throughout the blade when cooling channels was 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.

7. ACKOWLEDGEMENTS

The first author would 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.

8. 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.

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.

9. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.