NUMERICAL CHARACTERIZATION OF A JET IMPINGEMENT COOLING SYSTEM

Kacar E.N.[1]* and Erbay L.B.[2]
*Author for correspondence
[1]PhD Candidate, Department of Mechanical Engineering,
Eskisehir Osmangazi University, Turkey
E-mail: nadirkacar@gmail.com
[2]Professor, Department of Mechanical Engineering,
Eskisehir Osmangazi University, Turkey
E-mail: lberbay@ogu.edu.tr

ABSTRACT

In this study the parameters of jet impingement cooling system were determined and possible effects were observed. Single row of jet impinging to a flat target surface in a nozzle guide vane of a gas turbine engine were simulated due to the operating conditions. In the scope of geometry; metal which was in contact with the air at 1500 K and 800 kPa was cooled via the air at 500 K and 400 kPa. For specified problem; channel height, distance between jets and Reynolds number were chosen as parameters affecting cooling performance. Three different channel height value as 2D, 3D, 4D; three different distance between jets value as 4D, 5D, 6D and three different Re number value as 4000, 5000, 10000 were used for the solutions. As a result of 27 different combinations best cooling configuration was determined as 2D(channel height), 4D (jet spacing) with Re number as 10000.

NOMENCLATURE

CFD		Computational Fluid Dynamics
D	[m]	Jet Diameter
HTC	[W/m^{3}K]	Heat Transfer Coefficient
k		Turbulence Kinetic Energy
Ma		Mach Number
NGV		Nozzle Guide Vane
Nu		Nusselt Number
RANS		Reynolds Averaged Navier-Stokes
Re		Reynolds Number
x	[m]	Distance on x axis, between jet plane and target surface
z	[m]	Distance on z axis, between jets
y+		Dimensionless wall thickness
ε		Turbulence kinetic energy dissipation rate
ω		The specific rate of dissipation

INTRODUCTION

The turbine is a part of gas turbine engine which generates power from flowing hot air stream. So, turbine section is the most affected part from temperature level of combustor gases. Because of the increasing technology level and energy need of the World, combustion temperature at gas turbine engines is gradually increasing. On the other hand higher gas temperature means higher cooling requirement, since gas temperature is high enough to weaken, even melt, any superalloy material of the turbine. Advanced cooling techniques are used to satisfy extreme cooling demand. The cooling is mostly applied as internal cooling using flow manipulators like; ribs, pin fins, dimples. Moreover, there exist techniques like; film cooling, impingement cooling depending on high speed fluid release.

Impingement cooling is the most aggressive internal cooling technique used in gas turbines [4]. The method depends on impinging fluid towards hot surface from an internal cooling channel and supply cooling. Gas turbines use the air supplied from compressor stages as coolant. Theoretically impingement cooling system has complicated flow characteristics and specifications. In literature, wide variety of analyses and applications has been processed over years. Studies generally concentrated on single jets, one row of jets or staggered jets of multiple rows. Gardon and Corbonpue (1962) measured radial heat transfer coefficient gradient for a single jet [5]. According to the measurements they pointed out that the maximum heat transfer coefficient was at the stagnation point and the values were decreasing when the distance from stagnation point increases. For gas turbines, in addition to the jet position, the shape of target surface is also considered whether being flat plate or concave surface. Goodro et al. executed flat plate experiments which had verified the effect of hole spacing on staggered jet array impingement heat transfer. They investigated Nusselt number distribution using two different jet to jet distance plate at various Reynolds number and Mach number conditions. Their operation range differed from 8200 to 30500 for Re and 0.1 to 0.2 for Ma. Their results indicated that Nu values were observed at maximum value at the projection of jet center and decreased towards outer radius. Furthermore, secondary peaks existed at the intersection of jet zones [3]. Similarly, another researcher analyzed flat plate also but with a single row of jet. He prepared both CFD model and carried out the experiments for the same model. He used two Re level about 18000 and 45000 with two jet-to-jet distance and three jet to target surface distance. He also observed the effect of cross flow because of using totally confined channel with only jet inlets and one outlet. Along the channel, the area weighted heat transfer coefficient on the target surface was increasing but maximum value was decreasing [8]. On the other hand, concave surface simulations were used to simulate leading edge of turbine blade. Elebiary et al. carried out concave surface analyses and experiments. They used one row of impinging jets with changing quantities and differentiating flow direction. They both calculated and measured Nu due to changing Re. As a result of comparison they observed error factor in the range of 0.3% to 24.5% between analyses and experiments. Moreover, they indicated the effect of cross flow in the channel which was the dislocation of projection of jet center and high Nu area [2]. In another numerical study, the results were obtained due to changing Ma, jet diameter and distance to target surface for a single row of jet. They obtained maximum Nu values at maximum experimental subsonic speed (Ma=0.7) and jet diameter. Furthermore; due to the cross flow along the channel, maximum Nu that's been observed at the jet center became invisible [6].

In the present work, CFD studies are carried out to observe the effects of the parameters of impingement cooling system of a gas turbine engine. A single row of jets with flat target surface is used to simulate cooling at the suction side of each vane of NGV (Nozzle Guide Vane). The surface of NGV is in contact with the main flow at 1500 K and 800 kPa, which is cooled via the air from compressor at 500 K and 400 kPa. 27 combinations are prepared with three jet to target distance value (2D, 3D, 4D); three jet to jet distance value (4D, 5D, 6D) and three Re value (4000, 5000, 10000). As a result of calculations the best cooling configuration is determined as 2D (jet-to-target), 4D (jet-to-jet) and 10000 (Re).

NUMERICAL METHOD

Jet impingement cooling model is analyzed with CFD using Fluent. For Fluent analyses, before parametric study, it is important to determine the turbulence model and to validate the method of solution. Zuckerman and Lior stated although it was believed that k-ε model did not supply exact values for the jet impingement problem, it was frequently used due to being easily applicable and comparable with other models [10]. For the study, realizable k-ε turbulence model which solves two transport equations is used. This model is preferred because of using a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate with respect to k-ε turbulence model [9]. The term realizable meant that the model had satisfied certain mathematical constraints on the Reynolds stresses and it had been consistent with the physics of turbulent flows [1]. Furthermore, realizable k-ε model with "low" computational cost and "fair" heat transfer coefficient calculation capability with 15% error, seemed reasonable for analyses [10].

Validation

Ricklick [8] used a prismatic channel with flat target plate for experimental studies. Channel had one outlet at the end and inlets at the opposite surface of the target one. All other walls except the outlet were closed and heated with 10000 W/m2K flux. So, colder air jets was hitting hotter surface and total air was flowing through channel to outlet. Coolant air was at 300 K with flow rate of 0,00247 kg/s for each jet, so that average Re was 43000. Static inlet pressure for the jets were 111,7 kPa. Geometric specifications for the experimental setup were such that: the distance between jets were five times diameter (5D), channel width was 4D and channel height was 5D. The results of numerical calculations are compared by the heat transfer coefficient values at the symmetry plane of the target surface (Figure 1) for validation. Although the htc distribution is calculated as in the experimental one, through the channel the positions of jet centers are displaced, this can be a result of miscalculation of cross flow. This validation indicates that the analysis estimates htc values with an error of 10%-23%.

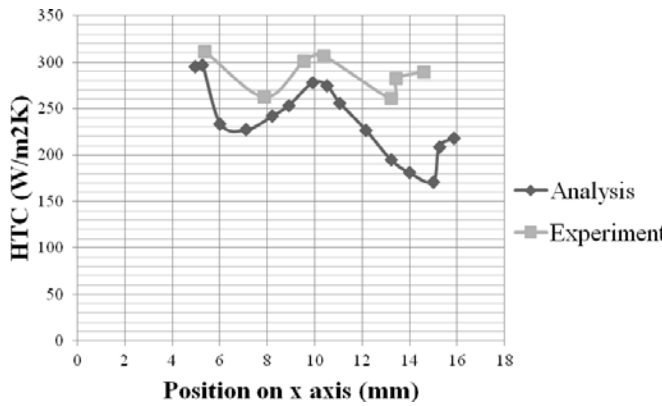

Figure 1 Heat transfer coefficient comparison (analysis-experimental) at the target surface

Parametric Model Specifications

The cooling model is a simulation of the impingement cooling system of an NGV of middle class gas turbine engine. This NGV is the first stage vane of engine and cooling air is supplied from the impingement insert to cooling channel of vane (Figure 2).

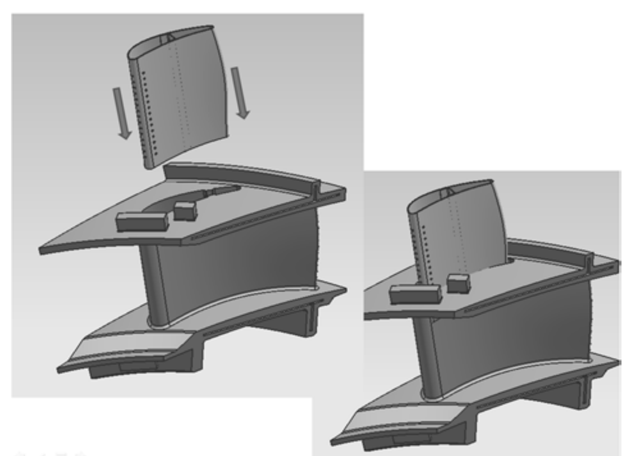

Figure 2 Impingement insert and vane

Air flow direction and final orientation can be seen in Figure 3 and 4. The chosen jet row which is at the suction side of vane close to the trailing edge is indicated in Figure 3 also. In Figure 4, it can be seen that cooling air fills the insert and then discharges through the space between vane wall and insert.

The cooling air is heated in this space before evacuated from trailing edge discharge holes.

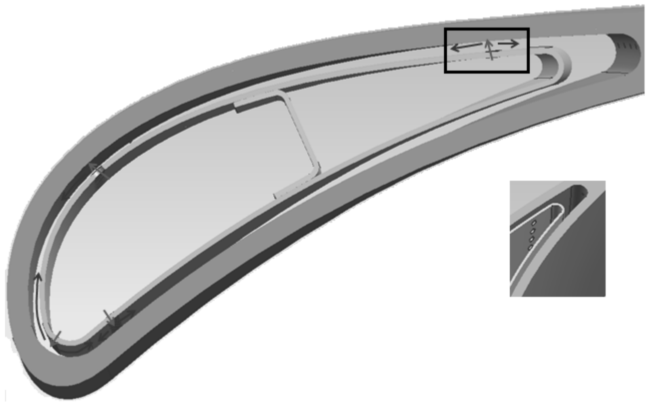

Figure 3 Simulated jet row in the vane

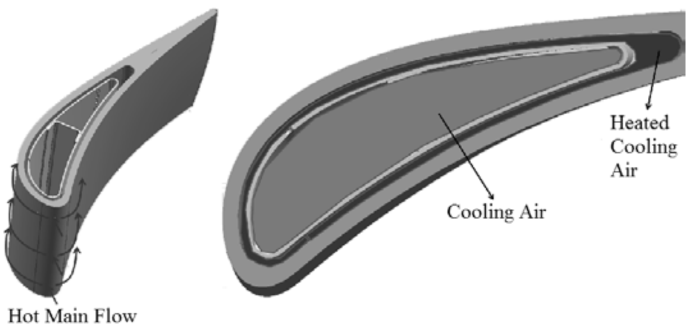

Figure 4 Flow scheme of the vane

For the indicated jet row; the target surface is assumed as flat plate and the channel height is the distance between vane wall and impingement insert. Side walls of the channel are open yet the ends are closed. Important boundary conditions for the problem are specified below:

- Metal temperature of outer surface of vane : 1250 K
- Metal temperature at the hub and tip of vane: 750 K
- Coolant air temperature: 500 K
- Air flow rate for a row of cooling jets: 0,693 g/s
- Jet numbers at one row: 7

Moreover the width of channel is simulated as 10D to prevent reverse flow towards the flow domain and jet inlets are defined at 1D distance from channel for developed flow. The meshed geometric model and CFD model denomination can be seen in Figure 5 and 6 respectively.

Enhanced wall functions are used for the calculations and because of that boundary layer mesh is generated. Analyses are performed with y+ values around 1, since viscous layer is one cell thickness at this value [1]. Additionally, for the industrial applications with Re number around 10000 and more, boundary layer thickness must be 1% thickness of each jet [7].

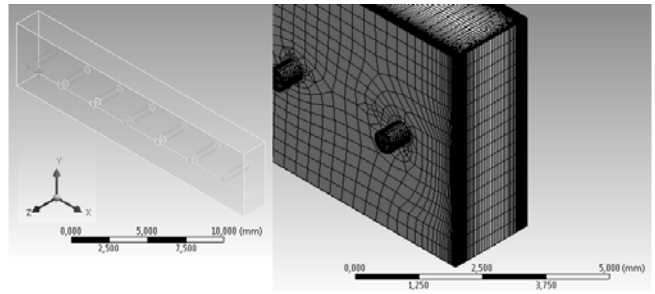

Figure 5 Meshed geometric model

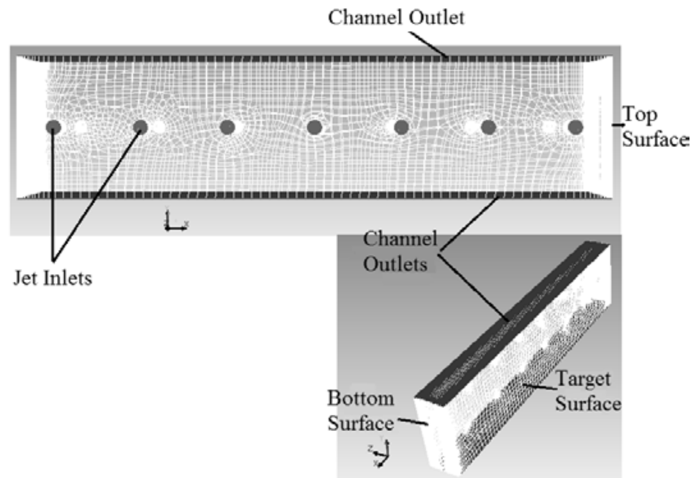

Figure 6 CFD Model Denomination

For the parametric study jet to jet distance, jet to target surface distance and Re number values are changed. Jet to jet distance values are differentiating as 4D, 5D and 6D; target surface distance has values of 2D, 3D and 4D; and finally Re values are 4000, 5000 and 10000. The values of Re are relatively low, since it is important to limit the Ma around 0,4 for uncompressible fluid conditions. Using the combinations of three different values of each parameter 27 analyses were performed.

RESULTS OF PARAMETRIC STUDY

The results of analyses can be investigated by the velocity contour at the symmetry plane of channel (Figure 7). According to Figure 7, important jet characteristics can be observed. Each jet hit the target and stagnation point exists. Moreover wall jets collide and create fountains. When the channel height decreases the fountains become bigger and solid. As a result of this, bigger vortexes are divided into smaller ones. Moreover, fountains are affected due to Re increase. When Re increases the fountains become bigger and average velocity increases too. In addition to wall jets moving between impinging jets, there exist free ones at the ends of channel which increase the htc values. When the distance between jets decreases, these free wall jets weaken and htc values at the channel ends decreases. Parameter variance changes velocity distribution of the air resulting in the change of heat transfer coefficient.

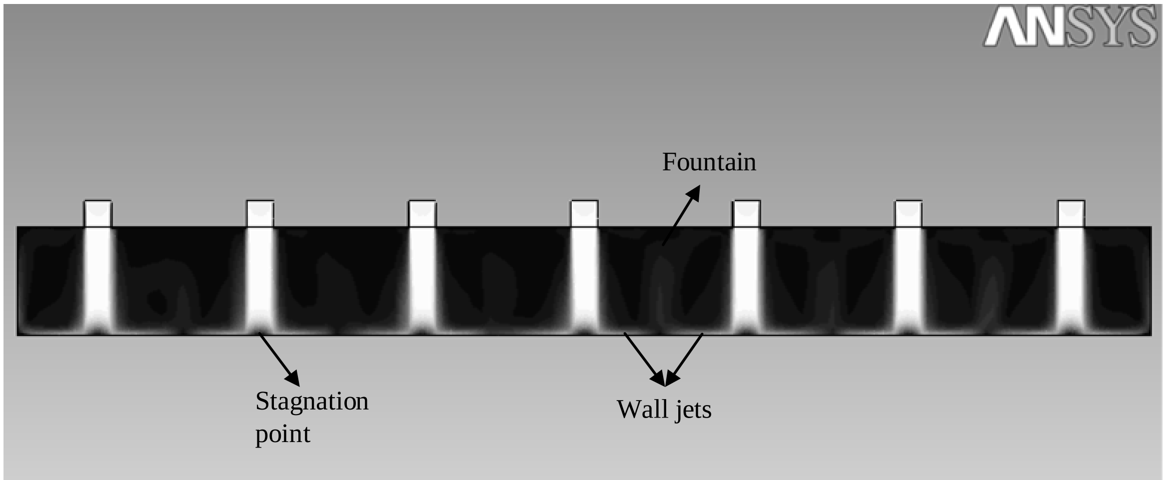

Figure 7 Velocity distribution at the channel symmetry plane

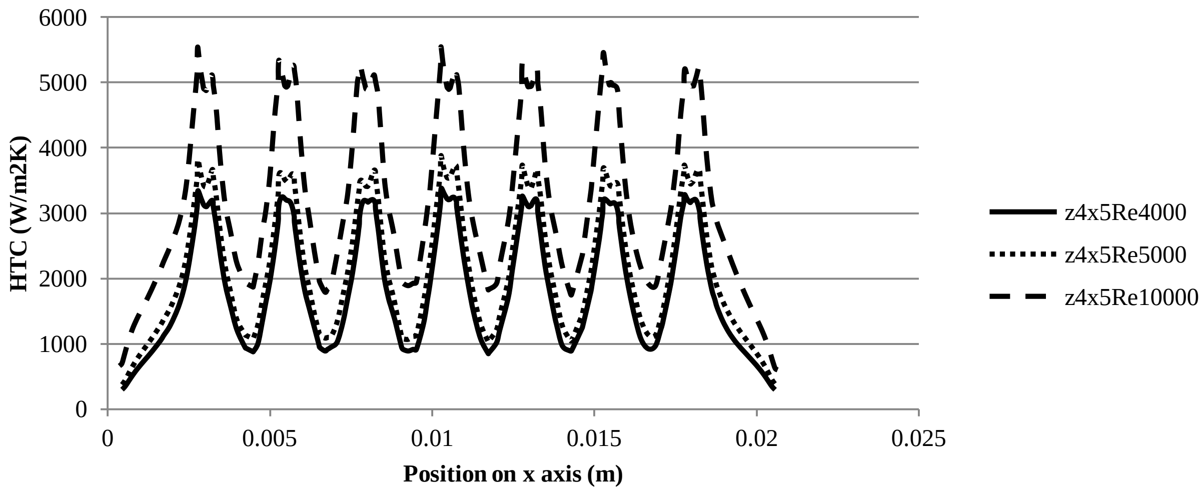

Figure 8 HTC values for different Re number at z=4D and x=5D

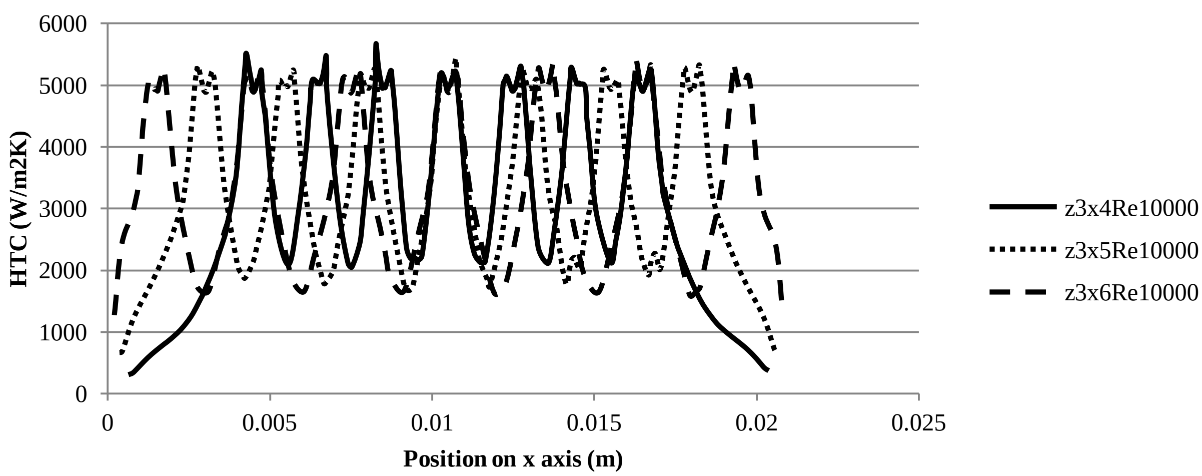

Figure 9 HTC values for different jet to jet distance at z=3D and Re=10000

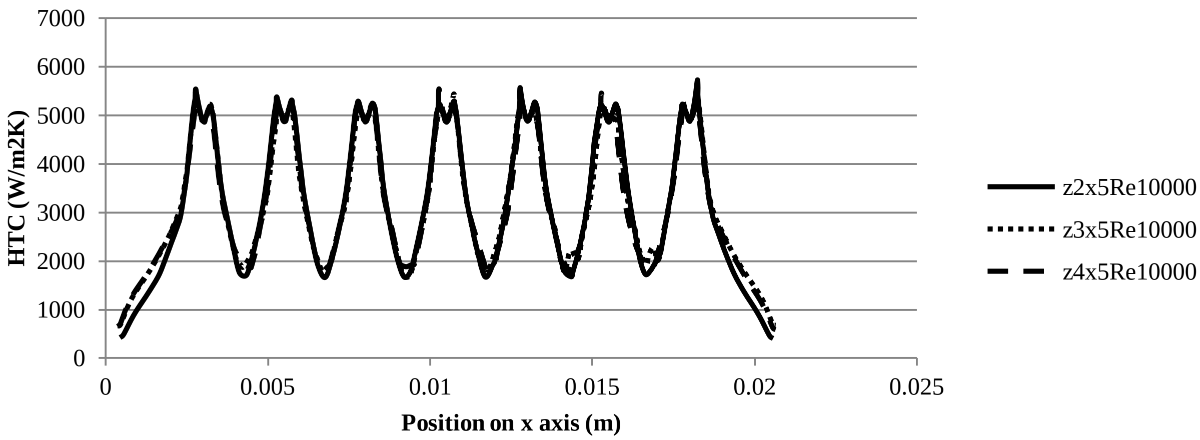

Figure 10 HTC values for different channel height at x=5D and Re=10000

Despite the 3D analyses, results of calculations are explained at symmetry line of channel along y axis (width). Because the side walls of channel are open, each jet hits the target surface with a center projection of just on the symmetry line. In other words jets do not displace along y axis. Firstly; in Figure 8, htc distribution can be seen due to several Re values when the jet to jet distance and channel height values are constant. If Re increases, htc values increases proportionally. HTC values are at maximum at the jet center (stagnation point). This maximum values are about 2,5-3,5 times bigger than the values of areas between jets. Furthermore, average values at Re=10000 are about 1,5 times higher than the values at Re=4000 and Re=5000. Secondly; htc distribution according to the different values of jet to jet distance can be seen in Figure 9. The length of channel is used constant during the analyses of variable jet to jet distance. When this distance decreases jets unite each other around the center of channel. Even if the positions of jet centers alter, change of htc distribution for each jet is negligible. Besides higher htc values are observed at lower peaks since collision of wall jets creates highly circulating zones.

Lastly, the effect of channel height on the htc distribution is observed in Figure 10. Similarly; the effects of channel height change on the maximum htc values are negligible, like the alteration of jet to jet distance. But as it can be seen; because of the solid fountains created at low channel height, the htc values at the zones between jets increase proportionally.

CONCLUSION

The scope of study is focused on impingement cooling system of a gas turbine engine. CFD analyses are carried out to find out the most effective configuration of three different parameters. For the validation of analyses, previous experimental data in the literature was analyzed and the results were compared. Due to the tolerable error values of analyses, the parametric study is performed. According to the parametric study, it is observed that htc values increases with increasing Re number. On the contrary, htc values decreases with increasing channel height and jet to jet distance values. The present study shows clearly that the effect of change of Reynolds number is dominant on other parameters. To conclude; maximum htc value is calculated as 5718 W/m2K for z=2D, x=4D and Re=10000 conditions which is the best cooling configuration.

REFERENCES

[1] Ansys Inc., 2010, *Ansys Fluent Theory Guide*, 720 p.

[2] Elebiary, K. and Taslim, M. E., 2011, *Experimental/Numerical crossover jet impingement in an airfoil leading-edge cooling channel*, ASME GT2011-46004

[3] Goodro, M., Park, J., Ligrani, P., Fox, M. and Moon, H., 2007, *Effect of hole spacing on jet array impingement heat transfer*, ASME GT2007-28292

[4] Han, J.C. and Wright, L.M., 2006, *Enhanced internal cooling of turbine blades and vanes*, The gas turbine handbook, 4.2.2.2, 321-346 p.

[5] Jambunathan, K., Lai, E., Moss, M.A. and Button, B.L., 1992, *A review of heat transfer data for single circular jet impingement*, International journal of heat and fluid flow, 13, 106-115 p.

[6] Liu, Z., Feng, Z. and Song, L., 2010, *Numerical study of flow and heat transfer of impingement cooling on model of turbine blade leading edge*, ASME GT2010-23711

[7] Martin, H., 1977, *Heat and mass transfer between impinging gas jets and solid surfaces*, Journal of material processing technology, 136, 1-3, 1-60 p.

[8] Lee Ricklick, M.A., 2006, *Characterization of an inline row impingement channel for turbine blade cooling applications*, PhD Thesis, University of Central Florida, 166 p.

[9] Versteeg, H.K. and Malalasekera, W., 2007, *An introduction to computational fluid dynamics-the finite volume method*, Pearson prectice hall, 503 p.

[10] Zuckerman, N. and Lior, N., 2006, *Jet impingement heat transfer: Physics, Correlations and numerical modeling*, Advances in heat transfer, 39, 565-631 p.