

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 2, February 2018

## Numerical and Experimental Modeling of Impingement Cooling: A Literature Review

Shantanu Vedpathak<sup>1</sup>, Dr. Ankur Saxena<sup>2</sup>, Dr. Kiran C. More<sup>3</sup>

P.G. Student, Department of Mechanical Engineering, D Y Patil School of Engineering Academy, Pune,  
Maharashtra, India<sup>1</sup>

Assistant Professor, Department of Mechanical Engineering, D Y Patil School of Engineering Academy, Pune,  
Maharashtra, India<sup>2</sup>

PG Coordinator, D.Y. Patil Institute of Technology, Ambi, Pune, Maharashtra, India<sup>3</sup>

**ABSTRACT:** Impingement cooling method utilizes high speed jet(s) to impinge on a high temperature surfaces for cooling. The same method could be used to for heating as well. In this journal, the research work in the impingement cooling had been discussed in detail. Computational and experimental methods had been widely used for studying the impingement cooling mechanisms. The standards for conducting experimental and numerical simulations had been identified from these literatures. Also, the factors that could impact the impingement cooling effectiveness such as jet-exit-to-impinging-surface-distance, number of jets, jet inclination angle, jet speed (characterized by the Reynolds Number) etc. had been studied from these literatures. Also, the settings for the CFD simulations such as boundary conditions, turbulence models etc. had been identified.

**KEYWORDS:** Impingement Cooling, CFD, High Speed Jets, Heat Removal Mechanisms

### I.INTRODUCTION

There have been various methods such as cross flow heat transfer, parallel flow heat transfer, natural convection heat transfer etc. for maintaining a heat sink such as electronic components at a desired temperature. The impingement cooling is one of such method for effective heat removal. Impingement cooling method utilizes high speed jet(s) to impinge on a high temperature surfaces for cooling. The same method could be used to for heating as well. It is important to understand the fluid flow pattern to design an optimal impingement cooling mechanism. Also, various factors that have strong influence on impingement cooling effectiveness such as jet-exit-to-impinging-surface-distance, number of jets, jet inclination angle, jet speed (characterized by the Reynolds Number) etc. must also be understood. So, prior to design an impingement cooling mechanism a detailed literature review of the existing research works must be performed.

### II.LITERATURE REVIEW

Ridvan Yakut [1] had investigated single jet impingement cooling on a heat sink with hexagonal shaped fins for various operating conditions. They had conducted experimental and numerical simulations for estimating heat transfer rate in terms of Reynolds number against the Nusselt number graph. For the CFD simulations, the authors had applied Realizable k- $\epsilon$  turbulence model in ANSYS FLUENT and they had observed good agreement between experimental and numerical results. They had obtained high heat transfer rate for  $Y/d = 1$  (impingement distance to the nozzle diameter ratio) among all configurations.

Based on a detailed literature review by M Behnia [2], the authors suggested that the heat transfer rate in the impingement region was sensitive to the fluid jet's turbulence intensity as well as the mean velocity profile. The authors had suggested to apply appropriate RANS modeling approach for accurate prediction of results from CFD

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 2, February 2018

simulations. They had performed with various turbulence models such as k- $\epsilon$  and V2F models along with different wall treatments. They had concluded that the V2F turbulence model provided better accuracy for entire operating range (Reynolds Number variations, Heat Sink Inclinations, Impingement Height Variations) of the experimental studies.

Most of the research on impingement cooling was performed with either Air or Water as fluid. However, Santosh Kumar Nayak [3] had conducted experimental studies to analyze the impact of dual phase (water + air) fluid jet as impingement cooling on a heat sink. They had varied the air and water injection pressure and thereby mass flow rate of the respective fluid in their study. They had reported a maximum cooling rate of 305.1 °C/s which were far higher than obtained from the single-phase jet impingement cooling methods.

Md Lokman Hosain [4] had applied impingement cooling method for Runout Table (ROT) cooling with water – below the boiling point temperature – as fluid. For CFD simulations, the author had explored 2-D axisymmetric as well 3D modeling approach with RANS turbulence models. The authors had conducted this study with single and two jets for identical operating conditions. For modeling the air (atmospheric fluid) and the water (jet fluid) interfaces in the CFD simulations, the Volume of Fluid (VoF) was chosen by the authors. Their CFD simulation results were compared with the analytical/empirical correlations and found to be acceptable.

The impingement/effusion cooling holes, arranged in a staggered pattern, on two parallel plates was studied by Zhang Jingzhou [5] with a focus on gas turbine cooling. With the holes are at 30° inclined to the surface, the author had varied the blowing ratio and the ratio of the center-to-center distance of adjacent holes to hole diameter. Based on the results obtained, they had observed that the decreasing the center-to-center distance of holes had improved wall cooling effectiveness. This could be applied to multi-jet impingement cooling.

There have been numerous research on heat removal using cross-flow (flow along the heat sink axial direction) as well as impingement cooling. Hussein M Maghrabie [6] had conducted CFD simulations for Jet Impingement in Cross Flow (JICF) using RNG k- $\epsilon$  turbulence model. This was cooling method was applied to a flat plate with rectangular shaped blocks attached. With this method, the authors had identified various vortex such as Lower and Upper Horseshoe Vortex (LHV & UHV), Side and Top Vortex (SV & TV) as well as Vortex Ring (VR). These vortices help in achieving higher heat transfer rate.

The application of jet impingement for turbine blade cooling was studied by Abdulla R Al Ali [7] using CFD simulations. In this case, the geometry was of the concave shaped surface rather than the flat plate as had been studied by various authors. The air injection velocity was varied from 5 – 65 m/s by the authors. They had conducted steady state 2-D simulations using ANSYS FLUENT with the k- $\epsilon$  turbulence model. Based on the temperature contour plots from the simulations, they concluded that the vortex created by the jet impingement on the turbine blade surfaces suppress the flow and thermal boundary layers due to the mainstream flow and hence helped in maintaining lower surface temperature.

An experimental investigation of the single submerged impinging jet was conducted by Carlo Carcasci [8]. For the heating elements, the authors had used two specimen – a smooth Inconel sheet and Stainless Steel Mesh. Their objective of the research was to obtain uniform heat flux condition on the specimen surface. By varying the impingement jet mass flow rate, they had conducted the experiments corresponding to Reynolds number of 10,000 to 40,000. From the results, they had concluded that the surface roughness played critical impact on the heat transfer rate as the Stainless Steel Mesh specimen had high surface Nusselt number as compared to the Inconel specimen.

Most of the available literatures on impingement cooling were performed for fluid jets that are normal to the impinging surface. Muhammad A R Sharif [9] had conducted numerical simulations using ANSYS FLUENT for varying jet inclination angle (90°, 80°, 75°, 70°, 60° and 45°) along with the variations on Jet exit Reynolds number, the normal distance between the jet exit and impingement surface as well as the distance between the jets. The results showed that for the jet inclination angle of 45° the heat transfer rate was more uniform. While for the jet inclination angle of 90°, the peak heat transfer rate at the stagnation zone was higher but lower heat transfer rates were observed in the remaining regions.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijrset.com](http://www.ijrset.com)

Vol. 7, Issue 2, February 2018

Siddique Mohd Umair [10] had performed numerical simulations using ANSYS CFX for studying the heat transfer enhancement from a pin-fin heat sink with an impinging air jet. In their study, the authors had varied the air jet turbulence intensity and found that it had a minimal impact on the surface heat transfer coefficient. They had conducted the simulations with three turbulence models – standard  $k-\epsilon$ , standard  $k-\omega$  and SST  $k-\omega$  – and concluded that the  $k-\omega$  turbulence model predictions were in good agreement with the experimental data.

Ajoy Debbarma [11] had investigated the rewetting of vertical oriented nuclear fuel rods with the help of impinging jets using CFD simulations in ANSYS CFX. The transient simulation was conducted by the authors with a time step of 1 second. They had maintained wall  $Y^+$  of  $\sim 10$  in their simulation to resolve the near wall boundary layer effects. G Srivalli [12] had investigated low-speed air jets – laminar flow regime  $Re = 100$  to  $500$  – using CFD simulations. Their model contained 5 jets impinging on a heated flat surface. When the jet exit to the impingement surface was varied, the authors had observed that heat transfer rate was affected.

N K Chougule [13] had conducted experimental and numerical studies on multi-air jet impingement on a heated flat plate for the flow conditions corresponding to  $Re = 7,000$ ,  $9,000$  and  $11,000$ . In their study, the model contained a total of 9 air jets arranged in  $3 \times 3$  positions. They had applied constant heat flux at the bottom of the heat sink plate while the side surfaces were maintained adiabatic. For performing the CFD simulations, the authors had generated block-structured meshes using ANSYS ICEM. They had concluded that the increase in the distance between jet exit plane to the impinging surface reduces the heat transfer rate. Also, they had observed that the lowest temperature zone had occurred at the stagnation zones for all configurations. This would lead to un-even / non-uniform heat transfer from the heat sink.

With the help of Instantaneous Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV), Juliana K Abrantes [14] had studied the swirling impinging jet over a heated flat surface. In order to generate the swirling flow, the air from the blower was made to pass through the swirl chamber – constructed with its top end closed and bottom end open and the cylindrical walls had 25 holes circumferentially. As the air passed through these holes tangentially, the required flow swirl was obtained. The authors had observed that the swirling jet produces more uniform local heat transfer coefficient on the impinging surface as compared to the non-swirl jets.

Jimmy Wayne Crafton [15] had conducted experimental studies to analyze the impact of the inclination angle of the flat plate over the jet impingement cooling. They had varied the inclination angle  $9^\circ$ ,  $15^\circ$ ,  $18^\circ$ ,  $30^\circ$  and  $45^\circ$ . In this study, two flow jet speed was considered – subsonic and supersonic. They had suggested that a system of shock and expansion wave (for supersonic jet condition) interaction with the impinging surface could potentially alter the pressure distribution.

The influence of curvature over the jet impingement cooling was studied using CFD simulations by A M Tahsini [16]. They had conducted the studies for three geometrical variants – flat plate, convex and concave impinging surface. The radius of curvature for convex and concave impinging surface was kept identical. The studies were extended for three different curvature ratios – 1.5, 2.0 and 2.5. For the identical operating conditions, the convex surface had the highest Nusselt number at the stagnation point followed by the flat plate and concave surface. For both convex and concave surfaces, the Nusselt number reduces significantly in the lateral direction as compared to the stagnation point. This was more severe in convex surfaces.

N Zuckerman [17] had conducted a detailed literature review on impingement cooling methods and the factors that could affect the heat transfer rate from this method. In this report, the author suggests the appropriate jet-exit-to-impinging-surface ratio based on the desired flow field. Also, in the summary of the turbulence models, the author recommends a  $v_2$ -f model for accurate prediction of impingement heat transfer coefficients in the CFD simulations.

The application of impinging jet on a heated flat plate with and without a porous layer was studied by Marcelo J S de Lemos [18]. In this research work, the porous medium permeability, the height of the porous medium along with the ratio of thermal conductivity were varied by the authors to investigate the impact on the impingement cooling. For the CFD simulations, they had employed a SIMPLE algorithm for the pressure-velocity coupling that arises for the

## International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 2, February 2018

incompressible flows. Based on their results, they had concluded that the high porosity limited the stagnation Nusselt number. The height of the porous medium was found to be affecting the local Nusselt number distribution. Overall, the configuration without the porous medium had high heat transfer rate as compared to the configurations with the presence of a porous medium.

In a study of impingement heat transfer on a heated flat surface, Jun-Bo Huang [19] had employed water with Al<sub>2</sub>O<sub>3</sub> nanoparticles as the impinging fluid. Their studies were conducted using CFD simulations in ANSYS FLUENT with a 2-Dimensional axisymmetric model. They had varied the nano-fluid particle volume fractions 0% (pure water), 1%, 3% and 5%. In order to model the combination of water and Al<sub>2</sub>O<sub>3</sub>, the mixture multiphase model was employed by the authors. With the addition of the high thermal conductivity nano-fluids, the overall thermal conductivity would increase and hence the high heat transfer rate. This trend was predicted by the authors in their study.

In their research work, Malladi R Ch Sastry [20] had predicted the pressure distribution on the plate under the impingement jet cooling. Their model had three slot-shaped jets with air as the fluid. The authors had conducted the studies for the flow conditions corresponding to  $Re = 5282$  to  $8450$  with a primary focus on estimating the pressure distribution on the impingement surface. With the primary pressure peaks were located at the stagnation point, a secondary pressure peak was observed between the jets. And this secondary peak was found to be independent of the Reynolds number while the jet-exit-to-impingement-surface distance was found to have influence over this.

The application jet impingement cooling for the aircraft wing' anti-icing was studied experimentally by Xueqin Bu [21]. So, a variable-curvature concave surface was considered by the authors for a high speed jet ( $Re = 21,021$  to  $Re = 85,340$ ) in their study. The authors had investigated the impact of the circumferential angle of the jet holes ( $-60^\circ$  to  $60^\circ$ ) on the piccolo tube. From the results, they had concluded that the decrease in the curvature radius and the increase in the jet impingement angle result in enhanced heat transfer.

Patricia Streufert [22] had investigated impingement heat transfer from a heated spheroid. These heated half-prolate spheroid were attached to a heated plate while the air at the ambient temperature was the impinging fluid. They had studied the spheroids for different ellipticity – defined as the ratio between spheroid's polar radius to equatorial radius. The authors had noticed a strong influence on heat transfer rate due to the spheroid's ellipticity. For the lower ellipticity, they had observed high heat transfer rate at the stagnation point.

D Sui [23] had analyzed the impingement heat transfer on a heat sink with the help of axial fans. With the axial fan speed was limited to 2577 rpm, the corresponding Reynolds number in their study was 9,000. The author had varied the fan-exit-to-flat-plate distance from 0.2 (very low clearance) to 2.0. In this model, there were no jets instead the high speed flow was made to impinge on the heat sink surface for heat removal. The authors had observed the pressure coefficient as well as Nusselt number was found to be decreasing with the increase in fan-exit-to-flat-plate distance.

Rajesh Kumar Panda [24] had applied the impingement cooling for a circular shaped heat sink. The authors had conducted experimental and computational studies for the flow conditions corresponding to Reynolds number 7,115 to 10674. In their study, the radiation heat transfer as well as natural convection effects was neglected. For the CFD simulations, velocity-inlet boundary condition was applied for modeling the jet inlet conditions. They had observed for that the closer positioning of the jets resulted in higher heat transfer rate.

Neil Zuckerman [25] had conducted 2-dimensional CFD simulations for modeling jet impingement on a cylindrical surface. In their study, the author had varied the number of nozzles from 2 to 8 to investigate the impact of heat transfer rate. For a radial surface, the authors had observed that the high heat transfer rate could be achieved for lower number of jets.

The heat removal mechanism using the impinging jets from nozzles that were positioned in a cylindrical plenum was studied by K Marzec [26] with the help of CFD simulations. The flow condition at the nozzle were corresponding to  $Re = 4,800$ . The 3-Dimensional, incompressible steady-state simulations were performed using ANSYS CFX. The authors had used SST  $K-\omega$  turbulence model in their simulations. A total of four nozzle geometry configuration – Short

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 2, February 2018

cylindrical nozzles, Long Cylindrical nozzles, Chamfered nozzles and countersunk nozzles. High heat transfer rate was observed for the short cylindrical nozzle and countersunk nozzle configurations.

Hwa-Chong Tien [27] had applied the impingement cooling mechanism for computer' CPU. The heat load for the study was 68.4 W, 80 W and 81.8 W with pin-finned heat sinks. The authors had conducted the research for various parameters such as fin height, fin width, heat sink' base plate thickness, the vertical distance between the nozzle and heat sink position ( $Z/D$  ratio). They had conducted this research work using CFD software IDEAS-ESC with  $k-\epsilon$  turbulence model. They had suggested a flow conditions corresponding to  $Re = 20,000 - 25,000$  at the impinging nozzle to obtain the comparable performances that were associated with the traditional cooling mechanisms.

Liquid jet impingement cooling method applied for Insulated Gate Bipolar Transistors (IGBT) of a Hybrid Automobile' inverters by S.V.J. Narumanchi [28]. The coolant for this study was a mixture of Water and Glycol (50%-50%). The authors had investigated this work for single phase as well as boiling mechanism (phase change due to the temperature exceeding the mixture saturation temperature). ANSYS FLUENT was chosen by the authors for performing all the CFD simulations in this research work. In the liquid jet impingement cooling, the material erosion must be accounted for while gas jet impingement cooling may not have such issue. Also, the pressure drop was critical for the liquid jet cooling as the increase in pumping power would lead to high capacity pumps.

For most of the jet impingement cooling mechanism, circular shaped nozzles had been used in general. However, Yavuzer Karakus [29] had investigated nozzles of multiple lobe shape for turbine blade' impingement cooling method. The authors had conducted experimental and numerical simulations for three nozzle configurations – 4 lobes, 6 lobes and 12 lobes. For the CFD simulations, the  $v_2-f$  turbulence model of Reynolds Averaged Navier Stokes (RANS) solver of ANSYS FLUENT was chosen by them.

Yu Rao [30] had conducted experimental liquid crystal thermography for investigating the heat transfer mechanism of multiple-impingement cooling. In their study, pin fins (fins with circular cross section) were attached to the flat plate. The authors had used 3CCD camera to capture the color changes, due to heat transfer mechanisms, on the target surfaces. These were compared against the CFD simulations that were performed using ANSYS CFX 14.0. They had studied three configurations – flat plate heat sink, heat sink with pin fins, heat sink with pin fins and effusion holes. The heat transfer rate, characterized by the Nusselt number in their study, was found to be improved by 6.3% for heat sink with pin fins and 25.3% for the heat sink with pin fins and effusion holes as compared to the flat plate heat sink. They had observed that the Nusselt Numbers reach a peak at the jet impingement location and decreases sharply then.

### III.CONCLUSION

The following points could be observed based on this literature survey on impingement cooling

- Realizable  $k-\epsilon$  turbulence model provided better accuracy in the impingement heat transfer simulations [1, 4]
- The impingement jet inclination plays an influential role in obtaining uniform heat transfer rate on the impinging surface [9]
- The swirl jet impingement produces more uniform heat transfer as compared to the non-swirl jet impingement [14]

### REFERENCES

- [1] Ridvan Yakut, Kenan Yakut, Faruk Yesildal, Altug Karabey, "Experimental and Numerical Investigations of Impingement Air Jet for a Heat Sink" IX International Conference on Computational Heat and Mass Transfer, Procedia Engineering, Volume 157, pp 3 – 12, 2016;
- [2] M. Behnia, A. Ooi, P. Gregory, "Prediction of Turbulent Heat Transfer in Impinging Jet Geometries" WIT Transactions on State of the Art in Science and Engineering, Volume 15, 2005;
- [3] Santosh Kumar Nayak, Purna Chandra Mishra, Sujay Kumar Singh Parashar, "Influence of Spray Characteristics on Heat Flux in Dual Phase Spray Impingement Cooling of Hot Surface" Alexandria Engineering Journal, Volume 55, pp 1995 – 2004, 2016;
- [4] Md Lokman Hosain, Rebei Bel Fdhila, Anders Daneryd, "Multi-Jet Impingement Cooling of a Hot Flat Steel Plate" The 6th International Conference on Applied Energy, Energy Procedia, Volume 61, pp 1835 – 1839, 2014;
- [5] Zhang Jingzhou, Xie Hao, Yang Chengfeng, "Numerical Study of Flow and Heat Transfer Characteristics of Impingement/Effusion Cooling" Chinese Journal of Aeronautics, Volume 22, pp 343 – 348, 2009;

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Issue 2, February 2018

- [6] Hussein M Maghrabie, M. Attalla, H. E. Fawaz, M. Khalil, "Numerical Investigation of Heat Transfer and Pressure Drop on In-Line Array of Heated Obstacles Cooled by Jet Impingement in Cross-Flow" Alexandria Engineering Journal, December 2016;
- [7] Abdulla A Al Ali, Isam Janajreh, "Numerical Simulation of Turbine Blade Cooling via Jet Impingement" The 7th International Conference on Applied Energy, Energy Procedia, Volume 75, pp 3220 – 3229, 2015;
- [8] Carlo Carcasci, Lorenzo Cocchi, Bruno Facchini, Daniele Massini, "Impingement Cooling Experimental Investigation using Different Heating Elements" 71st Conference of the Italian Thermal Machines Engineering Association, Energy Procedia, Volume 101, pp 18 – 25, 2016;
- [9] Muhammad A R Sharif, "Heat Transfer from an Isothermally Heated Flat Surface Due to Twin Oblique Slot-Jet Impingement" 5th BSME International Conference on Thermal Engineering, Procedia Engineering, Volume 56, pp 544 – 550, 2013;
- [10] Siddique Mohd Umair, Nitin Parashram Gulhane, "On Numerical Investigation of Heat Transfer Augmentation through Pin Fin Heat Sink by Laterally Impinging Air Jet" IX International Conference on Computational Heat and Mass Transfer, Procedia Engineering, Volume 157, pp 89 – 97, 2016;
- [11] Ajoy Debbarma Krishna Murari Pandey, Deepak Sharma, Gautam Choubey, "CFD Analysis of Rewitting Vertical Nuclear Fuel Rod by Dispersed Fluid Jet Impingement" Perspectives in Science, Volume 8, pp 110 – 112, 2016;
- [12] G. Srivalli, B. Raghavarao, M. R. Ch. Sastry, "Fluid Flow and Heat Transfer Analysis of Laminar Multiple Square Jets Impinging on a Flat Plate" International Journal of Engineering Research and Applications, Volume 6, Issue 6, pp 048 – 056, December 2012;
- [13] N. K. Chougule, G. V. Parishwad, P. R. Gore, S. Pagnis, S. N. Sapali, "CFD Analysis of Multi-jet Air Impingement on Flat Plate" Proceedings of the World Congress on Engineering, London, Volume III, July 2011;
- [14] Juliana K Abrantes, Luis Fernando, A. Azevedo, "Fluid Flow Characteristics of a Swirl Jet Impinging on a Flat Plate" 13th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal, pp 1 – 12, June 2006;
- [15] Jimmy Wayne Crafton, "The Impingement of Sonic and Sub-Sonic Jets onto a Flat Plate at Inclined Angles" Doctor of Philosophy Thesis, Purdue University, December 2004;
- [16] A. M. Tahsini, S. Tadayon Mousavi, "Laminar Impinging Jet Heat Transfer for Curved Plates" International Journal of Mechanical Aerospace Industrial Mechatronic and Manufacturing Engineering, Volume 6, Number 12, pp 2788 – 2793, 2012;
- [17] N Zuckerman, N Lior, "Jet Impingement Heat Transfer: Physics, Correlations and Numerical Modeling" Advances in Heat Transfer, Volume 39, pp 565 – 631, 2006;
- [18] Marcelo J S de Lemos, Cleges Fischer, "Thermal Analysis of an Impingement Jet on a Plate with and without a Porous Layer" Numerical Heat Transfer, Part A, 54, pp 1022 – 1041, 2008;
- [19] Jun-Bo Huang, Jiin-Yuh Jang, "Numerical Study of a Confined Axisymmetric Jet Impingement Heat Transfer with Nanofluids" Scientific Research, Volume 5, pp 69 – 74, 2013;
- [20] Malladi R Ch Sastry, B V S S S Prasad, A V S S K S Gupta, "Experimental and Computational Study of Fluid Flow on a Flat Plate with Three Rectangular Impinging Slot Jets" Indian Journal of Engineering & Materials Sciences, Volume 22, pp 631 – 640, December 2015;
- [21] Xueqin Bu, Long Peng, Guiping Lin, Lizhan Bai, Dongsheng Wen, "Experimental Study of Jet Impingement Heat Transfer on a Variable Curvature Concave Surface in a Wing Leading Edge" International Journal of Heat and Mass Transfer, Volume 90, pp 92 – 101, 2015;
- [22] Patricia Streufert, Terry X Yan, "Heat Transfer from an Impingement Jet on to a Heated Half-Prolate Spheroid Attached to a Heated Flat Plate" 2012 2nd International Conference on Environmental and Industrial Innovation, Singapore, Volume 35, pp 114 – 119, 2012
- [23] D Sui, T Kim, M L Xu, T J Lu, "Flow and Heat Transfer Characteristics of Impinging Axial Fan Flows on a Uniformly Heated Flat Plate" International Journal of Transport Phenomena, Volume 10, pp 353 – 363, 2008;
- [24] Rajesh Kumar Panda, B. V. S. S. S. Prasad, "Conjugate Heat Transfer from a Flat Plate with Shower Head Impinging Jets" Frontiers in Heat and Mass Transfer, Volume 2, pp 1- 10, 2011;
- [25] Neil Zuckerman, Noam Lior, "Radial Slot Jet Impingement Flow and Heat Transfer on a Cylindrical Target" Journal of Thermophysics and Heat Transfer, Volume 21, Number 3, pp 548 – 561, July – September 2007;
- [26] K Marzec, Kucaba-Pietal, "Heat Transfer Characteristics of an Impingement Cooling System with Different Nozzle Geometry" Journal of Physics: Conference Series 530, 2014;
- [27] Hwa-Chong Tien, Wei-Dong Huang, "Simulation and Assessment of Air Impingement Cooling on Squared Pin-Fin Heat Sinks Applied in Personal Computers" Journal of Marine Science and Technology, Volume 13, Number 1, pp 20-27, 2005;
- [28] S. V. J. Narumanchi, V. Hassani, D. Bharathan, "Modeling Single-Phase and Boiling Liquid Jet Impingement Cooling in Power Electronics" Technical Report, National Renewable Energy Laboratory, December 2005;
- [29] Yavuzer Karakus, Araz Jabbarh, Duygu Erdem, "Effect of Non-Circular Nozzle Shape on Impingement Cooling Characteristics" 7th European Conference for Aeronautics and Space Sciences, 2017;
- [30] Yu Rao, Yuyang Liu, "Experimental Investigation of Heat Transfer Characteristics in Jet Impingement Cooling System With Pin-Fins and Film Cooling Holes" Proceedings of Shanghai 2017 Global Power and Propulsion Forum, 30th October – 1st November 2017;