Numerical study on convective heat exchange between impinging gas jets and solid surfaces

Cite as: AIP Conference Proceedings **2449**, 050002 (2022); https://doi.org/10.1063/5.0090995
Published Online: 01 September 2022

Viktor Stoyanov and Valyo Nikolov

Numerical Study on Convective Heat Exchange between Impinging Gas Jets and Solid Surfaces

Viktor Stoyanov [a)] and Valyo Nikolov [b)]

Department of Transport and Aviation Equipment and Technologies,
Technical University of Sofia, Plovdiv Branch,
25 Tsanko Dyustabanov str., 4000 Plovdiv, Bulgaria.

[a)] Corresponding author: viktor.stoyanov@gmx.de
[b)] vnikolov@tu-plovdiv.bg

Abstract. When two areas of different gas pressure are connected through a nozzle, forming and propagating a gas jet with certain energy characteristics in the area with the lower pressure is observed. If a solid body is located in the area of the jet propagation, this particular case is defined as an impinging gas jet and heat transfer from the air jet to the solid body is observed. The aim of this work is to consider a specific case of heat transfer method at dimensionless nozzle and surface distances, respectively Z/D=10 under dimensionless jet Reynolds number Re=23300, using 3D engineering simulation models known as Computational Fluid Dynamics CFDs. Simulations in the commercial software product Star-CCM+ version 6.04.014, using two turbulence models - Shear Stress Transport SST and v^2f Low-Reynolds Number k-ε model are performed. The obtained results are presented with a dimensionless Nusselt number and its local validation determines the accuracy of the solver when using a digital twin describing a real measurement setting with known empirical measurements.

THEORETICAL PREREQUISITES

Fundamental experimental studies determining the temperature value on the surface of a dense solid body which is in a gas flow were realized in the 50s of the XX century by K Peri [5]. Later in the 1970s and 1980s, further analyzes and research in this area were conducted by H Martin [4], S Beltaos [10] and L W Florschuetz et al. [3]. H Martin [4] performed fundamental research and further developed the gas flow physics knowledge by proposing division of the impinging gas jet structure into three distinct flow regions: the free jet region, the stagnation flow region and the wall jet region [4]. The gas jet model proposed by H Martin [4] illustrates the airflow rate profile of an impinging gas jet in separate areas of distribution, in space, and in an area close to a solid body. The gas jet propagation speed directly depends on the coefficient of thermal conductivity in the area of contact between the impinging jet and the surface of a dense body which is heated. The model of H Martin [4] and the schematic separation of the impinging jet in different areas he proposed is illustrated in Figure 1 [4].

The Free Jet Region

In this area propagation with increasing gas jet rate is observed against the fluid from the surrounding space. In the area of the potential core Figure 1 to a length of 4 to 6 times the nozzle diameter, no changes are observed in terms

10th International Scientific Conference "TechSys 2021" – Engineering, Technologies and Systems
AIP Conf. Proc. 2449, 050002-1–050002-10; https://doi.org/10.1063/5.0090995
Published by AIP Publishing. 978-0-7354-4397-6/$30.00

of airflow rate and temperature which corresponds to the physical characteristics of the airflow at the Nozzle geometry outlet.

The Stagnation Flow Region

This area is characterized by the presence of a stagnation point. The pressure value has a local maximum at the stagnation point.

The Wall Jet Region

It is characterized by formation of a laminar boundary layer, which turns into turbulent flow at a distance of 1-2 times the diameter of the nozzle [4].

The characterization of the heat transfer of impinging jets and accuracy evaluation of the numerical analysis against the experimental results of the heat transfer process measurements and the efficiency of heat exchange processes [9] [11-19], is most often performed using the dimensionless number of Nusselt (Nu). In this study, the Nu was chosen as a dimensionless criterion for comparative analysis of the numerical study on convective heat transfer between impinging gas jets and solid surfaces.

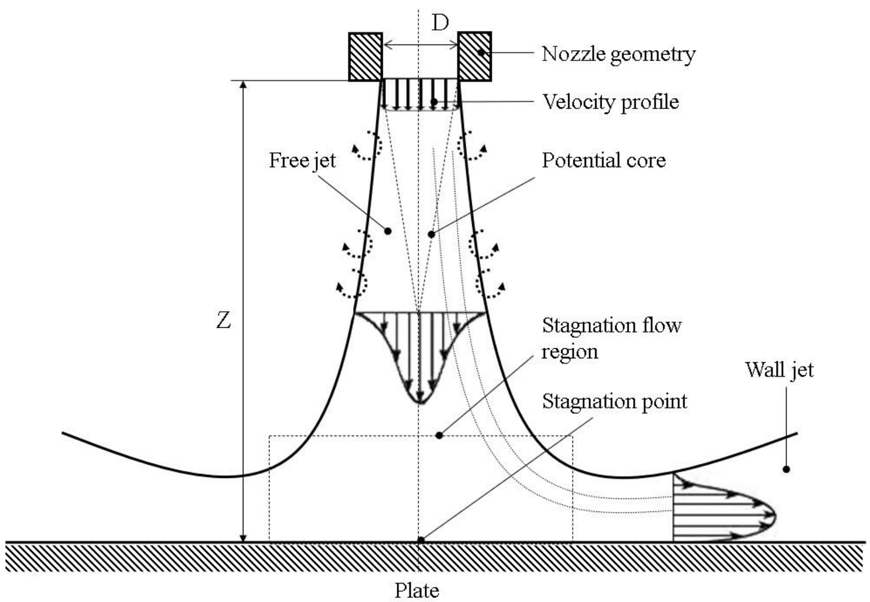

FIGURE 1. Schematic representation of a vertical impinging gas jet [4].

On the one hand, the transition to Industry 4.0 and the observed trends to reduce the new products development time and on the other hand the increasing complexity of the tasks solved with the help of numerical simulations necessitates in-depth analysis of the accuracy of various software programs used for modeling impinging gas jets. Despite the accumulated extensive practical experience in using impinging gas jets in our daily live, significant difference often occurs between the basic research results in this direction and the digital twins used. The differences in the results may be due to the choice of turbulent models of numerical simulations and the digital twin network parameters or the choice of proper software programs and in particular their program code for solving the equations describing the impinging gas jets with the help of numerical methods. In this regard a number of software program validations have been carried out in recent years. K. Thiel in [6] conducted a comparative analysis of three CFD programs, Fluent, CFX from ANSYS and Hybrid, using the following meshing tools: ICEM CFD, GAMBIT and Centaur. In this case, experimentally obtained NACA0012 values were investigated. The results show a difference between the values obtained from the comparative analysis for the different cases of validation of each of the software. In his publications S. Spring [9,11] presents the results of his research on the correlation between the simulated values of heat transfer obtained with the help of CFX-5.7.1 in different models of turbulence. S Spring used a CFX-5.7.1 digital twin to validate accuracy, including a numerical simulation of a single impinging gas jet and the resulting heat transfer from the contact between the gas jet and a flat plate. He analyzed the results obtained at different Z/D distances, including 6 and 12, at different Reynolds numbers. Z defines the distance between the nozzle outlet and a heat-conducting solid plate, and D defines the inner diameter of the nozzle. F. Ahmet et al. [12] used CFD simulations to examin the effect of the nozzle geometric parameters on the mode of the impinging jet spreading. An experimental case of the interaction between 225 impinging gas jets passing through nozzles with a diameter of 400 μm was considered by P. S. Penumadu and A. G. Rao [8]. To limit the overall dimensions of the study area, they choose to use a digital twin with symmetrical boundary conditions. The results of this study provide an insight into the physical processes of impinging gas jets propagation and their mutual influence. V. Stoyanov, V. Nikolov and M. Garcia [15] performed numerical study of heat transfer between impinging gas jets and solid surfaces at Z/D=2 and Re=23000, using Star-CCM+ software for this study and two different turbulent models. The obtained results were used to select turbulent models for further research. A. R. Salem, Nourin, M. Abousabae et al. [19] conducted experimental and numerical study of jet impingement cooling with Star-CCM+ software. The digital twins used in this study reproduce real areas of the Gas Turbine Blade geometry. F. Afroz and M. A. R. Sharif [16] used ANSYS Fluent CFD code for the numerical investigation of heat transfer from a plane surface at different values of Re=5000 to 25000. S. M. Simionescu, C. Bălan [18] conducted a numerical investigation on the convective heat exchange between an impinging gas jet and a structured solid surface with square-shaped grooves. In this study, the Fluent code was used in different turbulent models, as well as at Z/D=2 and Re=10000 and 20000. The results confirm the fact that the choice of turbulent model affects the accuracy of the numerical results obtained by using digital twins. Y. Lyu, J. Zhang, B. Wang et al [17] conducted convective heat transfer experimental and numerical study on flat and concave surfaces subjected to an impinging jet from round and lobed nozzle at Z/D=2, 4, 6 and 8, as well as Re=10000 and 20000. The results show a change in heat transfer efficiency, expressed by the dimensionless Nu, which is achieved with one nozzle at the expense of the other nozzle when changing the values of Z/D the efficiency changes. For this numerical investigation is used ANSYS Fluent 14.0 CFD code.

SELECTION OF TEST CASES

Determining the accuracy of the results obtained from the performed numerical study of impinging gas jet of solid surfaces is carried out by direct comparison of the values of dimensionless Nu obtained by experimental [1] and simulation way with Star-CCM+. The Nu appropriately represents the interrelation of geometric and energy parameters of a heat exchange process of impinging gas jet of solid surfaces. The Nu is often used in various scientific developments and analyses, as a dimensionless criterion for comparing the efficiency of heat transfer processes, including for evaluating the accuracy of results obtained by simulation against experimental results [12], [15-19]. In the considered experiment, the parameters describing the efficiency of the heat exchange process entirely depend on the nozzle geometry, the temperature of the solid in a gas flow, the distribution in the air jet space, the type of medium with which the experiment is conducted and its initial characteristics.

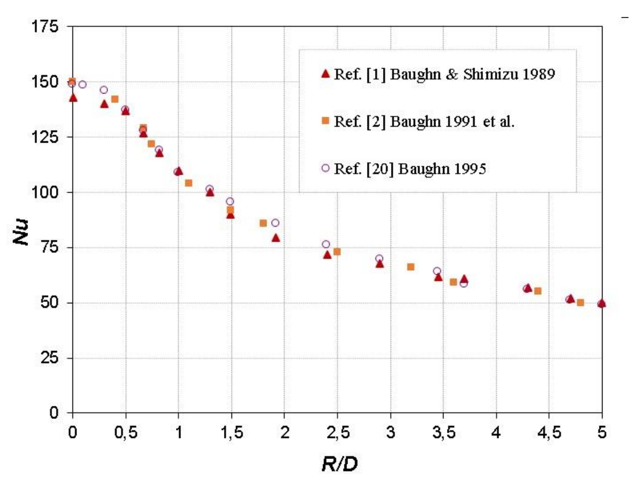

FIGURE 2. Local distribution of The Nusselt Number on a plate based on a test of impinging gas jet at *Z/D* = 10 [1-2] [20]

The investigated case corresponds to the experimental setup at a dimensionless number of Re=23300 and a dimensionless distance ratio of Z/D=10. In the Stagnation Flow Region, a strong initial maximum of Nu at a distance of 1D corresponding to one diameter and subsequent systematic decrease of the value of Nu is characteristic of all experiments performed at Re=23750, Z/D=6,10 and 14 [20] and similar at Re=23300, Z/D=6 [1-2]. To validate the STD-CCM+ CFD code, fundamental studies by S. Spring [11], F. Ahmet [12] and V. Stoyanov, V. Nikolov and M. Garcia [15] were used and the corresponding values of the local Nu at Re=23750 [20] and Re=23300 [1], obtained experimentally and visualized in Figure 2, where R defines the radius are on the inside of the nozzle outlet.

COMPUTATIONAL DETAILS

The commercial software code STAR-CCM+ 6.04.014 and a digital twin of the experimental model indicated in point 2 [2] [20] and illustrated in Figure 3 were used for the purposes of this work. In the current study of impinging gas jet of solid surfaces, a comparative analysis of the simulated values of a dimensionless Nu compared to the experimentally obtained Nu values was performed.

Domain and Boundary Conditions

During the simulation the boundary and initial conditions of the digital twin did not undergo changes. Reading the Nusselt local numbers was performed after reaching a quasi-stable equilibrium state of the iterations. Thus the inclusion of an additional numerical error in the final result of the reported Nu was limited. Two turbulence models were validated - SST and v^2f within this study. The choice of both turbulence models is based on a preliminary analysis of the research results at different Z/D ratio with software products such as ANSYS Fluent [7-8], [17], CFX [6] and STAR-CCM+ [15] which show satisfactory accuracy against other popular turbulence models such as k-ε and k-ω for the respective experimental set-up.

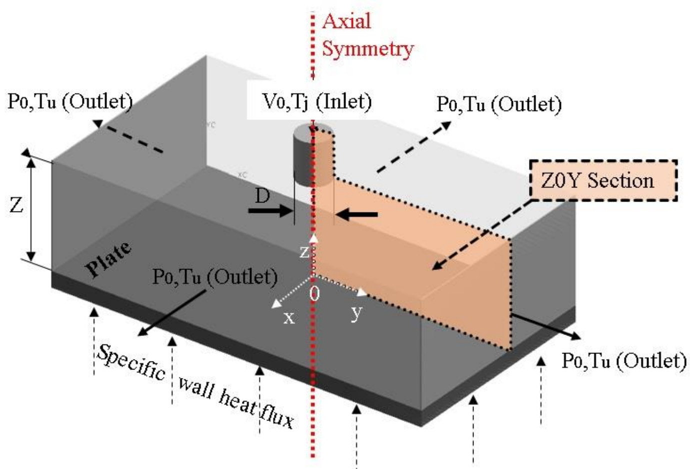

FIGURE 3. Numerical modeling and boundary conditions of the digital twin.

The digital twin shown in Figure 3 which serves as a basis for subsequent studies in this work is characterized by the following Domain and boundary conditions which are identical to the experimental setup of J. Baughin [20]. A circular cross section of a gas nozzle with diameter D=26 mm at initial air flow velocity $u_j = V_0$ at Re=23300 and kinematic viscosity at gas jet temperature T_j=293,15 K was used. The working medium in this system was air at an initial state corresponding to a static pressure p_u=101325 Pa and a temperature T_u=293,15 K. Impermeable solid body with thermally conductive properties was located parallel to the cross section of the nozzle outlet at a distance Z=10D. The initial temperature of the solid corresponded to the temperature of the air stream $T_j=T_u$=293,15 K. In turn, the solid body was exposed to external heat impact with specific wall heat flux = 300 W/m^2 which did not change during the study.

The numerical domain, fully corresponding to the experimental setup was characterized by axial symmetry of the system through the axis of the nozzle. This allowed for further changes in the numerical field of study with the purpose of using hardware resources efficiently and accelerating the time of obtaining the results of the digital twin investigation while maintaining the accuracy of the results.

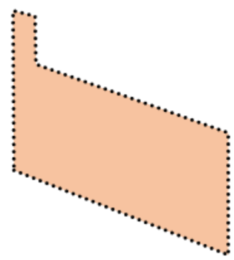

FIGURE 4. Two-dimensional view of the XOZ Section area.

The two-dimensional XOZ *Section* in Figure 3 is extrapolated independently from the basic digital twin and has the shape shown in Figure 4. Further modification of XOZ *Section* from Figure 4 was made to limit the influence of additional numerical errors in the field of axial symmetry of the system. The result of this modification is the conversion of a two-dimensional XOZ *Section* into a three-dimensional domain with a thickness Ds corresponding to the thickness of one cell of the numeral domain (Figure 5).

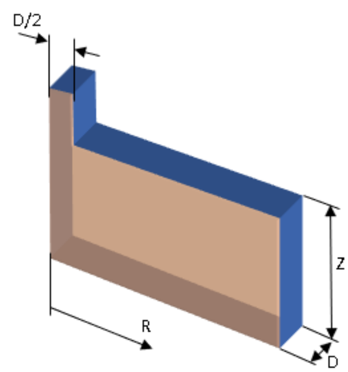

FIGURE 5. Three-dimensional view of the numerical area.

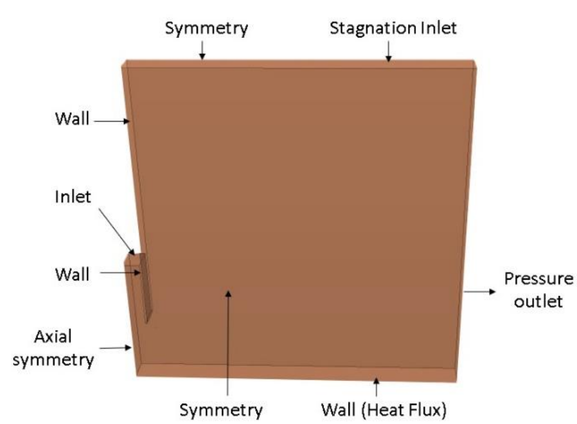

FIGURE 6. Boundary areas of the digital twin.

Then to the already defined initial requirements of the digital twin of fig. 5 the respective boundary areas were determined with their physical characteristics corresponding to experimental environment, respectively Wall, Achsen symmetry, Wall (Heat Flux), Inlet, Stagnation Inlet, pressure Outlet and Achsensymmetry (Figure 6).

Grid Generator

Computational meshes were generated by using *STAR-CCM+* version 6.04.014. In a similar way to other scientific studies [6, 7, 8], [15, 16, 17], *TRIMM* cells with a base cell length of 2 mm were used for the purposes of this development. The area near the wall (Figure 7 - "Wall Region") was meshed with a prismatic layer of 6,6 mm thickness in 200 layers with a coefficient of expansion of 1,03 and Dimensionless wall distance y+=0,06 (Figure 7 - "Section contact").

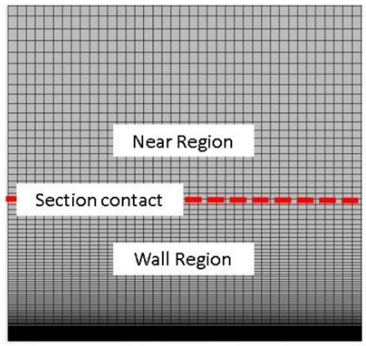

FIGURE 7. Section contact.

In the global computational area (Figure 8 - "Basis"), additional local areas were located to individually define the parameters of the mesh in them (Figure 7 - "Near Region", "Jet Nozzle Region", "Jet Region" and "Nozzle Extrud"). The selection of local areas for individual meshing was made on the basis of previous studies [15] which allows improvement of the obtained numerical results accuracy. In the contact regions between the different meshed areas, transition between the cells from a zone with smaller to a zone with larger elements with a factor of 2 to 1, respectively was defined (Figure 8).

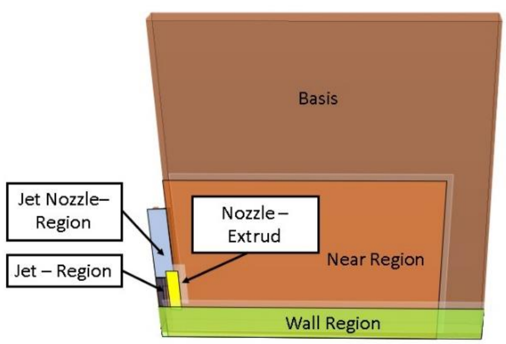

FIGURE 8. The location of the additional meshed areas.

The ratio of elements size in all areas relative to the element basic size, expressed as a percentage, is presented in Table 1.

TABLE 1. The ratio of element size in different areas to the element basic size of 2 mm.

Area	Ratio %
Basis	100
Surroundings	25
Inner jet tube	30
Ray environment	20
Pipe wall extrusion	10
Wall	10

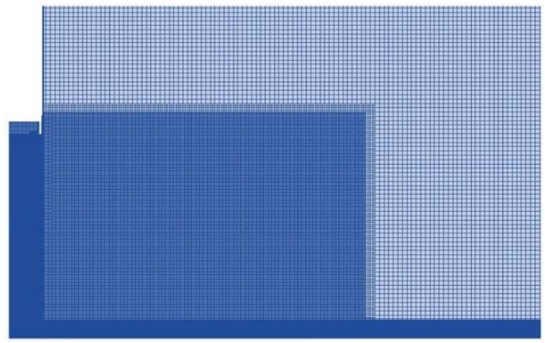

FIGURE 9. Three-dimentional view on a grid for the study of heat transfer between impinging gas jets and solid surfaces.

For the purposes of this work, the results of the mesh independent analysis were used to determine the independence of the result from the mesh size. After choosing the element base length, complete meshing of all areas of the twin was performed, the result of meshing the digital twin is presented in Figure 9, which includes a total of 597930 elements.

NUMERICAL PREDICTION AND EVALUATION OF TURBULENCE MODELS

The turbulence models selected in section 3.1 (v^2f and SST) were used to determine the local Nu required to perform accuracy analysis of the simulation results. Both turbulent models were integrated into the STAR-CCM + software product. To limit the influence of numerical errors in the validation of the results obtained from the simulation models, completely identical domain and boundary conditions of the digital twin were used in both turbulent models. The obtained results were validated with experimentally determined values of Nu at Re=23300, Z/D=10 [2, 20], whose numerical values are shown in Figure 10. Figure 10 also adds a result for the distribution of a dimensionless Nu obtained experimentally by the same author when using an identical experimental setup, but at Re=23750 [1].

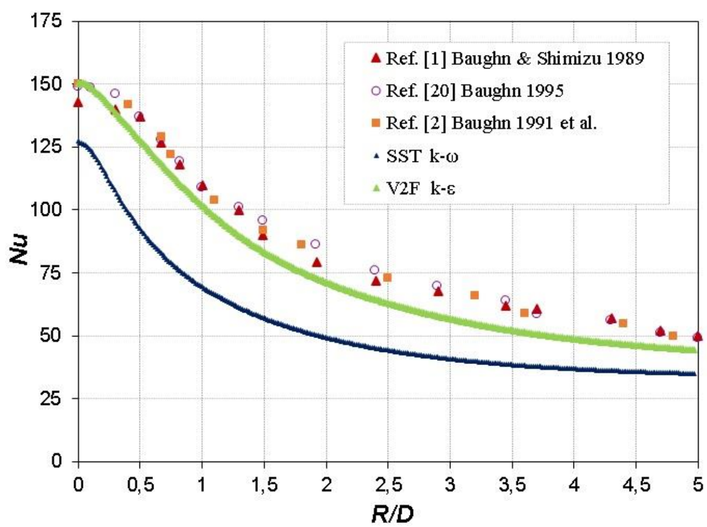

FIGURE 10. Local Nu distributions predicted by v^2f и SST turbulence models by Z/D=10 in comparison to experimental data from Ref. [1, 2, 20].

The turbulence models selected in section 3.1 (v^2f and SST) were used to determine the local Nu required to perform accuracy analysis of the simulation results. Both turbulent models were integrated into the STAR-CCM+ software product. To limit the influence of numerical errors in the validation of the results obtained from the simulation models, completely identical domain and boundary conditions of the digital twin were used in both turbulent models. The obtained results were validated with experimentally determined values of Nu at Re=23300, Z/D=10 [2, 20], whose numerical values are shown in Figure 10. Figure 10 also adds a result for the distribution of a dimensionless Nu obtained experimentally by the same author when using an identical experimental setup, but at Re=23750 [1].

Based on the obtained numerical results, a conclusion can be drawn that at the point of stagnation (R/D=0) where according to the experimental studies [1, 2] and [20] a maximum value of a Nu is expected, the v^2f model has better comparability with experimental results than the SST model. The obtained local value of Nu at Re=23000 and Z/D=10 by using the v^2f turbulent model is fully comparable with the experimental results [2] and [20]. At R/D=0, it was found that the SST model had a deviation of 18% from the experimental values of Nu at this particular point.

The numerical prediction of flow and heat transfer of the two turbulent models showed a partial deviation from the tendency for decreasing the value of local Nu as a function of the radial distance R/D observed in the experimental measurements as this is much more expressed for the SST model.

For the SST model in a range of R/D=0,1 to 1, was observed that the relative difference from the experimental values was not constant and changed, initially from 18 % at R/D=0,1 and reached a 40 % deviation in the experimental and numerical results at R/D=3. From R/D>3, a gradual decrease in the difference between the experimentally obtained Nu results and the corresponding results obtained by numerical methods using the SST model was observed. However, the deviations of the numerical results at R/D=5 from the experimental ones were still around 30%. Regarding the v^2f turbulent model, the observed differences from the experimental values vary up to a maximum of 9% in the region at R/D=0 to 1 and reach a local maximum of 18% at R/D=4,3.

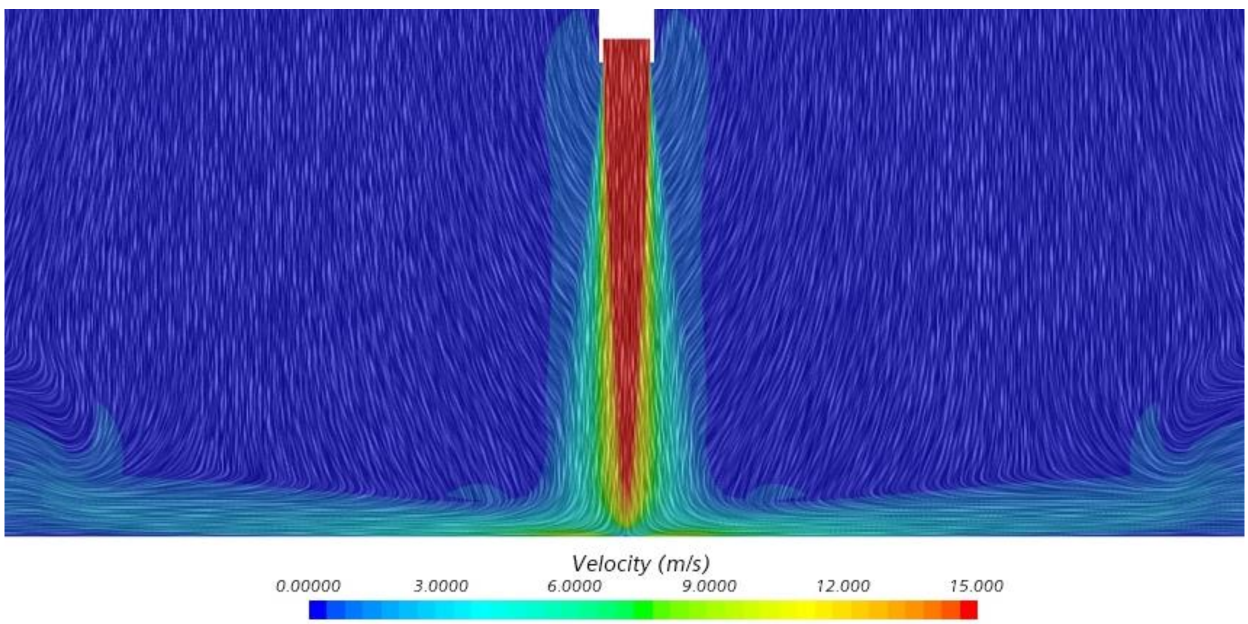

FIGURE 11. Visualization of the velocity distribution - magnitude velocity (turbulence model *v^2f*).

Both turbulent models v^2f and SST reproduce the effect of accelerating the initially stationary gas in the area near the nozzle which corresponds to the real situation and can be explained by the physical characteristics of the air mass. This effect, as well as the propagation of impinging gas jets in space and their deviation of the propagation direction due to the contact with a solid, is represented by the distribution profile of the velocity magnitude in Figure 11.

CONCLUSION

Despite the differences in the specific values of the Nu at Re=23300 in both turbulence models v^2f and SST no significant deviation of the relative tendency for distribution of local Nu as a function of the radial distance R/D was found. Initially found difference of Nu compared to the experimental values of [2] and [20] in the two turbulent models, respectively for SST is 15 % and for v^2f is 0,5 %. The ratio of the obtained differences of Nu compared to the experimental values is not only local, but rather an increase of this effect is observed in the numerical model in the range of impinging gas jet propagation near the surface of a solid body. It should be noted that the deviations of the

Nu results obtained using the SST model from the experimental values and shown in Figure 10 as a function of the radial distance R/D, in the range from R/D=0 to R/D=3 show deviations from 18 % to 40 %, respectively. There is a gradual reduction in the differences between the experimental and numerical results obtained using the SST model at R/D>3. In the range where heat transfer is expected to be at maximum efficiency, i.e., between R/D=0 to R/D=1, the results obtained with respect to local numbers of Nu using the v^2f model differ by 0,5 %, respectively (at the point of stagnation) up to 9 % of the experimental values. For the v^2f model, an average deviation of the numerical results with respect to the average number of Nusselt respectively 10,2 % was found in the range of the whole studied interval of R/D=0 to 5. In the same range, this value of the average deviation of numerical values from experimental values of Nu is 33,7%, respectively for the SST model.

ACKNOWLEDGMENTS

This work was supported by the European Regional Development Fund within the OP "Science and Education for Smart Growth" 2014-2020, Project Competence Centre "Smart Mechatronic, Eco- and Energy Saving Systems and Technologies", № BG05M2OP001-1.002-0023.

The authors would like to thank the Research and Development Sector at the Technical University of Sofia for the financial support.

REFERENCES

1. J. W. Baughin and S. Shimizu, "Heat transfer measurements from a surface with uniform heat flux and an impinging jet", Journal "Heat Transfer", Volume 111, pp.1096–1098, 1989.
2. J. W. Baughin, A. E. Hechanova and X. Yan, "An experimental study of entrainment effects on the heat transfer from a flat surface to a heated circular impinging jet", Journal "Heat Transfer", Volume 113, pp. 1023–1025, 1991.
3. W. Florschuetz, C. R. Truman, and D. E. Metzger, "Streamwise flow and heat transfer distributions for jet array impingement with crossflow", Journal "Heat Transfer", Volume 103, Issue 2, pp. 337-342, 1981.
4. H. Martin, "Heat and mass transfer between impinging gas jets and solid surfaces", "Advances in Heat Transfer", Academic Press, New York, Volume 13, pp. 1–60, 1977.
5. K. Perry, "Heat transfer by convection from a hot gas jet to a plane surface", Proceeding of Inst. Mech. Eng., Volume 168, pp. 775-784, 1954.
6. K. Thiel, "Vergleich des Lösungsverhalten der CFD - Programme: Fluent TM, CFXTM und Hybrid Graduate Thesis on the Institute of Aerospace Thermodynamics, University of Stuttgart, pp. 24-47, 2006.
7. Y. Q. Zu, Y. Y. Yan, and J. D. Maltson, "CFD prediction for confined impingement jet heat transfer using different turbulent models", Proceeding of ASME Turbo Expo 2009: "Power for Land, Sea and Air", Orlando, Florida, USA, GT2009-59488, 2007.
8. P. S. Penumadu and A. G. Rao, "Numerical investigations of heat transfer and pressure drop characteristics in multiple jet impingement system", Journal of Applied Thermal Engineering, Volume 110, pp. 1511–1524, 2017.
9. S. Spring, "Numerical Prediction of Jet Impingement Heat Transfer", PhD thesis, Institute of Aerospace Thermodynamics on the University of Stuttgart, ISBN 978-3-86853-811-3 35-54, 2010.
10. S. Beltaos, "Oblique impingement of circular turbulent jets", Journal of Hydraulic Research, Volume 14, pp. 17–36, 1976.
11. S. Spring, B. Weigand, W. Krebs and M. Hase, "CFD Heat Predictions of a Single Circular Jet Impinging with Grossflow", AIAA ASME Joint Thermophysics and Heat Transfer Conference, San Francisco, California, USA, 9 3589, 2006.
12. F. B. Ahmet, R. Tucholke, B. Weigand, and K. Meier, "Numerical investigation of heat transfer and pressure drop characteristics for different hole geometries of a turbine casing impingement cooling system", 14th International Heat Transfer Conference, Washington, DC, USA, IHTC14-22817 99-212, 2010.
13. S. Penchev, D. Seyzinski, J. Stanchev, "Modeling the flow in test section of wind tunnel UT-1", Journal of the Technical University at Plovdiv "Fundamental Sciences and Applications", Plovdiv, Bulgaria, Volume 13, Issue 8, pp. 82-86, 2006.
14. H. Panayotov and S. Hafizoğlu, "Airfoil aerodynamics investigation using automated CFD analysis", 6th Sc. Conf. BulTrans 2014, Sozopol, Bulgaria, Proceeding, pp. 86-90, 2014.

15. V. Stoyanov, V. Nikolov and M. Garcia, “Numerical study of heat transfer between impinging gas jets and solid surfaces”, IOP Conf. Ser.: Materials Science and Engineering, Volume 618, 012064, 2019.
16. F. Afroz, and M. A. R. Sharif, “Numerical investigation of heat transfer from a plane surface due to turbulent annular swirling jet impingement”, Journal of Thermal Sciences, Volume 151, 106257, 2020.
17. Y. Lyu, J. Zhang, B. Wang et al., “Convective heat transfer on flat and concave surfaces subjected to an impinging jet form lobed nozzle”, Science China Technological Sciences, Volume 63, pp. 116–127, 2020.
18. S. M. Simionescu, C. Balan, “CFD Study on Convective Heat Exchange between Impinging Gas Jets and Solid Surfaces”, Journal of Energy Procedia, Volume 85, pp. 481-488, 2016.
19. A. R. Salem, F. N. Nourin, M. Abousabae and R. S. Amano, “Experimental and Numerical Study of Jet Impingement Cooling for Improved Gas Turbine Blade Internal Cooling With In-Line and Staggered Nozzle Arrays”, ASME Journal of Energy Resources Technology, Volume 143, Issue 1, pp. 1-10, January 2021.
20. J. W. Baughin, “Liquid crystal methods for studying turbulent heat transfer”, Journal of Heat and Fluid Flow, Volume 16, pp. 365-375, 1995.