NUMERICAL STUDY OF HEAT TRANSFER ENHANCEMENT DUE TO THE USE OF FRACTAL-SHAPED DESIGN FOR IMPINGEMENT COOLING

by

***Lin CAI*[a,b*], *Zhuo LIU*[a], *Jianshu GAO*[b], and *Xiao LIU*[c,d]**

[a] School of Power and Mechanical Engineering, Wuhan University, Wuhan, China
[b] Key Laboratory of Transients in Hydraulic Machinery, Ministry of Education, Wuhan, China
[c] Department of Power and Energy Engineering, Harbin Engineering University, Harbin, China
[d] Division of Fluid Mechanics, Lund University, Lund, Sweden

Original scientific paper
https://doi.org/10.2298/TSCI17S1033C

This paper describes a numerical analysis of a heat transfer enhancement technique that introduces fractal-shaped design for impingement cooling. Based on the gas turbine combustion chamber cooling, a fractal-shaped nozzle is designed for the constant flow area in a single impingement cooling model. The incompressible Reynolds-averaged Navier-Stokes equations are applied to the system using CFD software. The numerical results are compared with the experiment results for array impingement cooling.

Key words: *Navier-Stokes equation, fractal-shaped, impingement cooling, CFD, heat transfer number*

Introduction

As the development of high energy ratios and low emissions for the modern gas turbines, the traditional film cooling is unable to meet their contemporary needs, and new technologies for the cooling the combustion chamber wall are required, see, for example, [1]. Because of its high convection heat transfer coefficient and its simple cooling structure, impingement cooling, which involves continuously adding a liquid or gas to a surface, is widely used in various fields, including the electronics, industrial manufacturing, and gas turbines [2]. To improve the cooling performance of the jet flow with the same coolant flow, the turbulence of the mainstream (or the flow near the target wall surface) needs to be enhanced. It has been reported that fractal design [3, 4] can produce higher intensity turbulence. For example, the fractal-shape design can enhance the turbulent strain contribution in the opposed jet [5]. The fractal-shaped orifices for the flow meters to enhance flow measuring techniques using the CFD method were investigated in [6]. The simulation result showed that the fractal-shaped design could accelerate flow recovery downstream. An experimental study of the heat transfer enhancement by applying a regular fractal grid to a single hole jet flow was carried out in [7]. The results showed that the fractal grid led to a non-uniform curvature of the jet shear layer, which caused instabilities near the nozzle exit and produced the vortices in the stream.

* Corresponding author, e-mail: cailin03313@whu.edu.cn

In our research, a fractal-shaped design is introduced to impingement the cooling in a gas turbine combustion chamber with the use of the CFD method. The fractal-shaped system used in this work has a multiple-holes structure instead of a grid structure.

The CFD modeling

Fractal design for the single jet cooling model

In our tests, Re_d = 10000 as all simulated conditions, and the ratio of the distances between jet nozzle and target plate to the diameter of the hole of the combustion chamber, denoted by H/d, is 4 (the parameters are similar to the results in [8]). Figure 1 shows the fractal-shaped design for the three models. The first model had a single hole with a diameter of 2 mm, the second model had four holes arranged around the center hole, and the third model has 21 holes. To keep the coolant flow rate constant, the inlet and outlet pressures were unchanged. The total area of all the holes for each model changed a little from one model to the next.

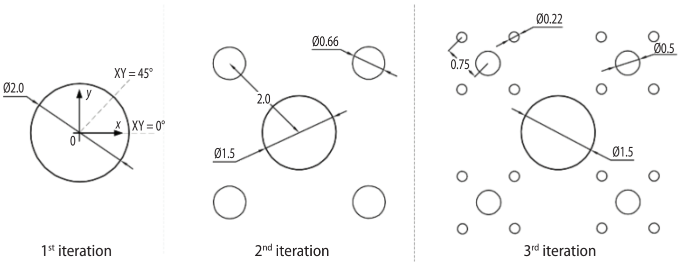

Figure1. Sketches of the three fractal-shaped models

Grid independent analysis and numerical verification

The CFD software FLUENT 16.2 was used to solve the steady time average equations, given by [8]:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\rho U_i \frac{\partial U_i}{\partial x_i} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho\overline{u_i^{'} u_j^{'}}\right] \tag{2}$$

$$\rho U_i \frac{\partial T}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\mu}{\mathrm{Pr}}\frac{\partial T}{\partial x_i} - \rho\overline{u_i^{'} T^{'}}\right) \tag{3}$$

where P is the time averaged pressure of the fluid, T – the temperature of the fluid, U_i – the velocity of the fluid, $u_i^{'}$ – the fluctuation components of velocity, and $T^{'}$ – the fluctuation components of temperature.

The SST k-ω turbulence model is used to close the transport equation. The PDE of the turbulence kinetic energy (TKE), k, and the specific dissipation rate, ω, are given by:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial k}{\partial x_i}\right) + G_k - Y_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_i}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_i}\right) + G_w - Y_w + D_k \tag{6}$$

where Γ_k and Γ_ω are the effective diffusivity terms, G_k and G_w – the generation terms, Y_k and Y_w – the dissipation terms, and D_k – the cross-diffusion term.

Numerical verification results

The total number of cells for the 1st-, 2nd-, and 3rd-iteration models is 1082720, 2323500, and 6190980, respectively. In order to verify the accuracy of the calculations, the array impingement cooling experiment data, see [8], is adopted. The symmetry boundary is used for the calculation, and the velocity inlet type is considered for the jet nozzle inlet. The number of grids for the numerical verification model is in the range from $3 \cdot 10^5$ to $1.1 \cdot 10^7$. The $\overline{\mathrm{Nu}}$ of the target wall is calculated directly. The conversion formula in [8] is used to obtain Sherwood number. Figure 2 displays the results with the experimental values for the different values H/d (changing from 1 to 10). This result is in a fairly good agreement with the experimental data. The maximum error is about 8.21% for $H/d = 10$ and 3.9% for $H/d = 4$. Thus, the mathematical models are reliable.

Results and discussion

When applying the fractal-shaped design for the single impingement cooling, we find that the potential cone is changed. The velocity ratio (VR) is the ratio of the inlet velocity of the model to the average inlet velocity of the first iteration model. The comparison of the VR of the three models is given in fig. 3. Changing from the 1st-, 2nd-, and 3rd-iteration models, the potential cone zone of each center hole becomes shorter in the jet direction. The VR of the center hole in the 3rd-iteration model is about 0.6.

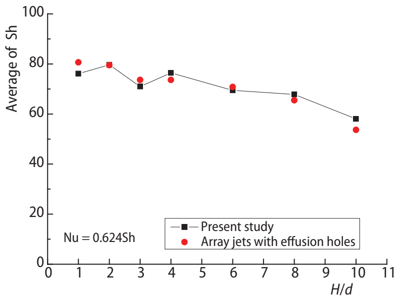

Figure 2. Numerical verification results of the array impingement cooling at $\mathrm{Re}_d = 10000$

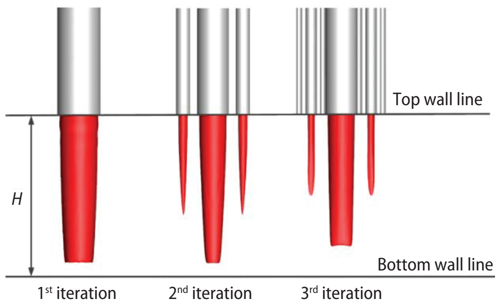

Figure 3. The surface of the VR (VR ≥ 0.99) of the three models

The top wall side surface is a wall type for the gas turbine combustion chamber impingement cooling. As the mainstream fluid jet moves toward the target plate, the surrounding fluid moving into the jet at high velocity generates a secondary vortex.

As shown in fig. 4, the secondary vortex created in the mainstream is loop-like, and a secondary vortex is found near the exit.

The local distribution of Nusselt number at the bottom line in the XY = 0° direction is illustrated in fig. 5.

The local distribution of Nusselt number at the bottom line in the XY = 45° direction is shown in fig. 6.

In our work, the heat transfer in some areas is enhanced by the fractal-shaped design, especially in the XY = 45° direction. The mechanism of heat transfer enhancement can be suc-

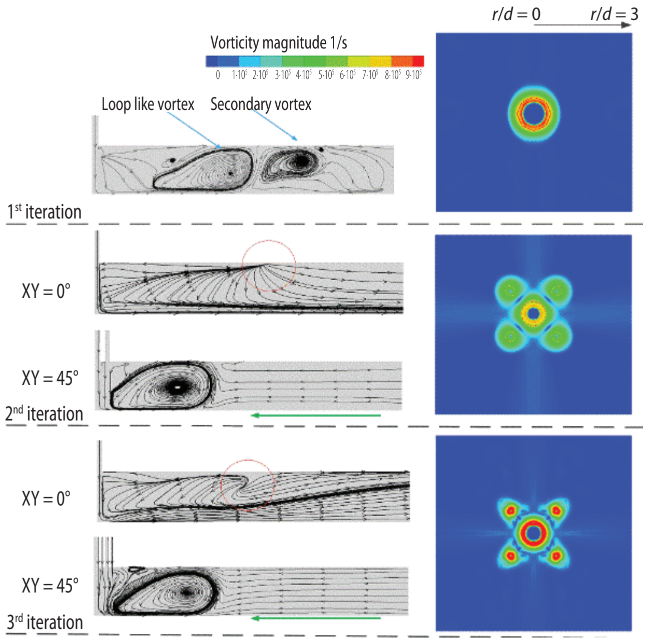

Figure 4. The streamline at XY = 0° and XY = 45° (left) and vorticity magnitude at the jet nozzle and target plates (right) (*for color image see journal web site*)

cessfully explained by TKE distribution. When r/d changes from 0 to 10, the TKE contours for the three models in the XY = 0° and XY = 45° directions were given in fig. 7.

When r/d changes from 0 to 3, the TKE contours for the three models in the XY = 0° and XY = 45° directions were given in fig. 8.

The TKE contours for the three models in the XY = 0° and XY = 45° directions are illustrated in fig. 9.

When r/d changes from 1 to 3, the bar charts of $\overline{\mathrm{Nu}}$ for the three models is displayed in fig. 10.

When r/d changes from 4 to 6, the bar charts of $\overline{\mathrm{Nu}}$ for the three models is displayed in fig. 11.

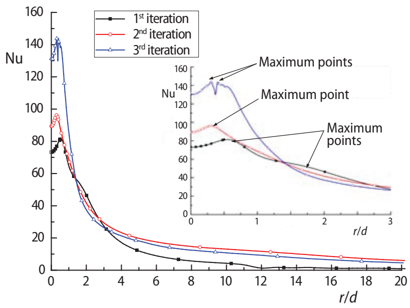

Figure 5. Local distribution of Nu at the bottom line in the XY = 0° direction

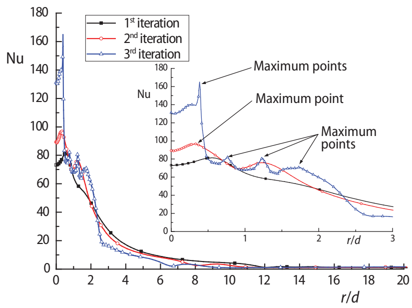

Figure 6. Local distribution of Nu at the bottom line in the XY = 45° direction

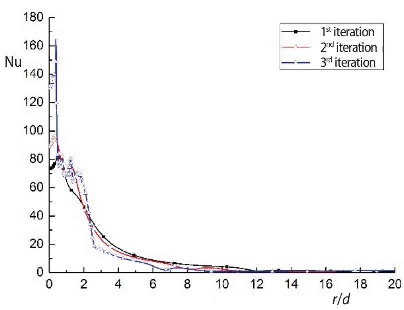

Figure 7. Local distribution of Nu at the bottom line in the XY = 45° direction

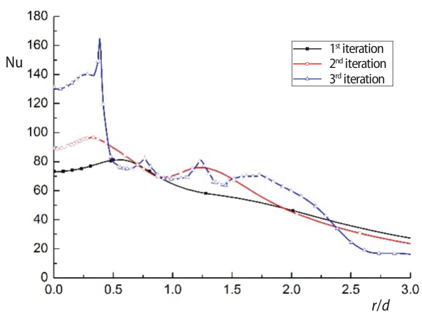

Figure 8. Local distribution of Nu at the bottom surface in the XY = 45° direction

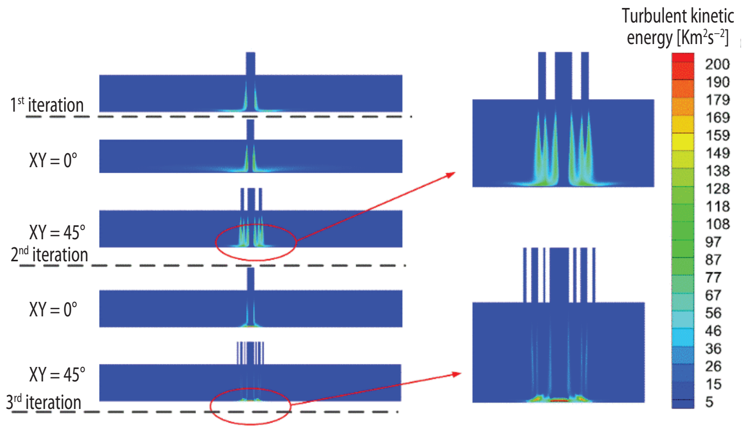

Figure 9. The TKE distributions of the three models in the XY = 0° and XY = 45° directions
(*for color image see journal web site*)

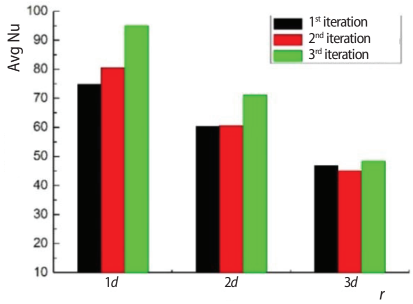

Figure 10. The bar chart of $\overline{\mathrm{Nu}}$ for three models

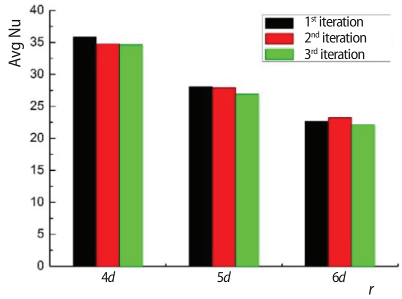

Figure 11. The bar chart of $\overline{\mathrm{Nu}}$ for three models

Conclusion

A fractal-shaped design for the impingement cooling of the gas turbine combustion chambers is considered in the present work. The CFD method was used to investigate the flow field at $\mathrm{Re}_d = 10000$ and $H/d = 4$. The fractal-hole models were adopted instead of a network structure. Making use of a constant flow under the same pressure drop, the total flow area of the fractal holes for each model is unchanged. The obtained results show that the length of the potential cone is reduced after introducing the fractal-shaped design. The fractal holes weaken the vortex in the 0° direction, while enhancing it in the 45° direction. Analysis of the TKE in the 0° and 45° directions showed that the TKE value in the stagnation zone increases with the change from the three models. The fractal-shaped nozzles change the mixing process and the entrainment effect of the main jets. The heat transfer in the stagnation zone is enhanced by the fractal-shaped design because it is related to the backflow caused by the entrainment of the main jets and the distribution of the TKE. It is shown that the effects of effusion holes and cross flow should be considered to handle the actual cooling of combustion chambers. The jet interference between array holes may bring new problems for the fractal-shaped design nozzles. Thus, it will be open a new perspective for introducing the fractal-shaped design to the array impingement or impingement/effusion cooling.

Acknowledgment

The present work is supported by the Fundamental Research Funds for the Central Universities (Grant No. 413000011, 413000062). Author Xiao Liu is supported by scholarship from China Scholarship Council which is gratefully acknowledged.

Nomenclature

d – diameter of circular nozzle, [m]
H – height of nozzles, [m]
Nu – Nusselt number, [–]
$\overline{\mathrm{Nu}}$ – average Nusselt nubmer, [–]
Re_d – Reynolds number, [–]
r – radius distance, [m]
XY – axis of Cartesian co-ordinates, [m]

References

[1] Womac, D. J., *et al*., Correlating Equations for Impingement Cooling of Small Heat Sources with Single Circular Liquid Jets, *Journal of Heat Transfer*, *115* (1993), 1, pp. 106-115
[2] Facchini, B., *et al*., Impingement Cooling for Modern Combustors: Experimental Analysis of Heat Transfer and Effectiveness. *Experiments in Fluids*, *40* (2006), 4, pp. 601-611
[3] Liu, S., *et al.*, Distribution of Primary Additional Errors in Fractal Encoding Method, *Multimedia Tools and Applications*, *76* (2017), 4, pp. 5787-5802
[4] Liu, S., *et al.*, Fractal Generation Method Based on Asymptote Family of Generalized Mandelbrot Set and Its Application, *Journal of Nonlinear Science and Applications*, *10* (2017), 3, pp. 1148-1161
[5] Geipel, P., *et al.*, Fractal-Generated Turbulence in Opposed Jet Flows, *Flow, Turbulence, Combustion*, *85* (2010), 3, pp. 397-419
[6] Elsaey, A., *et al.*, CFD Simulation of Fractal-Shaped Orifices for Flow Measurement Improvement, *Flow Measurement and Instrumentation*, *36* (2014), Apr., pp. 14-23
[7] Cafiero, G., *et al.*, Heat Transfer Enhancement of Impinging Jets with Fractal-Generated Turbulence, *International Journal of Heat and Mass Transfer*, *75* (2014), Aug., pp.173-183
[8] Rhee, D. H., *et al.*, Local Heat/Mass Transfer and Flow Characteristics of Array Impinging Jets with Effusion Holes Ejecting Spent Air, *International Journal of Heat and Mass Transfer*, *46* (2003), 6, pp. 1049-1061

Paper submitted: March 8, 2017
Paper revised: April 25, 2017
Paper accepted: May 8, 2017