Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

NUMERICAL STUDY ON THE EFFUSION COOLING PERFORMANCE OVER THE WALLS OF AN ANNULAR BURNER

Wei-Hsin SUN[1], Chao-Jun HUANG[2], Chen-Yuan WU[3] and Jr-Ming MIAO[4*]

[1] Ph.D Student, Graduate School of Defence Science Studies, Chung Cheng Institute of Technology, National Defence University, Taoyuan, Taiwan, R.O.C.
[2] Senior Specialist, Division of Mechanical Engineering, Chung-Shan Institute of Science and Technology, Taoyuan, Taiwan, R.O.C.
[3] Assistant Professor, Department of Mechanical Engineering, Air Force Institute of Technology, Kaoshiung, Taiwan, R.O.C.
[4] Professor, Department of Materials Engineering, National Pingtung University of Science and Technology, Pingtung, Taiwan, R.O.C.

*Corresponding author, E-mail address: jmmiao@mail.npust.edu.tw

ABSTRACT

The present study aims to investigate the cooling effectiveness of effusion cooling of a convex plate under the adiabatic and the conjugate heat transfer model. A computational fluid dynamic technique based on the control volume method is used to compute the detailed Nusselt number and cooling effectiveness distribution on the target surface by solving the steady, three-dimensional incompressible Reynolds-averaged Navier-Stoke's equations. The turbulence quantities are enclosed by the realizable κ-ε turbulence model with enhanced wall function. To investigate the film cooling effectiveness of effusion cooling mode under the effects of elliptic jet hole configuration and blowing ratio, combinations of three aspect ratios of AR=0.5, 1, 2 and four blowing ratios of BR=0.25, 0.50, 0.75, 1.00 are numerical conducted. The results show that with the increase of the aspect ratio of holes the film cooling effectiveness reduces for all the simulation cases. Also, the conjugate heat transfer model is better suitable for the application of real engineering problem as compared with the adiabatic heat transfer model. The injection hole with aspect ratio of 0.5 yields the best cooling effectiveness among the test modelled scenarios in ranges of tested blowing ratios.

NOMENCLATURE

AR aspect ratios
BR blowing ratios
U,u velocity
ρ density
t time
φ dependent Variable
Γ diffusion coefficient
S source term
η film cooling effectiveness
T temperature

Subscripts

c coolant
∞ mainstream
w wall

INTRODUCTION

For any aircraft, the engine holds incredible amount of importance, much like the heart does for the human body. Currently, the most widely used power source is the gas turbine engine but constant breakthroughs in engine technology have had a critical impact on the aviation industry. According to the theory of the Brayton cycle, in the hopes of enhancing the performance of an engine, one might increase the inlet temperature of the turbine sections to obtain a higher thermal efficiency and thrust. However, when the turbine inlet temperature is raised, the temperature of the combustion chamber must be raised as well. Thus, when considering this approach for enhancing thermal efficiency, it is important to pay attention to the design of the combustion chamber wall lining and the turbine blades to ensure that they can operate under high pressure and high temperature circumstances. Improved combustion efficiency, reduced exhaust pollution, uniform temperature distribution, and less weight, among others, are indicators of a well-designed combustion chamber. Modern day gas turbine temperatures have far surpassed the melting point of many metal materials. Therefore, the availability of high temperature alloy materials and the development of thermal protection cooling technologies have become important factors in the bid to reach performance targets. As the performance targets rise, with rising gas temperatures, however, room for developing high temperature alloy materials and the demand for this material has dropped. Instead the focus has fallen on development in thermal protection. In the area of thermal protection cooling, there are a few main methods of cooling: convection cooling, impingement cooling, internal rib cooling, and diffusion cooling. Many present day methods are combinations of the above listed methods. The jet impingement cooling method is one in which cooling fluid is thrown on the high temperature solid surface to dissipate heat and cool the surface.

The jet impingement heat transfer model has often been used for engineering issues regarding heat or mass transfer efficiency in cooling and heating. The scope of its use also includes electronic chip cooling, metal plate cooling, glass manufacturing, and textile and paper drying among others. In recent years, it has been used for cooling micro-electro-mechanical components (Myung and Issam, 2006), solar thermal absorbers (Roger and Buck, 2005), internal turbine blade cooling (Han, 2004), and combustor liner cooling (Facchini and Surace, 2006) among other technical uses. The jet impingement heat transfer method is useful in many engineering applications. Aside from that, the single-jet and multi-jets of fluid dynamics and heat transfer study have plenty of interesting characteristics worthy of in-depth study. Becko (1976) divides the jet impingement nozzle configuration into four different categories: a single round jet, arrays of round jet, a single slot jet, and arrays of slot jet.

Because single jet stagnation occurs at the highest heat or mass transfer coefficient, there are a considerable number of research results that focus on the stagnation point with the Nusselt number, Reynolds number and Prandtl number, among other flow transfer numbers (Neil md Noam, 2005). Huber's research group (1994) further pointed out some key factors that also influence the jet impact heat transfer method: wall-jet interaction, separation distance, adjacent jets and jet spacing, jet diameter, distance between jet and target plate among others. Lee and Lee (2000) focused on the effect of aspect ratio (AR); after testing various AR values of hole and spacing between target plate and orifice plate, under different cooling jet Reynolds numbers, there were some findings. For one, it was discovered that when the separation distance is smaller, higher AR of holes revealed better heat transfer distribution. On the other hand, when the separation distance was larger, the round hole jet (AR=1) had a better cooling performance than the high AR ellipse-shaped hole jet.

For the numerical simulation method for forecasting the accuracy of heat flow field characteristics, it is very important to select the appropriate turbulence model. Of these turbulence models for impingement cooling, Neil and Noam (2005), Roger and Buck (2005), Coussirat et al. (2005) compared the realizable k-ε turbulence model, standard k-ε turbulence model, standard k-ω turbulence model, Reynolds stress model turbulence model, SST-k-ω turbulence model for differences. Roger and Buck (2005) believes that using the realizable k-ε turbulence model generated better data that were more in line with predicted results, as well as having better coverage of the convective heat transfer performance predictions. Coussirat et al. (2005) pointed out that the realizable k-ε turbulence model fit the impingement surface as predicted by the Nusselt number, and that the standard k-ε turbulence model or the standard k-ω turbulence model fit the partial impact surface as predicted by Nusselt number. Lanyaa and Deborah (2005) compared the Yang-Shih turbulence model with the standard k-ε turbulence model under conditions in which the cross-flow had different angled impact holes with experimental measurements to explore error value. The above-mentioned turbulence models cause numerical unavoidable shock. However, by using multi-grid or fine-grid treatment, their problem can be prevented. Many studies have tended to explore the theoretical point of view when discussing impingement cooling flow field characteristics and heat transfer conjugate analysis. R&D engineers can use this as reference for designing combustion wall lining with enhanced cooling properties. In practical engineering application, Li et al. (2005, 2006), Gordon and Levy (2005) have successfully conducted a numerical simulation analysis of the flow field characteristics and cooling efficiency of the combustion chamber. These results can accurately predict the combustion chamber temperature and hot spot positions. This has helped to prevent lining overheating as well as greatly saved considerable time and funding for R&D.

As can be seen from the above studies, the majority of past researchers adopt experimental measurements to explore relevant parameters. Even though data from actual engineering problems and academics are used, they can not accurately predict three-dimensional flow field characteristics and fully clarify the complex phenomenon of thermal convection. In recent years, following the progress of computer technology and ever-increasing numerical accuracy, numerical simulation can now complement the lack of experimental results. In view of this, the presented study uses a ring-shaped combustion chamber simulation for testing CFD methods to evaluate the different cooling models and the protection they offer.

RESEARCH MODEL DESCRIPTION

This study explores how different blowing ratio conditions and shapes of the cooling hole affect effusion cooling performance of combustion chamber lining by three-dimensional complex flow field numerical characteristics analysis. This study uses models from physics of which, the mainstream hot flow model for turbine engine combustion chamber and the effusion cooling plate are located on the outer ring, as shown in Figure 1. A 12 ° fan shaped access is the defined boundary of computational domain, with the aspect ratio of hole (AR) is fixed at three numbers, AR=0.5, AR=1, and AR=2, and the hydraulic diameter of the hole is set at 13 mm. This arrangement is fixed at the injection hole to compare film cooling effectiveness as well as to test out the effect of hole's configuration.

GOVERNING EQUATION

For fluid domain, the governing equation used in this study is three-dimensional Reynolds-averaged Navier-Stokes equation included of conservation of mass, momentum and energy. The general form of Navier-Stokes equation is as follows:

$$\frac{\partial(\rho\phi)}{\partial t} + \mathrm{div}(\rho \boldsymbol{u} \phi) = \mathrm{div}(\Gamma \ \mathrm{grad}\ \phi) + S \qquad (1)$$

where S represents the source term which is ignored in present work. The Reynolds stress term in the momentum equation is treated with the reliable k-ε turbulence model. For solid domain, the heat conduction Fourier equation is solved to obtain the temperature distributions.

RESEARCH METHODS

Numerical Methods

The three-dimensional Navier-Stokes equations are solved by the commercial software FLUENT with control volume method. The working fluid for hot gas and coolant stream is air and assumes as ideal gas. The convection term is discretized with the second order upwind scheme, while the diffusion term with the central differencing scheme. As for the coupling of velocity and the pressure in the momentum equation, the SIMPLEC (semi-implicit method for pressure-linked equations consistent) method is used. The AMG scheme is also implemented for acceleration of convergence.

Boundary Conditions

In the calculation of convective cooling performance with the adiabatic heat transfer model and conjugate heat transfer model, the entrances of and coolant flow is set to "Mass Flow Inlet" patterns with specified mass flow rate and static temperature, while the outlet side of the mainstream duct is set to the "Pressure Outlet" pattern. The static pressure is set at 0 Pa, and the reference to a fixed operating pressure of 1atm. The mainstream condition is fixed at the entrance with a speed of U_∞ = 80 m/s, temperature T_∞ = 1596 K, density ρ_∞ = 5.088 kg/m^3; coolant flow is fixed at the temperature T_c = 800 K, density ρ_c = 10.853 kg/m^3 (Hu and Ji, 2004). As the speed of the coolant flow, U_c, is calculated from the BR (Blowing Ratio), it is defined as follows:

$$BR = \frac{\rho_c U_c}{\rho_\infty U_\infty} \tag{2}$$

Because the 12° fan model is built as the computational domain, shown in Fig. 1, the two-sides of boundary faces in circumferential direction are set as the periodic boundary condition. All the solid walls are assumed as no-slip and adiabatic condition except for interface between the fluid medium and solid medium in the conjugate heat transfer model. The thermal condition is set as "coupled" type based on energy balance.

Convergence Criteria

For the numerical modelling of steady-state flow fields, setting the convergence standard is an important step. With regard to related researches on hybrid impingement/effusion cooling problem, the convergence standard selected by this study is set to when the relative residual value of mass continuity is smaller than 10^{-5} in two sequencing iteration. The numerical calculation work s in this study was processed by a personal computer with a Pentium(R) D-3.0Ghz CPU and 2048 MB of RAM. The number of calculation iterations necessary to finish a single case is approximately 4000. The necessary time is approximately 48 hours.

GRID SYSTEM

The quality and number of grids is strongly related to the accuracy of numerical modelling on thermal-flow field capturing and the physical implementation. The grid systems are constructed by the ICEM/CFD software. To ensure the high orthogonality of the grids, the computational domain is filled with multi-block and body fitted Hexagonal cells. The consideration is that the grid placement of the various blocks inside the computational domain needs to capture actual thermal-flow field phenomenon into account. As there are more intense flow fields changes at the impingement cooling flow ducts and the intersection area between the main cross flow and the cooling flow, local refinement is performed on the grid placement in these areas. Figure 2 are surface grid distribution on the impingement hole plate and the lining wall for elliptic hole with aspect ratio of AR=0.5. The O-type grids are used at the impingement holes plate and the jet flow exit area, while H-type grids are used at every other location.

In order to confirm the grid independence of the calculation results, and taking the research focus on the average Nusselt value of the testing plate surface produced through each jet flow into consideration, grid adjusting is mainly performed at areas close to each cooling jet hole and the main flow path area. A grid number of 1.39 million is set as the standard, and the grid number is reduced by 35% to approximately 910,000 and increased by 42% to approximately 1.98 million to produce three different grid systems. Independent grid testing is performed to compare the area-average Nusselt number results for tested runs with different grid systems. The difference between predicted area-average Nusselt number is less than 3% for the grid system of 1.39 million numbers and the 1.98 million numbers. Thus, the grid system with 1.39 million of cells is used to conduct further runs, based on the work station simulation efficiency.

RESULTS AND DISCUSSION

As effusion cooling mode combines the protective methods of impingement cooling and film cooling is widely used to increase the heat-transmission amount in hot heat flux components in gas turbine engine. The schematic view of the cooling structure is shown in Figure 3. Advantages of this hybrid cooling method include of a higher cooling effectiveness, evenly-distributed wall temperatures, and low pressure losses [6]. This study utilizes air as the working fluid, and explores the influence of impingement hole configuration with aspect ratios of AR=0.5, 1, and 2 on the development of thermal-fluid flow field and surface Nusselt number distribution. In should be noted that the arrangement type in adjacent row of holes is parallel in mainstream direction. For three temperature heat convective problems, the film cooling effectiveness η is defined as:

$$\eta = \frac{T_\infty - T_w}{T_\infty - T_c} \tag{3}$$

where T_∞ is the hot mainstream temperature, T_w is the wall temperature and T_c is the coolant temperature.

Adiabatic Heat Transfer Model

It can be discovered from observing Figures 4(a)~(c) that the lining wall temperature in region of $3 \leqq x/a \leqq 5$ is significantly higher than other positions when the effusion hole has an aspect ratio of AR=0.5 and the blowing ratio is BR=0.2 The reason is that the coolant jets ejected from the elliptic holes with various aspect ratios is less strength to defect with the cross mainstream which causes very low amounts of coolant flow to flow over these area. Thus, the predicted film cooling effectiveness is relatively low. However, it can be discovered for tested configurations of elliptic hole that, with the increase in the blowing ratio, coolant jets have enough momentum to penetrate into the high-temperature cross mainstream, and the interaction forms a low-temperature film layer against the inner wall to achieve the designed cooling protection effect. It can be discovered from comparing of Figures 4(a)~(c) that, under same blowing ratios, the lining wall temperature distributions between holes in laterally direction increases with increasing of the AR value of elliptic holes. Besides, the cooling streams can more effectively flow out from the effusion holes for low AR elliptic hole even with less blowing ratios thus have a better cooling performance along the streamwise direction. This phenomenon attributes to the findings that effusion hole's shape indeed affects the distributions of adiabatic wall temperature over lining wall especially at the medium blowing ratios.

Figures 5 (a)~(c) presents the detailed velocity contours in the X-Z section through second row of impingement holes for tested models with aspect ratio of effusion holes as AR=0.5, 1, and 2, respectively under various blowing ratios. In the impingement chamber, the impingement jets are no longer behaviour as free jets. The deflection of individual jet is due to one ends of chamber is closed and the deflection level of jets is influenced by the pressure drop between the impingement hole and beneath effusion hole for all tested aspect ratios of holes. Once the coolant streams ejected from the effusion holes into the combustion chamber, the thermal-flow field of multiple jets are resulting from the momentum and energy balance of jets and incoming cross mainstream.

It can be clearly observed from Figure 5(a) that when the hole with aspect ratio (AR) of 0.5 and BR=0.25 there is very few cooling flows existed at the leading portion of lining wall (i.e. $3 \leqq x/a \leqq 9$). When the blowing ratio gradually rises, impingement coolant jets have enough momentum to against the high-temperature cross mainstream. A visible layer of low-temperature flow structure is developed from the leading edge of lining wall to reduce the lining wall temperature and achieve a protection effect. Similar observations can be found for others tested conditions in which the effusion holes with aspect ratio of 1.0 and 2.0, shown in Fig. 5(b) and (c).

Figures 6(a)~(c) respectively presented the predicted laterally averaged η distribution on the lining wall along the streamwise direction for effusion holes with aspect ratio of 0.5, 1.0, and 2.0 under different blowing ratios. Downstream of x/a=5, the laterally averaged film cooling effectiveness is varied in ranges of 0.6 to 0.8 regardless of the shape of effusion hole. The local peaks in each curve are caused from the direct interaction of individual jets with the cross mainstream. It is obvious that the turbulence effect will enhance the convective heat transfer effect. Results also indicated that the curve of laterally averaged film cooling effectiveness is not monotonically increased with .the blowing ratios as expected. This is due to the coolant jets may penetrate into the core of mainstream to loss the designed film covering protection capacity.

Figure 7 clearly shows the relation between area-averaged film cooling effectiveness on the lining wall with the blowing ratio for three tested effusion hole's configurations. For test cases of hole's aspect ratio of AR=0.5, an increase of blowing ratio from 0.25 to 0.5 obviously enhanced the film cooling effectiveness from 0.26 to 0.46, further increasing the blowing ratio to 0.75 and 1.0 resulted in a descending in film cooling effectiveness. When the aspect ratio of holes is changed from AR=0.5 to AR=1.0, the predicted results shows that the area-averaged film cooling effectiveness over the lining walls seems kept a reverse fashion with the blowing ratios. It means that the area-averaged film cooling effectiveness is not positive augmented with the blowing ration. Similar statement can be applied for the test cases with highest aspect ratio of holes in present work. Generally, the test parameter combination of AR=0.5 and BR=0.75 results in the highest area averaged film cooling effectiveness among all tested cases conducted with adiabatic heat transfer model.

Conjugate Heat Transfer Model

Typically, analysis on three temperature thermal-flow problems obtains only the convective heat transfer coefficient and the adiabatic film cooling effectiveness distribution information by ignore of the solid heat diffusion process, thus being insufficient in the designing stage. Aforementioned approach is widely applied in the cooling design of gas turbine blades due to a thin layer of thermal barrier coating material is covered on the outer surface. However, this approach is rarely applied for the lining wall in the combustion chamber to be an effective protection mode. Besides, the effusion cooling mode combines of internal impingement cooling method and the external film cooling method to keep both sides of the wall temperature below of melting temperature. Therefore, performing numerical simulations with Conjugate Heat Transfer Model, or CHT, is necessary and the results can be used to coincide with actual engineering problems. In present work, the selection of solid metal for lining wall is stainless steel with a thickness of 3.5mm.

Figure 8(a)~(c) displayed the predicted static temperature over the lining wall with three aspect ratios of elliptic hole under four difference blowing ratios. As can be observed from Figure 8(a)~(c), due to taking solid heat conduction into consideration, the wall temperature is more evenly distributed compared to that of Figure 4 under the same hole geometric and blowing ratio. Besides, the results from CHT model reveal that the effusion cooling mode and can provide superior cooling protection. This is due to the hybrid effect of internal cooling and external cooling on removing the heat flux from dual sides of lining wall.

Figure 9 shows the predicted relationship between the area-averaged η values with the blowing ratio for three different aspect ratios of elliptic hole with the CHT model. The results with adiabatic heat transfer model are also plotted for comparison. The predicted results with CHT model are always higher than those with adiabatic heat transfer model under the same blowing ratio. When the aspect ratio of elliptic hole is fixed, the area-averaged η values gradually increased with the increasing of the blowing ratio as expected. Comparison of results for three different tested configuration of effusion cooling holes, it is obvious that the elliptic hole with aspect ratio of 0.5 is the best selection under present tested operational conditions.

CONCLUSION

In this study, three different length-width aspect ratio of jet hole's structures employed to effusion cooling protection of lining wall of a circular combustion chamber under different blowing ratios are modelled. The three-dimensional thermal-flow field structure of multiple jets with the adiabatic heat transfer model and conjugate heat transfer model were presented and compared. For adiabatic heat transfer modelling, the predicted effusion cooling protection effect decreases with the increase in the blowing ratio due to the reliance of the combustion chamber wall on only external film cooling characteristics. As the blowing ratio increases, this penetration effect of jets is more significant and results in the descending in area-averaged film cooling effectiveness. However, the effusion cooling mode, if considered with conjugate heat transfer modeling, it was observed that in the testing limits of the jet hole length-width aspect ratios, when the blowing ratio increased to 0.5 from 0.25, the cooling protection effect greatly increases. However, if further increased to 0.75 or 1.0, the effect was not significant. To jet hole configurations, in general, AR=0.5 shows the best performance for the investigated cases in this work.

ACKNOWLEDGEMENTS

The authors deeply appreciated for the financial support from National Science Council of Taiwan under grant numbers of *NSC 93-2212-E-014-010* and *NSC 98-2218-E-344-001-MY2.*

REFERENCES

MYUNG, K.S. and ISSAM, M., (2006), "Experimental and numerical investigation of single-phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme," *Int. J. Heat Mass Transfer*, Vol. 49, pp. 682-694.

ROGER, M. and BUCK, R., (2005), "Numerical and experimental investigation of a multiple air jet cooling system for application in a solar thermal receiver," *ASME J. Heat Transfer*, Vol. 127, No.8, pp. 863-876.

HAN, J.C., (2004), "Recent studies in turbine blade cooling," *Int. J. Rotating Machinery*, Vol. 10, No. 6, pp. 443-457.

FACCHINI, B. and SURACE, M., (2006), "Impingement cooling for modern combustors: experimental analysis of heat transfer and effectiveness," *Experiments in Fluids*, Vol. 40, No.4, pp. 601-611.

BECKO Y., (1976), "Impingement cooling – a review," Von Karman Institute for Fluid Dynamic, Lecture Series 83, Turbine Blade Cooling, 1976.

NEIL, Z. and NOAM, L., (2005), "Impingement heat transfer: correlations and numerical modeling," *ASME J. Heat Transfer*, Vol. 127, No.5, pp. 544-552.

HUBER A.M. and VISKANTA, R., (1994), "Effect of jet-jet spacing on convective heat transfer to confined, impinging arrays of axisymmetric air jets," *Int. J. Heat Mass Transfer*, Vol. 37, No.18, pp. 2859-2869.

HUBER, A.M. and VISKANTA, R., (1994), "Comparison of convective heat transfer to perimeter and center jet in a confined, impinging arrays of axisymmetric air jets," *Int. J. Heat Mass Transfer*, Vol. 37, No.18, pp. 3025-3030.

LEE, J. and LEE, S.J., (2000), "The effect of nozzle aspect ratio on stagnation characteristics of elliptic impinging jet," *Int. J. Heat Mass Transfer*, Vol. 43, pp. 555-575.

COUSSIRANT, M., BEECK, J. van, MESTRES, M., EGUSQUIZA, E., BUCHLIN, J.M., and VALERO, C., (2005), "Computational fluid dynamics modeling of impinging gas-jet systems: II. application to an industrial cooling system device," *ASME J. Heat Transfer*, Vol. 127, pp. 704-713.

LANYAA, A.E. and DEBORAH, A.K., (2005), "Numerical investigation of jet impingement cooling with cross flow- comparison of YANG-SHIH and standard k-ε turbulence models," *Numerical Heat Transfer, Part A*, Vol. 47, pp. 441-469.

LI, L., LIU, T., and PENG, X.F., (2005), "Flow characteristics in an annular burner with fully film cooling," *Applied Thermal Engineering*, Vol. 25, Issues 17-18, pp. 3013-3024.

LI, L., PENG, X.F., and LIU, T., (2006), "Combustion and cooling performance in an aero-engine annular combustor," *Applied Thermal Engineering*, Vol. 26, Issue 16, pp. 1771-1779.

GORDON, R. and LEVY, Y., (2005), "Optimization of wall cooling in gas turbine combustor through three-dimensional numerical simulation," *ASME J. Heat Transfer*, Vol. 127, No. 4, pp. 704-723.

HU, Y. and JI, H., (2004), "Numerical study of the effect of blowing angle on cooling effectiveness of an effusion cooling," *Proceedings of ASME Turbo Expo*, Vienna, Austria, pp. 1-8.

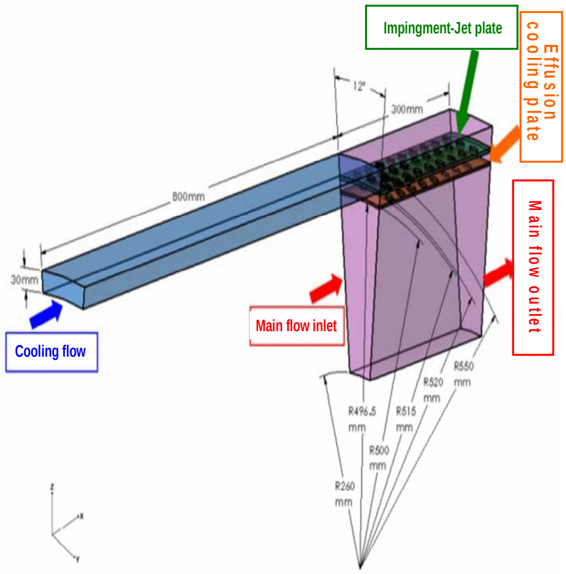
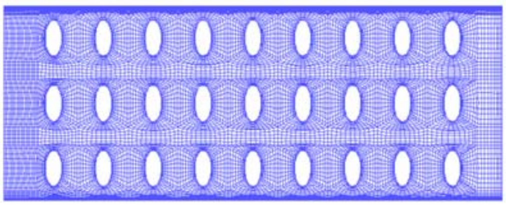
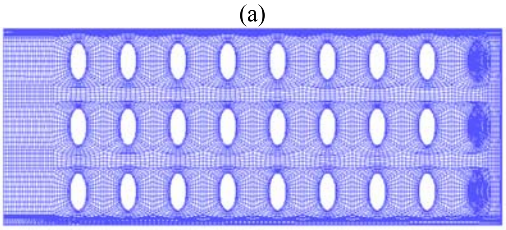


Figure 1: A schematic view of the computational domain for effusion cooling lining wall in a circular combustion chamber.



(a)



(b)

Figure 2: Surface grid distribution for elliptic hole with aspect ratio (AR) of 0.5 on ; (a) impingement jet hole plate; (b) lining wall.

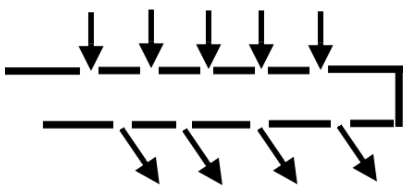


Figure 3: Schematic view of the effusion cooling model.

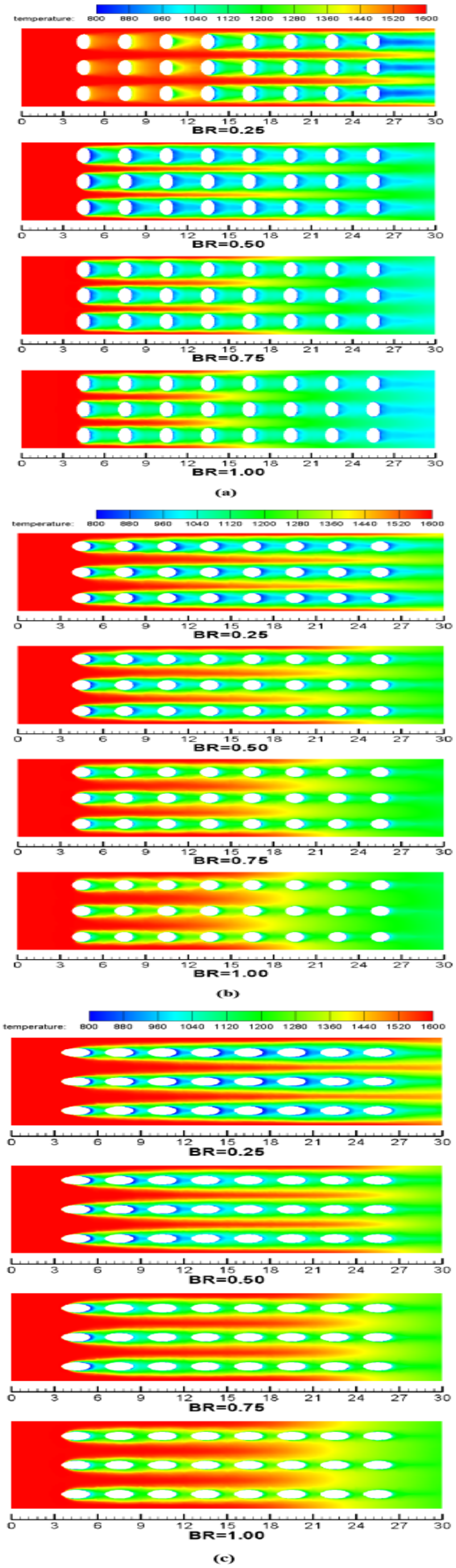


Figure 4: Predicted wall temperature distributions on lining wall under different blowing ratios with adiabatic heat transfer model: (a) AR=0.5, (b) AR=1, (c) AR=2.

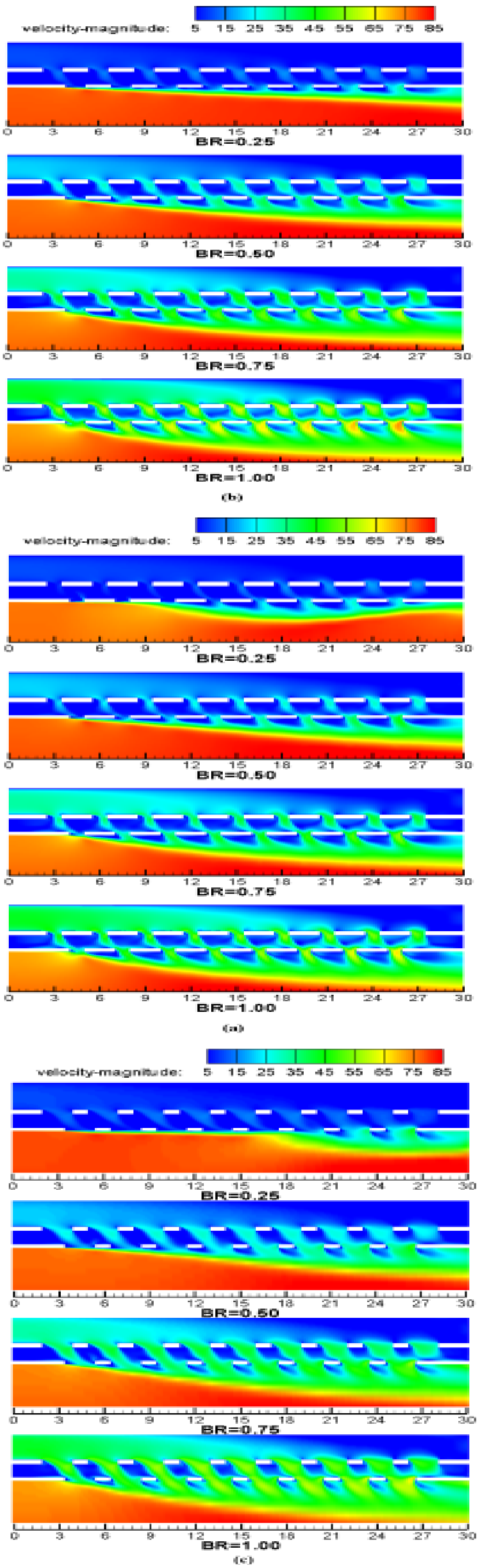
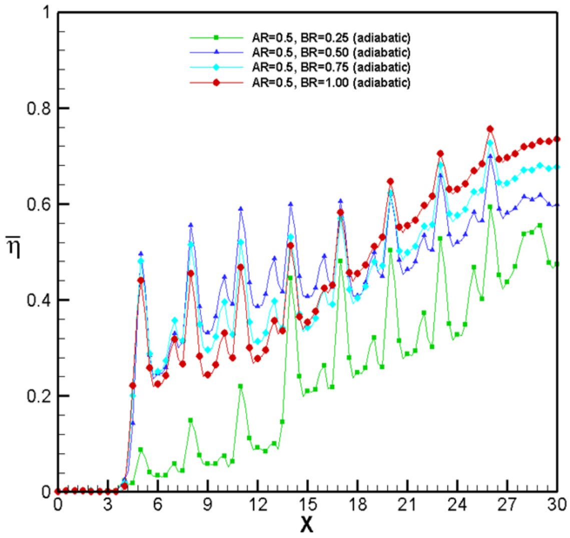
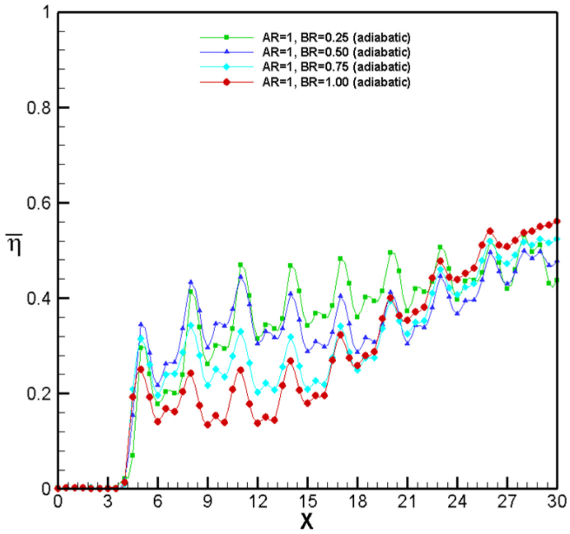


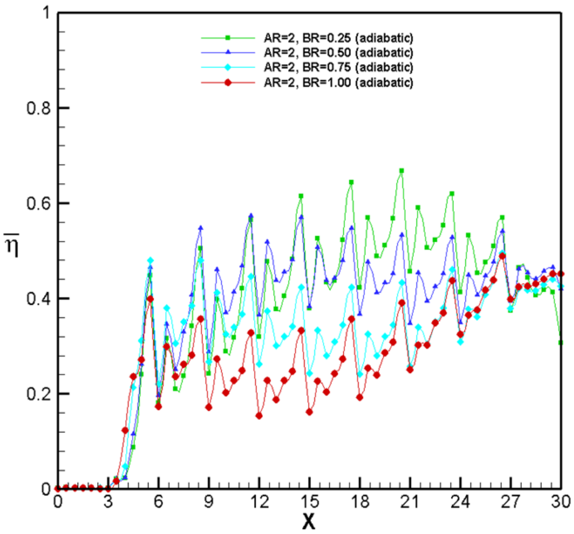
Figure 5: Velocity contours in the X-Z section along middle row of effusion holes under different blowing ratios with adiabatic heat transfer model; (a) AR=0.5, (b) AR=1, (c) AR=2.



(a)



(b)



(c)

Figure 6: Predicted laterally averaged η distribution on the lining wall along the stream-wise direction under different blowing ratios; (a) AR=0.5, (b) AR=1, (c) AR=2.

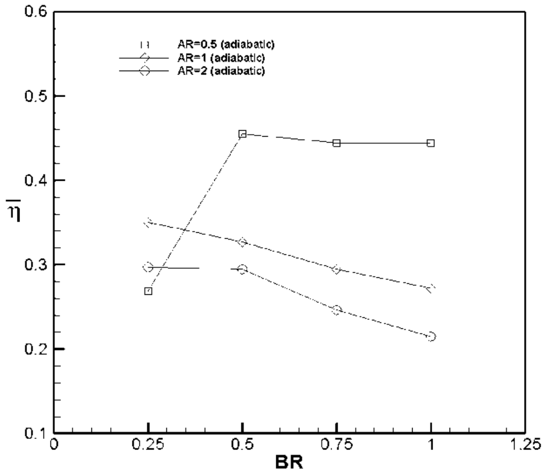
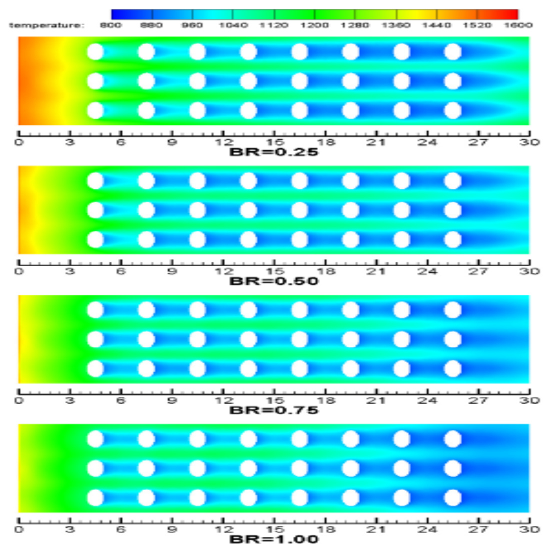
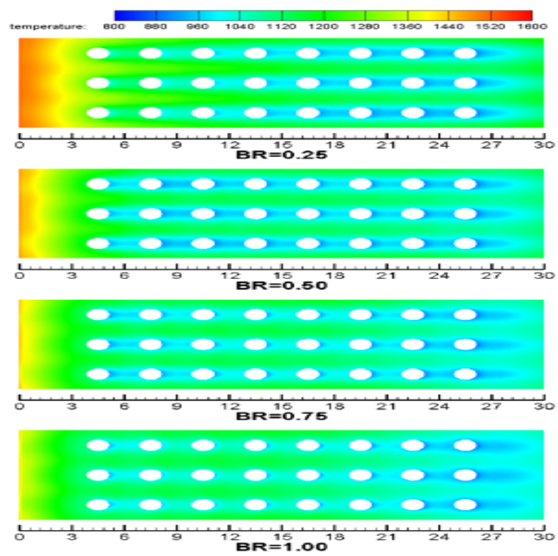


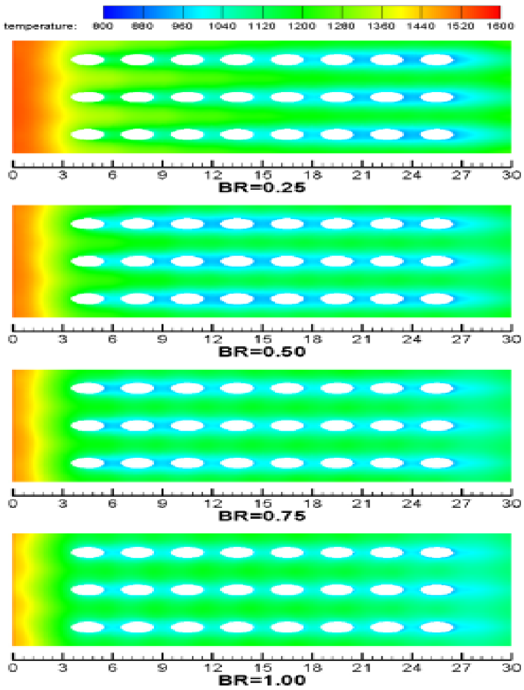
Figure 7: Relationship between area-averaged η values and blowing ratios for three tested elliptic hole's aspect ratios (adiabatic heat transfer model).



(a)



(b)



(c)

Figure 8: Predicted wall temperature distributions on lining wall under different blowing ratios with conjugate heat transfer model: (a) AR=0.5, (b) AR=1, (c) AR=2

.

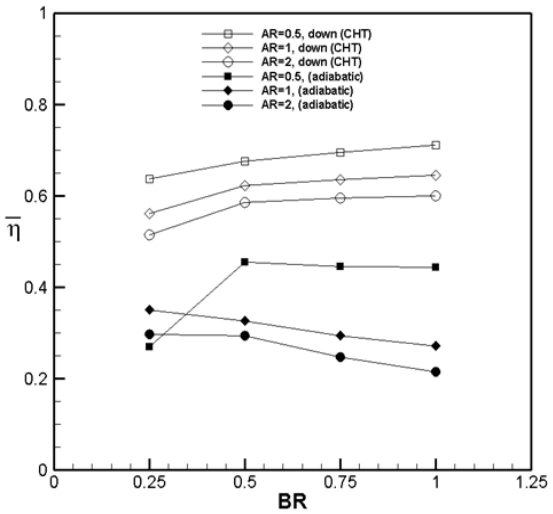


Figure 9: Relationship between area-averaged η values and blowing ratios for three tested elliptic holes (conjugate heat transfer model).