Penerbit Akademia Baru

Journal of Advanced Research Design

Journal homepage:
https://akademiabaru.com/submit/index.php/ard
ISSN: 2289-7984

Analysis of External Film Cooling Effectiveness using Compound Cooling Holes

Muhammad Faizal Shaari[1], Nor Azwadi Che Sidik[1,*]

[1] Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

ARTICLE INFO

Article history:
Received 22 September 2021
Received in revised form 27 January 2022
Accepted 11 February 2022
Available online 21 February 2022

Keywords:
Gas turbine; turbine technology; material properties

ABSTRACT

By using the well-known Brayton cycle, great turbine industries are always in a revolution phase to further improve the nowadays technology. One of the ways to improve turbine performance is by increasing the turbine inlet temperature. However, the turbine inlet temperature increment creates a harsh environment for the components of the combustor. This is due to the high temperature of air exceeding the melting point of material inside the combustor. So the proper cooling technique needs to be implemented. Compound cooling holes are a useful way to this achievement. This study was accomplished to investigate the effects of cylindrical and row compound cooling holes with alignment angles of 30 degrees and beta of 0,10,20,30 and 40 degrees. This model was simulated and analyzed with a commercial finite volume package ANSYS FLUENT 14.0. Analysis of the findings of the study showed that using the row compound cooling holes of 30 degrees provides better cooling performance than other degrees. The film cooling effectiveness is also higher when blowing ratio is 1.5 compared to 1.

1. Introduction

Gas turbine technology has played a vital role in today's industrial technology as this technology has continued to be in demand in today's world. They are commonly used in the aircraft industry and also as the power generation industry. The turbine technology has also been used as a power plant for ships [1,2].

The developments of turbine industries continue to be researched by engineer all over the world. One of the fundamentals that has been pursued is striving for higher engine efficiencies and power-to-weight ratio. The basic way to approach this problem is by using a Brayton. Based on the Brayton cycle, to increase the efficiency of the turbine, the inlet temperature of the turbine inlet temperature should be increased as state by Salimi *et al.,* [3.4].

In the 1960s, material properties limited gas turbine firing temperature and turbine blade temperature to around 1100°C. In the advanced gas turbine today, the temperature of the turbine inlet can be high as 1500°C to 2000°C [13,14]. The pattern on the turbine inlet temperature will continue to increase in future.

* *Corresponding author.*
E-mail address: azwadi@utm.my

However, the operating temperature as high as that cannot be used based on existing material, [5] Lu (2007). The higher temperature also will give a negative effect of thermal stresses to material of turbine [6,7,8] Han *et al*., (2000). Therefore, it is highly unlikely that the material that used in inlet turbine can withstand the very high temperature from inlet liquid.

In general, gas turbine cooling is achieved by bleeding a part of cool air from the compressor and used it to lower the temperature of the fluid stream inside the combustor chamber [9,10]. The cooling air will flow through cooling passage inside the combustor chamber. These passages will specifically design to improve the efficiency of cooling inside the combustor chamber.

The high temperature of the hot air which flows into the inlet of combustor of the turbine will damage the condition of the end wall of the combustion chamber [11,12]. The film cooling has been applied to increase the effectiveness of film cooling of the turbine [15]. The coolant air will come from the bleed off air from compressor. It is important because the lower turbine efficiency result of more air use for cooling and lead to lowering thrust power of the engine. The aim of this study is to find out the effects of compound cooling holes on the thermal characteristics of external film cooling on the surface. The study also will investigate the analysis of temperature contour and film cooling effectiveness based on film cooling of compound cooling hole. The study also will cover about the relation of blowing ratio to the cooling performance of film cooling of turbine engine.

2. Methodology

The basic governing equations for Computational Fluid Dynamics technique are continuity equation and momentum equation. Energy equation is utilized for the temperature. The equation was derived for the rate of increase of energy of a fluid particle per unit volume, which is given by Eq. (1),

$$\bar{u}_i \frac{\partial \theta}{\partial x_i} + \frac{\partial}{\partial x_i} u_i \theta = 0 \tag{1}$$

2.1 CFD Process

The process to start simulation using CFD program have involved several steps of processes. For the first process, the design model geometry was build and was mesh before the boundary condition can be set up. Next the operating condition is defined before the solving and iteration process took place. One the simulation program is complete; the results can be obtained in form of contour, vector and graph. Consequently, analysis is done on the results obtained and finally the findings are compared or validated against established journals. The steps in CFD process was shown in Figure 1.

Journal of Advanced Research Design
Volume 89, Issue 1 (2022) 1-14

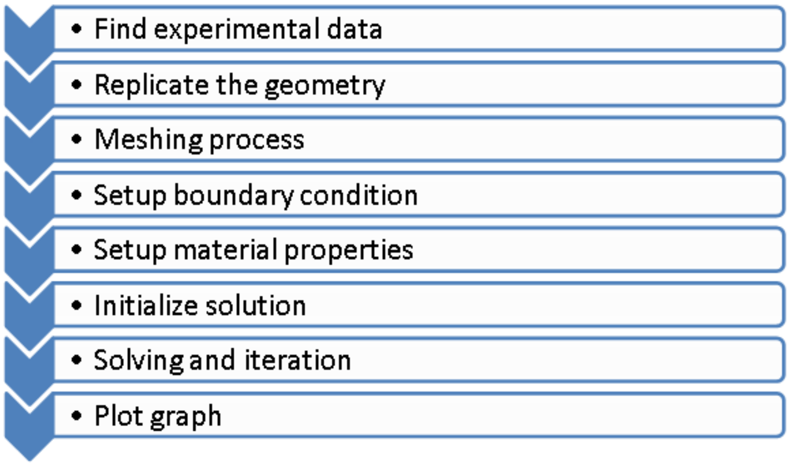


Fig. 1. Step in CFD process

2.2 Geometry Modelling

The simple combustor model was used to help gain better understanding on the proposed problem. The geometry size and measurement is shown as Figure 2-4, while the details specification of the combustor simulator is shown in Table 1-4.

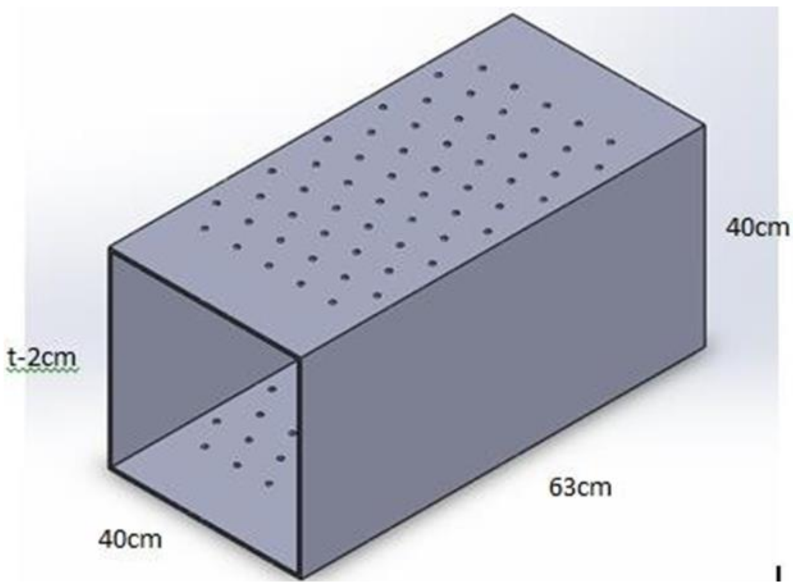


Fig. 2. Combustor model for simulator

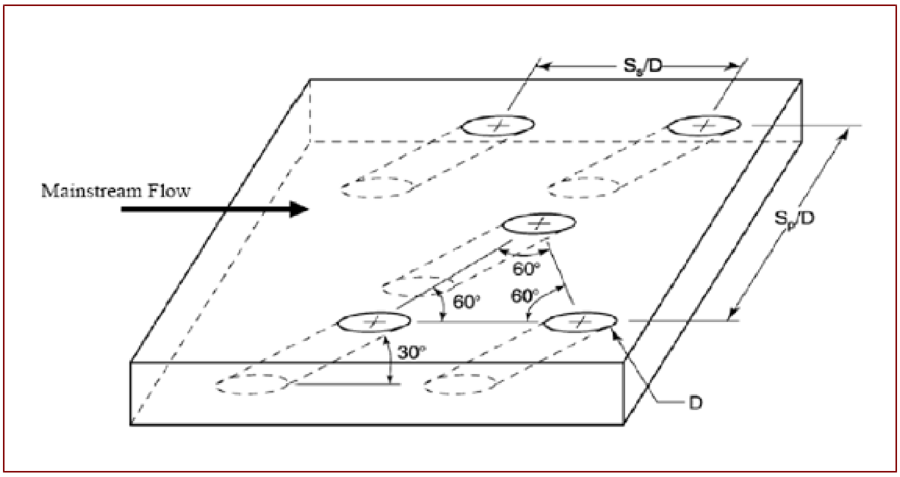


Fig. 3. The compound cooling hole arrangement

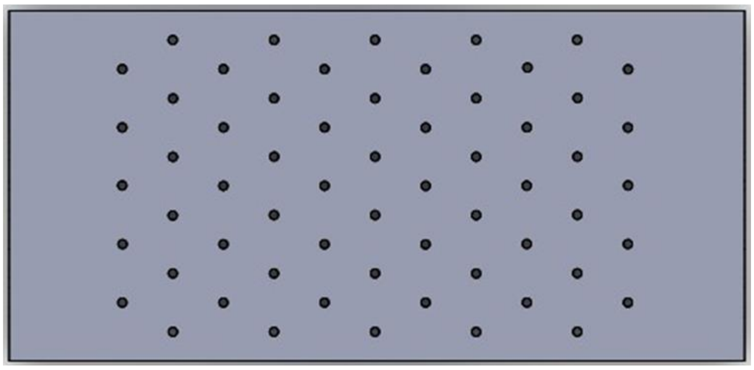


Fig. 4. Cooling holes arrangement on panel

Table 1
Overall Cooling Holes Specification

Hole Length	2.5 cm
Hole Diameter	0.8 cm
Angle of Triangle Hole	60 deg

Table 2
Overall Simulation Specifications

Case	Inclination Angle(α)	Orientation Angle (β)
A	0	0
B	30	0
C	30	10
D	30	20
E	30	30
F	30	40

Table 3
Operating Condition for Main flow

Main flow Pressure (Kilo Pascal)	98.82
Main flow Temperature (Kelvin)	332
Main flow Velocity(m/s)	1.62
Main flow Density (Kg/m2)	1.22

Table 4
Inlet Flow Conditions

Parameter	Location	Combustor Simulator
Density (Kg/m2)	Cooling panels	1.22
Velocity(m/s) M=1	Cooling panels	1.62
Velocity(m/s) M=1.5	Cooling panels	2.43
Temperature (Kelvin)	Cooling panels	295.5

Firstly, software of Ansys 14.0 was used to build the simulated model. Then the entire build model was meshed. One of the important aspects in meshing is number of nodes in each cell. It is important in order to get the solution of the flow directly. To get the more accurate result high number of meshed is used. The geometry of model that will be used is build using Workbench14.0. The geometry of the model in workbench is shown below in Figure 5.

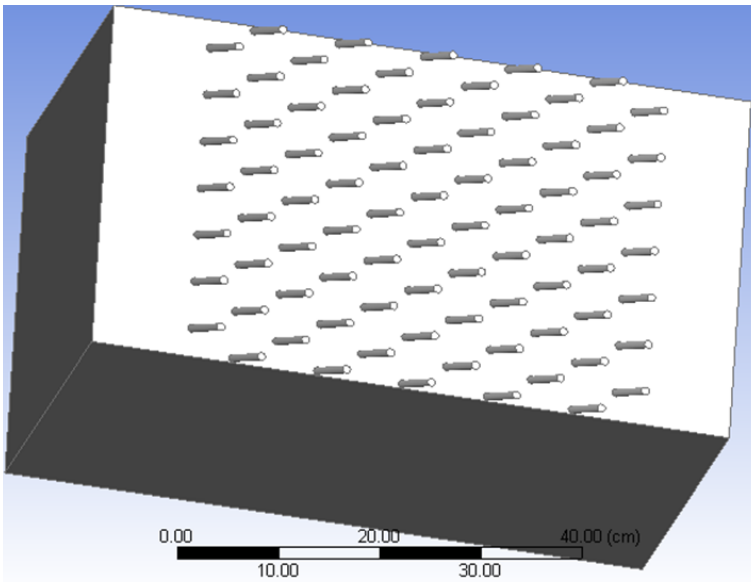


Fig. 5. The combustor model geometry in Workbench

2.3 Meshing

At first, the simulated model was constructed with Ansys 14.0 software. Then all available surfaces were meshed. After that, the meshed model was derived and was used by fluent software. This created model was concerned with the geometry of the cooling holes and the primary geometry of the combustion chamber. Then, all surfaces were synthesized together and create a complete geometry of combustor simulator. While, the baseline case, compound cooling holes with alignment angles of 0 degree and 30 degree were completely symmetric along the x-y and x-z plane, the compound cooling holes with alignment angles of 0 degree to 40 degree, were just symmetric along the x-y plane. The finishing model with mesh process is shown in Figure 6.

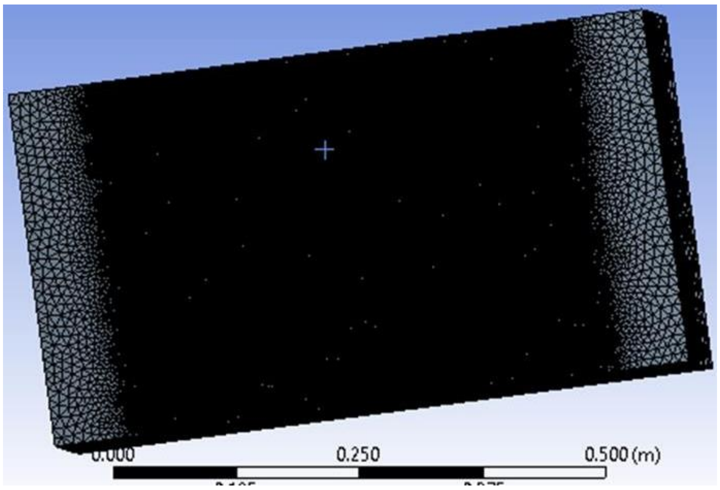


Fig. 6. The finishing meshing model

2.4 Boundary Condition

Boundary conditions specify the flow and thermal variables on the boundaries of the physical model. They are, therefore, a critical component of the fluent simulations and it is important that they are specified appropriately. Figure 7 shows the selected boundary conditions that used for the current case of study.

Mass flow boundary conditions can be used in the Fluent to provide a prescribed mass flow rate or mass flux distribution at an inlet. Specifying the mass flux permits the total pressure to vary in response to the interior solution. The function of the mass flow inlet boundary condition is defined for the fluent software in the form of free transition from the stagnation condition to the inlet condition.

Pressure outlet boundary conditions require the specification of a static pressure at the outlet boundary. The value of the specified static pressure is used only while the flow is subsonic. All other flow quantities are extrapolated from the interior. A set of back flow conditions is also specified should the flow reverse direction at the pressure outlet boundary during the solution process. Several options in Fluent exist, where a radial equilibrium outlet boundary condition can be used and a target mass flow rate for pressure outlets can be specified.

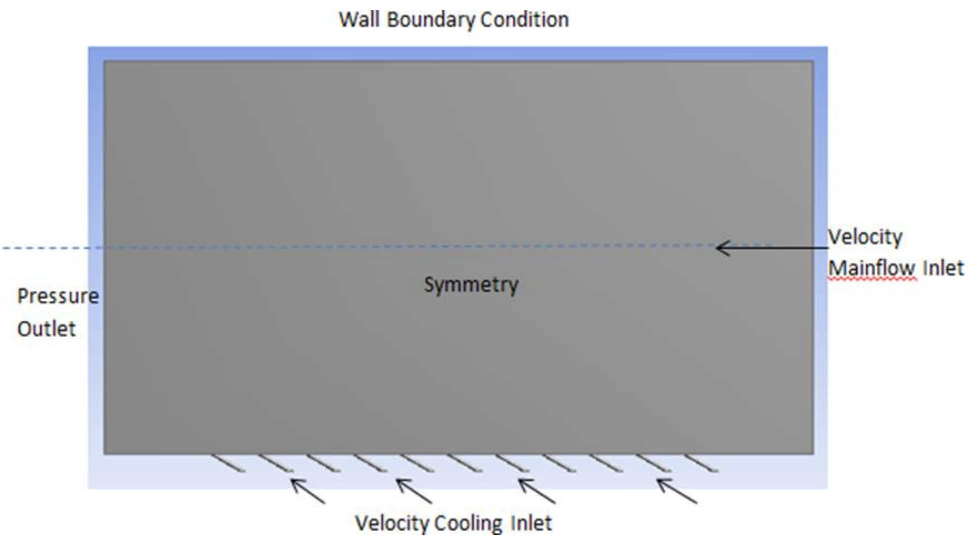


Fig. 7. Applied boundary condition on model

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but at an initial velocity component in terms of the translational or rotational motion of the wall boundary, or model is specified by the shear specification. For the definition of the current case of study and according to the condition of the model it is required to consider the thermal boundary condition. In these situations, it is needed to define the thermal boundary condition near the sided walls.

2.5 Measure Quantity

To understand the thermal and flow field results, non-dimensional quantities should be introduced. At the first step, film cooling effectiveness is defined. This non- dimensional quantity is given as Eq. (2),

$$\eta = \frac{T_\infty - T}{T_\infty - T_c} \tag{1}$$

where,
$\eta =$ *the cooling effectiveness*
$T =$ *the local temperature*
$T_\infty =$ *temperature of mainstream*
$T_C =$ *temperature of coolant*

2.6 Fluent Setup

For the simulation process, Ansys fluent was used to conduct the simulation for the model. Firstly, the meshing geometry will be loaded using Fluent. For the Fluent starting setup, the double precision method was used. The processing option chosen is parallel by using 4 numbers of processes as shown in Figure 8. This was done to speed up the time for the simulation to take place.

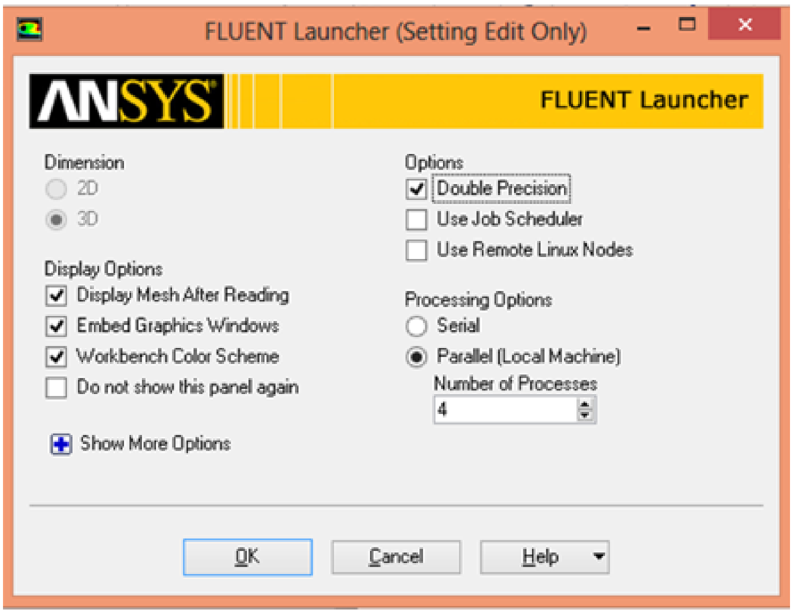


Fig. 8. The fluent launcher setting

For the general setup, the pressure-based density is used as shown in Figure 9. For this type of solver velocity field is obtained from the momentum equation. Mass conservation is achieved by solving a pressure correction equation. The pressure-based solver (segregated) is applicable for a wide range of flow regimes from low speed incompressible flow to high-speed compressible flow. It also allows flexibility in the solution procedure – damping of all equations separately.

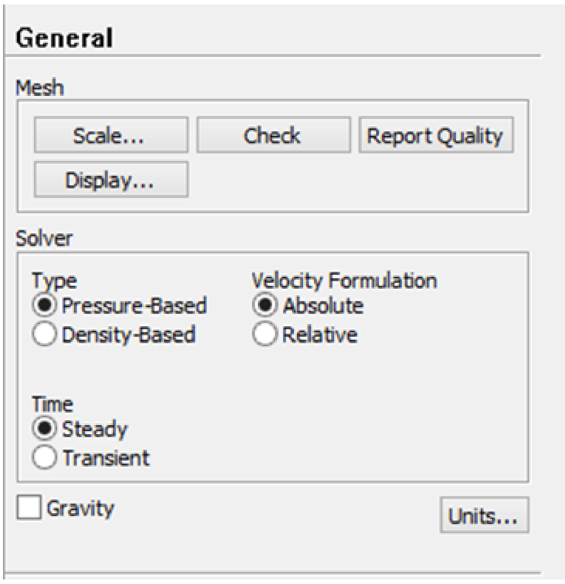


Fig. 9. The solver setting for fluent

As the simulation is to study the effect of different compound angle model and the effect of the different blowing ratio to the cooling performance, the fluent will be setup accordingly. To study out about the effect of angle of orientation angle, the different geometry was built and mesh in first part of the simulation. For the effect of blowing ratio, the variable that will be manipulated is the velocity of the air entering the cooling holes. This situation can be simulated by changing the value of the velocity of cooling inlet in the boundary condition setup earlier during the meshing process. The manipulated velocity was shown in Figure 10 to 14.

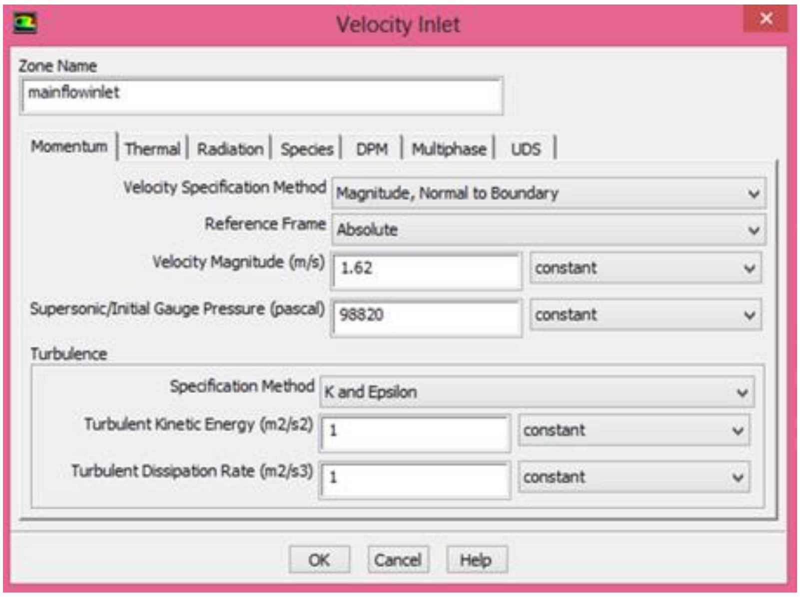


Fig. 10. Velocity profile for mainflow inlet

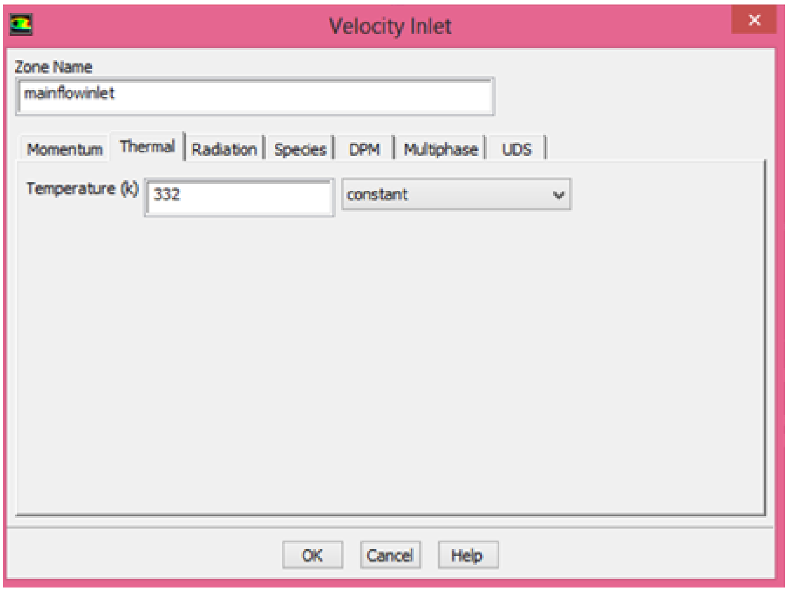


Fig. 11. Temperature profile for mainflow inlet

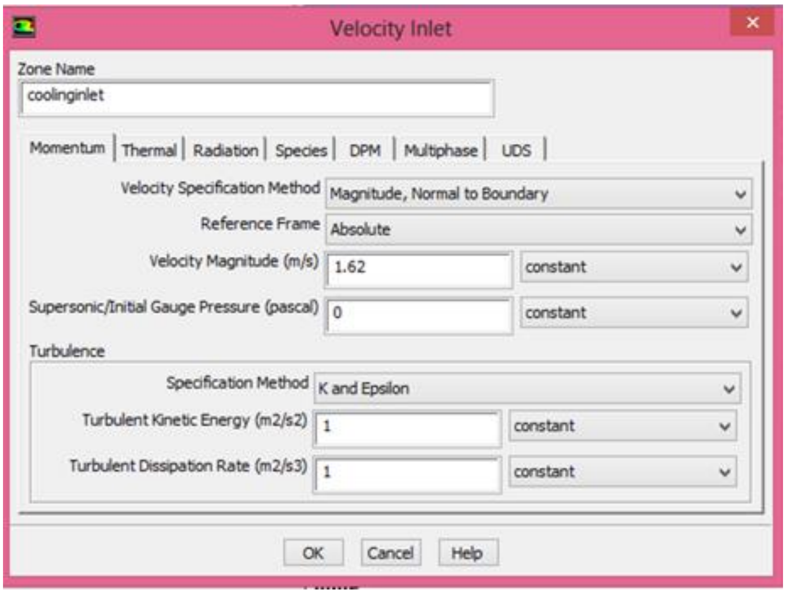


Fig. 12. Velocity profile for cooling inlet for M=1

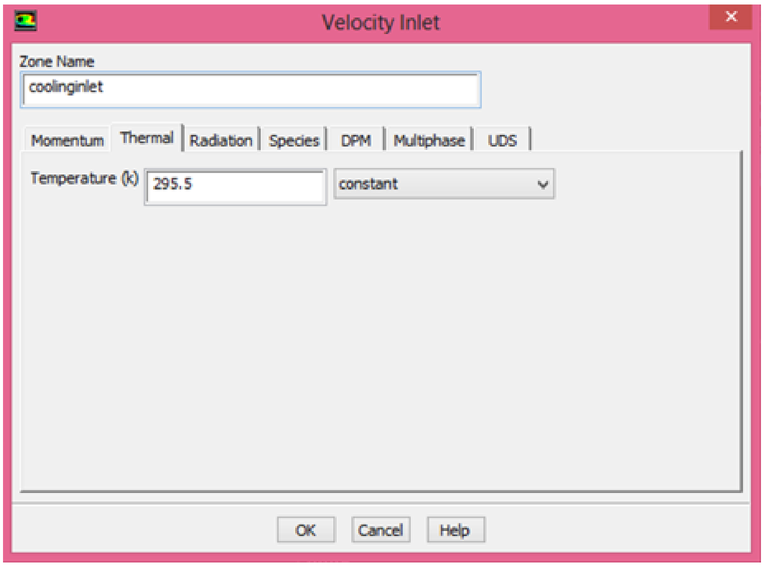


Fig. 13. Temperature profile for cooling inlet

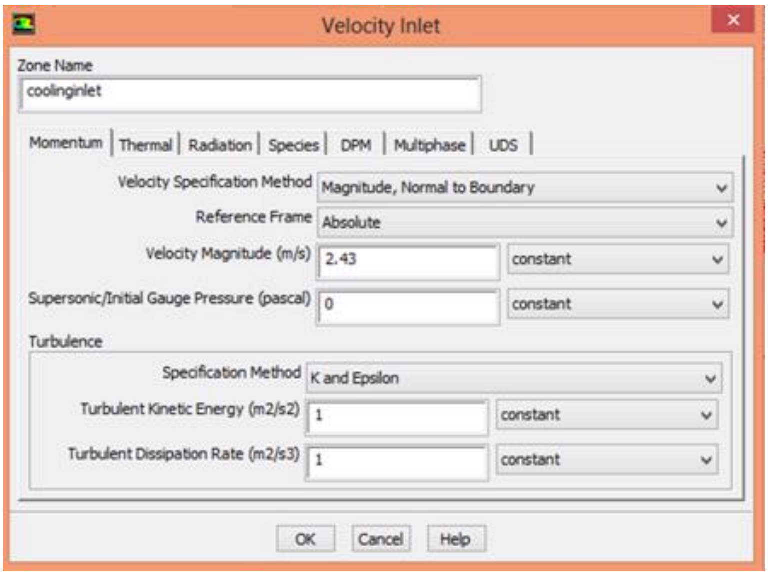


Fig. 14. Velocity profile for cooling inlet for M=1.5

3. Results and Discussion

The contour of temperature represents the distribution values of temperature across selected faces. For the cooling holes specification, the angle of the coolant air that enters the combustor will affect the coverage of the cool air around the surface of the wall.

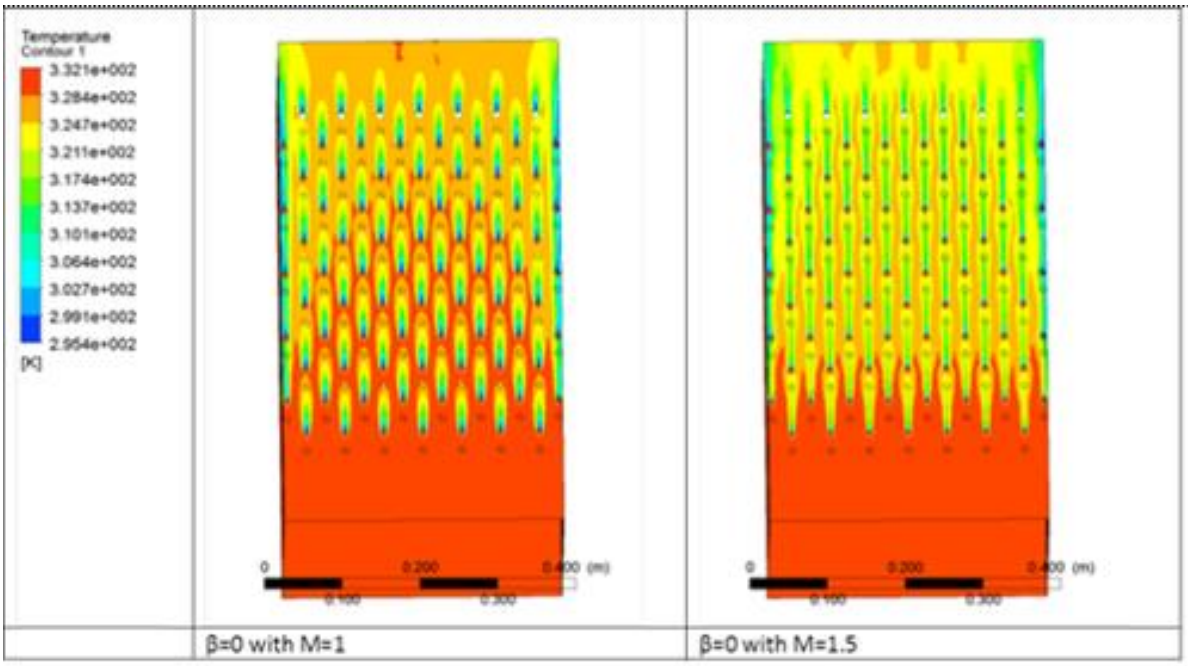


Fig. 15. Contour temperature for β =0

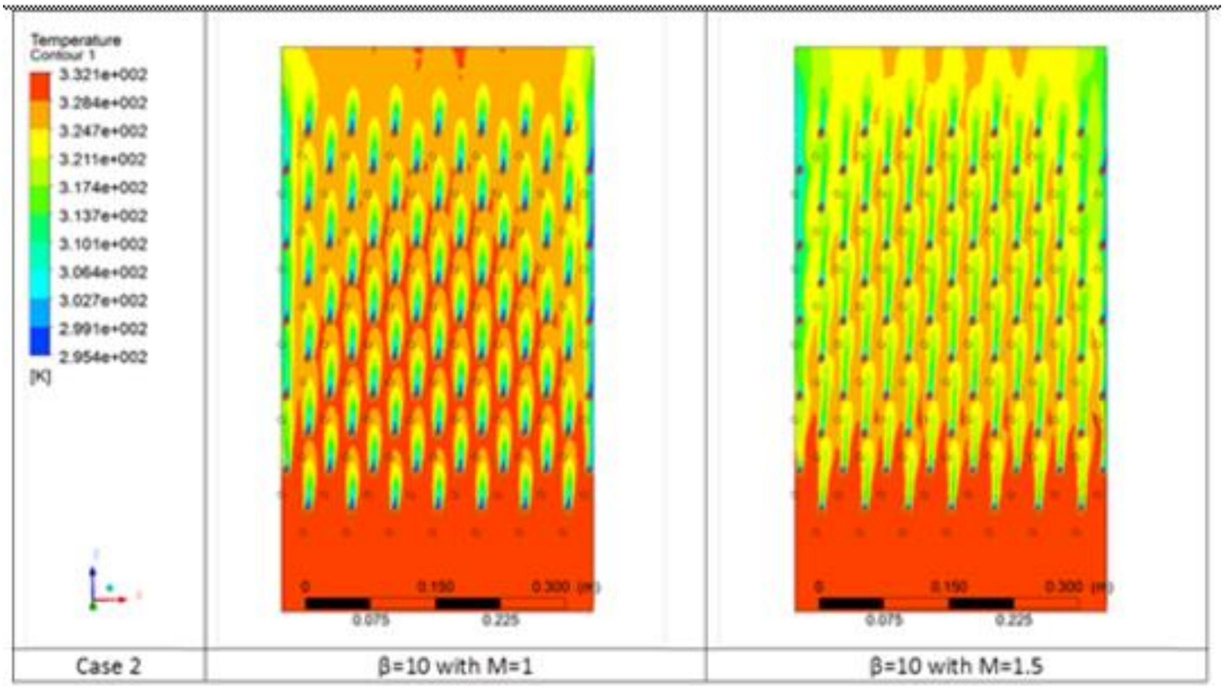
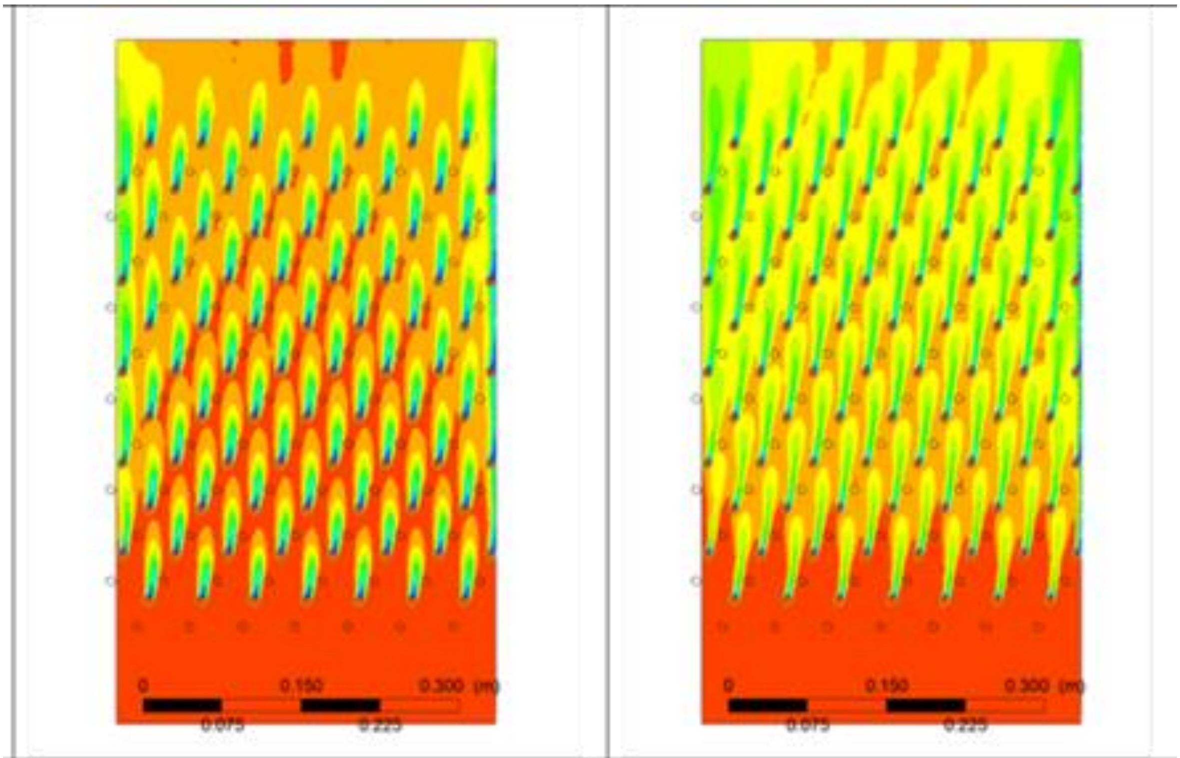
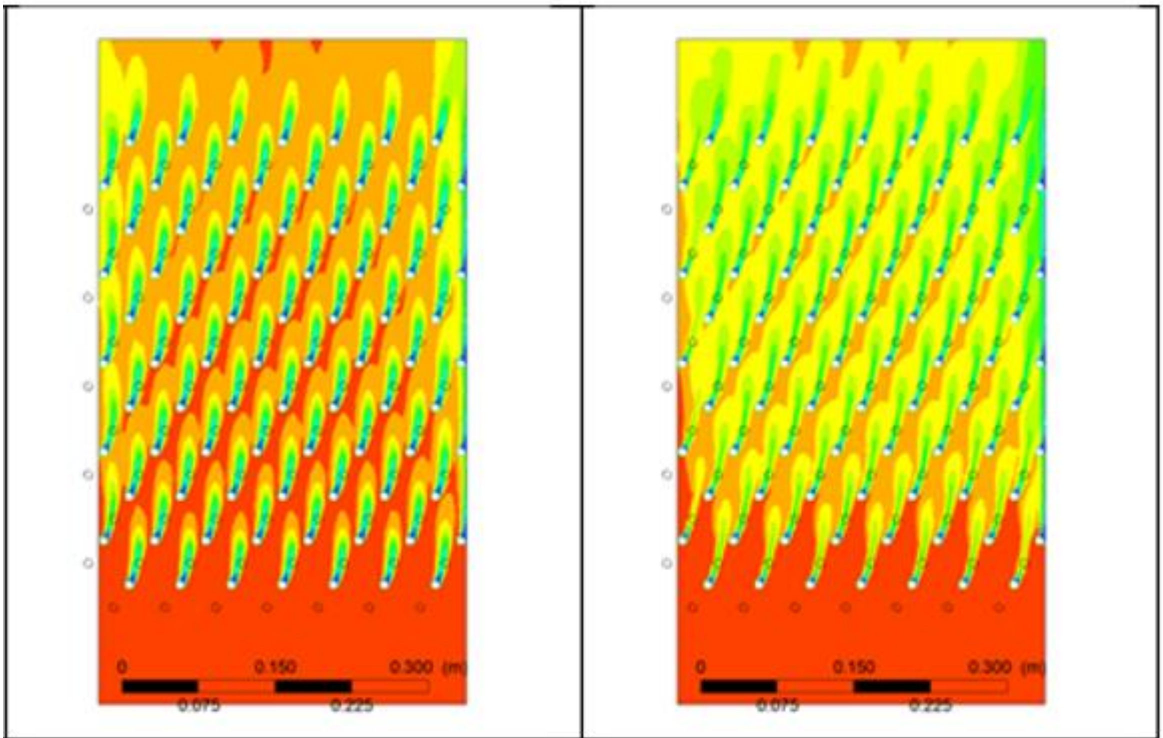


Fig. 16. Contour temperature for β =10



β=20 with M=1	β=20 with M=1.5

Fig. 17. Contour temperature for β =20



β=30 with M=1	β=30 with M=1.5

Fig. 18. Contour temperature for β =30

Journal of Advanced Research Design
Volume 89, Issue 1 (2022) 1-14

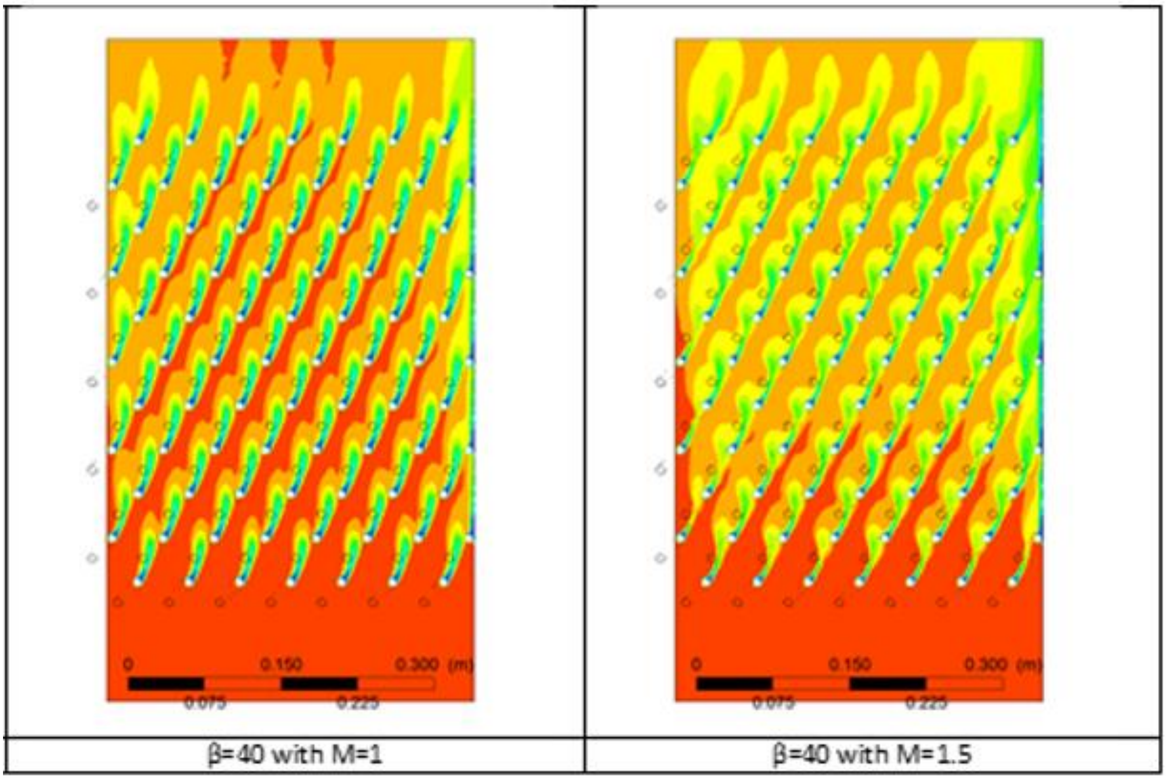


Fig. 19. Contour temperature for β =40

From the result (Figure 15-19) it shown that the compound cooling hole has temperature coverage area based on temperature contour coverage of the wall compare to conventional cooling hole. From the contour, it can be seen that the cool air that enters the combustor chamber not only provides a protective layer for the angle of injection but also covers the surrounding area. This shows that compound cooling holes provide a better layer of protectiveness compared to one direction of conventional cooling holes. For the different orientation value of β**,** it's shown that the β with 30 degrees cover more area compared to other orientation angles for low blowing ratio. The effect if temperature coverage also increases as the value of the blowing ratio increases.

4. Conclusion

This research was done to get a better understanding of the film cooling effectiveness and the effects of row compound cooling holes with a range of alignment angles of 0 degrees, 10 degrees, 20 degrees, 30 degrees, and 40 degrees to the combustor wall of a gas turbine engine. This data will allow designers to better compare different cooling hole geometries to increase the cooling performance for the external film cooling. An optimized design of cooling holes will help to maximize the effectiveness of cooling along the combustor end wall surface and prevent premature wear in this area.

Film cooling effectiveness increment was performed by coolant velocity ratio enhancement. As the blowing ratio increased, we can see from the temperature contour that it helped to protect the surface of the wall better. Compound cooling holes with an alignment of 30 degrees give better cooling distribution to the wall of the combustor compared to the other study alignment. Using staggered rows arrangement of compound angle injection cooling holes tends to provide better and more uniform cooling than that obtained when inline compound angle holes were used. At high blowing ratios and high angles, the compound angle has a noticeable effect on cooling on the surface of the wall.

Acknowledgement
This research was not funded by any grant.

References

[1] Ammar, Nader R., and N. F. S. H. Alshammari. "Overview of the green hydrogen applications in marine power plants onboard ships." *Int. J. Multidiscip. Curr. Res.* 6, no. 01 (2018). https://doi.org/10.14741/ijmcr.v6i01.10912
[2] Bolbot, Victor, Nikoletta L. Trivyza, Gerasimos Theotokatos, Evangelos Boulougouris, Athanasios Rentizelas, and Dracos Vassalos. "Cruise ships power plant optimisation and comparative analysis." *Energy* 196 (2020): 117061. https://doi.org/10.1016/j.energy.2020.117061
[3] Salimi, Shahin. "Numerical Analysis of Film Cooling Effectiveness Using Compound Cooling Holes at the End of Gas Turbine Engine Combustor." PhD diss., Universiti Teknologi Malaysia, 2014.
[4] Salim, Mohammad Saad, Muhammad Saeed, and Man-Hoe Kim. "Performance analysis of the supercritical carbon dioxide re-compression brayton cycle." *Applied Sciences* 10, no. 3 (2020): 1129. https://doi.org/10.3390/app10031129
[5] Lu, Yiping. *Effect of hole configurations on film cooling from cylindrical inclined holes for the application to gas turbine blades*. Louisiana State University and Agricultural & Mechanical College, 2007.
[6] Han, Je-Chin, Sandip Dutta, and Srinath Ekkad. *Gas turbine heat transfer and cooling technology*. CRC press, 2012. https://doi.org/10.1201/b13616
[7] Vaferi, Kourosh, Sahar Nekahi, Mohammad Vajdi, Farhad Sadegh Moghanlou, Mohammadreza Shokouhimehr, Amir Motallebzadeh, Jianjun Sha, and Mehdi Shahedi Asl. "Heat transfer, thermal stress and failure analyses in a TiB2 gas turbine stator blade." *Ceramics International* 45, no. 15 (2019): 19331-19339. https://doi.org/10.1016/j.ceramint.2019.06.184
[8] Kadir, Ali, Anwar O. Bég, Mohammed El Gendy, Tasveer A. Bég, and M. D. Shamshuddin. "Computational fluid dynamic and thermal stress analysis of coatings for high-temperature corrosion protection of aerospace gas turbine blades." *Heat Transfer—Asian Research* 48, no. 6 (2019): 2302-2328. https://doi.org/10.1002/htj.21493
[9] Baakeem, Saleh S., Jamel Orfi, and Hany Al-Ansary. "Performance improvement of gas turbine power plants by utilizing turbine inlet air-cooling (TIAC) technologies in Riyadh, Saudi Arabia." *Applied Thermal Engineering* 138 (2018): 417-432. https://doi.org/10.1016/j.applthermaleng.2018.04.018
[10] Hackney, Rick, Theoklis Nikolaidis, and Alvise Pellegrini. "A method for modelling compressor bleed in gas turbine analysis software." *Applied Thermal Engineering* 172 (2020): 115087. https://doi.org/10.1016/j.applthermaleng.2020.115087
[11] Funke, HH-W., Nils Beckmann, and Sylvester Abanteriba. "An overview on dry low NOx micromix combustor development for hydrogen-rich gas turbine applications." *International Journal of Hydrogen Energy* 44, no. 13 (2019): 6978-6990. https://doi.org/10.1016/j.ijhydene.2019.01.161
[12] Sharma, Saurabh, Paramvir Singh, Ashish Gupta, Arindrajit Chowdhury, Bhupendra Khandelwal, and Sudarshan Kumar. "Distributed combustion mode in a can-type gas turbine combustor–A numerical and experimental study." *Applied energy* 277 (2020): 115573. https://doi.org/10.1016/j.apenergy.2020.115573
[13] Han, Je-Chin. "Advanced cooling in gas turbines 2016 Max Jakob memorial award paper." *Journal of Heat Transfer* 140, no. 11 (2018): 113001. https://doi.org/10.1115/1.4039644
[14] Wang, Jiangjiang, Zepeng Han, and Zhimin Guan. "Hybrid solar-assisted combined cooling, heating, and power systems: A review." *Renewable and Sustainable Energy Reviews* 133 (2020): 110256. https://doi.org/10.1016/j.rser.2020.110256
[15] Zeng, Lingyu, Pingting Chen, Xueying Li, Jing Ren, and Hongde Jiang. "Influence of simplifications of blade in gas turbine on film cooling performance." *Applied Thermal Engineering* 128 (2018): 877-886. https://doi.org/10.1016/j.applthermaleng.2017.09.008