Jurnal
Teknologi

Full Paper

CFD ANALYSIS OF FIRST STAGE NOZZLE COOLING OPTIMIZATION IN POWER STATION GAS TURBINE

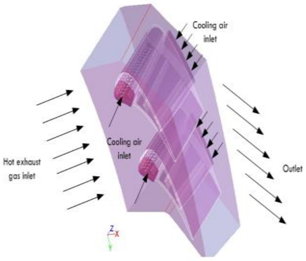
Hasril Hasini[a*], Siti Sarah Ain Fadhil[a], Mohd Nasharuddin Mohd Jaafar[a], Norhazwani Abdul Malek[a], Mohd. Haffis Ujir[b]

[a]Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000, Kajang, Selangor, Malaysia
[b]TNB Research, No 1, Jalan Ayer Itam, Kawasan Institusi Penyelidikan, 43000, Kajang, Selangor, Malaysia

Article history
Received
15 January 2015
Received in revised form
30 April 2015
Accepted
31 May 2015

*Corresponding author
hasril@uniten.edu.my

Graphical abstract



Abstract

Computational Fluid Dynamics analysis on First Stage Nozzle in full scale multi-stage power station gas turbine has been carried out. The main aim is to investigate the turbine thermal performance when cooling rate decreases at certain level. All calculations were executed using commercial CFD code, ANSYS FLUENT which is able to accurately predict the flow and conjugate heat transfer problem as demonstrated in this investigation. The modelling of gas turbine nozzle is assisted by geometric cloud data obtained from 3D scan. Preliminary calculation shows that at the given worst case scenario for, the maximum thermal stress experienced by the component is within the maximum yield strength of the nozzle material. However, the safety margin between the predicted stress and maximum allowable stress is very small.

Keywords: First stage nozzle, CFD, cooling optimization, gas turbine

1.0 INTRODUCTION

Power station gas turbine plays an important role in power generation industry and it is one of the most widely used prime movers in the energy sector. With the recent advancement in the gas turbine business, engineers and designers is pushing the limits of operating temperature to much higher temperature. This is made possible by efficient cooling system adopted into the design of gas turbine nozzle and blade. Combination of both internal and film cooling studies requires demanding task and CFD has been used as one of the tool to calculate both the internal and external parameters on blade cooling [1]. Studies on gas turbine performance are abundant in the open literature and they can be classified in wide range of categories. With respect to numerical heat transfer calculation, many studies are done focusing on film cooling [2-5] and internal cooling of blade. Attempt was also made to investigate the cooling effectiveness of blade in steam turbine to further explore the possibility of increasing working steam temperature [6-7]. In this paper, evaluation of the existing cooling system in first stage nozzle (FSN) is investigated using CFD. The preliminary investigation is required for further test and optimization on the existing system when the supply of cooling mass is reduced. In order to gain an insight into possible reduction in cooling air, a few cases are tested and the effect to the cooling pathlines and surface temperature distribution is analysed.

2.0 MODEL DESCRIPTIONS

In a gas turbine, the nozzle and blade sections cooling and sealing arrangement is shown in Figure 1. Hot combustion gas resulted from combustion in gas turbine exits through a transition piece before entering the first stage nozzle. At this section, the hot exhaust gas temperature is the highest and the temperature magnitude can be approximated in the range of

1500-1600 K. In the current gas turbine under investigation, the 1st stage nozzle (FSN), 1st stage shroud (FSS) and 1st stage blade (FSB) are cooled by the 17th extraction air. In addition, these components are also cooled by compressor discharge air. The 2nd stage nozzle (SSN) on the other hand, is cooled by the 13th stage extraction air while the cooling of 2nd stage blade (SSB) is provided by similar source as in FSN, FSS and FSB. For the 3rd stage nozzle (TSN), the cooling is provided by the 9th stage extraction air. The last turbine stage (3rd stage blade (TSB)) however was not provided with the cooling system due to the fact that the exhaust gas temperature reaching this stage is significantly lower than the first two stages and thus the existing blade material is capable of resisting thermal load from the residual hot exhaust gas temperature. All dimensions of the nozzle were obtained from scanned data carried out using a 3D scanner. For the blade cooling path, the approximate dimension was made by x-ray image which was done after completion of full 3D scan of the blade. However, it was found that the cloud data and x-ray image provided was not fully complete due to the difficulties in getting the exact coordinate on regions where geometries are extremely complex and where equipment access to the regions was not possible. In this case, assumption on missing profile has to be made for model completion. The geometric configuration of the FSN is shown in Figure 2. The hot exhaust path can be clearly seen in between the two nozzles. Cooling air enters the blade from the side in two opposite directions. One flow through the gaps near the leading edge while the other through the longer cooling hole near the trailing edge. The first cooling path exits the nozzle through an array of cooling exit hole located at the leading edge. This is important to provide film cooling on both sides of the surfaces. On the other hand, the second cooling hole exits the nozzle through an array of cooling exit hole located on the suction and pressure surfaces halfway through the nozzle length.

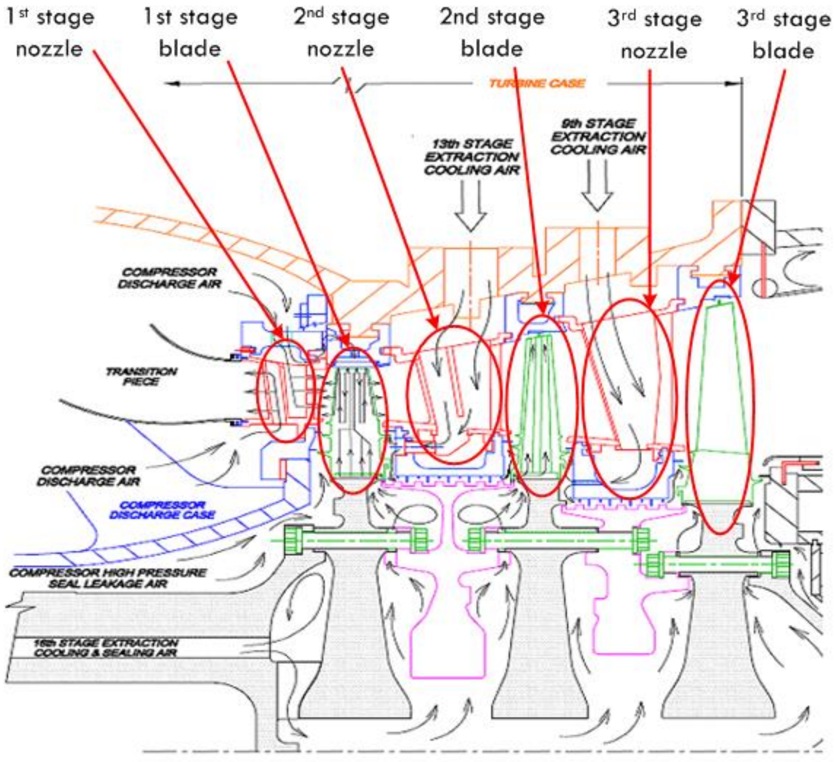


Figure 1 Turbine section, sealing and cooling air flow

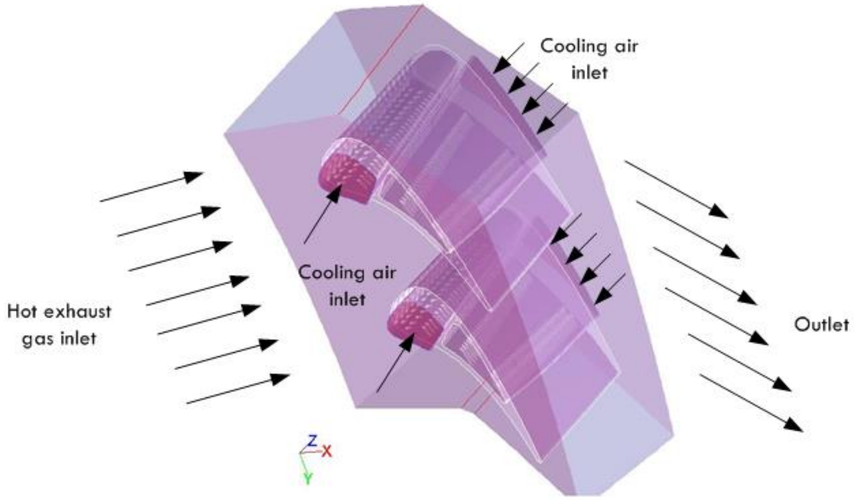


Figure 2 Model of first stage nozzle (FSN)

2.1 Mesh Generation

Generating a good mesh for flow and conjugate heat transfer calculation is extremely important to ensure sufficient solution accuracy. In this work, the flow domain is made by the inclusion of hot exhaust gas entering and leaving the blade. This is made possible by assuming wind tunnel like regions for the hot exhaust pass. Tetrahedral grid was adopted throughout the regions is illustrated in Figure 3. Higher mesh concentration is applied in the cooling air path where the interaction between the cooling air and blade solid is extremely important. Due to the size difference between the cooling path and the whole hot gas flow domain, the mesh size increases drastically and in this work the number of mesh generated can be approximated at 4.5 million. Grid dependency study shows that the current mesh density is sufficient to resolve the flow field with good accuracy.

2.2 Model Assumption and Boundary Conditions

In this work, the standard Navier-Stokes equations are solved within the framework of FLUENT. The turbulence flow was solved using the standard k-ε model. To resolve the flow field, SIMPLE method is employed. The interaction of heat between hot exhaust gas, blade solid and cooling is made possible using the conjugate heat transfer (CHT) method. The CHT assumes conduction of heat through solids, coupled with convective heat transfer in a fluid. One of the biggest difficult encountered in the modelling process is the determination of correct boundary conditions at the inlet. Due to the unavailability of actual measurement of flow entering the first stage blade zone, the mass flow of hot exhaust gas was estimated based on the information from the operation manual. In this case, the inlet temperature of the hot exhaust gas is assumed to be in the range of 900 – 1500 K while the cooling air inlet is assumed to be at a temperature slightly higher than ambient at 623 K.

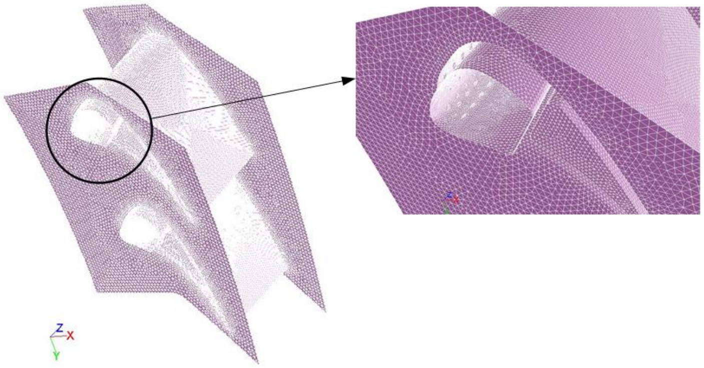


Figure 3 Mesh scheme for FSN

3.0 RESULTS AND DISCUSSION

In order to investigate the effect of variation of hot exhaust gas and cooling air to nozzle surface, several cases were numerically tested. The analysis was carried out based on normalized mass flow rate for both hot exhaust and cooling air. In this case, the mass of hot exhaust gas is made dimensionless by dividing with the base mass flow.

$$m_{HA,norm} = \frac{m_{HA}}{m_{HA,base}} \quad (1)$$

The same normalized mass is also applied to the cooling air and the normalized cooling air ratio is defined as

$$m_{CA,norm} = \frac{m_{CA}}{m_{CA,base}} \quad (2)$$

3.1 Flow Pathlines

The velocity of hot exhaust gas passing through the nozzle is very high with near supersonic in magnitude. The pathlines of the cooling air entering through gap near the leading edge is shown in Figure 4. It can be seen that the air leaving the nozzle through the holes at the leading edge which subsequently act as film cooling for the nozzle as shown in Figure 5.

3.2 Surface Temperature Distribution

Three different cases were simulated where the cooling mass flow entering the nozzle is reduced by 25% and 50% respectively. At 100% cooling air, the surface temperature distribution is shown in Figure 6(a). The maximum temperature is predicted at approximately 1,700K and is located in regions close to the side plate. The lowest temperature region is recorded at the leading edge as a result of film cooling when cooling air exits through the leading edges and swept in the main gas stream close the nozzle surface. When the cooling air is reduced by 25%, the overall surface temperature increases and this is shown in Figure 6(b). Similarly, when the cooling air is further reduced by 50%, the capability of reducing heat at the surface reduces. Thus regions of high temperature increase as shown in Figure 6(c). Despite the increase in the high temperature regions, the maximum temperature remains constant at approximately 1,700K. Figure 7 illustrates comparison of the pressure surface temperature distribution between the three cases of 100%, 75% and 50% cooling air. Comparison between the three cases show increased region of higher temperature spots for cases where cooling air is reduced. The effectiveness of film cooling is evident from these figures which show drastic reduction in temperature at pressure surface in all cases. Table 1 shows the summary of the average exhaust gas temperature entering and leaving the FSN stage. The exhaust gas temperature leaving the nozzle stage is the highest when rate of cooling air is the lowest. Nevertheless, the change in temperature between the nozzles does not differ much and it varies from 191 K to 205 K.

Table 1 Quantitative temperature of hot exhaust gas entering and leaving FSN

Cooling air variation	Avg. temp. hot exhaust gas (K) in	out	Delta T, hot pass(K)
1.00	1,700	1,495	205
0.75	1,700	1,504	196
0.50	1,700	1,509	191

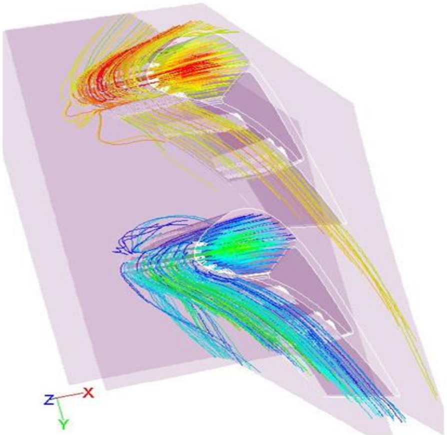


Figure 4 Cooling pathlines through gaps in FSN leading edge

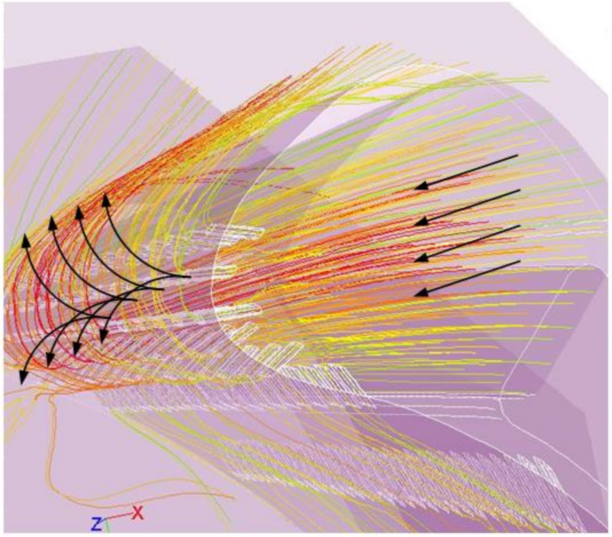
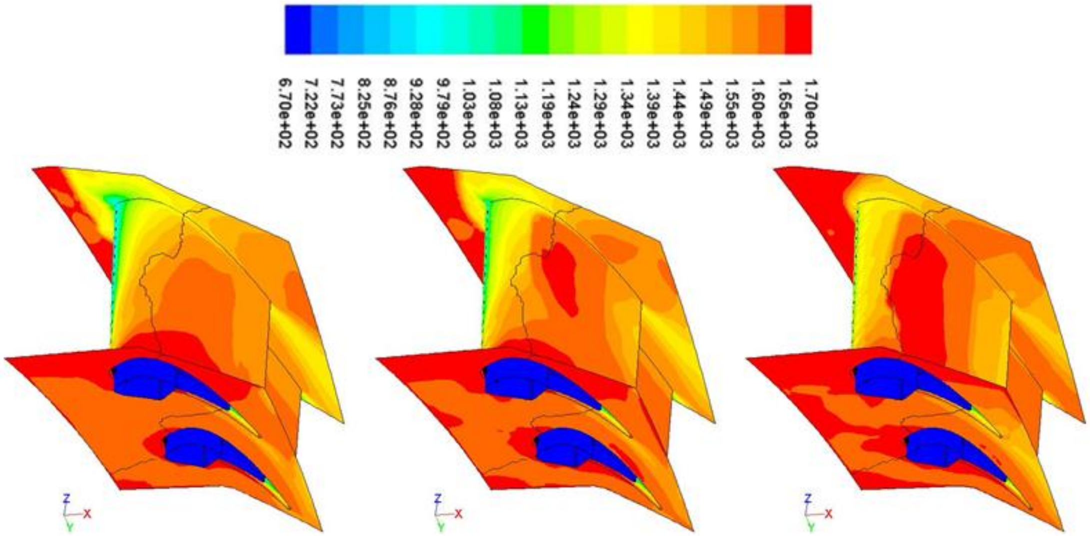


Figure 5 Detail cooling pathline leaving FSN leading edge

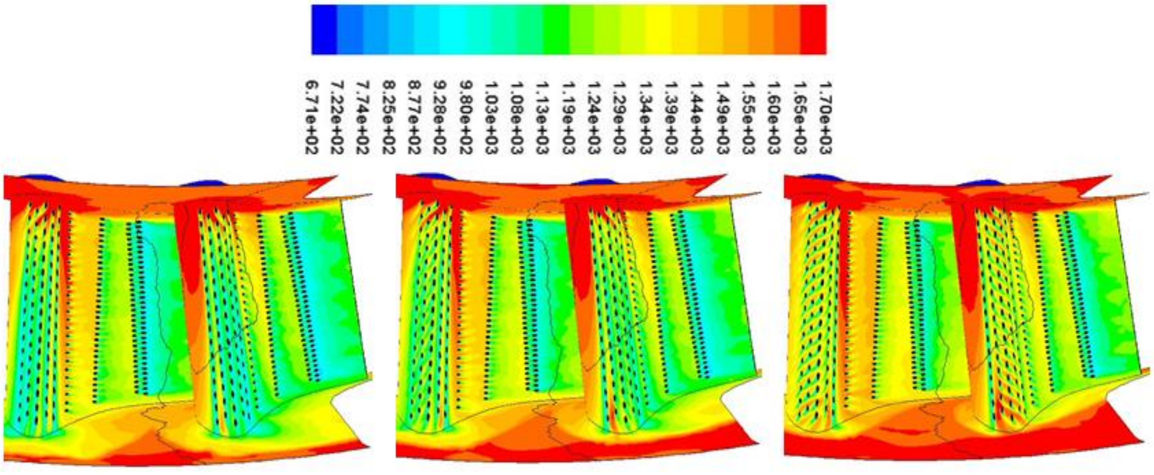
4.0 CONCLUSION

CFD simulation of conjugate heat transfer behaviour in FSN of power station gas turbine has been performed. The ability of commercial code in simulating flow and heat transfer in complex nozzle cooling path show reasonable accuracy with the flow pathlines exiting through leading edge, forming film cooling for the surface. In addition, the effect of reduction in cooling flow to surface temperature profile is also investigated. The cooling air channel in the FSN is critical to ensure effective film cooling on both the suction and pressure surfaces of the nozzle. Reduction in cooling air mass flow increases the surface temperature distribution but the maximum temperature of the surface does not differ much.



(a) HA=1.0, CA=1.0 (b) HA=1.0, CA=0.75 (c) HA=1.0, CA=0.50

Figure 6 Suction surface temperature distribution on FSN; (a) cooling mass =1.0, (b) cooling mass = 0.75 and (c) cooling mass = 0.50



(a) HA=1.0, CA=1.0 (b) HA=1.0, CA=0.75 (c) HA=1.0, CA=0.50

Figure 7 Pressure surface temperature distribution on FSN; (a) cooling mass =1.0, (b) cooling mass = 0.75 and (c) cooling mass = 0.50 (front view facing leading edge)

References

[1] Dyson, T. E., Bogard, D. G. and Bradshaw, S. D. 2014. Evaluation of CFD Simulations of Film Cooling Performance on a Turbine Vane Including Conjugate Heat Transfer Effects. *International J. Heat and Fluid Flow*. 50: 279-286.

[2] Moritz, N., Kusterer, K., Bohn, D., Sugimoto, T., Tanaka, R. and Taniguchi, T. 2013. Conjugate Calculation of a Film-cooled Blade for Improvement of the Leading Edge Cooling Configuration. *Propulsion and Power Research*. 2: 1-9.

[3] Gao, Z., Narzary D. P. and Han, J.-C. 2008. Film Cooling on a Gas Turbine Blade Pressure Side or Suction Side with Axial Shaped Holes. *International J. Heat and Mass Transfer*. 51: 2139-2152.

[4] Silieti, M., Kassab, A. J. and Divo, E. 2009. Film Cooling Effectiveness: Comparison of Adiabatic and Conjugate Heat Transfer CFD Met alodels. *International J. Thermal Sciences*. 48: 2237-2248.

[5] Albeirutty, M. H., Alghamdi, A. S. and Najjar, Y. S. 2004. Heat Transfer Analysis for a Multistage Gas Turbine Using Different Blade Cooling Scheme. *Applied Thermal Engineering*. 24: 563-577.

[6] Nowak, G., Wroblewski, W. and Nowak, I. 2012. Convective Cooling Optimization of a Blade for Supercritical Steam Turbine. *International J. Heat and Mass Transfer*. 55: 4511-4520.

[7] Wroblewski, W. 2013. Numerical Evaluation of the Blade Cooling for the Supercritical Steam Turbine. *Applied Thermal Engineering*. 51: 953-962.