

Research Article

Inlet Flux Influence to Aerodynamic Performance of Steam Turbine Vane Cascade

¹Feng Zi-Ming, ¹Ding Huanhuan, ³Li Chunhong and ²Xu Ping

¹School of Mechanical Science and Engineering, Northeast Petroleum University, Daqing, Heilongjiang 163318, China

²Daqing Oilfield Co., Ltd., Production Plant 3, Daqing, Heilongjiang 163000, China

³Petroleum Engineering Research Institute of Daqing Oilfield Company Ltd., Daqing 163453, China

Abstract: In order to improve the aerodynamic performance and the performance of off-design working condition, it is necessary to research the inlet flow rate influence to overall performance of steam turbine. This study uses CFD technology to numerical simulation the flow field of nozzle cascades passage at 80, 100, 120% flow rate, respectively. The simulation results indicates that the various flow loss of nozzle have a little change at 100% design working condition, have decreased at 80% design working condition and have increased at 120% design working condition.

Keywords: Aerodynamic, CFD simulation, nozzle, steam turbine

INTRODUCTION

In recent several decades, our steam turbine industry had fast development. With the fast and stable development of our economic the power demand was increased year by year, so improving the total aerodynamic performance of steam turbine were aim of the engineers of turbomachinery (Karamanis *et al.*, 1999; Wang *et al.*, 1994). At the same time, the periodic change of electricity consumption was bigger than before, such as the seasonal demand of enterprise or the fluctuation of electricity consumption by users, the stability requirement of steam turbine's off-design working condition was more and more rigorous than before. So we must research the off-design working performance of steam turbine cascades. Every country engineering had conducted many work to improve the cascade efficient (Murt *et al.*, 1994; Yamamoto *et al.*, 1994). This study mainly researches the influence to cascade aerodynamic performance by inlet flow rate change and the results can provide the theoretical support to improve the aerodynamic efficient of cascade.

NUMERICAL SIMULATION METHOD

Sixty MW super-critical steam turbine nozzle cascade was selected as the numerical simulation geometric model, the CFD simulation software was ANSYS-CFX. The blade geometric model was shown in Fig. 1. The mesh grid model was shown in Fig. 2. The total mesh sum was 420000 and the minimum

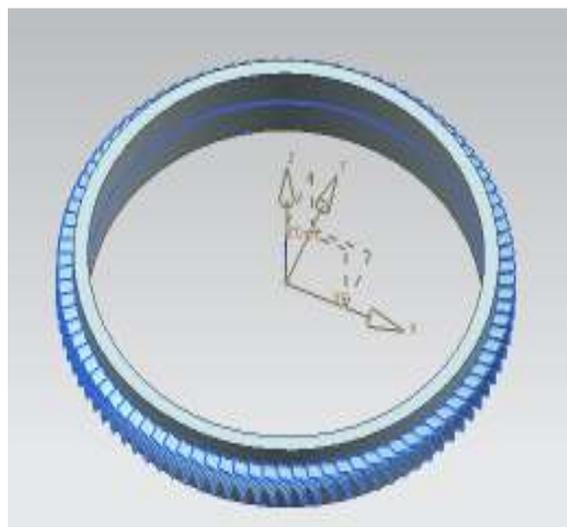


Fig. 1: Geometry model

mesh angle was 36.47. It was indicated the orthogonality was fine. The maximum length-width ratio was 127.73 that also reached the demand of CFX and there were no negative mesh. Maximum expansion ratio is 2.56. The mesh grid model was reached the demand of CFD simulation of the flow field of nozzle cascade. The boundary was as follow; inlet total pressure was 103100 Pa, inlet total temperature was 316 k, outlet static pressure was 0.1 MPa, inlet flow rate were 80, 100 and 120%, respectively design flow rate. The turbulence model was S-A model.

Corresponding Author: Feng Zi-Ming, School of Mechanical Science and Engineering, Northeast Petroleum University, Daqing, Heilongjiang 163318, China

This work is licensed under a Creative Commons Attribution 4.0 International License (URL: <http://creativecommons.org/licenses/by/4.0/>).

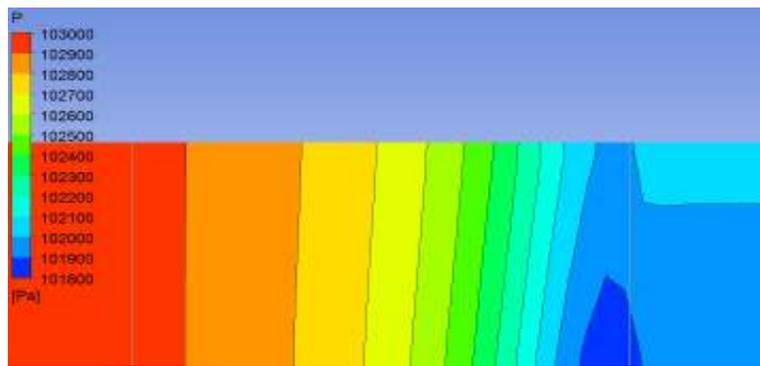
RESULTS AND DISCUSSION

Pitch-averaged static pressure coefficient distribution along blade height: This study adopts aft-

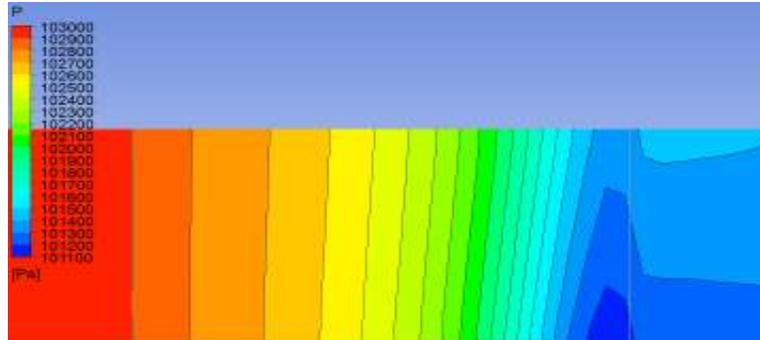


Fig. 2: Grid mesh model

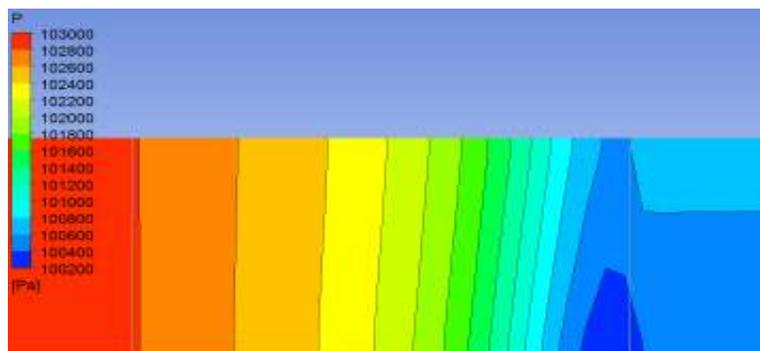
loaded blade profile as researched geometric model. The simulation results indicated that there is adverse pressure gradient at the 80% relative axial position, the adverse pressure gradient is very short that can weaken the secondary flow. In the three flow rate working-condition, all the three pitch-averaged static pressure isolines distribution are nearly similar to each other and are mainly parallel with each other. It indicates that the main flow are all laminar flow with little flow loss. After 80% relative axial position, passage flow begin to be turbulence with bigger flow loss because there appears adverse pressure gradient. As Fig. 3 shows that



(a)



(b)



(c)

Fig. 3: Pitch-averaged static pressure isoline distribution, (a) 80% design flow rate, (b) 100% design flow rate, (c) 120% design flow rate

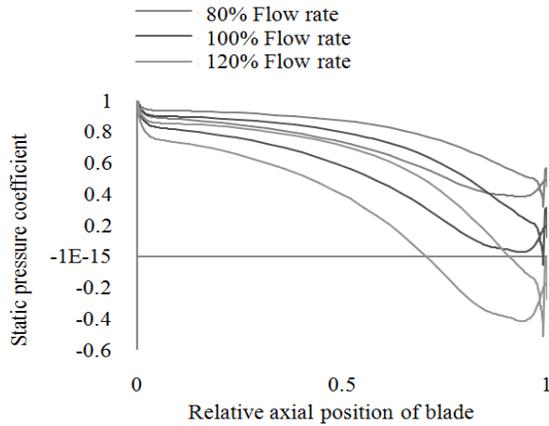


Fig. 4: Influence of variant flow rate to static pressure coefficient distribution at middle blade profile

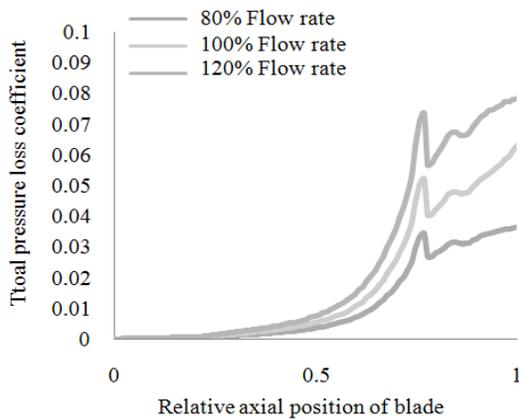


Fig. 5: Total pressure loss coefficient distribution along axial direction

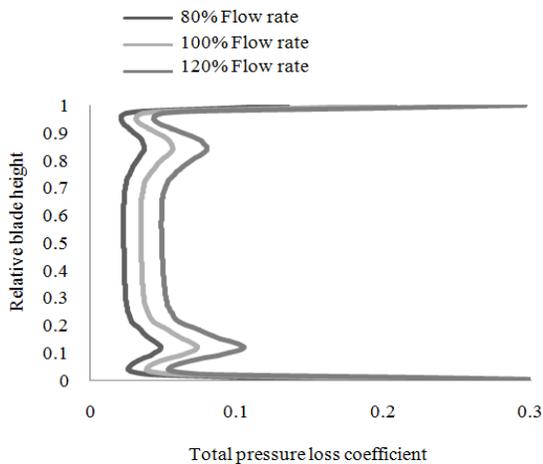


Fig. 6: Total pressure loss coefficient distribution along spanwise

the passage flow is smooth and steady at design flow rate and the other two passage flow property of 80 and 120% design are close to the simulation results of design working condition. It indicates that this nozzle

cascade has good adaptation and can be working with high efficient at wide off-design working condition with flow rate change.

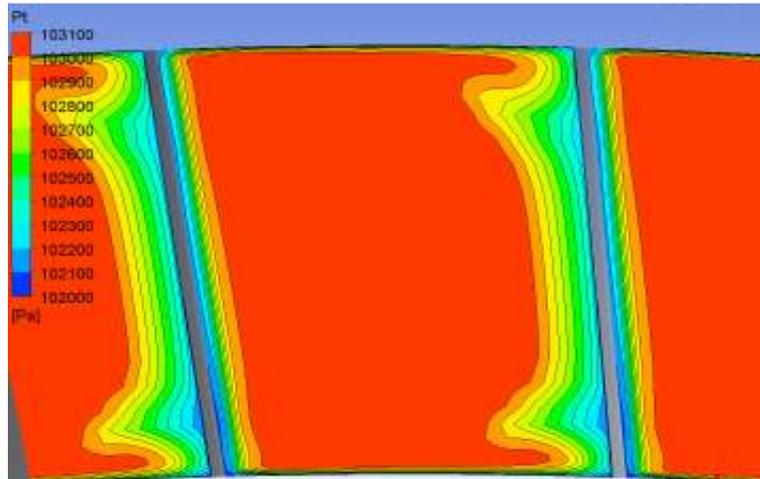
Influent of rate if flow change to static pressure coefficient at middle blade profile: Figure 4 shows that the static pressure coefficient of 80, 100 and 120%, respectively design flow rate distribute along blade profile at 50% blade height.

At the middle of blade height, the static pressure coefficient decreases as the flow rate increasing. The static pressure coefficient is decreased from 0.45 to 0.15 with 80 to 100% design flow rate and is decreased from 0.1 to -0.3 with 100 to 120% design flow rate.

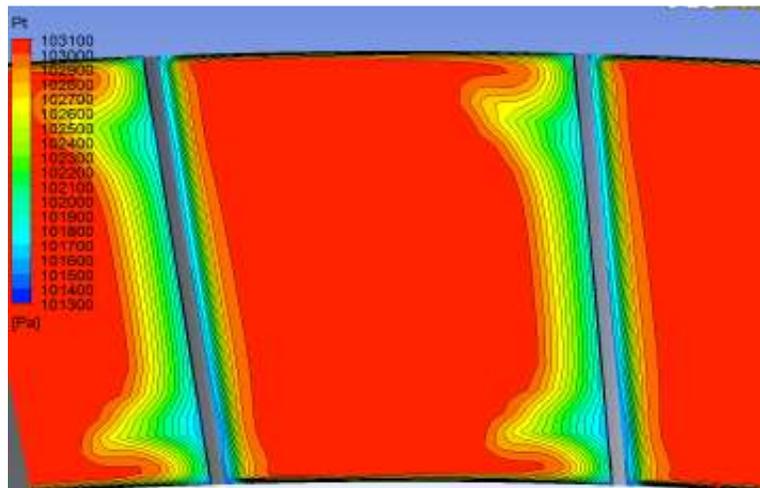
Total pressure loss coefficient distribution along axial direction: As Fig. 5 shows that before 60% relative axial position, the total pressure loss coefficients of 80, 100 and 120%, respectively design flow rate have the similar change law and little difference. At 120% design working condition, the total pressure loss coefficient is 7.8%, its efficient is 92.2%. At 100% design working condition, the total pressure loss coefficient is 6.3%, its efficient is 93.7%. At 80% design working condition, the total pressure loss coefficient is 3.7%, its efficient is 96.3%. It is obvious that the flow loss of 80% design flow rate is smaller than the other two working condition, but the smaller flow rate will not provide enough steam to the next cascade stage.

Pitch-averaged total pressure loss coefficient distribution along spanwise at cascade outlet: The pitch-averaged total pressure loss coefficient distribution along spanwise at cascade outlet is shown in the Fig. 6. Comparing with the three curves from Fig. 6, it is clear that there are three regions, root of blade, middle of blade and tip of blade, from root to tip blade. The total flow loss curve can be divided into five regions, there is a high flow loss region at the up and down endwalls, it is mainly induced by the frictional loss with boundary layer flow. There is also a high flow loss region near the up and down endwalls about 10% relative blade height, it is mainly induced by the up and down passage vortexes. The passage flow loss of most middle blade region is smaller than other region, it is mainly boundary layer loss on the blade walls, this part of flow loss is smaller than other region but occupies more region. In the nozzle cascade, the main flow loss is boundary layer flow loss, the other flow loss are made of passage vortex loss and wake fall off vortex loss and secondary flow loss.

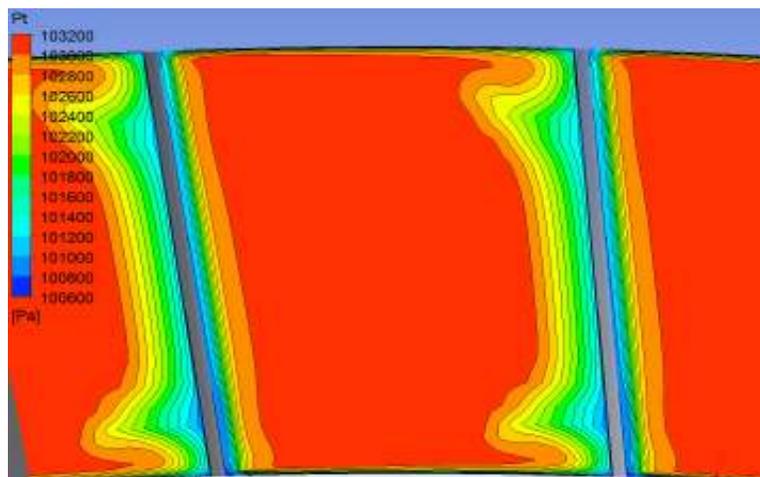
The cross-section total pressure loss isoline at cascade outlet is shown in Fig. 7. It is can be seen from Fig. 7 that the down fields of three flow rate have the similar total pressure distribution, the waking flow fields have the similar geometric type, the high flow loss are all located in the waking flow region and endwall region, the three fields are all typical



(a)



(b)



(c)

Fig. 7: Total pressure loss coefficient isoline distribution on the outlet cross-section, (a) 80% design flow rate, (b) 100% design flow rate, (c) 120% design flow rate

downstream field structure of annular cascade. At 8% blade height apart from up and down endwalls, there are respectively a high flow loss region, the flow loss maximum value increases as the increasing of flow rate.

CONCLUSION

The CFD simulation of nozzle cascade at off-design and design working condition indicates; at the design working condition, all kinds of aerodynamic performance parameters are all good for the nozzle cascade and all kinds of flow loss are not large, it indicates the prototype blade cascade has excellent aerodynamic performance. At smaller flow rate working condition, the boundary layer flow and secondary flow loss are all decreased, but too low flow rate will not provide enough demand steam for the next cascade stage and will influent the power of steam turbine. At bigger flow rate working condition, the boundary layer flow loss and secondary flow loss are all increased and will outstanding decrease the total system efficient.

By the analysis and research of all kinds of aerodynamic performance parameters in the cascade flow field at different flow rate, in the 80~120% design working condition region, no air flow block up phenomenon is founded, the up and down passage vortexes have small change, the three waking fall off

vortex have similar constructor, it is indicated that the blade cascade adopted by this study has excellent aerodynamic performance in off-design working condition.

REFERENCES

- Karamanis, N., R.F. Martinez-Botas and C.C. Su, 1999. Detailed flow measurements at the exit of a mixed flow turbine under steady flow conditions. Proceeding of the International Gas Turbine and Aeroengine Congress and Exhibition. Indianapolis, Indiana, ASME-GT-342, 1: 1-9.
- Murt, G.V., Y. Ramana and N. Venkatrayulu, 1994. Proceeding of the International Gas Turbine and Aeroengine Congress and Exposition, 132: 1-9.
- Wang, Z.Q., J.X. Su and J.J. Zhong, 1994. New progress of investigation into mechanism of reducing energy loss in cascades with curved and twisted blades. *J. Eng. Thermophys.*, 15(2): 40-44.
- Yamamoto, A., J. Tominaga, T. Matsunuma and E. Ota, 1994. Detailed measurements of three-dimensional flows and losses inside an axial flow turbine rotor. Proceeding of the International Gas Turbine and Aeroengine Congress and Exposition, 24: 1-9.