Investigation on heat transfer characteristic and optimization of the cooling air inlet for the twin-web turbine disk

Mengchuang ZHANG*, Wenxuan GOU, Qin YAO, Lei Li, Zhufeng YUE
Department of Engineering Mechanics, Northwestern Polytechnical University (Chang'an Campus), Xi'an, P.R. China.

*zmc@mail.nwpu.edu.cn,

Abstract. With a higher operation temperature, the conventional aero-turbine single web disk (SWD) has reached its limits. The twin-web disk (TWD) has been designed as a breakthrough, which has an expected performance in weight loss, strength and heat transfer efficiency. However, the lack of investigation on the position of the cooling air inlet is slowing down further application of TWD. Therefore, for a further study, inlet position optimization with maximum average Nusselt number is conducted for TWD flow structure study. The average Nusselt number result shows that the TWD has a better performance in heat transfer. All the works, including modeling and analyzing, can be referred for engineering design. And the conclusions obtained in this paper could be valuable for the future improvement of the TWD.

1. Introduction
Currently, higher turbine inlet temperature (TIT) has reached 1600K and expected to be pushed toward 2000K in the future, which is beyond the limits of the most parts of aero-engine turbines, especially the first stage turbine (i.e., high pressure turbine, or HPT) disk [1]. The conventional aero-turbine single web disk (SWD) has reached its limit of the AN2 (turbine annulus area multiplied by speed squared, unit m²(r/min)²) and thermal loads. As a breakthrough, the twin-web disk (TWD) has been designed with an expected performance in weight loss, strength and heat transfer efficiency. Therefore, it has been proven to be the future trend of the high pressure turbine disk (HPT) by the U.S. IHPTET program [2, 3].

Many researchers have studied on the SWD cavity and its discoidal rotor-stator systems using experimental or numerical methods [4-6]. Harmand et al. [7] reviewed many convinced numerical technologies during the past decades. In the reference [8-10], the decoupled and conjugate methods are widely used. Harmand [7] indicated that the conjugate heat transfer method is an efficient and accurate coupling analysis numerical method and has been widely used in the numerical study on the heat transfer characteristics of gas turbine disk [11-13] and other parts of the gas turbine engine [14-16] with a desired accuracy.

In recent years, optimization based on thermo-fluid analysis is becoming increasingly popular in engineering design [17, 18] and turbine disk[19]. In some cases, evolutionary algorithms are used to ensure reaching the global optimum. Besides, thermo-fluid analysis often includes the solution of high nonlinear equations, the cost of time is huge and unacceptable. As a lower fidelity computational model, two-dimensional model is used widely in aerodynamic optimization [20, 21].
Therefore, the two-dimensional model and structural mesh, which can reduce the cost of the optimization searching by a large margin, are used to find the optimum cooling air inlet position. For the purpose of comparing with the SWD in [22], the models in this study have the same size and boundary conditions. All the works, including modeling and analyzing, can be referred for engineering design.

2. Numerical method

2.1. Geometry model and grid

The 2D dimensional model (fluid region) is shown in Fig. 1 accordingly to Ding’s experiment [22]. It is unable to conduct the calculation of 2D model by ANSYS CFX directly. A thin model with only single layer grid in z direction is often used as a solution. Structural grid is also shown in Fig. 1. To ensure that solutions yield sufficient accuracy within CFX, a mesh dependency study has been performed. The average total pressure value and the average temperature are set as the baseline for the mesh independence study based on reference [23]. It is found that the mesh constitutes a satisfying compromise between the duration and the accuracy of the calculation for optimization purpose.

The heat transfer on the rim surface consists of two parts: the heat conduction from the heated blade and the heat convection from the high temperature gas. A constant and uniform heat flux is applied here. The detailed boundary conditions are listed in Table 1. The layers of near wall mesh are defined as 7 to satisfy the requirement.

![Figure 1. 2D dimensional model for optimization](image)

**Table 1. Boundary condition of the calculation model**

| Parameters                  | Symbols         | Given values |
|-----------------------------|-----------------|--------------|
| Working condition           | PRES_RIM        | 340 MPa      |
| Working rotating speed      | RPM             | 14000 rev / min |
| Thermo-fluid boundary       | T_IN            | 600 K        |
| Temperature at rim          | T_OUT           | 850 K        |
| Mass flow rate of inlet     | MFLOW_INLET     | 500 kg/h     |
| Average static pressure of outlet1 | P_OUTLET1 | 6 atm        |
| Average static pressure of outlet2 | P_OUTLET2 | 8 atm        |

2.2. Turbulence models

The SST turbulence model is considered a good quality in interface data transition and therefore offers a good compromise between accuracy and calculation cost [7]. In order to obtain a reasonable
computational result, \( Y^+ \) is set in a range from 10 to 30 based on the near-wall grid requirement by SST model. Rotational Reynolds number is defined as:

\[
Re_\omega = \frac{\rho \omega R^2}{\mu}
\]

Where \( \rho \) is the density of the air, \( \omega \) is the rotational speed of the disk, \( R \) is the radius, \( \mu \) is the dynamic viscosity of the air.

2.3. Experiments verification

Shuiting et al. [24] have conducted an experiment to investigate the flow and heat transfer on rotor-stator system with pre-swirl angle. Fig.2 shows the configuration of a SWD in the experiment. The data of the SWD sample can be easily obtained from the reference. 1/6 of the rotational computational model in 6 cases (mass flow rate from 200kg/h to 450kg/h) are analyzed for heat transfer by the numerical method above. The numerical result for the temperature distribution of computational SWD (\( \dot{m} = 450kg/h \)) is shown in Fig.3.

![Figure 2. Schematic configuration of a TWD in Shuiting’s experiment](image)

Fig.4 shows the numerical and experimental results in all the cases. The most difference between the numerical and experimental results is the region of disk hub in the case 1 (\( \dot{m} = 200kg/h \)). Case 6 (\( \dot{m} = 200kg/h \)) is the most accurate model with the only 1K of the maximum deviation. Therefore, larger mass flow rate (\( \dot{m} = 500kg/h \)) is used in this paper in order to get expected accuracy.

![Figure 3. The temperature distribution of computational SWD (\( \dot{m} = 450kg/h \))](image)
3. Optimization models

3.1. Design variable and object function
The dimensionless inlet position $x/R$ is used as the design variable in a range of $0 \sim 0.9$. The object is the maximum average Nusselt number $Nu_{ave}$:

$$Nu_{ave} = \frac{\alpha_{av} R}{\lambda} \quad (1)$$

Where $\alpha_{av}$ is the average heat transfer coefficient, $R$ is the disk radius, and $\lambda$ is the thermal conductivity of the static fluid. As the only unknown variable, the average heat transfer coefficient $\alpha_{av}$ can be obtained by the ANSYS CFX. Then the optimization problem can be defined as Maximize $Nu_{ave}$.

3.2. Optimization technique
Evolution optimization algorithm is a stochastic global search method. The algorithm is operated on a population of potential solutions applying the principle of survival of the fittest to generate better approximations. For each generation, mutation is introduced to reduce the possibility of local optimal search. The evolution strategy here is based on the works of Rechenberg and Schwefel which mutates designs by adding a normally distributed random value to each design variable. The mutation strength (standard deviation of the normal distribution) is self-adaptive and changes during the optimization process [25].

3.3. Optimization process
ISIGHT optimization software with ANSYS CFX are adopted here to find optimum designs. The automatic process control is implemented in the windows batch file and windows executable files generated by FORTRAN. The geometry of the model is changed with the changes of geometrical parameters input by the ISIGHT optimizer.

4. Results
Included mesh generation, CFD analysis, using the 32 core CPU, every optimized step last for about 1.5 min in total. The mean iterations of the computation for thermal-fluid using ANSYS CFX is about 60 and more than 60% of the total time cost. After 51 optimized iterations, the final global optimal results are obtained by evolution optimization algorithm. Fig. 5 shows the optimization history of
object and design variables, which can be easily seen the characteristic of the evolution optimization algorithm. The optimum (x/R = 0.432, \( \text{Nu}_{\text{ave}} = 2853.1 \)) is also shown in Fig. 12 in green color.

![Graph](image)

**Figure 5.** The optimization history of object (a) \( \text{Nu}_{\text{ave}} \); (b) dimensionless inlet position x/R

In order to study the sensibility of different inlet positions, the grids and the velocity streamline of 5 models are shown in Fig.6 and 7 respectively. It is obvious that higher inlet position form more vortex in the cavity of TWD, but the velocity loss is also higher. The two aspects brought by the higher inlet position has opposite function on Nusselt number, while the optimum x/R = 0.432 satisfies both aspects.

The \( \text{Nu}_{\text{ave}} \) of TWD with the sequential dimensionless inlet position x/R is plotted in Fig.8, along with the \( \text{Nu}_{\text{ave}} \) data of SWD from the reference [22]. It shows that the optimum inlet positions of SWD and TWD are different. TWD has a higher heat transfer performance in the region of disk hub and a global higher \( \text{Nu}_{\text{ave}} \) in disk surface.

![Graph](image)

**Figure 6.** The grids of 5 models with different inlet position (* - the optimum)

![Graph](image)

**Figure 7.** Velocity streamline study of 5 models with different inlet position (* - the optimum)
5. Conclusions
In this paper, a global optimization is conducted with the object of the maximum average Nusselt number. The results show that x = 0.432 is the best position. The Nu_{ave} of TWD and SWD with the sequential dimensionless inlet position x/R also indicate that TWD has a better performance in heat transfer. The works in this paper could be valuable for the future improvement of the TWD.

Acknowledgment
This work is supported by China Postdoctoral Science Foundation(Grant No. 2014M562281), National Natural Science Foundation of China (Grant No. 51205315) and Aerospace Technology Support Foundation(2014-HT-XGD).

Reference
[1] LIAO G, WANG X, LI J, et al. Numerical investigation on the flow and heat transfer in a rotor-stator disc cavity [J]. Applied Thermal Engineering, 2015, 87(0): 10-23.
[2] CAIRO R R, SARGENT K A. Twin web disk: A step beyond convention [J]. Journal of Engineering for Gas Turbines and Power-Transactions of the Asme, 2002, 124(2): 298-302.
[3] SHEN X L, DONG S J. Structure Optimization and Welding Residual Stress Analysis of Twin-Web Turbine Disc [J]. Advanced Materials Research, 2013, 622-623(4).
[4] KARABAY H, WILSON M, OWEN J M. Predictions of effect of swirl on flow and heat transfer in a rotating cavity [J]. International Journal of Heat & Fluid Flow, 2001, 22(1): 143-55.
[5] TRAN N, CHANG Y-J, TENG J-T, et al. Numerical and Experimental Investigations on Heat Transfer of Aluminum Microchannel Heat Sinks with Different Channel Depths [J]. International Journal of Mechanical Engineering and Robotics Research, 2015, 4(3): 204.
[6] POUNTNEY O J, SANGAN C M, LOCK G D, et al. Effect of Ingestion on Temperature of Turbine Disks [J]. Journal of Turbomachinery, 2013, 135(5): 1790-1.
[7] HARMAND S, PELL J, PONCET S, et al. Review of fluid flow and convective heat transfer within rotating disk cavities with impinging jet [J]. International Journal of Thermal Sciences, 2013, 67(5): 1–30.
[8] SMITH P E J, MUGGLESTONE J, THAM K M, et al. Conjugate Heat Transfer CFD Analysis in Turbine Disc Cavities; proceedings of the ASME Turbo Expo 2012: Turbine Technical Conference and Exposition, F, 2012 [C].
[9] OKITA Y, YAMAWAKI S. Conjugate Heat Transfer Analysis of Turbine Rotor-Stator System;
proceedings of the ASME Turbo Expo 2002: Power for Land, Sea, and Air, F, 2002 [C].

[10] ANDREI L, ANDREINI A, FACCHINI B, et al. A Decoupled CHT Procedure: Application and Validation on a Gas Turbine Vane with Different Cooling Configurations ☆ [J]. Energy Procedia, 2014, 1087–96.

[11] ANDREINI A, SOGHE R D, FACCHINI B. Turbine Stator Well CFD Studies: Effects of Coolant Supply Geometry on Cavity Sealing Performance [J]. Journal of Turbomachinery, 2009, 133(2): 841-53.

[12] GANINE V, JAVIYA U, HILLS N, et al. Coupled Fluid-Structure Transient Thermal Analysis of a Gas Turbine Internal Air System With Multiple Cavities [J]. Journal of Engineering for Gas Turbines & Power, 2012, 134(10): 2167-77.

[13] SUN Z, CHEW J W, HILLS N J, et al. Coupled Aerothermomechanical Simulation for a Turbine Disk Through a Full Transient Cycle [J]. Journal of Turbomachinery, 2011, 134(1): 112-21.

[14] HEIDMANN J D, KASSAB A J, DIVO E A, et al. Conjugate Heat Transfer Effects on a Realistic Film-Cooled Turbine Vane; proceedings of the ASME Turbo Expo 2003, collocated with the 2003 International Joint Power Generation Conference, F, 2003 [C].

[15] KIM K M, PARK J S, DONG H L, et al. Analysis of conjugated heat transfer, stress and failure in a gas turbine blade with circular cooling passages [J]. Engineering Failure Analysis, 2011, 18(4): 1212–22.

[16] DING S, LI G, LUO B. Active Control Thermal-Loading Method to Ameliorate Stress in Aeroengine Turbine Disk [J]. Journal of Thermophysics & Heat Transfer, 2013, 27(2): 274-85.

[17] ALEXANDROV N M, LEWIS R M, GUMBERT C R, et al. Approximation and Model Management in Aerodynamic Optimization With Variable-Fidelity Models [J]. Journal of Aircraft, 2001, 38(6): 1093-101.

[18] WANG H, ZHU X, DU Z. Aerodynamic optimization for low pressure turbine exhaust hood using Kriging surrogate model ☆ [J]. International Communications in Heat & Mass Transfer, 2010, 37(8): 998-1003.

[19] ZHANG M, GOU W, LI L, et al. Multidisciplinary design and optimization of the twin-web turbine disk [J]. Structural & Multidisciplinary Optimization, 2016, 53(5): 1-13.

[20] AIAAA. Two-dimensional high-lift aerodynamic optimization using the continuous adjoint method [J]. Aiaa Paper, 2000,

[21] KOLLA M L, YOKOTA J W, LASSALINE J V, et al. Stowability Constraint Within a Two-Dimensional Aerodynamic Optimization Method [J]. Journal of Aircraft, 2012, 46(2): 696-9.

[22] DING S T, ZHANG G, YE L I, et al. Sensibility analysis of the air inlet position in the rotating cavity [J]. Journal of Aerospace Power, 2011, 26(8): 1681-7.

[23] XIE G, LIU J, LIGRANI P M, et al. Flow structure and heat transfer in a square passage with offset mid-truncated ribs [J]. International Journal of Heat & Mass Transfer, 2014, 71(4): 44–56.

[24] SHUITING L X X G T Z D. Flow and heat transfer in rotor-stator system with pre-swirl angle of 30° [J]. Journal of Beijing University of Aeronautics & Astronautics, 2007, 33(3): 290-3.

[25] FONSECA C M, FLEMING P J. An Overview of Evolutionary Algorithms in Multiobjective Optimization [J]. Evolutionary Computation, 1999, 3(1): 1--16.