NUMERICAL ANALYSIS OF STEAM FLOW IN STEAM TURBINE NOZZLE BLADE CASCADE

Abhijit J. Patil
Department of Mechanical Engineering
Rajarambapu Institute of Technology
Rajaramnagar, India.
abhijitpatil12@outlook.com

Sharad D. Patil
Department of Mechanical Engineering
Rajarambapu Institute of Technology
Rajaramnagar, India
sharad.patil@ritindia.edu

Abstract— **During the operation of the low-pressure steam turbine, condensation phenomenon is observed in stationary and rotating blade cascade. In this paper Numerical method was employed to simulate steam flow in linear blade cascade in order to study the surface pressure distribution and flow field characteristics. Initially, 2D Dykas cascade results validated with the k-ω SST viscous turbulence model for vapour liquid mixture, then the 3D cascade model was simulated with IAPWS IF-97 model. From the comparative study of both models, IAPWS IF-97 model wasquite easy to implementduring the simulation which gives more accurate blade surface pressure distribution according to standards. Also, it helps in the design process of steam turbine blades.**

Keywords— Nozzle blade cascaade; CDF; Condensing flow

I. INTRODUCTION

The search for model and predict the performance of steam turbine under off-design and abnormal operation is ongoing worldwide. Various methods used to model axial gas and steam turbines, such as Stodola's ellipse law, based on the analysis of velocity triangles, and in other cases making use of CFD analyses. Models of steam turbine blades vary from Simple to Complex. The simple model requires few inputs for design, and complex models require detail knowledge of loss contributors as well as each blade passage details.

Use of multi-stage axial steam turbine is found in power generation plants. In large steam turbines, steam enters as dry saturated vapour but exhausts to the condenser as a two-phase mixture of saturated vapour and small liquid droplets. When high-pressure temperature steam flows through such stages of the steam turbine, its pressure and temperature drop sharply. Due to the high velocity of steam with the bulk flow, a thermodynamic equilibrium is not achieved and non-equilibrium condensation with the presence of water droplets in gas flow results in major losses as follows.

1. Thermodynamic loss: This occurs due to heat transfer between the liquid phase and gaseous phase involve with entropy generation.
2. Aerodynamic loss: This occurs due to surface boundary loss, pressure side boundary loss and mixing loss downstream of the trailing edge.
3. Erosion loss: This occurs due to the impact of water droplets on the blade surface.

These losses result in decrease the overall performance of the steam turbine. To overcome these losses shape optimization of nozzle cascade has been done worldwide, which reduces the droplet formation as well droplet size of condensed vapour.

The presence of the liquid phase in the low-pressure (LP) steam turbines is responsible for energy losses; it generates thermodynamic and aerodynamic losses, as well as erosion in the turbine components and blade failures due to the corrosion effects. Consequently, it results in a reduction of efficiency and reliability of the steam turbine stages. Therefore, an accurate prediction of the loss mechanism due to the steam condensation is important. The steam properties (steam purity) and the flow unsteadiness, play an important role in the processes of liquid phase formation and fog or coarse droplets formation. It may have some influence on the condensation shock formulation and the generation of shock waves under conditions of supercooled steam flow. [1]

The condensation process is very complicated because heat and mass transfer between the droplet and vapour will take place. The two-phase flow will occur after the droplets emerge. The droplet formation is so rapid that it is difficult to measure by experiment method. In order to understand the phenomenon of steam flow through numerical simulation with phase change and two-phase flow model theories are done. [2] However, still the result calculated with the ideal gas model is large deviates with the comparison of experimental results. To obtain better results through the simulation flow parameters need to specify properly so that the correct estimation of thermodynamic losses were done. Therefore, for modelling of these flows a real gas equation of state for steam has to be used. Hence it is necessary to implement IAPWS IF-97 model for steam flow through nozzle blade cascade for the numerical simulation.

Compressive literature survey will be made by using paper published in the International journals like Heat and Mass Transfer, Applied Thermal Engineering, Journal of Turbomachinery, International Journal of Thermal Sciences etc. Manuals of ANSYS, Solid Works and books related to blade design of steam turbine also been studied.SlawomirDykas et al. [1] carried out experimental as well as a numerical study of condensing non-equilibrium steam flow through cascade. The aim was to present the steam condensation while flowing through cascade experimentally on a setup developed in Silesion University laboratory. While they have carried out numerical study also and came to state that problems existing during the measurement of non-equilibrium condensing steam flow.Guojie Zhang et al. [2] firstly studied the condensing flow results of Dykas's experimental as well as numerical model. To get the better results they have been modified the numerical model by using UDF in ANSYS FLUENT. From modified model, they came to state that better pressure distribution was obtained than the original model. Also, shock angle predicted by the modified model was almost closer to the experimental results obtained from Dykas setup.

N. R. Abadi et al. [3] carried out CFD based nozzle blade shape optimization with the consideration of condensing two-phase model for the reduction of aerodynamic as well as thermodynamic losses due to the formation of droplets. Shape optimization was mainly focused for two reasons one for Reduction in Entropy Generation and second for the Reduction of maximum droplet diameter. By implementation of this optimization thermodynamic as well as aerodynamic losses were reduced and maximum 2.1% turbine blade efficiency is achieved.W. F. fuls [4] developed a methodology to model a turbine stage. Also, Classical nozzle theory is used to model a turbine stage as a series of two nozzles. Thermo-hydraulic network solver flownex was used accurate prediction of turbine's off-design operation. This method is helpful to obtain the inner stage process conditions.

S. Plerret et al. [5]developed a method for designing of the steam turbine blade with minimum time by the use of Artificial Neural Network with the database of Navier-Stokes solver. To achieve aerodynamic and mechanically efficient blade large numbers of iterations need to carry out which require more time also require a large cost. Hence they developed above method in order to reduce design time as well cost.Bo Song et al. [6] carried out an experimental test on three sets of steam turbine nozzle blade cascade for prediction of the solidity of the blade. Solidity plays a role while estimating profile losses in steam turbine blade design. From experimental testing, they found that reduction in profile losses achieved by the reduction in the solidity of the cascade. Also, the stated lower solidity reduces cost and gain an increase in efficiency. Experimental tests carried out for a broad range of incidence angle, while at each incidence angle exit Mach number is tested.

Chao Zhou et al. [7] conducted experimental, numerical and analytical methods to study the effects of the blade trailing edge thickness on the profile loss of steam turbine. Flow distribution along the passage of trailing edge plane is non-uniform. The experimental results show that the profile loss first decreases and then increases as the trailing edge thickness increases. Also increasing the trailing edge thickness from 1.9% pitch to 2.8% pitch has a small effect on the blade profile losses.D. Corriveau et al. [8]describe an experimental study on the influence of loading distribution on the performance of HP turbine blades. Measurements have been presented for the aerodynamic performance of three blades at the off-design incidence and Mach number.Lin Cai et al. [9]studied numerically about the steam flow properties through supersonic steam ejector. They developed a Real gas model and compared with the Ideal gas model. They found that Real gas model in CDF is predicting more accurate than the Ideal gas model.

Assim H. Yousif et al. [10] studied experimentally and numerically about the effect of exit pressure variation in low-pressure steam turbine cascade with the two-phase flow of saturated vapour and fine water droplet model. They found that high rate of expansion has been increased by continue reducing the back pressure.

II. NUMERICAL ANALYSIS

A. Geometry details

In order to simulate results of both models the stationary blade cascade developed by S. Dykas et al.[1] is used. Geometry dimensions are as shown in Figure 4.1. The linear cascade consists 4 blades which gave 3 blade-to-blade passages. The blade axial chord is 173.97 mm and the blade pitch is 91.74 mm. The width of the cascade is 110 mm.

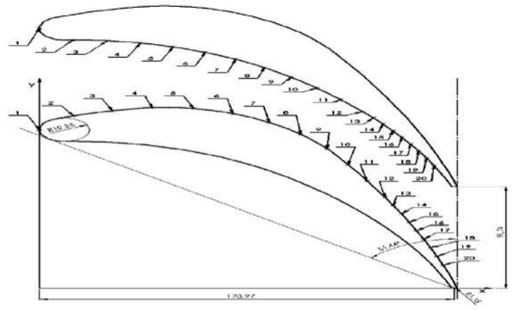

Fig. 1 Geometry of Dykas stator blade.

B. Mesh and boundary conditions

The nozzle cascade of Dykas is in three dimensions;in order to save the time of computation 2D simplification is considered. There are four statorblades in the experiment of Dykas. In this paper, only one passage including two blades is modelled by employing a periodic boundary on upper and lower boundary of flow domain. Boundary conditions and the unstructured mesh is generated for the computational domain is shown in Fig. 2. Boundary details are as maintained in Table 1.

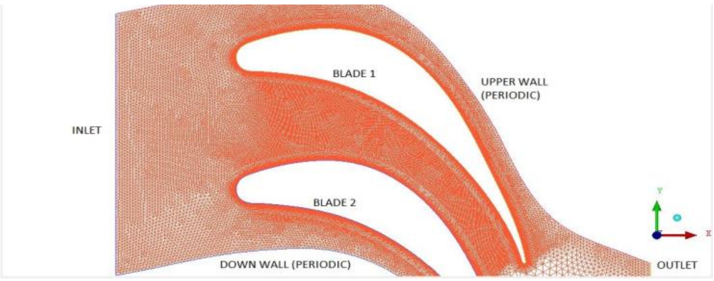

Fig. 2 Mesh and Boundary condition of Dykas cascade

TABLE I. BOUNDARY CONDITION

Sr. No	Boundary	Type of boundary condition	Magnitude	Temperature
1.	Inlet	Pressure Inlet	89000Pa (a)	373 K
2.	Outlet	Pressure Outlet	39000 Pa (a)	300 K
3.	Up Wall	Periodic Boundary	Translational	
5.	Down Wall	Periodic Boundary	Translational	
6.	Blades	Wall	No slip condition.	

Flow domain is meshed using ICEM CFD. In order to achieve fine mesh, element size is selected as 1 mm; also unstructured patch dependent mesh is created with inflations around blades. For the spacing of first layer inflationY+ value is calculated using fluid density, freestream velocity, characteristic length and the absolute viscosity. From the calculations, Y+ value came below 1mm and first layer cell height came as 0.2mm. 16 inflation layers created around the blades with first layer height as 0.2mm and growth ratio as 1.2.

C. Turbulence model

According to Dykas et al.[1] two-equation k-ω SST viscous turbulence model is selected with vapour/liquid mixture. Also, two-phase non-equilibrium flow was assumed, the volume occupied by droplets is negligibly small. The interaction betweenthe droplets is not taken into account in the model. The heatexchange between the liquid phase and the solid boundary as wellas the velocity slip between vapour and the liquid phase areneglected. The homogeneous condensation phenomenon ismodelled on the basis of the classical nucleation theory and thecontinuous droplet growth model.[1]

After 2D simulation of turbulence model,3D geometry meshed using ICEM CFD and simulation run with the ANSYS CFX with the homogeneous mixture and IAPWS IF-97 as material properties.

III. RESULTS AND DISCUSSION

A. Numerical Validation

In order to validate results with Dykas[1] simulation run with URANS equation coupled with two-equation k-ω SST viscous turbulence for vapour/liquid mixture. With the available boundary condition outlet mach observed as 1.3, also pressure and mach number counters show the same natureas obtained by Dykas et al.[1] Validation variation found about 6% due to the slightly geometrical variation and mesh type. Mach number and pressure counters are as shown in Figure.3a, 3b and Figure. 4a, 4b.

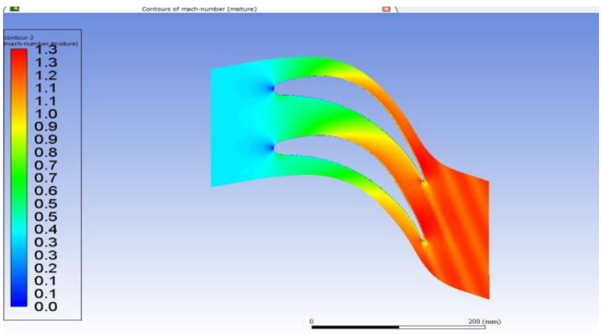

Fig.3a Mach No. distribution with 2D simulation

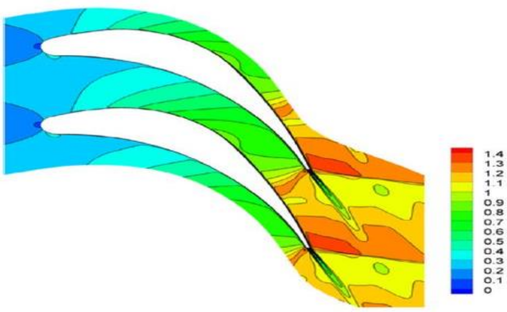

Fig.3bMach No. distributionby Dykas et al.[1]

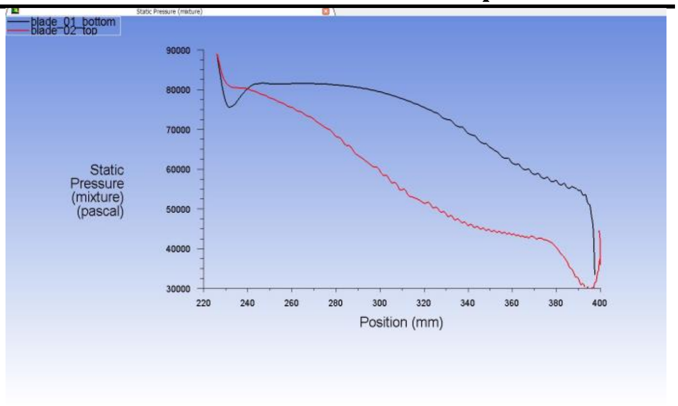

Fig. 4a Static pressure Distribution with 2D simulation

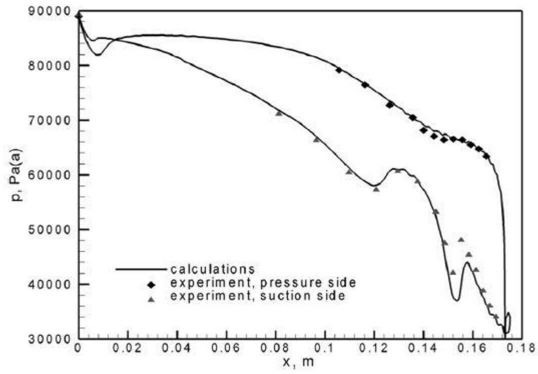

Fig. 4bStatic pressure Distribution by Dykas et al.[1]

B. IAPWS IF-97 Model

In order to verify the accuracy of IAPWS IF-97 model 3D cascade model is simulated in ANSYS CFX with the homogeneous mixture and IAPWS IF-97 as material properties. The geometry and boundary conditions of the cascade were taken to be same as the Dykas simulation, Figure. 5 shows better pressure distribution over the pressure distribution curve obtained in 2D Dykas et al.[1] simulation.

From static pressure distribution curve, it also clears that IAPWS IF97 model gives 10% accurate results at trailing edge of the blade than the two-phase model.

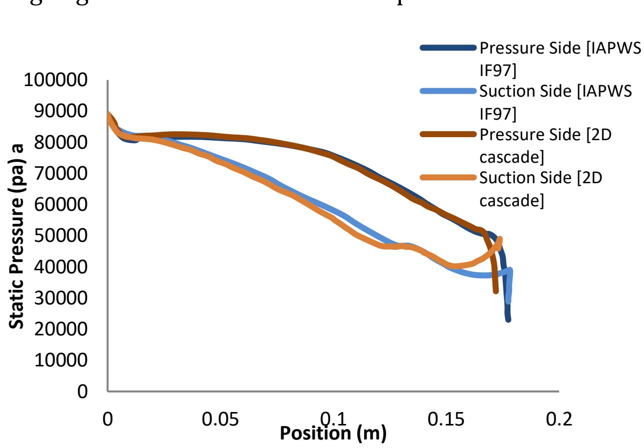

Fig. 5 Static pressure distribution

IV. CONCLUSIONS

The fluid flow properties are important for simulating steam flow problems, specifically when steam properties are far away from the critical region, so IAPWS IF-97is used as equation of state. The IAPWS IF-97 database comparatively represents an accurate equation of state for water and steam properties.

Also from the calculations of flow field of steam turbine, it clears that water liquid mass fraction in cascade has been successfully simulated with the use of IAPWS IF-97 model so that pressure distribution along the surface gives the correct value.

REFERENCES

[1] SławomirDykas, MirosławMajkut, Michal Strozik&KrystianSmołka,"Experimental study of condensing steam flow in nozzles and linear blade cascade", International Journal of Heat and Mass Transfer, vol. 80, pp. 50-57, Sept 2014.

[2] Guojie Zhang, Sen Zhang, Zhongning Zhou, Yimin Li, Liang Wang & Cong Liu,"Numerical study of condensing flow based on the modified model", Applied Thermal Engineering, vol. 127, pp. 1206-1214, Aug 2017.

[3] S.M.A. Noori Rahim Abadi, A. Ahmadpour& S.M.N.R. Abadi, J.P. Meyer,"CFD-based shape optimization of steam turbine blade cascade in transonic two-phase flows", Applied Thermal Engineering, vol. 112, pp. 1575-1589, Oct 2016.

[4] W.F. Fuls,"Accurate stage-by-stage modelling of axial turbines using an appropriate nozzle analogy with minimal geometric data", Applied Thermal Engineering, vol. 116, pp. 134-146, Jan 2017.

[5] S. Pierret& R. A. Van den Braembussche,"Turbomachinery Blade Design Using a Navier-Stokes Solver and Artificial Neural Network", Journal of Turbomachinery, vol. 121, pp. 326-332, April 1999.

[6] Bo Song, Wing F. Ng, Joseph A. Cotroneo, Douglas C. Hofer & Gunnar Siden,"Aerodynamic Design and Testing of Three Low Solidity Steam Turbine Nozzle Cascades", Journal of Turbomachinery, vol. 129, pp. 62-71, Jan 2007.

[7] Chao Zhou, Howard Hodson& Christoph Himmel,"The Effects of Trailing Edge Thickness on the Losses of Ultrahigh Lift Low-Pressure Turbine Blades", Journal of Turbomachinery, vol. 136, pp. 081011.1-081011.9, Aug 2014.

[8] D. Corriveau& S. A. Sjolander,"Influence of Loading Distribution on the Off-Design Performance of High-Pressure Turbine Blades", Journal of Turbomachinery, vol. 129, pp. 563-571, July 2007.

[9] Lin Cai& Miao He,"A Numerical Study on the Supersonic Steam Ejector Use inSteam Turbine System", Hindawi Publishing Corporation, vol. 651483, Dec 2012.

[10] Assim H. Yousif, Amer M. Al-Dabagh&Reyadh Ch. Al-Zuhairy, "Non-equilibrium spontaneous condensation in transonic steam flow." International Journal of Thermal Sciences, vol. 68, pp. 32-41, Feb 2013.