Numerical Analysis of Flow Induced Vibrations of a Low-Pressure Steam Turbine Rotating Blade

T. Molale 1*, N. Ahmed 1 and M. Bhamjee 1

1Department of Mechanical Engineering Science, University of Johannesburg, South Africa. Corresponding Author; molale@lagrangeengineers.com

Abstract-
This paper presents the results of a numerical synthesis and characterization of vibrations of low-pressure steam turbine last stage rotating blades. A Fluid Structure Interaction (FSI) study is carried out using ANSYS Fluent 18.1 and ANSYS Mechanical 18.1. Using a one way coupling between ANSYS Fluent and ANSYS Mechanical, it was possible to link the two systems and allow pressure force calculations from Fluent to be used for the blade excitation in ANSYS Mechanical. The result of simulation shows that the blade exhibits vibrations which are characterized by amplitude modulation. An approximation for the equation of motion along the axial, radial and tangential direction was uncovered and fit. The three approximations for the blade vibrations show a good agreement with results from ANSYS Mechanical. The effect of liquid mass (droplets) in the flow in the blade vibration was also investigated numerically. It is shown that increase in liquid mass is directly correlated with increase in the amplitude of the vibrations but has no effect on the frequency of the vibrations.

1. Introduction

As steam is expanded in the low-pressure steam turbine, condensation takes place and tiny droplets form and bind together to form bigger droplets through nucleation [1]. The flow of droplets and vapour moves along the steam turbine at high speed to the exhaust section of the low-pressure turbine [1]. This flow collides with stationary and rotating blades causing excitation of the blades [1]. This investigation seeks to characterize flow induced vibrations of low-pressure steam turbine blade during steady state operation.

In a low-pressure steam turbine, steam flow causes vibrations on the blades and simultaneously the structural vibrations modify the flow structure, a phenomenon commonly referred to as Fluid Structure Interaction (FSI) [1-3]. The modelling approach for FSI involves linking the fluid flow model with the structural model system using a coupling mechanism [12]. A common time step, sharing of data between both models and synchronization is required, leading to high demands in computing resources [1-3]. A common practice within the research community has been to consider the steam flow model and the structural model as decoupled systems [1-3].

One difficulty in the analysis of vibration of low-pressure steam turbine blades is to accurately account for excitation force. Experimental data on low pressure steam turbine flows is rarely available in the open literature making validation of numerical models for steam turbine models a difficult task. At the Institute of Thermal Turbomachinery and Machinery Laboratory (ITSM), several researchers have carried out numerical modelling of two-phase flows in a low-pressure steam turbine rig [2-4]. The focus of the work at ITSM has been in understanding flow characteristics of wet steam using an experimental rig and Computational Fluid Dynamics
(CFD) simulations [2-4]. Another major wet steam modelling study was done by researchers from several institutions for the International Wet Steam Modelling Project (IWSMP) [5]. In the IWSMP project, different groups carried out an evaluation of wet steam models implemented in common CFD codes. The work by IWSMP showed that CFD results from several codes had noticeable variances [5]. In addition, some of the codes could not reproduce satisfactory results across all nozzles that were being considered for the study [5]. A calibration was required to bring some results into agreement with other codes and experimental results [5]. More work is required to improve wet steam modelling for a consensus on best practice around the subject to emerge.

Booysen et al. [6] investigated low cycle fatigue of low-pressure steam turbine blades caused by start-up resonance conditions. To simulate resonance conditions, Booysen et al. [6] uses a sinusoidal to estimate time varying steam force over the surface of the blade. Furthermore, Booysen et al [6] experimentally determined and documented the blade material as chromium martensitic stainless steel X22CrMoV12-1. In addition to low cycle fatigue as investigated by [6], low pressure steam turbine blades experience high cycle fatigue due to small strains encountered during the steady state operating conditions.

Several explanations for pressure force fluctuations during steady state operating conditions of low-pressure steam turbine blades include turbulent flow effects, rotating instability [7] and tip clearance effects [8].

In this work, a numerical investigation of flow induced vibrations at steady state operating conditions in a 600MW low pressure steam turbine blade is carried out by taking into account FSI. Focus is placed on characterizing vibrations in order to uncover high cycle fatigue of low-pressure steam turbine blades. Condensing steam flow conditions are considered in line with prevailing plant conditions in a low-pressure steam turbine. Using ANSYS Fluent 18.1, a homogeneous two-phase flow of vapour and droplets is modelled. A free standing low-pressure steam turbine blade is modelled using ANSYS Mechanical 18.1.

2. Methodology

ANSYS Fluent and ANSYS Mechanical was used for modeling wet steam flow and blade structural system respectively. The Fluent condensation theory is based on the classical non-isothermal nucleation theory described in [9]. In this theory, condensation is known to be caused by binding together of vapour nuclei [9].

2.1 Wet Steam Theory

To model condensation, Fluent uses the pre-conditioned Navier-Stokes for compressible flow given here in vector form as:

$$\frac{\partial W}{\partial \tau} \int_V Q dV + \hat{f}[F - G] \cdot dA = \int_V H dV$$

where \( Q, W, F \) and \( G \) are vectors \( \{p, u, v, w, T\}^T \), \( \{\rho \rho u \rho v \rho w \rho E\}^T \), \( \{\rho v (\rho u v + p\hat{\tau}) (\rho v w + p\hat{j}) (\rho v E + p\hat{v})\}^T \) and \( \{0 \, \tau_{xi} \, \tau_{yi} \, \tau_{zi} \, (\tau_{ij} v_j + q)\}^T \) respectively and \( E \) is the total energy of the system [12].

The \( \frac{\partial W}{\partial \tau} \) is a Jacobian matrix given by:
and vector $H$ contains the source terms for body forces and energy sources [12].

Two additional transport equations are required for modeling the evolution of number of droplets and conservation of the mass fraction of liquid phase. The first such transport equation governs the mass fraction of the liquid phase and is given by:

$$\frac{\partial \rho \beta}{\partial t} + \nabla \cdot (\rho \vec{v} \beta) = \Gamma$$

(3)

where $\Gamma$ is the mass generation rate due to condensation given by Eq. 4 and $\rho$ is the density of the mixture, see Eq. 9 [12]. In classical nucleation theory, the mass generation rate $\Gamma$ is given by:

$$\Gamma = \frac{4}{3} \pi \rho_l I r_s^3 + 4 \pi \rho_l \eta \bar{r}^2 \frac{\partial \bar{r}}{\partial t}$$

(4)

Droplets larger than the Kelvin-Helmholtz critical radius $r_s$ grow while smaller droplets evaporate [12]. $I$ is the average nucleation rate defined by Eq. 8, $\bar{r}$ is the average droplet radius, $\rho_l$ is the density of the liquid phase and $\frac{\partial \bar{r}}{\partial t}$ is the growth rate of the droplets given by the modified form of Gyamarthy’s formula in Young [10]:

$$r_s = \frac{2\sigma}{\rho_l R \ln(S)}$$

(5)

Where the super saturation ratio $S$, is given by [10, 12]:

$$S = \frac{\rho}{\rho_{sat}(T)}$$

(6)

The second equation which governs the evolution of droplet numbers is given by [12]:

$$\frac{\partial \rho \eta}{\partial t} + \nabla \cdot (\rho \vec{v} \eta) = \rho I$$

(7)

where $I$ is the average nucleation rate given by Eq. 8, mixture density $\rho$ is given by Eq. 9 and number of droplets per unit volume $\eta$ by Eq. 10 [12]:

$$I = \frac{q_v}{(1+\beta)} \left( \frac{\rho_v}{\rho_l} \right)^2 \sqrt{\frac{2\sigma}{M_n}} \frac{4\pi r_s^2 \sigma}{3K_b T} \exp\left(-\frac{4\pi r_s^2 \sigma}{3K_b T}\right)$$

(8)

$$\rho = \frac{\rho_v}{(1-\beta)}$$

(9)
\[ \eta = \frac{\beta}{(1-\beta)v_d(\rho_l/\rho_v)} \quad (10) \]

Non-isothermal correction factor \( \theta \), is given by [12]:

\[ \theta = \frac{2(\gamma-1)}{(\gamma+1)} \left( \frac{h_{lv}}{RT} \right) \left( \frac{h_{lv}}{RT} = 0.5 \right) \quad (11) \]

The average droplet volume \( V_d \) of a droplet of average radius \( \bar{r}_d \) is given by [12]:

\[ V_d = \frac{4}{3} \pi \bar{r}_d^3 \quad (12) \]

2.2 Numerical Scheme

A one-way fluid structure interaction analysis between wet-steam and blade was setup using system coupling feature available in ANSYS Workbench. Pressure force from wet-steam flow is used as an excitation force in the structural system. Simulation of condensation process was controlled by switching on and off, the “Wet Steam” equations in the calculation controls in Fluent. The wet-steam model in Fluent allows for heat and mass transfer between the liquid phase and vapour phase [12]. Finite Element Model (FEM) and transient structural analysis was carried out using ANSYS Mechanical 18.1.

2.3 Fluid Flow Model

ANSYS Bladegen was used to design a 3D model of the blade and flow path. Bladegen allows for the parameter based design of blade and flow path with options for refining the initial design. To simplify the model, cyclic symmetry was assumed and only one blade and one fluid path modelled. Gravitational effects on the flow were assumed to be negligible and therefore not included in the calculations.

The fluid model was meshed using 110,350 3D hexahedral elements with 120,992 nodes. Mesh refinement near the blade surface was applied to the blade model to improve accuracy of the results. The mesh was configured as dynamic with the blade and main flow passage set as deforming. A wet-steam model available in ANSYS Fluent was used for the modelling of the two phase flow conditions available in a low pressure steam turbine. With this model, it was possible to solve heat and mass transfer equations between vapour phase and liquid phase. The realizable \( k-\varepsilon \) turbulence model with standard wall functions was applied for all analysis.

2.4 Blade Model In Fluent

In Fluent, a rotating low-pressure steam turbine blade was modelled as a rotating wall with its rotational velocity set to 3000 rpm in line with steady state operating speed of the steam turbine under investigation. The temperature of the blade was set to match the fluid at inlet of computational domain. To reduce computational cost, no heat transfer between the blade and fluid was included in the calculations. The blade was modelled as non-slip wall with standard roughness models [11] applied.

2.5 Condensing Steam Flow Modelling

Modelling condensing steam flow presents some computational difficulties. The main difficulty arise from numerical modelling of mass transfer between liquid phase and vapour phase.
Another difficulty involves modelling of droplet trajectories and droplet shapes. Fluent makes a number of assumptions [12] about the condensation process including that:

(a) The flow is homogeneous
(b) The droplet growth is based on average mean radius
(c) The droplets are spherical in shape.
(d) The droplets are surrounded by infinite vapor space
(e) The droplets and vapour travel at the same velocity

A transient simulation of a two phase flow through a rotating low pressure steam turbine blade was carried out using ANSYS Fluent. To guide the solution to convergence, wet steam equations were initially not included in the solution. This allowed the solution to be approximated before more computationally demanding calculations were included in the calculations. The focus of the CFD analysis was in determining the total pressure over the surface of the blade which was then used to calculate the excitation force required for the FEA. A time step of 0.001s was used for calculations in ANSYS Fluent.

### 2.6 Boundary Conditions

The following boundary conditions were set for the fluid model. The mean total pressure at inlet is 18793Pa, inlet temperature is 335K, inlet wetness fraction is 0.0794, outlet mean back pressure is 11924Pa, outlet temperature is 330.96K and outlet wetness fraction is 0.083.

### 2.7 Finite Element Modelling

To simulate vibration and stresses on the blade, a finite element model was developed using ANSYS Mechanical 18.1. As in Fluent, only one blade was modeled and cyclic symmetry assumed. The blade hub was setup as a fixed support in ANSYS Mechanical. The pressure surface, suction surface and shroud surface was setup as a Fluid Structure Interface. This allowed for the fluid pressure to be imported directly from Fluent and into Mechanical and to be used as excitation force.

The two models were synchronized through system coupling and a common time step of 0.001s used for all calculations in Fluent and Mechanical. At the end of each time step, fluid pressure data from fluent is imported into ANSYS Mechanical and the solution advances on ANSYS mechanical for each time step. When the calculation in Mechanical is complete, the time step is advanced and Fluent recalculate new pressure values and they are imported into Mechanical. This process was continued until enough data was available for analysis.

### 2.8 Blade Material

The blade material chromium martensitic stainless steel X22CrMoV12-1 was added to ANSYS Mechanical’s material library. These blade material properties were used for all structural calculations. The blades’ material composition and properties were assumed constant throughout the blade.

### 2.9 Blade Excitation

A transient analysis was setup in Mechanical and Fluent systems to investigate the response of the blade structural system. The excitation force on the blade was simulated from the steam
pressure force calculations in Fluent and imported into Mechanical at each time step. This setup allowed for variation in excitation force on the blade body.

The blade hub was setup as a fixed support in ANSYS Mechanical. When setup in this manner, ANSYS Mechanical puts a no displacement constraint on the hub and the hub is fixed to a rotor with no gap or relative motion. As a result, the blade root was not included in the modelling or calculations.

3. Result and discussions

The vibration of a low-pressure blade set with 32 blades was simulated using ANSYS Mechanical. The blades were assumed to be equally placed at 11.25 degrees from each other. The blade’s rotation was set at 3000rpm in line with the steady state operating speeds of the steam turbine. As the blades rotates, it washes across condensing steam flow. The unsteady flow effects of the axially travelling steam introduce time varying steam pressure force on the surface of the blades. In addition, the airfoil design of the blade causes the pressure surface of the blade to experience greater force than the suction surface leading to small deflection of the blade. To measure the deflection of the blade, a probe tool was used in Mechanical to record the time history of the position of blade tip during the transient analysis. Fig. 2-4 shows the tangential, axial and radial position of the blade tip during 0.12s time history.

3.1 Wetness effects

Three case studies involving setting different values of inlet liquid mass fraction were considered for analyzing effects of increased liquid mass on the vibration of the low-pressure steam turbine blade. In all three cases, the flow temperature at inlet, flow pressure at inlet, flow pressure at outlet were kept the same. The only parameter that was varied for the three cases was the wetness fraction at inlet. The results of the non-directional deformation recorded from the blade shroud are shown in Fig. 1.

![Figure 1: Comparison of tangential deformation to when different wetness factors at inlet are considered](image)

In the three wetness effects investigations, the highest deformation is noticed when in the inlet wetness fraction is 0.0794 and the lowest deformation is noticed when the inlet wetness fraction
is 0.005. In all the three cases, the frequency of the vibration is not affected by the variation in the wetness fraction at the inlet. A direct correlation between increased liquid mass and increase in the amplitude of the tangential deformation is noticed.

3.2 Equation of Motion
In this section, the blade deflection at the tip is characterized using the Finite Element Analysis and an equation of motion of the blade tip is approximated. The deflection of the blade is considered in the tangential, axial and radial directions. A case where the steam is considered to enter the flow at a wetness fraction of 0.0794 is considered.

3.3 Tangential Motion
The blade’s tangential vibrations exhibit characteristics of an amplitude modulated vibration. The equation of motion is approximated by the following function;

\[ y(t) = 3.5 \cdot (0.017 \cos(2\pi f_c t)) \cdot (0.2 \sin(2\pi f_m t) + 1) + c \]

Where the \( y(t) \) is the tangential displacement of the blade tip with respect to the blade’s equilibrium position, \( f_c = 137 \text{Hz} \) is the carrier frequency of the tangential mode of vibration, \( f_m = 15 \text{Hz} \) is the modulating frequency of excitation force and \( c = -0.047 \text{mm} \) is position at which the vibrations are centered about. The estimation of the tangential vibration using the equation of motion shows good agreement with the results from ANSYS Mechanical.

3.4 Axial Motion
The blade’s axial vibration exhibit amplitude modulated vibration characteristics. The equation of motion of the blade tip axial displacement is approximated by the following function;

\[ z(t) = 3.5 \cdot (0.014 \cos(2\pi f_c t - 91)) \cdot (0.2 \sin(2\pi f_m t + 40) + 1) + c \]
Where the $z(t)$ is the axial displacement of the blade tip with respect to the blade’s equilibrium position, $f_c=137\text{Hz}$ is carrier frequency of the axial mode of vibration, $f_m=15\text{Hz}$ is the modulating frequency of excitation force and $c=0.055\text{mm}$ is position at which the vibrations are centered about. The estimation of the axial vibration using the equation of motion shows good agreement with the results from ANSYS Mechanical.

![Axial Deflection of Blade Tip When Inlet Wetness is 0.0794](image)

Figure 3: Comparison of axial deflection using FEM and estimation using equation of motion

### 3.5 Radial Motion

The blade’s radial vibration exhibits sinusoidal vibrations upper bounded by a cycloid function. The equation of motion of the blade tip radial displacement is approximated by the following function;

$$x(t) = [0.042 \cos(2\pi f_c t - 91.2) + 0.005]. [0.15 \sin(2\pi f_m t) + 1] + c$$

Where the $x(t)$ is the radial displacement of the blade tip with respect to the blade’s equilibrium position, $f_c=137\text{Hz}$, $f_m=8\text{Hz}$ and $c=0.0008\text{mm}$. The estimation of the radial vibration using the equation of motion shows good agreement with the results from ANSYS Mechanical.
4. Conclusion
In this paper, we carried out the numerical synthesis of the vibration of the low-pressure rotor blade of a 600MW steam turbine rotating blade. Wet steam simulation was done using the ANSYS Fluent 18.1 inbuilt wet steam model. The numerical simulation shows that there is direct correlation between the increase in liquid mass and increase in strains on the rotating blades.

The work also revealed the equation of motion of the blade tip vibrations under wet steam conditions. More work is needed to define the equation of motion in general sense using physical quantities. The work did not analyze various flow effects with could explain the excitation of the blade during the steady state operating conditions of a steam turbine. More work is needed to understand high cycle fatigue that may result from the vibrations uncovered in this work.

Acknowledgements
The Centre for High Performance Computing (CHPC) is acknowledged for the use of the CHPC facilities. Opinions expressed and conclusions arrived at, are those of the authors and are not necessarily to be attributed to the University of Johannesburg or the CHPC.

Reference

[1] Xie, D., Yu, X., Li, W., Hou, Y., Shi, Y., and Cai, S., (2010). Numerical Simulation of Water Droplets Deposition on the Last-State Stationary Blade of Steam Turbine. Energy and Power Engineering, 2, pp. 248-253. DOI: 10.4236/epe.2010.24036

[2] Gröbel, M., Starzmann, J., Schatz, M., Eberle, T., Vogt, D.M., and Sieverding, F., (2014). Two-Phase Flow Modeling and Measurements in Low-Pressure Turbines: Part 1 — Numerical Validation of Wet Steam Models and Turbine Modeling. Journal of Engineering for Gas Turbines and Power, ASME Paper No. GT2014-25244.

[3] Schatz, M., Eberle, T., Gröbel, M., Starzmann, J., Vogt, D.M., and Sürken, N., (2014). Two-Phase Flow Modeling and Measurements in Low-Pressure Turbines: Part 2 — Turbine Wetness Measurement and Comparison to CFD-Predictions. Journal of
Engineering for Gas Turbines and Power, ASME Paper No. GT2014-25245

[4] Starzmann, J., Kaluza, P., Casey, M.V., and Sieverding F., (2013). “On Kinematic Relaxation and Deposition of Water Droplets in the Last Stages of Low Pressure Steam Turbines,” Journal of Turbomachinery, ASME Paper No. GT2013-95179.

[5] Starzmann, J., Hughes, F.R., Schuster, S., White, A.J., Halama, J., Hric, V., Kolovratník, M., Lee, H., Sova, L., Šťastný, M., Grübel, M., Schatz, M., Vogt, D.M., Patel, Y., Patel, G., Turunen-Saaresti, T., Gribin, V., Tishchenko, V., Gavrilov, I., Kim, C., Baek, J., Wu, X., Yang, J., Dykas, S., Wróblewski, W., Yamamoto, S., Feng, Z., and Li, L., (2018). Results of the International Wet Steam Modeling Project,” Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 0(0),pp. 1-21.

[6] Booyesen, C., Heyns, P.S., Hindley, M.P. and Scheepers, R., (2015). Fatigue life assessment of a low pressure steam turbine blade during transient resonant conditions using a probabilistic approach. International Journal of Fatigue, 73, pp. 17–26

[7] Garg, D.K., Chambalwar, S., Sarode, J., and Dhanopia, A., (2015). Investigation of Rotating Instability in the Last Stage of Low Pressure Turbine during Low Volume Flow Operation. International Journal of Recent Advances in Mechanical Engineering, 4(2), DOI: 10.14810/ijmech.2015.4204

[8] Sun T., Petrie-Repar P., and Qi D., (2017). Investigation of Tip Clearance Flow Effects on an Open 3D Steam Turbine Flutter Test Case. ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition, 8, ASME Paper No. GT2017-64021

[9] Wyslouzil, B.E., and Seinfeld, J.H., (1992). Nonisothermal homogeneous nucleation. The Journal of Chemical Physics, 97(4),DOI: 10.1063/1.463055

[10] Young, J. B.,(1992). Two-dimensional, nonequilibrium wet-steam calculations for nozzles and turbine cascades. Journal of Turbomachinery, 114, pp. 569–578

[11] Sommerfeld M., and Huber N. (1999). Experimental analysis and modeling of particle-wall collisions. Int. J. Multiphase Flow. 25. 1457—1489.

[12] “ANSYS Fluent 12.0/12.1 Documentation”, from http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node337.htm