Cavitation Draft Tube Analysis of Francis Turbine Loads Variations: A Case Study from Cirata Hydropower Plant

R T Sibuea\textsuperscript{1}, H Mirmanto\textsuperscript{1,2}\\
\textsuperscript{1} Pembangkitan Jawa Bali, West Java 40558, Indonesia \textsuperscript{2}Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Kampus Keputih-Sukolilo, Surabaya 60111, East Java, Indonesia
\textsuperscript{a}risma@ptpjb.com \textsuperscript{b}bsamir@me.its.ac.id

Abstract. Hydropower plant (PLTA) is a power plant of EBT (Renewable Energy), which utilizes water as a primary energy source. PLTA has a very important role in balancing the stability of network systems. In improving network system stability, hydropower is operated in a wide load range, frequent start-stop, and varying loads. Other functions are also as a buffer of peak load and back up system in case of interference in the network system. Francis turbine is the most widely used turbine and has a maximum efficiency of 93-95\%. Problems related to the operation of hydropower are operational instability, cavitation, vortex breakdown, pressure shocks, vibration and noise that sometimes cause turbine system failure. In relation to the above conditions then in this thesis will be studied how the occurrence of cavitation due to variations loading. Case study conducted in this research is at Cirata hydropower plant with capacity of 126 MW per unit, which has type Francis turbine vertical shaft (variation of load starts 40\% and 100\%). The research method is done numerically, with modeling and flow simulation using ANSYS CFX 18.2. The results of this study show that at 40\% load operating conditions there is a very low-pressure area (under vapor pressure) and the emergence of a vortex rope that causes cavitation in the draft tube area, especially in the elbow region. While at 100\% load operating conditions only slightly visible low-pressure area and no vortex rope appears.

1 Introduction
PLTA Cirata is a hydroelectric power plant that utilizes water from Citarum river flow as primary energy. The construction was divided into two stages, Cirata I was built in January 1984, it has 4 units, and started to operate in September 1988, and Cirata II also has 4 Units, started operating in 1997. The installed capacity of UP Cirata is 1008 MW, consists of 8 Units and each of them has a capacity of 126 MW. The Cirata hydropower plant is the largest in South East Asia. PLTA Cirata has an important role in maintaining the balance or stability of the 500 kV Jawa Bali system. PLTA Cirata is operated under fluctuating and frequent start-stop conditions, Load Frequency Control, and Governor Free, to improve the stability of 500 kV network systems that currently have high loads. Thus, the reliability of the generating unit is required in the operating conditions that change frequently to maintain the system frequency balance. However, in certain operating conditions, there are frequent emergences of excessive vibration accompanied by a noise that alleged the occurrence of the cavitation phenomenon. Cavitation in hydraulic machinery presents unwanted consequences such as
flow instabilities, excessive vibrations, damage to material surfaces, and degradation of machine performance [1]. The spiral-type vortex rope occurs typically at the "part-load condition" (below 60% of the full load or so), whose strong unsteady motion is associated with severe low-frequency pressure fluctuations that may damage the turbine's normal operation and even the safety of the whole power station [2]. The purpose of this research is to know the vortex rope, which can cause the cavitation phenomenon at part load operating conditions and resulted in pitting on the draft tube [3]. Flow characteristics that occur in the draft tube are simulated using CFD commercial software. To know the vortex rope that happened then done simulation unsteady or transient by using the model of turbulence Detached Eddy Simulation (DES) [4].

2. Numerical Method

The computational domain comprises three sub-domains: distributor (stationary domain-I), runner (rotating domain), and draft tube (stationary domain-II). Guide Vane and draft tube are located upstream and downstream the runner, respectively. The numerical model of the Francis turbine is shown in Fig. 1. Both stationary domains are connected with the runner using a general grid interface (GGI) type interface. The runner is attached with guide vane and draft tube by frozen rotor interface. The distributor is 24 guide vanes. The runner includes a hub, shroud, 16 full-length blades. The draft tube is an elbow with varying cross-section connected at the runner outlet. The inlet boundary condition was prescribed velocity inlet at the guide vane and the outlet boundary condition was the pressure outlet at the draft tube.

A CFD Simulation software was used to create the numerical model, generated a hexahedral mesh, simulated the flow domain, and analyzed the results. The mesh was created using a three-dimensional structured multi-block technique with meshing software independently in each subdomain. Fig. 2 shows the hexahedral mesh developed in the turbine. A continuous mesh from the guide vane inlet to the runner inlet was constructed in the distributor. The total number of mesh made is about 3 million nodes for the guide vane there is 282,624 nodes, runner 904,800 nodes, 1,871,240 nodes for the draft tube.

The simulation in this study focuses on the condition of part-load operation (40%) and full load operation. At part-load operation with the opening angle of guide vane, \( \alpha = 15^\circ \); turbine rotation, \( n = 187.5 \text{ rpm} \); discharge, \( Q = 51.4 \text{ m}^3/\text{s} \). At full load operation with the opening angle of guide vane, \( \alpha = 30^\circ \); turbine rotation, \( n = 187.5 \text{ rpm} \); discharge, \( Q = 121 \text{ m}^3/\text{s} \). The time step of the simulation is 0.00178 s.
Figure 2. Hexahedral mesh of Francis turbine

Run definition method at ANSYS CFX 18.2 can be seen at Tabel 1.

**Tabel 1. Run Definition Method at ANSYS CFX 18.2**

| Parameter                | Deskripsi                                                                 |
|--------------------------|---------------------------------------------------------------------------|
| Simulated components     | Domain 1: Stationary - guide vane                                           |
|                          | Domain 2: Rotating - runner blade                                          |
|                          | Domain 3: Stationary, draft tube                                           |
|                          | All domains have 0 atm as reference pressure                               |
| Grid type                | Hexahedral                                                                |
| Simulation type          | Steady dan Unsteady (Transient)                                           |
| Interfaces               | Rotational periodicity, Mesh connection: GGI                              |
|                          | Interface 1 and 2: General connection, transient rotor stator with automatic pitch change, Mesh connection: GGI |
|                          | Mixing model : Frozen Rotor                                               |
| Boundary conditions Inlet| Mass flow rate: 51.4 m³/s, 121 m³/s, turbulence intensity 5%              |
|                          | Cylindrical Velocity Components 40 %: (axial 0 m/s, radial -4 m/s theta -10 m/s) |
|                          | Cylindrical Velocity Components 100 %: (axial 0 m/s, radial -2.1 m/s theta -8.4 m/s) |
|                          | Outlet: relative pressure (0 Pa) and Pressure profile blend : 0.05        |
| Fluid                    | Walls: Smooth walls with no slip condition                                 |
| Solver control           | Water atau air pada 25°C                                                  |
|                          | Advection Scheme: High resolution                                         |
|                          | Turbulence numerics: First order                                          |
|                          | Transient scheme: Second order backward Euker                            |
| Convergence control      | Maximum coefficient loops : 2, Residual target (RMS): 1E-5               |
| Turbulence model         | k-Epsilon, RNG k-Epsilon, Shear Stress Transport dan Detached Eddy Simulation |

The parameter data was obtained from the numerical simulation, which is used for the calculation of hydraulic efficiency. Hydraulic Efficiency is the ratio between the energy that is generated by the turbine with the energy of water coming from the potential energy. Hydraulic efficiency is an indicator used as a validation guide to compare the simulation results on some turbulence models [5]. Table 1 shows the actual and numerical (standard k-ε, RNG k-ε, SST k-ω, and DES) at part-load operating conditions and full load operating conditions. Based on table 1 below, it can be seen that the smallest hydraulic efficiency difference is found in the DES turbulence model, which is 0.07% for 40% load.
operating conditions and -1.18% for 100% load operating conditions. The minimum difference between actual and numerical hydraulic efficiency is in the detached eddy simulation (DES) turbulence model.

### Table 2. Experimental and numerical hydraulic efficiency at part-load operating condition

| Operating Condition | Hydraulic efficiency, $\eta_h$, % | Actual | Numerical | Standard k-$\varepsilon$ (steady) | RNG k-$\varepsilon$ (steady) | SST k-$\omega$ (steady) | DES SST (Transient) |
|---------------------|----------------------------------|--------|----------|---------------------------------|-----------------------------|------------------------|--------------------|
| GVO, ($\alpha^\circ$) | Q, m$^3$/s | 15$^\circ$ | 51.40 | 84.77% | 82.97% | 83.08% | 82.67% | 84.84% |
| 30$^\circ$ | 121.00 | 92.89% | 83.06% | 83.07% | 90.56% | 90.69% | 91.71% |

### 3. Result and discussion

This chapter explains the cavitation phenomenon that occurs on draft tube Francis turbines in different operating conditions with 40% and 100% of the full load using ANSYS CFX 18.2 software. The conducted simulation uses a steady-state and transient simulation. To predict the flow characteristics that are close to the actual, steady-state simulation was conducted in this study using 3 (three) turbulent models of RANS (Reynolds Averaged Navier Stokes) namely Standard k-$\varepsilon$, RNG (Renormalization Group) k-$\varepsilon$, and SST (Shear Stress Transport) k-$\omega$ while unsteady (transient) simulation uses the DES (Detached Eddy Simulation) turbulence. Transient simulations were carried out to well predict the flow characteristics in the form of vortex rope especially in Cirata hydropower turbines.

Before analyzing the flow characteristics that occur in the draft tube, the simulation results are validated first by comparing the operating parameters and turbine performance in a certain operating condition with the actual data at the time of the commissioning unit. The comparison of actual data and numerical simulations resulted in turbulence model simulation with the smallest error value. The result can be used as an analysis of the flow characteristics that occur namely pressure distribution, velocity distribution, vortex flow, velocity vector, and vortex rope on the draft tube component.

To get the simulation results that are close to the actual condition, the numerical simulation in this study was done by setting the mass flow rate parameters in the boundary condition inlet on the guide vane domain and the guide vane opening angle. For example, to set the opening angle of the guide vane 15$^\circ$ for the mass flow rate at 40% load and the opening angle of the guide vane 30$^\circ$ for the mass flow rate at a load of 100%.

The numerical result of the pressure distribution contour on the mid-section part along the x-y plane ($z = 0.0$ mm) under 40% load condition from each turbulence model will be analyzed as follows:
Figure 3. Pressure distribution contour on 40% load

Steady simulation results with the standard turbulence model k-\(\varepsilon\), RNG k-\(\varepsilon\), and SST k-\(\omega\) (figure 3) show different low-pressure areas, where the standard k-\(\varepsilon\) and SST k-\(\omega\) are symmetrical at the exit runner with pressure values > 3170 Pa. Meanwhile, the low-pressure area in the k-\(\varepsilon\) RNG turbulence model appears to control the outer diameter part with a lower pressure value (<3170 Pa). Pressure recovery that occurs in the RNG turbulence model k-\(\varepsilon\) is slower than the other two models (standard k-\(\varepsilon\) and SST k-\(\omega\)). This is shown by the longer color contour degradation. On the outer diameter of the elbow draft tube side all turbulence models show a higher-pressure area, this is indicated by the orange color covering the walls. In this area, the bubbles will burst and cause erosion.

In transient simulations with the DES turbulence model, the condition of the low-pressure area (under vapor pressure) clearly shows the exit runner area in dark blue. The low-pressure area in the DES simulation appears to have a wider area compared to the steady simulation with the RNG turbulence model k-\(\varepsilon\). The contour of the low-pressure region in this simulation is located inside the vortex rope. Then the flow moves across the channel to the elbow with increasing pressure. If the area being passed has a pressure exceeding the vapor pressure, then the vapor bubbles will burst and cause erosion in the channel wall. This can be shown in both models where the higher-pressure area is marked by the degradation of the light blue to orange. However, the pressure recovery of the DES simulation results is faster with the outer elbow symmetry position than the RNG turbulence model k-\(\varepsilon\). So this area is predicted to cause more significant material damage.
Figure 4. Pressure distribution contour on 100% load

Steady simulation results with standard turbulence models k-ε, RNG k-ε, and SST k-ω (figure 4) show different areas of low-pressure, where the standard k-ε and RNG k-ε are in the middle of the draft tube area and near trailing edge runner with a pressure value (<3170 Pa). While the low-pressure area in the SST k-model turbulence model appears slightly near the lower end of the trailing edge with a lower pressure value (<3170 Pa) while in the middle of the draft tube the pressure value is> 3170 Pa. Pressure recovery that occurs under load operating condition 100% faster than part-load condition (40%). This is shown by the red contour at the elbow position, which has reached a pressure above the atmospheric pressure. On the outer diameter of the elbow draft tube all turbulence models show a higher-pressure area, this is shown in red covering the wall. In these areas, the steam bubbles burst and cause erosion. However, in a 100% operating condition with a very low-pressure region, the potential for the steam bubbles to burst and cause erosion becomes smaller.

In the unsteady (transient) simulation results with the turbulence model DES (figure 4), the condition of low-pressure under vapor pressure looks very small in the middle part of the draft tube and in the draft tube inlet during trailing. The low-pressure area is marked by a dark blue contour. The low-pressure area in the DES turbulence simulation model is smaller than the steady simulation of the standard turbulence model k-ε and RNG k-ε.
Figure 5. Velocity Vector on 40% load

Figure 5 shows the velocity vector in the x-y midsection plane and section A-A after passing the runner at the turbine nominal rotation and operating condition of 40% load. Analysis in the x-y midsection plane shows the velocity vector flow in the identical draft tube. Where the flow after exiting the runner (inlet draft tube) has high turbulence intensity. This is characterized by the shape of the velocity vector dominated by the longitudinal direction with a higher velocity than other locations. The shape of this vector indicates that the flow at the inlet draft tube forms a vortex at a very high velocity and moves towards the elbow with the conical edge becoming increasingly conical. The vortex at the draft tube inlet is clearer when seen in section A-A. Inside the vortex flow with high velocity, the pressure becomes very low. If the pressure is lower than the vapor pressure, it causes cavitation. Considering that the form of the inner elbow has a large adverse pressure gradient, the flow tends towards the outer elbow accompanied by an increase in pressure (pressure recovery). Furthermore, the flow moves towards the exit draft tube with turbulence intensity that is getting weaker. This is indicated by the velocity vector flow from the elbow to the exit, which appears to have a more transversal shape.

In operating condition with a load of 40%, all turbulence model simulations can be observed showing a wide mixing region of high & low momentum region that is the area where the collision of the opposite flow is between upward and downward flow. This is indicated by the high head loss in the draft tube so that the pressure recovery in the draft tube is less optimal. As a result of less optimal pressure recovery, the hydraulic efficiency is much lower than the efficiency value at the best efficiency point (BEP).

The standard k-ε and the SST k-ω models have a vortex core shape right in the center diameter of the draft tube (symmetry), where the middle part looks brighter and the part near the wall looks darker. This indicates that the flow in the middle has a velocity towards the transversal higher than the flow near the wall. This means that the secondary flow formed in the draft tube has a greater head loss near the wall. Whereas the shape of the vortex core in the k-model and DES RNG turbulence models did not appear symmetrical. The velocity vector in section A-A in the k-R RNG simulation has a vortex on the outer diameter of the draft tube while the DES simulation is on the inner diameter of the draft tube. From the numerical results, it is seen secondary flow in the midsection plane x-y (z = 0.0 mm), that when compared to the results of RNG simulation k-ε and DES, the DES simulation has a smaller area. This causes the value of hydraulic efficiency with the DES turbulence model of 84.84% greater than the RNG turbulence model k-ε of 83.08%.
Figure 6. Velocity Vector on 100% load

Figure 6 shows the velocity vector in the midsection plane x-y and section A-A after passing the runner at the nominal rotation of the turbine and 100% load operating condition. Analysis in the x-y midsection plane shows the velocity vector flow in the identical draft tube. Where the flow after exiting the runner (inlet draft tube) has low turbulence intensity. This is indicated by the shape of the velocity vector, which is dominated by transversal direction with a higher velocity than other locations. This vector shape indicates that the flow in the draft tube inlet moves towards the outlet draft tube at extremely high velocity and moves towards the elbow without a vortex. In the absence of flow vortex with high velocity, therefore low-pressure (under vapor pressure) does not occur at full load. If the pressure is lower than the vapor pressure, it causes cavitation. Considering that the form of the inner elbow has a large adverse pressure gradient, the flow tends towards the outer elbow with an increase in pressure (pressure recovery). Furthermore, the flow moves towards the exit draft tube with turbulence intensity that is getting weaker. This is indicated by the velocity vector flow from the elbow to the exit, which appears to have a more transversal shape.

In 100% load operating condition, all turbulence model simulations can be observed showing that there is no mixing region of high & low momentum region, i.e. the area of collision of opposing flow between an upward and downward flow. This is indicated by the low head loss in the draft tube so that the pressure recovery in the draft tube runs fast. Due to the fast pressure recovery, hydraulic efficiency is higher than the efficiency value of fewer than 40% load operating conditions.

All of four simulations model has the form of a vortex core right in the center diameter of the draft tube (symmetry), which appears brighter in the entire section A-A. This indicates that the dominant flow has a velocity toward transversal. From the numerical results seen secondary flow in the midsection plane x-y (z = 0.0 mm), the DES simulation results when compared to have more vortex areas in the elbow area while for the wake-shaped steady simulation. Of the four simulation models, the DES numerical results are slightly brighter. This causes the value of hydraulic efficiency with the DES turbulence model of 91.71% greater than the three simulations of the standard turbulence model k-\(\varepsilon\), RNG k-\(\varepsilon\), and SST k-\(\omega\) with hydraulic efficiency values of 90.38%, 90, 56%, and 90.69%.
Vortex flow (velocity streamline) simulation results on various turbulence models all show identical flow forms. The shape of the velocity streamline indicates the existence of a rotating vortex rope in the draft tube. Vortex flow on part-load is in the form of a spiral with a helical angle. The flow in the walls of the draft tube has a higher longitudinal velocity than from the cone draft tube to the elbow draft tube. This is seen in red in the area of the draft tube inlet and gradually decreases towards the exit draft tube, which is marked by the degradation of the color towards dark blue.

Steady simulation results with the standard turbulence model k-\(\varepsilon\) (figure 4.9), RNG k-\(\varepsilon\) (figure 4.10), and SST k-\(\omega\) (figure 4.11) show an eccentric vortex flow in a spiral shape with a large helical angle in the center of the draft tube. Flow at the center of vortex flow has an irregular (turbulent) shape. The vortex flow diameter looks smaller in the k-standard standard turbulence model while in the k-R RNG turbulence model and the k-S SST are greater.

The unsteady (transient) simulation with the DES turbulence model (figure 4.12) shows an eccentric vortex flow in a spiral shape with a large helical angle in the middle of the draft tube compared to steady simulation. Flow at the center of vortex flow has an irregular (turbulent) shape.

**Figure 7.** Vortex Flow on 40% load
Figure 8. Vortex Flow on 100% load

All of the vortex flow (velocity streamline) simulation results on various turbulence models show identical flow forms. The shape of the velocity streamline indicates the existence of a rotating vortex rope in the draft tube. Vortex flow on part-load is in the form of a spiral with a helical angle. The flow in the walls of the draft tube has a higher longitudinal velocity from the cone to the elbow draft tube. This is indicated by red color in the area of the draft tube inlet and gradually decreases towards the exit draft tube, which is marked by the degradation of the color towards dark blue.

Steady simulation results with the standard turbulence model k-\(\varepsilon\) (figure 4.9), RNG k-\(\varepsilon\) (figure 4.10), and SST k-\(\omega\) (figure 4.11) show an eccentric vortex flow in a spiral shape with a large helical angle in the center of the draft tube. The flow at the center of vortex flow has an irregular (turbulent) shape. The vortex flow diameter looks smaller in the k-\(\varepsilon\) standard turbulence model but greater in the k-R RNG turbulence model and the k-S SST.

The result unsteady (transient) simulation with the DES turbulence model (figure 4.12) shows an eccentric vortex flow in a spiral shape with a large helical angle in the middle of the draft tube compared to steady simulation. The flow at the center of vortex flow has an irregular (turbulent) shape.

All vortex flow (velocity streamline) simulation results on various turbulence models show identical flow. The shape of the velocity streamline indicates the absence of a rotating vortex rope in the draft tube. The flow in the walls of the draft tube has a higher longitudinal velocity from the cone draft tube to the elbow draft tube. This can be seen as red in the center draft tube area until it passes through the elbow side (bottom) and gradually decreases towards the exit draft tube, which is marked by the degradation of the color towards dark blue.

Simulation results with the four standard turbulence models k-\(\varepsilon\), RNG k-\(\varepsilon\), SST k-\(\omega\) and DES (figure 8) show the absence of eccentric vortex flow in the form of a spiral in the middle of the draft tube.

Vortex rope observation was performed using the iso surface method on the fluid pressure vapor value in the draft tube domain. The results of this transient simulation show the occurrence of vortex rope in the cone area until the elbow draft tube (see Figure 9). This vortex rope is a flow that has a low-pressure and the potential for cavitation. This vortex rope can cause undesirable operating characteristics such as decreased efficiency, noise, vibration, variations in power output, runner
vertical movements, and pressure fluctuations in draft tubes. The transient simulation results at 100% load do not indicate no-vortex rope. Therefore, it is not shown in this paper.

![Vortex Rope with DES turbulence model on 40% load](image)

**Figure 9.** Vortex Rope with DES turbulence model on 40% load

4. **Conclusions**

The flow characteristics of a Francis turbine can be analyzed numerically using the RANS equation with an appropriate turbulent model. Pressure distribution under operating conditions of 40% load (part-load) shows a low-pressure area under vapor pressure (<3170 Pa) in the middle of the cone draft tube so that there is a potential for cavitation that results to damage draft tube material. The distribution of pressure under 100% load operating conditions shows a small area of low-pressure under vapor pressure (<3170 Pa) in the middle of the cone draft tube so that the potential for cavitation that results in damage to the draft tube material is very low. Vortex rope appears at 40% load operating conditions (part-load) while in operating conditions 100% load does not appear vortex rope. From the results of numerical simulations show that transient simulations with the DES turbulence model can be used for vortex rope analysis that occurs in the draft of the Francis turbine tube.

5. **References**

[1] X. Escaler, E. Egusquiza, M. Farhat, F. Avellan and M. Coussirat 2006 *Detection of cavitation in hydraulic turbines (Mechanical Systems and Signal Processing)* p 983–1007.

[2] R. K. Zhang, J. Z. Wu, S. Y. Chen, Y. L. Wu and S. H. Liu 2009 *Characteristics and Control of the Draft-Tube Flow in Part-Load Francis Turbine* (Journal of Fluids Engineering, vol. 131) p 14

[3] K. Patel, V. Chauhan, Y. Desai 2011 *Development of Francis Turbine using Computational Fluid Dynamics* (India: IIT Madras)

[4] H. Foroutan and S. Yavuzkurt 2014 *Flow in the Simplified Draft Tube of a Francis Turbine Operating at Partial Load—Part II: Control of the Vortex Rope* (Journal of Applied Mechanics)

[5] R. Goyal, C. Trivedi, B. K. Gandhi and M. J. Cervantes 2017 *Numerical Simulation and Validation of a High Head Model Francis Turbine at Part Load Operating Condition* J. Inst. Eng. India Ser. C.