

PAPER • OPEN ACCESS

Modal analysis of Kaplan turbine in Haditha hydropower plant using ANSYS and SolidWorks

To cite this article: Abdulmuttalib A. Muhsen *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1105** 012056

View the [article online](#) for updates and enhancements.

Modal analysis of Kaplan turbine in Haditha hydropower plant using ANSYS and SolidWorks

Abdulmuttalib A. MUHSEN^{1,2}, Ali A. Resan AL-MALIK³, Bashar H. ATTIYA², Omar Farhan AL-HARDANEE⁴ and Karam Aied ABDALAZIZE²

¹ Faculty of civil and Transport Engineering, Poznan University of Technology, Poznan, Poland

² Haditha Hydropower Station, Ministry of Electricity, Haditha, Iraq

³ College of Engineering, Misan University, Misan, Iraq

⁴ College of Engineering, Ankara Yıldırım Beyazıt University, Ankara, Turkey

Abstract. In this study, numerical analysis is conducted to investigate the failure modes in Kaplan turbine. All necessary steps for Kaplan turbine failure analysis are presented in this work using the modal analysis computational approach. The modal behavior analysis is carried out on a model of an existing Kaplan turbine blade, which is based on the existing turbine used in Haditha hydropower plant in Iraq. This work investigates the modal behavior of the blade of interest, which aid in predicting structural damage initiation. The Kaplan turbine blade is designed using the commercial software ANSYS and SolidWorks. To simulate the blade in operation, the blade is fixed from one end, and all degrees of freedom are measured. Moreover, the turbine blade is moved and rotated to simulate multiple operational conditions. Both mode shapes and natural frequencies are predicted and analyzed using the two aforementioned commercial software and the numerical formula involving the arrest Lanczos method. It is clear from the results that the natural frequency of the specified mode shape does not match with the natural frequency of the runner blade. Hence, there is no failure due to resonance phenomenon in this specific Kaplan turbine. The future work must investigate other aspect of the failure modes in such turbine, such as unbalance dynamic loading. The Results obtained from this study will help study the different possibilities for detecting the failure of the Kaplan blade by examining the modal behavior of the blade.

Keywords: Kaplan turbine; Modal analysis; Failure analysis; ANSYS; Solid Works.

1 Introduction:

Vibrations induced by transient flow behaviors and its adverse effects are critical problem in hydropower plants. This kind of adverse fluctuations can affect the structural integrity of the hydro unit in terms of the components fatigue as well as the unit stability and performance. As soon as, the operating frequencies match with the unit resonance frequency, the whole unit operational mode would become unstable, and one example of such operational issues is seen in Haditha hydropower plant. Haditha plant is the second largest hydropower plant in Iraq. The plant is located at Euphrates river in the west part of the country, which contains six vertical types Kaplan turbines with a 110 MWe per turbine. Haditha plant operating conditions is highly fluctuated due to the fact that it is the only fully operated plant in the west region of Iraq. This fact forces the plant to operate at highly fluctuated and wide operational regimes, which in most cases lies out of the optimum operational



zones, to provide continues power supply to the customers. Severe cavitation which results in high structural vibration levels because of the high dynamic loading, and runner blade erosion damages have been occurred due to these operating conditions. Moreover, structural cracks have been observed at the runner blades, and these issues increases with the continues off-design operation of the units. The aforesaid operational demands made several companies to be forces to operate at low water levels which is associated with low efficiency, making the turbine suffer from high dynamic loading and severe cavitation erosion [1].

Operating units at extreme off-design conditions cause blade damages, wear in various turbine parts, and eventually fatigue damage. Figure 1 shows an example in the structural damages that Haditha plant turbines suffered from. Kaplan turbines are widely used in low to medium water heads with high capability to produce power. This is due to the high efficiency in their entire operational range [2,3], and the reason comes from the ability of such turbine to change the angle of blades to adapt to the flow and head requirements to maintain high efficiency. Nonetheless, some failure cases have been reported in Kaplan turbines [4-6]. Several studies [7,8] have shown that the water transient behavior can causes high dynamic loading which might induce high-stress concentration at the blade's root area, hence give high potential to cause cracks to occur there. Moreover, if the metal structure has an accumulated problem in this area, the degradation process can be substantially accelerated. Frunzaverde *et al.* [4] reported a situation where a crack showed up at the advanced side origin location of the turbine blade due to high-stress concentrations.

These large fractures usually stem from micro fractures, which typically are undetected by the present non-destructive techniques and condition monitoring systems, and they will continually expand. Otherwise detected in time, sever structural failure in the turbine might occur. In the early studies on this topic, cracks were typically assumed to be absolutely open, like a narrow vacant space [9,10] However, it was demonstrated that the natural frequency decrease because of a practical split is always smaller than the one forecasted via making use of an absolutely open fracture [9,11].

This is brought on by the nonlinear behavior induced by the contact between the two fracture faces during the resonance cycle. One of the most significant impacts of the nonlinear practices is that the fractured framework's robust regularities shift substantially towards higher frequency. Bovsunovsky *et al.* [9] were among the first writers to acquire an estimated frequency solution of a beam with a crack considering the nonlinear aspects. According to whether the nonlinear behavior is thought about, the fracture simulation approaches are typically split right into two categories: direct method and a nonlinear process. For the immediate process, the crack is assumed to be as a slim void or low stiffness material. The force acting on the contact pair along with the regular at a discrete time immediately is calculated for the nonlinear approach.

In this paper, the modal analysis of one blade of a Kaplan turbine (see figure 2) is conducted using ANSYS software, and then compared with the modal analysis of the same turbine's blade by using SolidWorks software. The aim is to predict the behavior of the blade under the influence of different outside conditions and predict the failure mode. A Mathematical Modal Analysis is carried out for the blade by using the two software. Numerical outcomes have been contrasted as well as talked about. Results obtained allow analyzing the possibilities of predicting this type of crack by examining Kaplan turbine blades' modal behavior.



Figure 1. Damaged runner blades.



Figure 2. Runner blade - Kaplan turbine.

2 Numerical Approach:

Every structure can be understood as a superposition of masses, dampers and spring attached in between them. The matrix form of the structural formula of a structure based on a dynamic load [12] is the following (equation (1)):

$$[M]\ddot{x}(t) + [C]\dot{x}(t) + [K]x(t) = F(t), \text{ where } x = \begin{Bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{Bmatrix} \tag{1}$$

$\ddot{x}(t), \dot{x}(t), x(t)$ stands for displacement, velocity, and acceleration in the n points (degrees of freedom (DOF)) within the time domain. The [M], [C] and [K] are, respectively, the matrices of mass, dampers, and springs. These parameters represent the relation between the various DOFs within the predefined structure. F(t) is a vector that stand for the force applied in each DOF in the time domain.

By the implementation of the Fourier Transform (FT) to equation (1), the Frequency Response Function (FRF) is given by:

$$\{X(j\omega)\} = [H(j\omega)] \hat{A} \cdot \{F(j\omega)\} \tag{2}$$

here $\{X(j\omega)\}$ and $\{F(j\omega)\}$ stands for the equivalent vectors x(t) and F(t) within the frequency domain . As a result, [H (jω)] represents the FRF. This FRF can be also expressed as shown below:

$$[H(j\omega)] = \sum_{r=1}^N \frac{j2\omega_r Q_r \{\vartheta\}_r \{\vartheta\}_r^t}{(\theta_r^2 + \omega_r^2 - \omega^2) - 2\theta_r j\omega} \tag{3}$$

In the above equation, N stands for the number of the different vibration modes, and from the above equation the modal parameters can be computed. Those parameters include natural frequency (ω_r), the damping factor (θ_r) which governs the amplitude of the structural vibration in the natural frequency, the mode shapes ($\{\vartheta\}_r$) which describe the deformation profile and finally, the scaling factor (Q_r) which represents a fixed factor for every mode r. Moreover, the natural frequency (ω_r) depends on both modal stiffness (K_r) and also modal mass (M_r) of every mode-shape r, which is depicted in equation (4), as shown below:

$$\omega_r = \sqrt{\frac{K_r}{M_r}} \tag{4}$$

In the case of Experimental Modal Analysis (EMA), the two vectors $\{X(j\omega)\}$ and $\{F(j\omega)\}$ are acquired at the same time with multiple transducers, which is typically accelerometers for the response part, and impact hammers to take care of the force. Hence, the FRF can be acquired and thus, consequently the

modal parameters can be extracted [13]. The experimental part is out of the scope of the current study; however, it will be included in the future work to expand the outcomes of this work.

3 Numerical simulations (Results and discussion)

Numerical approaches based on finite element method modeling has been used in this study to analyze the Kaplan runner blade deformation modes. In this part, a description is given for the necessary steps implemented in the modal analysis simulation. The modal analysis is carried out on the Kaplan turbine blade (see figure 2). The Kaplan turbine blade was fixed from one end and all degrees of freedom on this end were taken, meanwhile the runner blade was allowed to move and rotate to simulate actual runner blade operational environment. Mode shapes and natural frequencies are computed in ANSYS and SolidWorks with the numerical formulation of the direct solver.

3.1 Calculation the corresponding working frequency:

The presence of the fluid exerted significant effects on the natural frequencies of the blade working inside the path. Rotating frequency of the blade [14-16] is given by equation (5):

$$f = \frac{n}{60} \quad (5)$$

For Haditha turbine case, the rated speed n is 100 rpm, the number of the blade (Z_1) is six, and the number of the wicket gate (Z_2) is twenty four, the frequency of blade in the operating flow path would be calculated as follows:

Blade frequency: $f_1 = f \times Z_1 = 100/60 \times 6 = 10$ Hz
 Wicket gate frequency: $f_2 = f \times Z_2 = 100/60 \times 24 = 40$ Hz
 Haditha turbine operates at 100 RPM, so the corresponding working frequency will be 10 Hz. In all simulations the blade material properties were taken from the original supplier of Haditha plant turbine. The runner blade is made of EN 1.4313 stainless steel. The mechanical properties of this type of stainless steel is tabulated in table1. The vanes are manufactured from-cast stainless steel containing 13 percent of chromium and 6 percent of nickel. They may resist abrasion and mechanical and dynamic straining under any operational conditions.

EN 1.4313 hardened steel is martensitic tempered steel figured for essential shaping into fashioned items. 1.4313 is the EN numeric assignment for this material. X3CrNiMo13-4 is the EN compound designation. It has a modestly high base expense among the created martensitic hardened steels.

The synthesis of EN 1.4313 treated steel is striking for containing nearly high nickel (Ni) measures and chromium (Cr). Nickel is principally used to accomplish a particular microstructure. Furthermore, it beneficially affects mechanical properties and specific kinds of consumption. Chromium is the characterizing alloying component of treated steel. Higher chromium content gives extra consumption obstruction. [17].

Table 1. Mechanical properties

Elastic (Young's, Tensile) Modulus	200 GPa, 28 x 10⁶ psi
Fatigue Strength	340 to 510 MPa, 50 to 73 x 10 ³ psi
Poisson's Ratio	0.28
Shear Strength	460 to 600 MPa, 66 to 87 x 10 ³ psi
Tensile Strength: Yield (Proof)	580 to 910 MPa, 84 to 130 x 10 ³ psi
Elongation at Break	12 to 17 %
Impact Strength: V-Notched Charpy	55 to 70 J, 41 to 51 ft-lb
Shear Modulus	76 GPa, 11 x 10 ⁶ psi
Tensile Strength: Ultimate (UTS)	750 to 1000 MPa, 110 to 150 x 10 ³ psi

3.2 Modal analysis results with ANSYS:

The modal analysis for the hydro turbine blade is executed by ANSYS Workbench (see figure 3). Modal analysis is a technique to study the dynamic characteristics of a structure under vibrational

excitation. Natural frequencies, mode shapes, and mode vectors of a system can be determined using modal analysis. A graphical variation of the number of modes vs. the frequency can also be obtained from ANSYS Workbench. The mode shapes are observed, which provide a comprehensive picture of deformations occurring. The material properties are assigned to the blade and the boundary conditions are defined (see figure 4), the blade is fixed from one end using a fixed support (blue part in figure 4). This condition prevents the movement of the surface in space. The mesh structure for the is generated automatically using ANSYS built-in meshing package, with a maximum size of the element of 5 mm. The mesh density of 524806 elements was used for this study due to computational limitation. The mesh structure is depicted in figure 5.

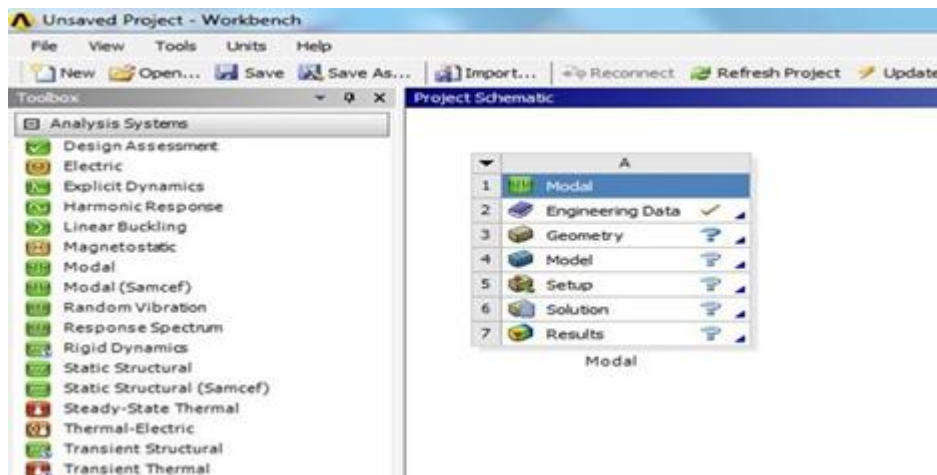


Figure 3. Graphical environment of ANSYS Workbench.

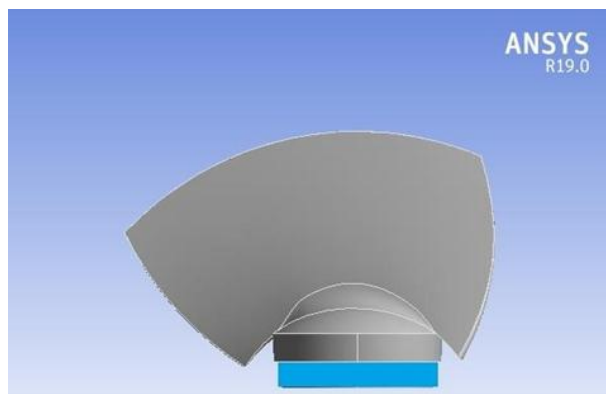


Figure 4. Boundary conditions.

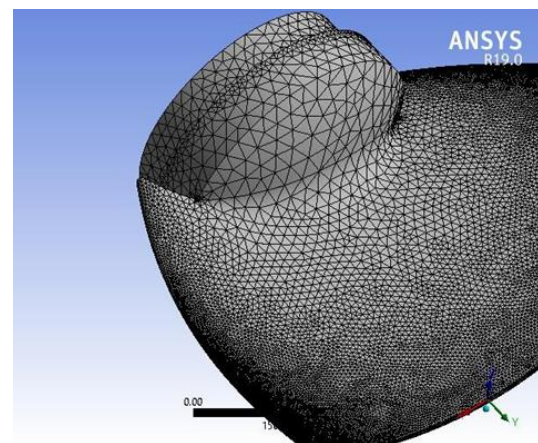


Figure 5. Mesh structural details

Figure 6 and Table 2 shows the simulation results that were predicted using ANSYS software. The figure depicts the six modal shapes that corresponds to six natural frequencies. The six natural frequencies of the runner blade are detected between 189.1 and 584.1Hz (see table 2). The mode-shapes associated to every of those natural frequencies are shown in figure 6. In general, the results show a trend of increasing deformation with higher frequencies and a higher distribution in the zone of the tip-lip of the runner blade. The results indicate higher deformation at the fourth mode (figure 6d) and the sixth mode (figure 6 f). However, the results indicate that mode-shapes does not change considerably for the first three modes. It is worth to mention that the influence of the computational time of the analysis is crucial when a range of frequencies or number of mode shapes is specified. Also, the type of the solver will affect the predicted results.

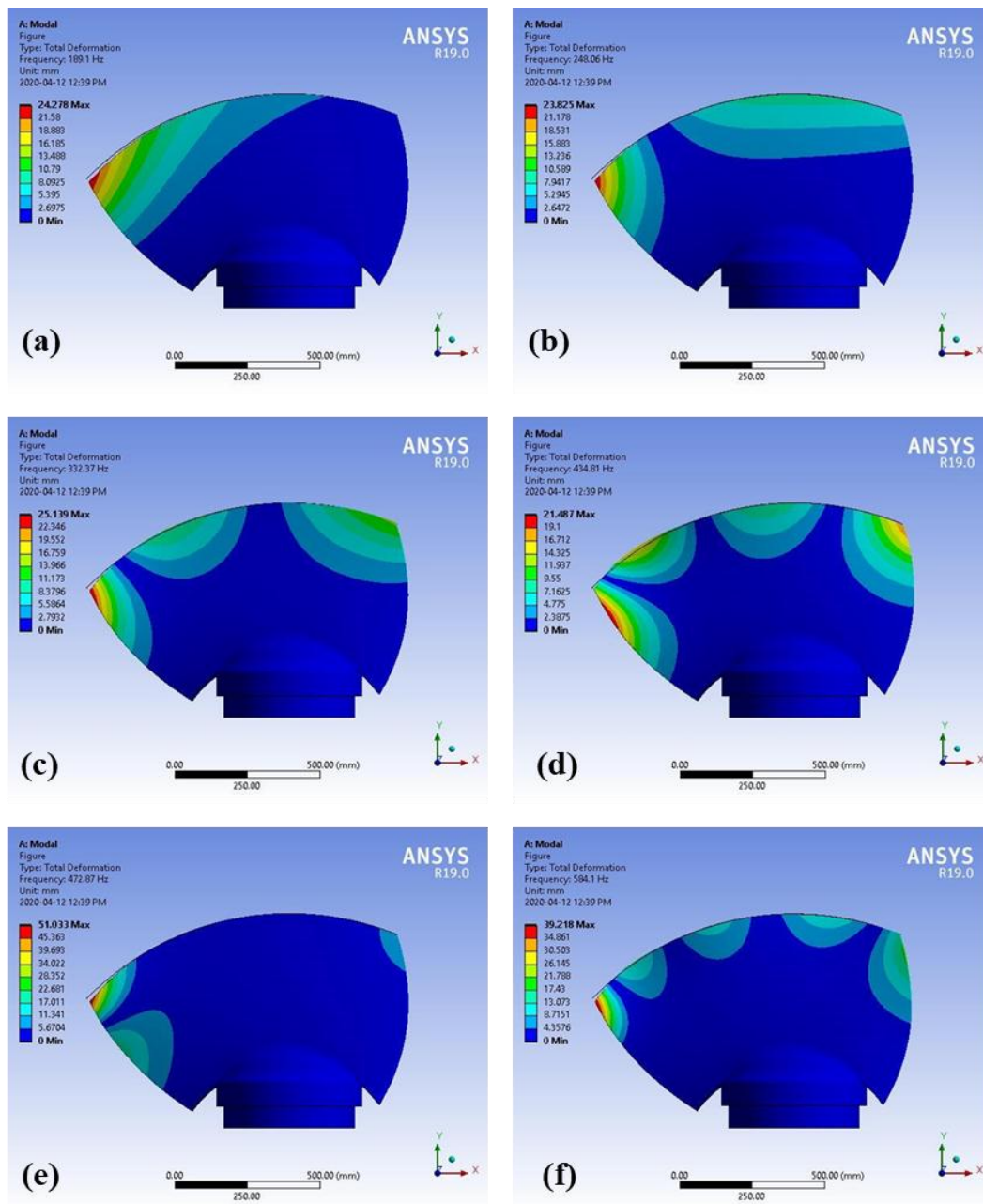


Figure 6. Runner blade mode shape at natural frequency: (a) 189.1 Hz, (b) 248.06 Hz , (c) 332.37 Hz , (d) 434.81 Hz, (e) 472.87 Hz, (f) 584.1 Hz. ANSYS data.

Table 2.The first six natural frequencies (ANSYS).

Mode shape	1	2	3	4	5	6
Frequency (Hz)	189.1	248.06	332.37	434.81	472.87	584.1

3.3 Modal analysis results with SolidWorks:

The modal analysis for the Haditha turbine blade is furthermore performed by SolidWorks (see figure 7). The aim here is to compare the two software results and what kind of discrepancies will be detected. Solidworks simulation is a multi-discipline computer-Aided Engineering (CAE) tool that enables users to simulate the physical behavior and allows users to improve the design. Solidworks simulation can predict how a system will behave in the real world by calculating stresses, deflections, frequencies, heat transfer paths, etc. The material properties are assigned to the blade and boundary conditions are defined.

The fixed end boundary condition is shown in blue (see table 3). This condition prevents the movement of the surface in space. The meshing process is done by the SolidWorks and the mesh on the blade surface is generated automatically. It is noted that the mesh size in this case is lower than the case of ANSYS software (see table 4).

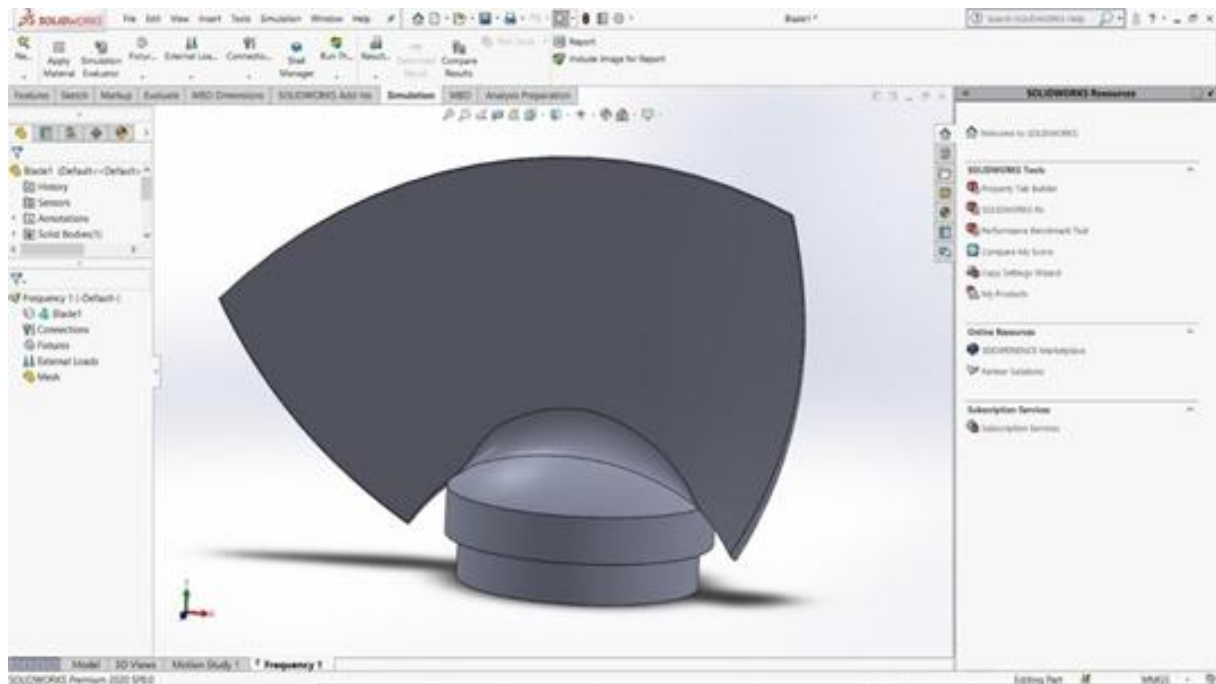


Figure 7. Graphical environment of SolidWorks.

Table.3. Boundary condition.

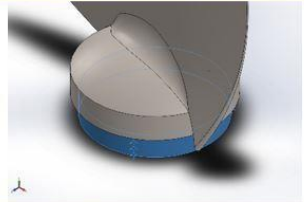
Fixture name	Fixture Image	Fixture Details
Fixed-1		Entities: Type: 1 face(s) Fixed Geometry

Table 4. SolidWorks mesh parameters.

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Jacobian points for High quality mesh	16 Points
Tolerance	1.43727 mm
Mesh Quality	High
Total Nodes	27175
Total Elements	16487

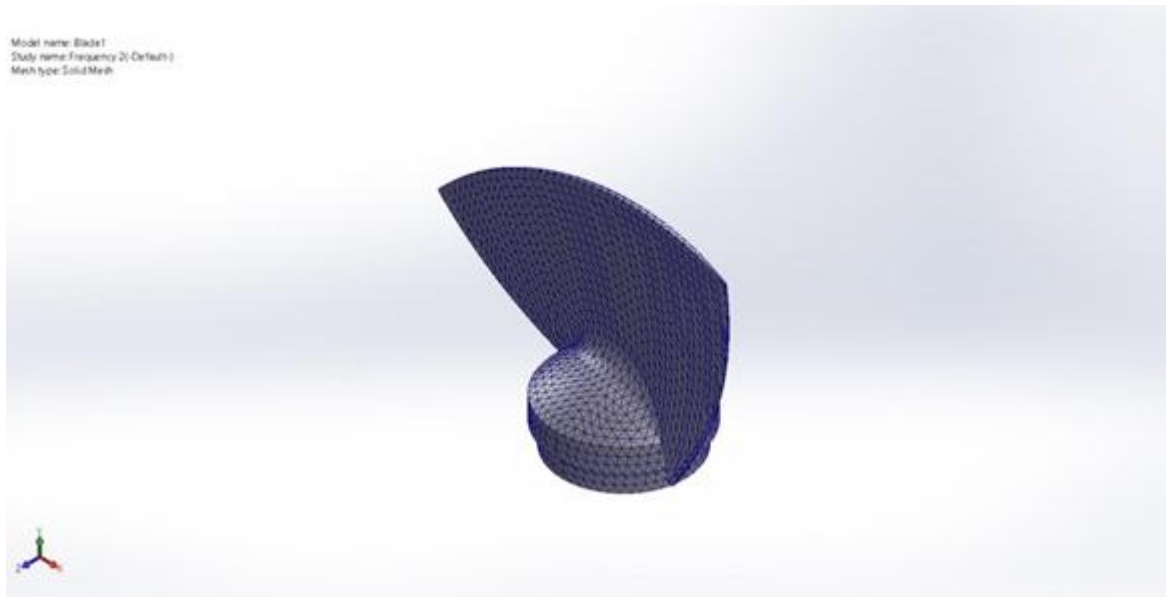


Figure 8. SolidWorks mesh.

Figure 9 and table 5 shows the simulation results that were predicted using SolidWorks software. Again, the modal analysis is carried out by SolidWorks and mode shapes and natural frequencies are also predicted similar to what conducted in ANSYS. The first six mode shapes are shown with the corresponding six natural frequencies as shown in the table 5. The results show similar trends as the ones shown in the ANSYS results (see figure 6), and generally the deformation mode shows higher intensity with larger distribution on the blade surface as the natural frequencies of the modes increases. Higher modal intensity shapes are associated with the higher frequencies. Results obtained from SolidWorks modal analysis shows a higher deformation mode than the one predicted by ANSYS (for example, see figures 9 d and f) and figures 6 d and f. This might be due to the fact that the used mesh within Solidworks is very coarse mesh when it compared to the mesh used in ANSYS. This will have its effect on the accuracy of the predicted modal shapes.

The modal analysis shows no resonance in any of the six mode shapes. The turbine operates at 100 rpm, so the corresponding working frequency is 10 Hz. Moreover, the natural frequency of all mode shape does not match with the natural frequency of the runner blade. Thus, no resonance was detected in the data from the modal analysis.

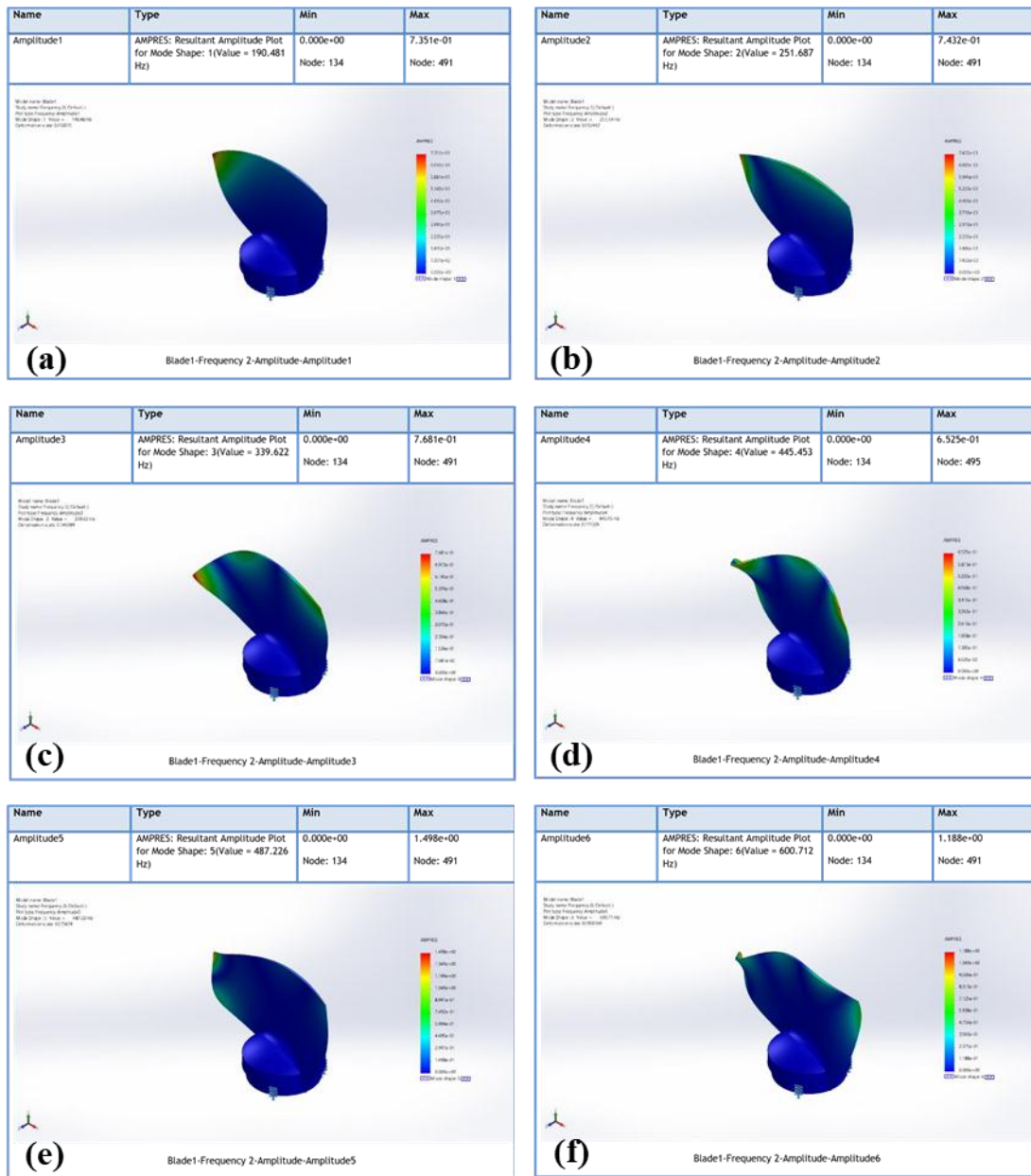


Figure 9. Runner blade mode shape at natural frequency: (a) 190.48 Hz, (b) 251.69 Hz, (c) 339.62 Hz, (d) 445.45 Hz, (e) 487.23 Hz, (f) 600.71 Hz. SolidWorks data.

Table 5. The first six natural frequencies (SolidWorks).

Mode shape	1	2	3	4	5	6
Frequency (Hz)	190.48	251.69	339.62	445.45	487.23	600.71

4 Conclusion:

In this study, the modal behavior of Haditha hydropower plant Kaplan turbine has been investigated. To conduct this investigation, two different commercial software were used (ANSYS & SolidWorks) to conduct the numerical simulations with a blade model mimicking the original runner blade at Haditha plant in Iraq. The aim was to give an insight and understand the unusual failure modes that occurred in Haditha plant Kaplan turbine. Usually, Kaplan turbine blades are prone to have fatigue issues due to high hydraulic loads that is the results of the wide operational modes associated with this type of hydro turbines. The blade root zone is normally associated with high-stress levels and it is easy for the cracks to initiated from it. In the case of Haditha plant, a big fracture occurred on one blade starting from the leading-edge side root hole. It is not clearly understood why this type of failure occurred in the plant, and this study tries to shed some light on this subject by investigating the modal behavior to see if any resonance might happen during operation of the unit.

In both programs, the modal analysis of the hydro turbine blade was executed, and the first six mode shapes and its corresponding natural frequencies were computed. Mode shapes behavior for both software results is very close in the general behavior. Higher natural frequencies were associated with higher deformation shapes which is related with the modal shapes at each frequency. Natural frequencies are very close in the case of the two numerical results for both programs. The deviation in the results between the two software might be related to the nature of the mesh used in each one of them. Higher mesh density was used in the ANSYS simulation compared to lower one in the case of SolidWorks analysis. The numerical investigations show no sign of resonance in any of the six mode shapes and the associated frequencies. The natural frequency of all mode shape does not match with the natural frequency of the Kaplan runner blade. Hence, we conclude that failure mode due to such phenomena is not possible according to the available data. Moreover, the blade acts as a fixed blade during the modal analysis, which in some cases were associated with high deformation at the blade tip. However, the results shown that it was within safe limits at the edges of the runner blade for all mode shapes.

In the future work, the authors will conduct an experimental and numerical analysis to investigate the effect of an existing crack on the natural frequencies and mode-shapes between a damaged blade and undamaged one. The blade with the fatigue crack is currently in Haditha plant and can be used to conduct the experimental part. The modal shapes and natural frequencies will be quantified and analyzed to evaluate the differences between the two different blades.

References:

- [1] Muhsen, A. A., Szymański, G. M., Mankhi, T. A., & Attiya, B. (2018). *Selecting the most efficient maintenance approach using AHP multiple criteria decisions making at Haditha hydropower plant*. Zeszyty Naukowe Politechniki Poznańskiej. Organizacja i Zarządzanie.
- [2] B.G. Mulu, P.P. Jonsson, M.J. Cervantes, 2012, Experimental investigation of a Kaplan draft tube–Part I: best efficiency point, *Appl. Energy*. 93 (2012) 695–706.
- [3] P.P. Jonsson, B.G. Mulu, M.J. Cervantes, Experimental investigation of a Kaplan draft tube–Part II: off-design conditions, *Appl. Energy* 94 71–83.
- [4] D. Frunzaverde, V. Campian, D. Nedelcu, G.-R. Gillich, G. Marginean, 2010, *Failure analysis of a Kaplan turbine runner blade by metallographic and numerical methods*, in: Proc. 7th WSEAS Int. Conf. FLUID Mech. (FLUIDS'10), Univ. Cambridge, UK, , pp. 60–67.
- [5] Y. Luo, Z. Wang, J. Zeng, J. Lin, 2010, Fatigue of piston rod caused by unsteady, unbalanced, unsynchronized blade torques in a Kaplan turbine, *Eng. Fail. Anal.* 17, 192–199.
- [6] G. Urquiza, J.C. Garcia, J.G. Gonzalez, L. Castro, J.A. Rodriguez, M.A. 2014, Basurto-Pensado, O.F. Mendoza, Failure analysis of a hydraulic Kaplan turbine shaft, *Eng. Fail. Anal.* 41, 108–117.
- [7] L. Zhou, Z. Wang, R. Xiao, Y. Luo, Analysis of dynamic stresses in Kaplan turbine blades, *Eng. Comput.* 24 (2007) 753–762.

- [8] Z.W. Wang, Y.Y. Luo, L.J. Zhou, R.F. Xiao, G.J. Peng, 2008, Computation of dynamic stresses in piston rods caused by unsteady hydraulic loads, *Eng. Fail. Anal.* 15, 28–37.
- [9] G.-L. Qian, S.-N. Gu, J.-S. Jiang, 1990, The dynamic behaviour and crack detection of a beam with a crack, *J. Sound Vib.* 138 233–243, [https://doi.org/10.1016/0022-460X\(90\)90540-G](https://doi.org/10.1016/0022-460X(90)90540-G).
- [10] W.M. Ostachowicz, M. Krawczuk, 1991, Analysis of the effect of cracks on the natural frequencies of a cantilever beam, *J. Sound Vib.* 150 191–201.
- [11] A.P. Bovsunovsky, V.V. Matveev, 2000, Analytical approach to the determination of dynamic characteristics of a beam with a closing crack, *J. Sound Vib.* 235, 415–434.
- [12] R.D. Blevins, 2001, *Formulas for Natural Frequency and Mode Shape*, Krieger, Publishing Company,.
- [13] Zhang, M., Valentin, D., Valero, C., Presas, A., Egusquiza, M., & Egusquiza, E., 2020. Experimental and numerical investigation on the influence of a large crack on the modal behaviour of a Kaplan turbine blade. *Engineering Failure Analysis*, 109, 104389.
- [14] Abubakar, M., Badshah, S., Ahmad, T., & Rahman, N. 2014. Modelling and Analysis of a very Low Head Kaplan Turbine Runner Blades for Rural Area of Punjab. *International Journal of Scientific & Engineering Research*, 5(7), 905-912.
- [15] Lixia, Z., Wei, Z., & Zhaohong, Y. 2010, Modal analysis of Francis turbine blade based on fluid-structure interaction theorem. *International Journal of Modelling, Identification and Control*, 10(1-2), 101-105.
- [16] Lengvarský, P., Bocko, J., & Hagara, M. 2013, Modal analysis of titan cantilever beam using ANSYS and SolidWorks. *American Journal of Mechanical Engineering*, 1(7), 271-275.
- [17] INGRA Co., 1985, *Operation & Maintenance Instructions Manual Book 8*.