STRUCTURAL ANALYSIS OF HYDROPOWER STOPLOG: A CASE STUDY OF SETI HYDROPOWER PROJECT

Sagar Timilsina[1*], Paras Katuwal[1]
[1]Department of Mechanical Engineering, Pashchimanchal Campus, IOE, Tribhuvan University, Pokhara, Nepal
[2]Department of Civil Engineering, Pashchimanchal Campus, IOE
Tribhuvan University, Pokhara, Nepal
*sgrtml2057@gmail.com

(Manuscript Received 30th August, 2024; Revised 29th September, 2024; Accepted 30th September, 2024)

Abstract

Hydropower stoplogs are essential components in managing water flow within hydropower facilities, playing a critical role in ensuring safe operations and maintenance. Despite advancements in technology, existing designs face persistent challenges related to efficiency, environmental impact, and maintenance needs, highlighting a significant research gap in optimizing stoplog design. This study aims to address these issues by employing Finite Element Analysis (FEA) to evaluate stress distributions and material behavior in stoplog design, thereby enhancing overall performance. The methodology involved a comprehensive literature review, analytical design calculations, and the development of a detailed Computer-Aided Design (CAD) model, followed by iterative static structural analysis using ANSYS Mechanical. Key findings revealed that the integration of stiffeners and support systems significantly improved the stoplog's performance, achieving a maximum central deflection of 11.226 mm—just within the allowable limit of 11.25 mm—and a maximum stress of 241.18 MPa, well below permissible limits. These results confirm that the optimized stoplog design is both safe and efficient, illustrating the effectiveness of FEA in advancing hydropower infrastructure.

Keywords: ANSYS Mechanical, CAD, FEA, MIT, Structural analysis

1. Introduction

Hydropower stoplogs are large, removable barriers used to control water flow in hydropower facilities (Bureau of Indian Standards, 2003). They help manage water levels, facilitate maintenance, and ensure safety by isolating sections of the system. Generally, a stoplog consists of a structural steel frame with vertical end girders and horizontal girders placed between them. The stoplog's dimensions and the design water pressure determine the spacing. Secure riveting or welding holds the frame together in one piece.

The concerned stoplog of this research study is located in the undersluice area of the hydropower project. The features of the undersluice area openings are tabulated below in Table 1.

Table 1 : Site Parameter

Clear Opening	9m x7.553m
Designed Water Level	588.5 masl
Gate Invert Level	573.3 masl
Surge Head	0 % taken
Design Head	15.20 m

Despite significant advancements in hydropower technology, existing hydropower stoplogs often face critical challenges related to efficiency, environmental impact, and maintenance. These challenges include optimizing stoplog design for varying water flow conditions, minimizing ecological disruption, and ensuring long-term durability. Current research lacks comprehensive solutions that address these issues holistically. The need exists to develop innovative hydropower stoplog systems that enhance operational efficiency, reduce environmental impacts, and lower maintenance costs while integrating seamlessly with existing infrastructure.

The research regarding stoplogs are not carried out efficiently and the designation of stoplogs are executed only on paper. In conventional method the stoplogs are generally designed on the basic of hydrostatic load bearing ability considered a stoplog as an integrated body. But in fact, the stress on body varies from point to point so the implementation of finite elemental study method is essential for both safety and for less material consumption (Bendsoe & Kikuchi, 1988).

Finite element analysis offers significant advantages in modeling complex geometries and evaluating stress distributions, which are crucial for the optimized design of structures such as hydropower stoplogs (Chen & Liu, 2015). Finite Element Analysis (FEA) is frequently preferred over traditional analytical methodologies for designing hydropower structures due to many major benefits. FEA is more successful at modelling complex geometries and sophisticated boundary conditions, which is critical for the thorough design of hydropower stoplog. It provides a detailed view of stress and strain distributions, allowing for precise characterization of material behavior and load responses, including nonlinearities. Furthermore, FEA may conduct dynamic calculations to evaluate the influence of changing load circumstances and optimize design parameters. This comprehensive methodology aids in detecting important stress spots and maintaining structural integrity that analytical methods may ignore or oversimplify.

2. Methodology

Firstly, previous research on hydropower gates and stoplogs was reviewed, focusing on analytical design methods and computational analyses. With adequate site data and information, we identified and defined the necessary design variables. Following this, preliminary calculations were performed through analytical design. A CAD model was developed, incorporating appropriate constraints and boundary conditions for structural analysis. The results of the static structural analysis were evaluated, and if satisfactory, conclusions were drawn. In cases where the results were not adequate, we revisited the computational model or analytical calculations for refinement. This iterative approach ensured a robust and effective design process.

The research methodology used for the following study is shown in figure 1.

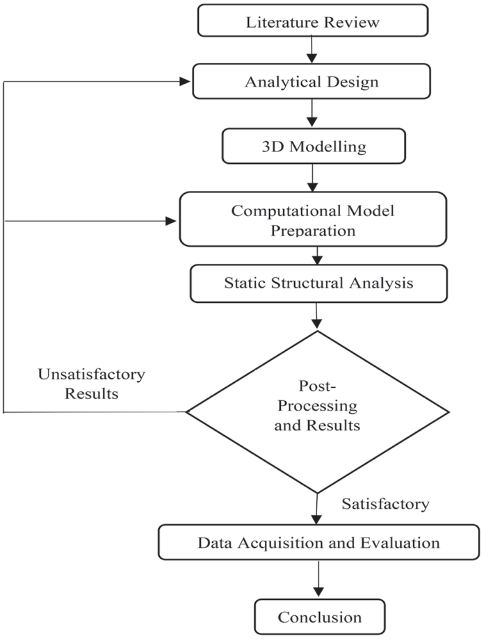

Figure 1 : Methodology

Material Selection and Allowable Stress and Deformation Calculation

The material used for the fabrication of the stoplog is E350B0 and its properties are tabulated in the Table 2.

Table 2 : Material Properties

Steel Grade	E350B0
Density	7850 kg/m3
Young's Modulus	210 GPa
Poisson's Ratio	0.3
Ultimate Tensile Strength (f_u)	490 MPa
Yeild Strength (f_y)	350 MPa

For the computation of allowable stress and deflection, IS 4622 (2003): Recommendations for Structural Design of Fixed-Wheel Gates is followed.

As per the table at Annex B of the code the permissible stress differs depending on wet/dry/accessible/inaccessible condition. For stoplogs stresses given under dry and accessible conditions is applied since they are stored above water level and are only occasionally used i.e.

Permissible Stress (S)=0.75xf_y x e

= 0.75 x 350 MPa x 1

= 262.5 MPa

Where, e= Welding Efficiency= 1 for shop weld [cl-5.2.6.1]

And according to cl-5.3.5 of the code, maximum deflection of the gate under normal conditions of the loading shall be limited to 1/800 of the span (clear opening span). i.e.

Maximum Deflection= (1/800) x L

= (1/800) x 9000 mm

= 11.25 mm

2.2 Model Preparation and Meshing

For the mechanical design, firstly, the CAD model was prepared in Solid works software using the geometrical data from site observation.

Due to the large height of the stoplog, the design was divided into three stages to make the installation process more manageable. The first design iteration was conducted using Ansys

Mechanical for Finite Element Analysis (FEA) to identify areas with high stress concentrations, significant stress gradients, and large deformations. Based on these findings, we incorporated additional supports and stiffening elements, such as stiffeners and back cover plates. Several subsequent design iterations were performed to assess how these modifications improved the structural behavior of the stoplog under the intended design conditions. Additionally, two vertical beams were added with a 4-meter span to facilitate the hoisting process and reduce deflection in the center.

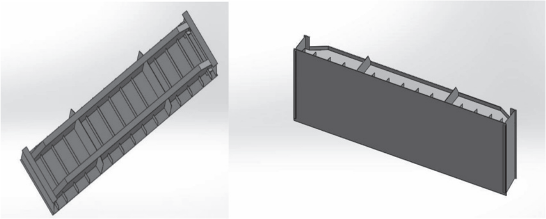

Figure 2 : Stoplog CAD Model

The bottom stage of the three-stage stoplog model, which experiences the highest hydrostatic forces, was analyzed using Ansys. As a result, the two upper stages are expected to remain within safe limits for allowable stress and total deformation.

All geometries were meshed using ANSYS Mechanical tools. The precision of computational analysis relies heavily on mesh size and boundary conditions (Chen & Liu, 2015). Generally, a finer mesh leads to more accurate results, while precise boundary conditions are essential for relevant outcomes. However, there is a point where further mesh refinement does not significantly alter the results. To establish a mesh-independent solution, a Mesh Independence Test was performed to identify a mesh size that produces stable and consistent results. For the selection of appropriate mesh size, Mesh Independence Test (MIT) was performed taking size of mesh as independent parameter and the maximum deflection at skin plate as dependent variable. The mesh size was reduced till a convergence was seen on the output deflection result.

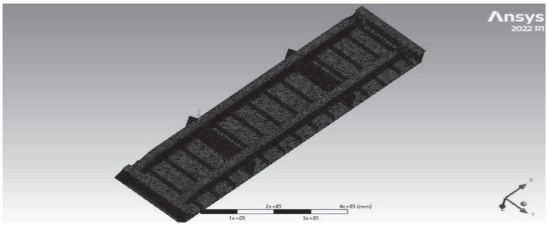

Figure 3 : Mesh Model

2.3 Boundary conditions and Load Constraints

The sealing of Undersluice Stoplog is downstream so the seal stopper acts as a fixed support for analysis in finite element solver-Ansys. Whereas the back of the skin plate is subjected to hold the total hydrostatic pressure. All the strength calculations were performed based on the maximum hydrostatic pressure in the system.

Calculation for hydrostatic pressure:
Total head=15.2m
Hydrostatic Pressure
(P)=dxgxh=1000x9.81x15.2= 0.149MPa.

3. Results

3.1 Mesh Independence Test

Numbers of iteration were carried out to optimize the model. The maximum deflection obtained on mesh independent test is displayed in the Table 3 and Figure 6. From Figure 6, the circled mesh element corresponding to mesh size of 37.5 mm was chosen for the static structural analysis. So deflection and stress results represented are retained from mesh size 37.5mm.

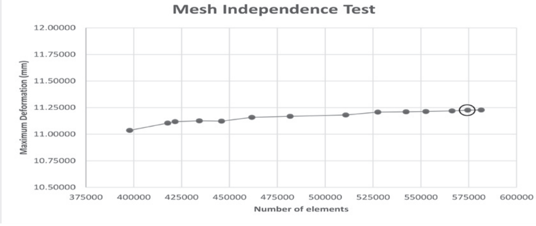

Figure 4 : Mesh Independence Test Results

3.2 Structural Analysis

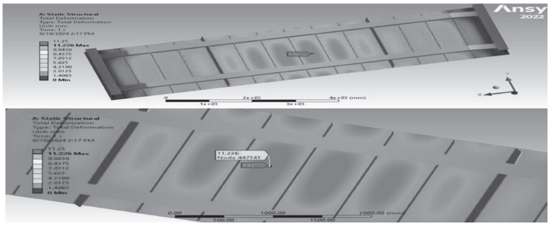

Figure 5 : Deformation results across Stoplog

The results of the finite element analysis (FEA) of the hydropower stoplog indicate that the design is both structurally sound and efficient under specified loading conditions. The observed central deflection of 11.226 mm is just below the allowable maximum of 11.25 mm, demonstrating that the stoplog is operating within acceptable limits. This close margin highlights the effectiveness of the design, ensuring safety while maximizing material efficiency.

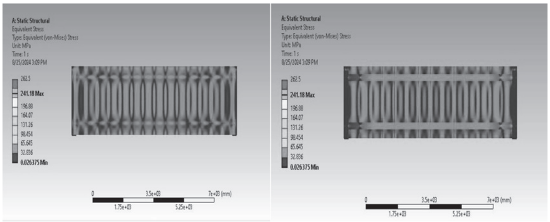

Figure 6 : Stress Distribution across Stoplog

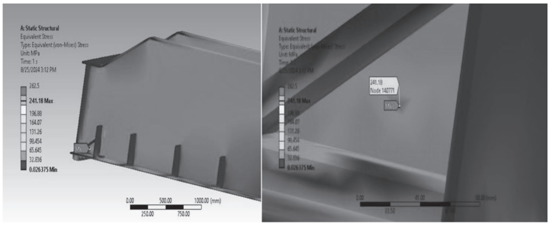

Figure 7 : Region of Maximum Stress

The stress distribution analysis reveals a maximum stress of 241.18 MPa, localized primarily in the boundary area, which is well within the permissible limits. This indicates that the design can withstand the hydrostatic forces without risk of failure, confirming its reliability in real-world applications.

The significance of these results lies in their contribution to improving hydropower stoplog designs. By employing FEA, the study identifies critical stress concentrations and deformation patterns that traditional methods might overlook. This approach not only enhances safety and performance but also paves the way for future innovations, such as optimized designs and material use.

4. Conclusion

In conclusion, the finite element analysis of the hydropower stoplog indicates that the design is both safe and efficient under the specified conditions. The mesh independence test confirmed that a mesh size of 37.5 mm provides reliable results for deflection and stress calculations. The central deflection of 11.226 mm, while close to the maximum allowable deflection of 11.25 mm, is within acceptable limits, demonstrating that the stoplog's structural performance is nearly optimal. The stress distribution analysis shows that the maximum stress of 241.18 MPa is localized in the boundary area, well within the permissible stress limits. The incorporation of stiffeners effectively reduces central deflection by increasing rigidity and reducing bending moments. Further improvements to the hydropower stoplog could include improving materials by investigating alternatives for increased durability and performing dynamic load studies to accommodate for changing water conditions and seismic impacts. Furthermore, adopting a modular design would make travel and installation easier, while upgrading finite element analysis techniques could provide deeper insights into structural relationships, ultimately improving the stoplog's integrity, efficiency, and durability.

Acknowledgement

Authors would like to acknowledge Seti Khola Hydropower Project for providing base data of Seti Khola Hydropower Project. The authors would also like to thank Machhapuchhre Metal & Machinery (P.) Ltd. for necessary support.

References

Bureau of Indian Standards. (2003). Recommendations for structural design of fixed-wheel gates (IS 4622:2003). Bureau of Indian Standards.

Chen, X., & Liu, Y. (2015). Finite element modeling and simulation with ANSYS Workbench (1st ed.). CRC Press. https://doi.org/10.1201/b17284

Bendsoe, M. P., & Kikuchi, N. (1988). Generating optimal topologies in structural design using a homogenization method. Computer Methods in Applied Mechanics and Engineering, 71(1), 197–224.

APPENDIX

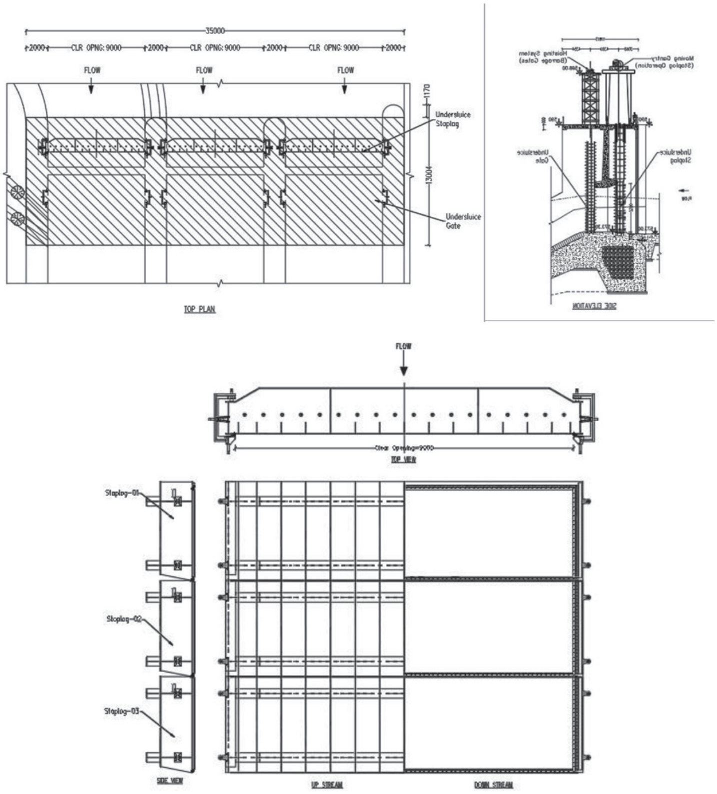

Figure 8 : Shop Drawing of Stoplog

Table 3: Final Model Parameters Based on Analysis

S. N.	Part	Material Section	Section (Thickness in mm)	Material Grade
1.	Skin Plate	MS Plate	12	E350 B0
2.	Side Vertical Girder	MS Plate	12	
3.	Back Cover Plate	MS Plate	20	
4.	Horizontal T Girder	MS Plate (T Section)	Flange=20 Web=12	
5.	Vertical Stiffener	MS Plate	10	
6.1	Sliding Base Plate	MS Plate	47	
6.2	Sliding Pad	MS Plate	10	
7.	Horizontal Girder Base	MS Plate	20	
8.	Bracing Stiffener	MS Plate	16	