Finite Element Analysis of a Photovoltaic Pumped Hydroelectric Storage (PHES) system

Anyanime Tim Umoette[1]

Department of Electrical and Electronic Engineering,
Akwa Ibom State University, Ikot Akpaden, Nigeria
libertycoast@yahoo.com

Nnamonso O. Akpabio[2]

Department Of Computer Engineering
Federal Polytechnic, Ukana,
Akwa Ibom State Nigeria

Okon Aniekan Akpan[3]

Department of Mechanical and Aerospace Engineering
University of Uyo, Akwa Ibom State Nigeria
drokonaniekan@gmail.com

***Abstract*— In this work, the Finite Element Analysis for Photovoltaic Pumped Hydroelectric Storage (PHES) system is presented. The FEA was conducted using ANSYS 2019 to evaluate the structural integrity of the turbine and water stand under operational loads for the pumped water storage solar hydropower plant. The analysis focused on determining the total deformation and maximum principal stress distribution within the system components.The deformation plots of Turbines 1 through 6 revealed moderate displacement concentrated near blade tips and hub edges locations typically susceptible to torsional loading. Meanwhile, the maximum principal stress plots indicated that the blade roots and shaft coupling zones experienced the highest stress intensities. The water stand also showed moderate deformation and stress concentrations at its anchoring points. The structural integrity, as visualized in the various results displayed, supports the mechanical feasibility of the design, suggesting that all turbines can withstand expected hydraulic loads with minimal risk of failure under steady-state operation. However, regular monitoring and reinforcement of stress-prone zones are advisable.**

Keywords*— *Finite Element Analysis (FEA), Maximum Principal Stress Distribution, Hydropower, Photovoltaic, deformation contour plot, Pumped Hydroelectric Storage, static structural analysis

1. Introduction

Photovoltaic Pumped Hydroelectric Storage (PHES) system is a power system that requires pumping and storage of large volume of water to upper reservoir and utilizing the stored water to drive the hydro turbine for the generation of the electrical energy that eventually get delivered to the load [1,2,3]. The water volume and involvement the movement of water mass to drive the turbine brings up the need for thorough analysis of the system to ensure the safety and durability of the system [4,5,6].

Notably, Finite Element Analysis (FEA) approach can be used to carry out fine-grained analysis on the system components so as to ensure they meet the required standard both at the design time and in the course of operation of the system [7,8,9]. Also, the FAE will make it cheaper to study the system under different operating conditions, such as different load level, different weather conditions, among others without spending money on building the actual facility for the experimental evaluation [10,11,12]. For the PHES, the FAE will also help to obtain requisite insight that will help to optimize the design, enhance safety, and reduce the design time, among other benefits. Accordingly, this study is focused on presenting the Finite Element Analysis (FEA) of a Photovoltaic Pumped Hydroelectric Storage (PHES) system using a case study system for the design calculations and result presentations.

2. Methodology: The Finite Element Analysis (FEA) of the system

In this work, the focus is to conduct Finite Element Analysis (FEA) ANSYS 2019 to evaluate the structural integrity of the turbine and water stand under operational loads for a pumped water storage solar hydropower plant [13,14]. The analysis focused on determining the total deformation and maximum principal stress distribution within the system components [15,16]. The simulation was achieved with the following steps;

A. Import the Geometry (Water Wheel)

1. Open **ANSYS Workbench**.
2. Drag **Static Structural** from the Toolbox into the Project Schematic.
3. Right-click **Geometry → Import Geometry → Browse**.
    - **i.** Import the **.STEP / .IGES / Parasolid (.x_t)** file you saved from SolidWorks.

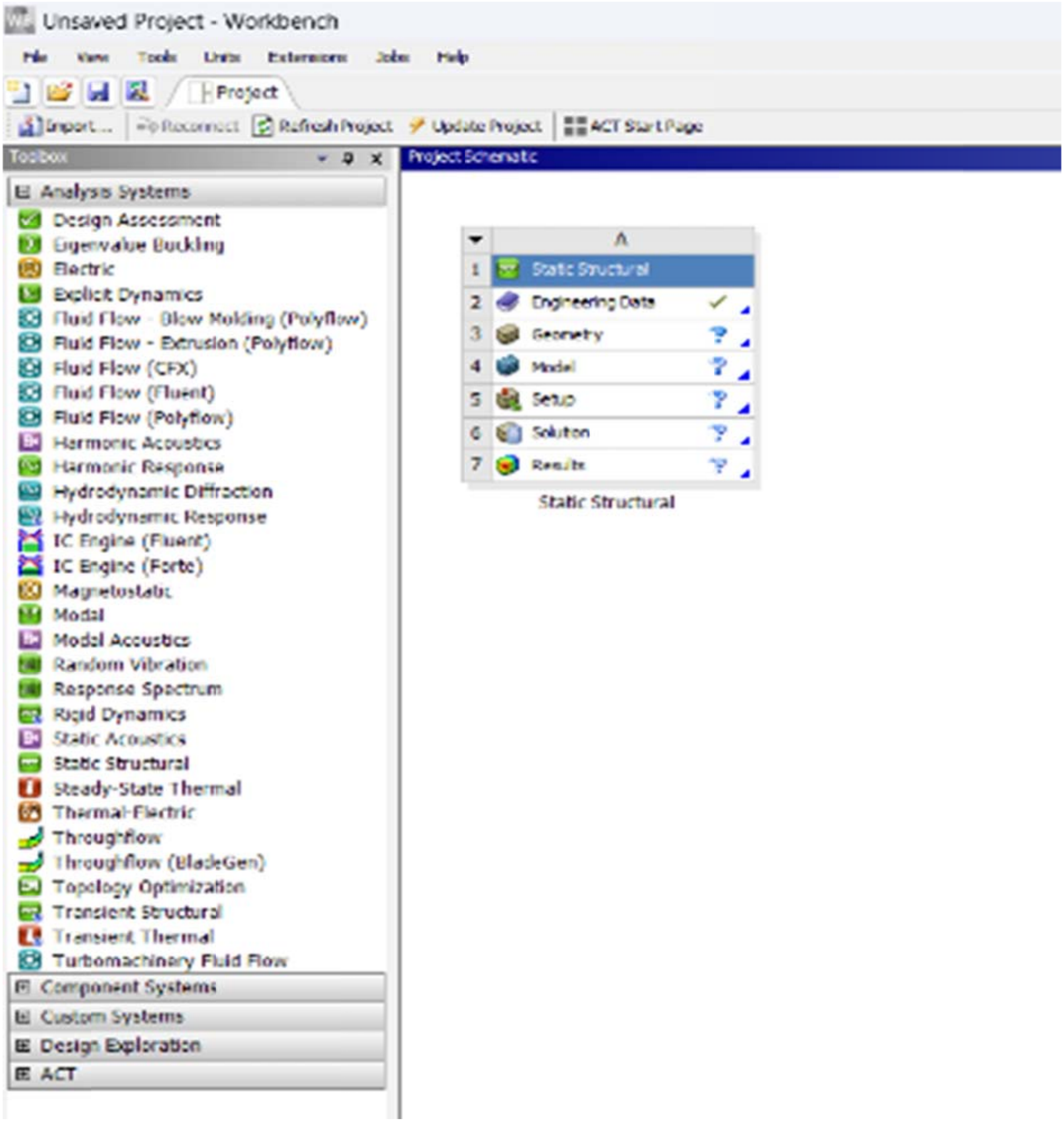

Figure 1: Importing Design Geometry to ANSYS Workbench.

B. Define Engineering Data (Material)

1. Double-click **Engineering Data**.
2. Choose or create your material:
    - **i.** Example: Aluminum, etc.
    - **ii.** Input properties: Elastic modulus (E), Poisson's ratio (ν), Density (ρ), Yield Strength.
3. Assign this material to your blade.

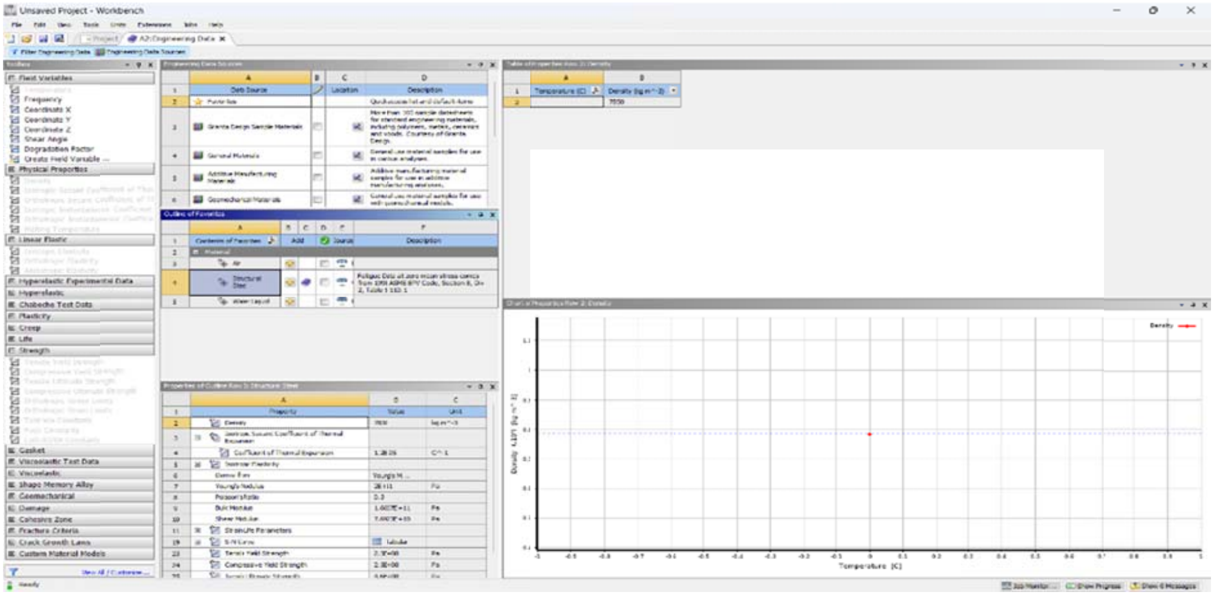

Figure 2: Assigning Material to Geometry for ANSYS Simulation.

C. Geometry Cleanup (optional)

1. If geometry is complex, you can simplify in **DesignModeler**:
   i. Remove small holes/fillets that won't affect results.
   ii. Keep only the blade and hub for faster computation.

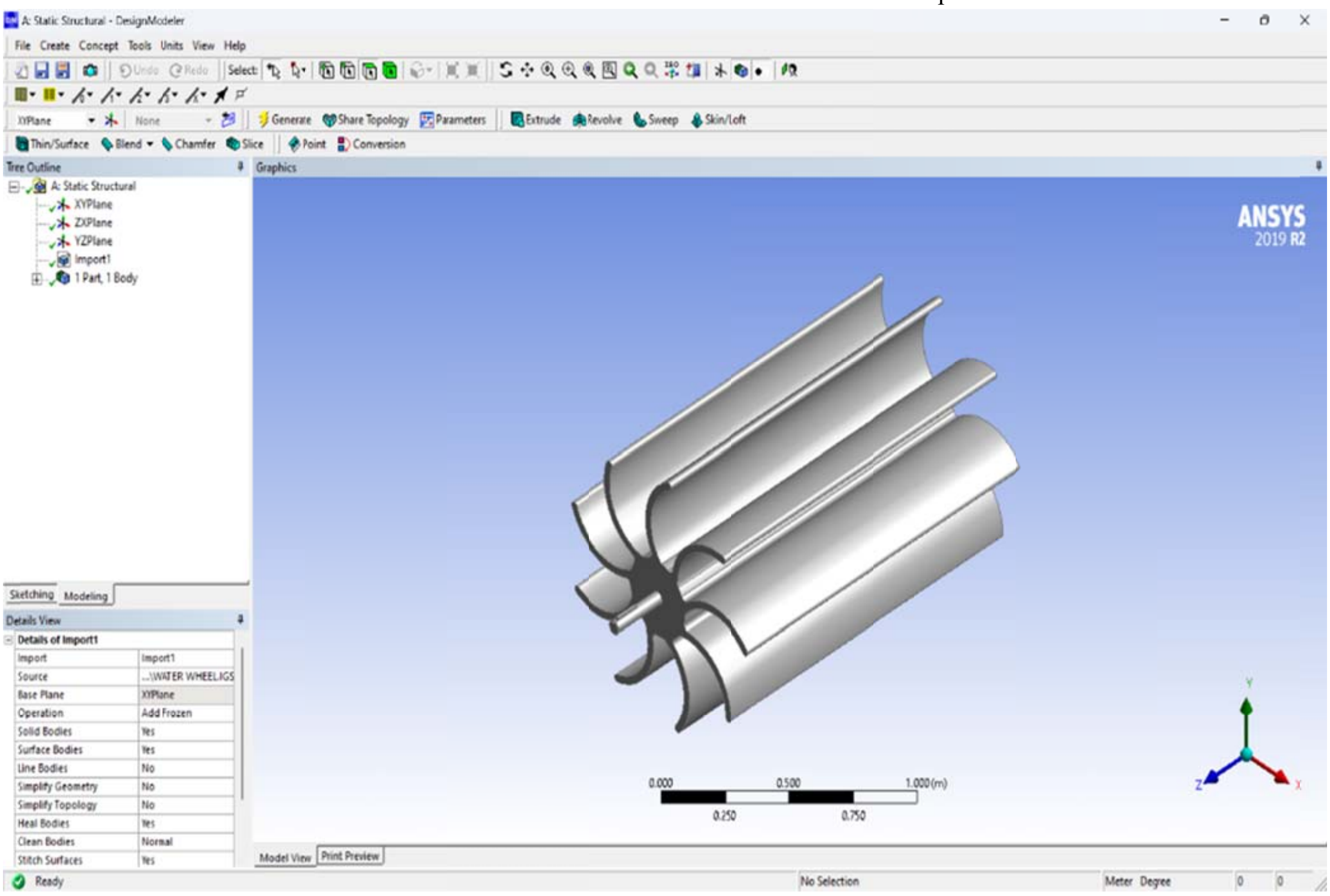

Figure 3: Cleaning up Geometry for ANSYS Simulation.

D. Model Setup

1. Double-click **Model** to open ANSYS Mechanical.
2. In the **Tree Outline**, ensure:
   i. The blade/hub are present.
   ii. Material assignment is correct.

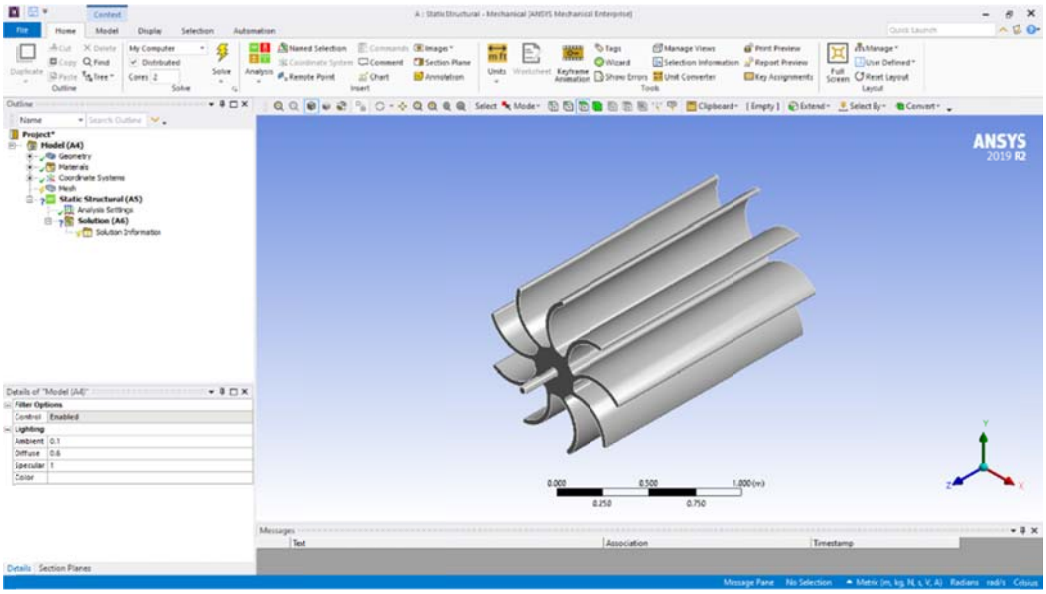

Figure 4: Setting up Geometry Model for ANSYS Simulation.

E. Apply Connections (if needed)

1. If multiple parts exist:
   - i. **Automatic Contacts** → generate bonded contacts between them.
2. If it's one solid body, you can skip this.

F. Apply Boundary Conditions

This depends on what you want to simulate:

1. Fixed Support (shaft mounting)

i. Select faces of the shaft/hub → Insert → Fixed Support.

2. Loads

i. **Pressure load (from water flow):**
   - i. Select blade surfaces exposed to water.
   - ii. Insert → Pressure → Apply magnitude (Pa) (171.07 kPa).

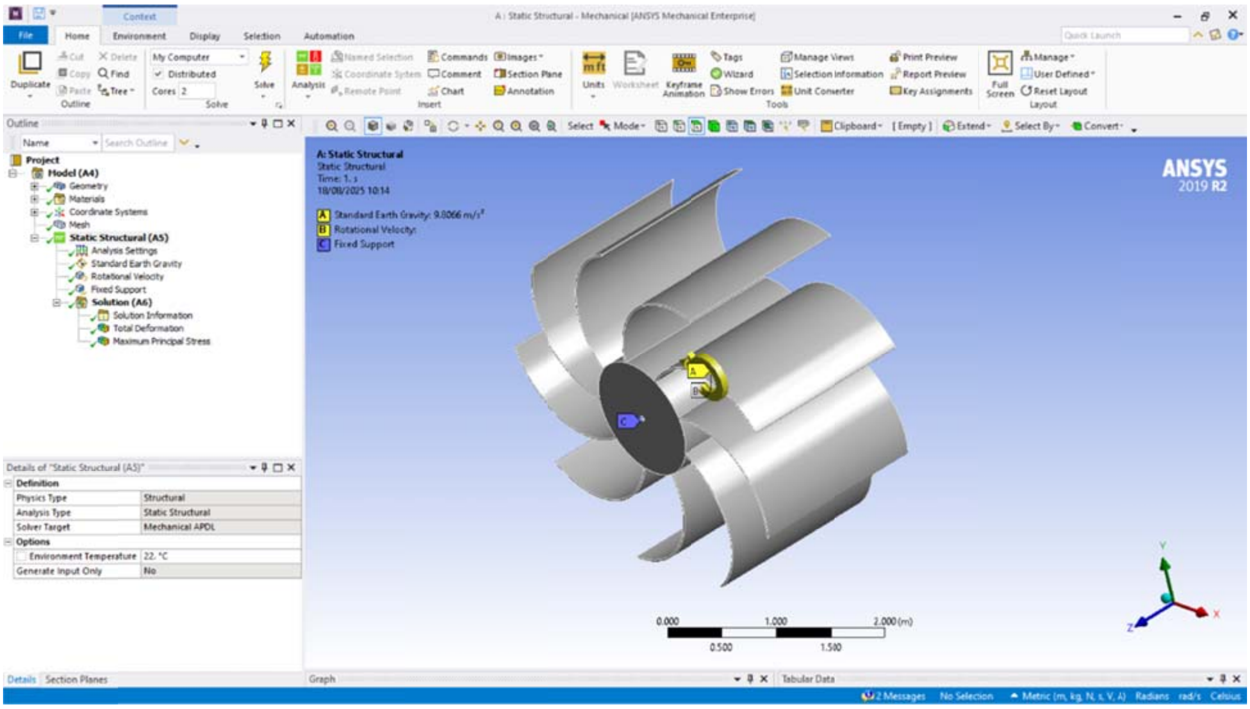

Figure 5: Applying Boundary Conditions for ANSYS Simulation.

G. Mesh Generation

i. Right-click **Mesh → Insert → Method**.
   - i. Choose **Tetrahedrons** (default) or Hex if structured mesh is possible.

ii. Use **Refinement** on blades (thin curved surfaces).

iii. Right-click Mesh → **Generate Mesh**.

i. Start coarse → refine if stress results vary too much.

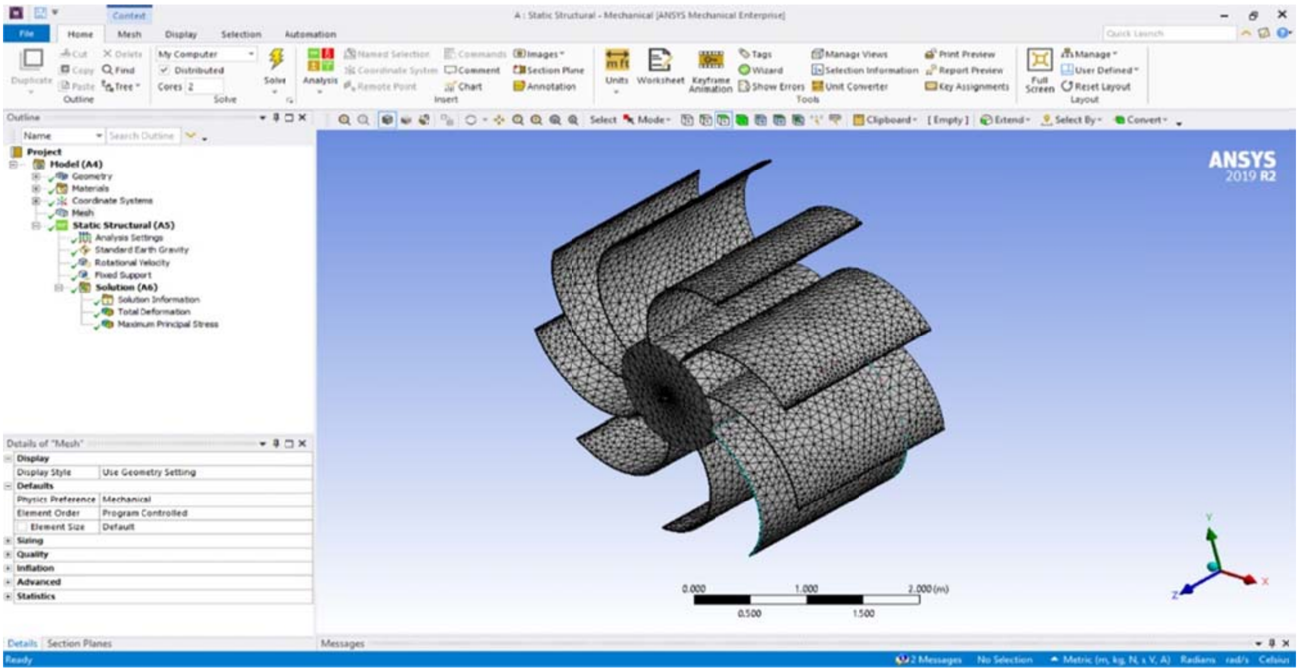

Figure 6: Generating Mesh for ANSYS Simulation.

H. Solve

1. Right-click **Solution** → **Solve**.
2. ANSYS will compute stresses, strains, and deformations.

I. Post-Processing (Results)

Insert the following:

i. **Total Deformation** → shows displacement of blades.

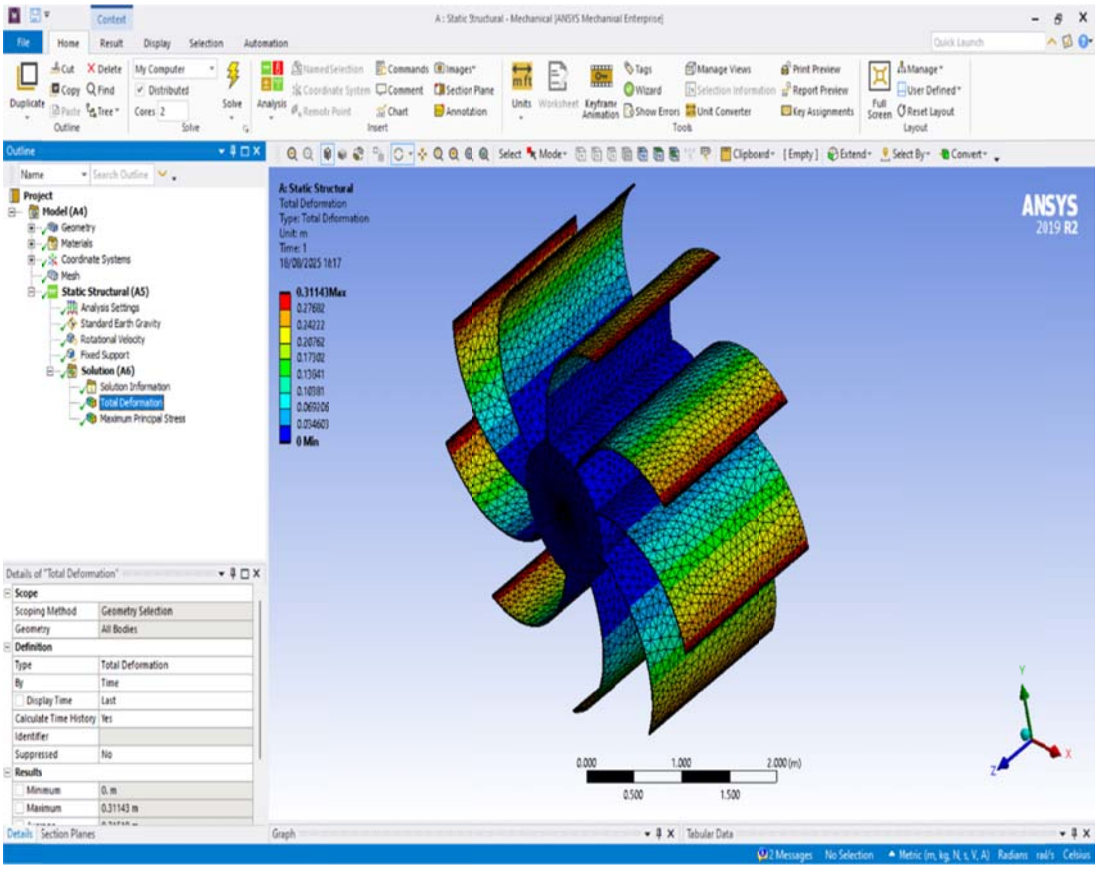

Figure 7: Structural Deformation Output FEA Simulation.

ii. **Equivalent (von-Mises) Stress** → check stress distribution.

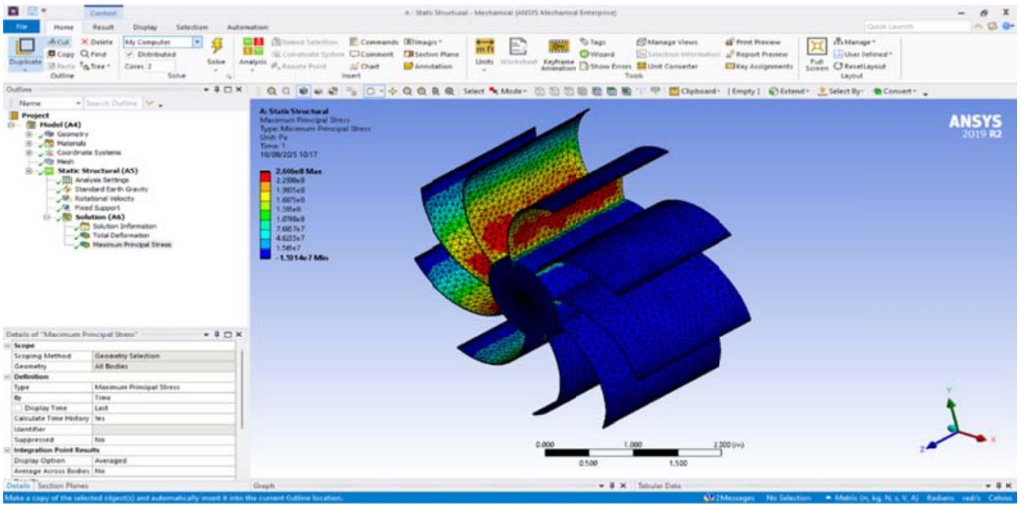

Figure 8: Equivalent (von-Mises) Stress Output FEA Simulation

J. Validate

i. Check if max stress < material yield strength (safe).

ii. If deformation is too high → redesign or change material.

iii. Perform a **mesh refinement study** to ensure accuracy.

K. The input parameters for the ANSYS FEA Simulation

i. The key input parameters for the ANSYS FEA Simulation are presented in Table 1 while the key output parameters expected from the **ANSYS FEA** simulation are presented in Table 2.

Table 1: Input parameters for ANSYS FEA Simulation

S/N	Parameter name	Symbol	Unit	Value or range of values used
1	Geometry file type	–	–	.STEP / .IGES / Parasolid (.x_t)
2	Material	–	–	Example: Aluminum
3	Elastic modulus	E	Pa	User-defined (material property)
4	Poisson's ratio	ν	–	User-defined (material property)
5	Density	ρ	kg/m³	User-defined (material property)
6	Yield strength	σ_y	Pa	User-defined (material property)
7	Boundary condition (support)	–	–	Fixed Support (shaft/hub faces)
8	Applied pressure load	p	kPa	171.07
9	Mesh type	–	–	Tetrahedron / Hexahedron
10	Mesh refinement	–	–	Coarse → refined (on blades)

Table 2: Output parameters for ANSYS FEA Simulation

S/N	Parameter name	Symbol	Unit
1	Total deformation	δ_total	m
2	Directional deformation	δ_dir	m
3	Equivalent (von-Mises) stress	σ_eq	Pa
4	Safety factor	SF	–
5	Reaction forces at supports	F_react	N
6	Reaction moments at supports	M_react	N·m

3. Results and discussion

The CAD Drawing of Turbine 1 and Turbine 6 showing their key geometric dimensions are presented in Figure 9 and figure 10 respectively.

The results of the total deformation contour plot of Turbines 1 to turbine 6 obtained from the static structural analysis as simulated in ANSYS 2019 are presented in Figure 11 to Figure 16.

The results of the Maximum Principal Stress Distribution in Turbines 1 to turbine 6 obtained from the static structural analysis as simulated in ANSYS 2019 are presented in Figure 17 to Figure 22.

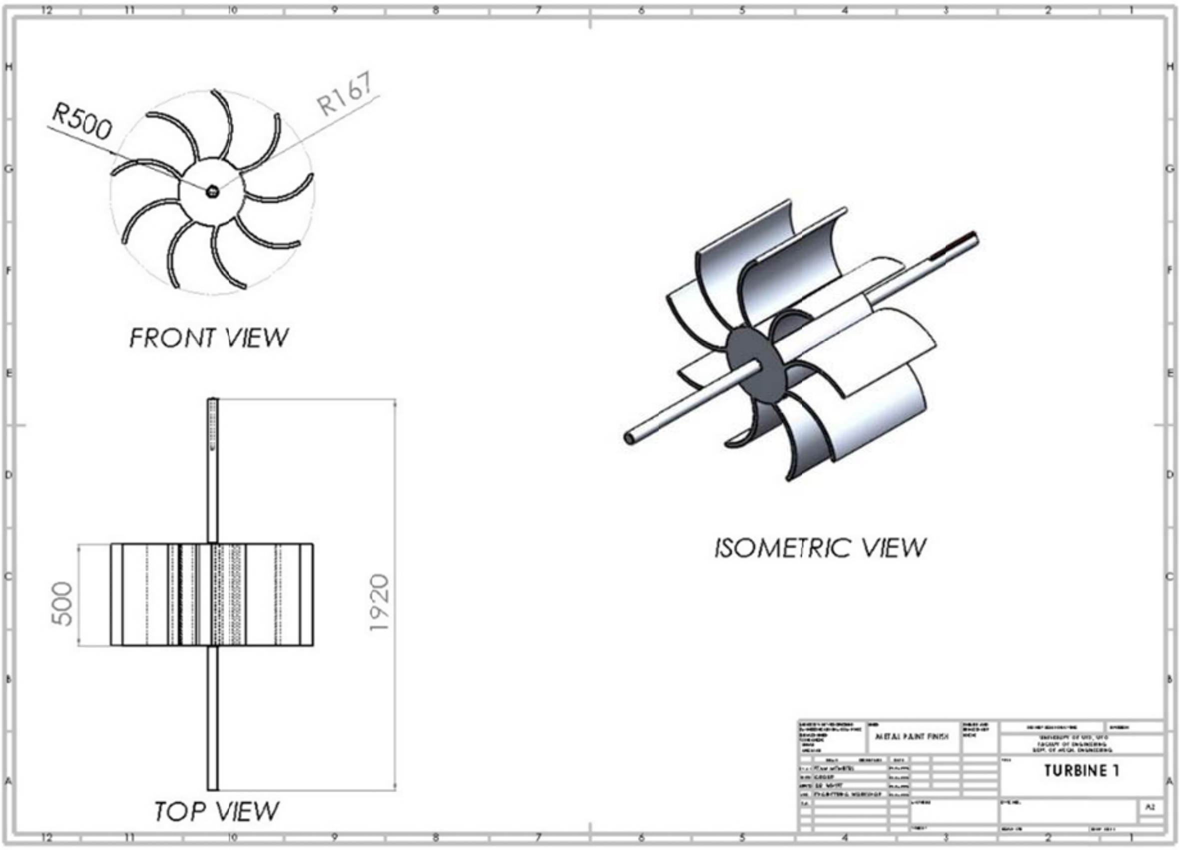

Figure 9: The CAD Drawing of Turbine 1 Showing Key Geometric Dimensions

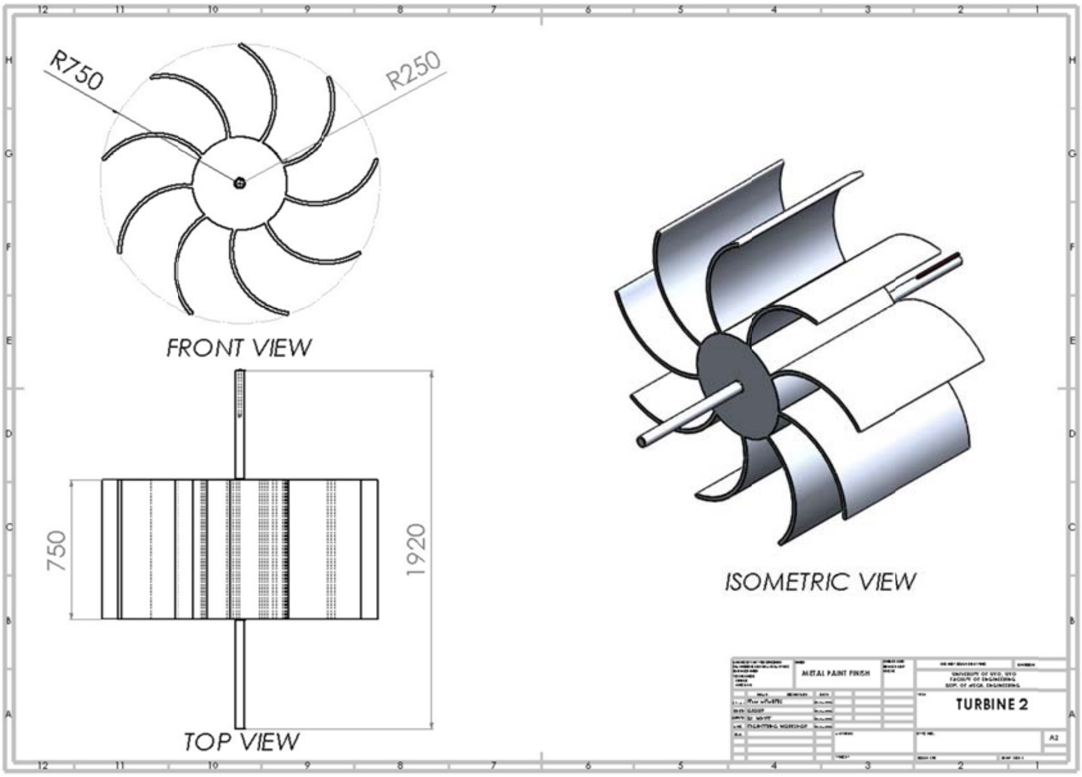

Figure 10: The CAD Drawing of Turbine 6 Showing Key Geometric Dimensions

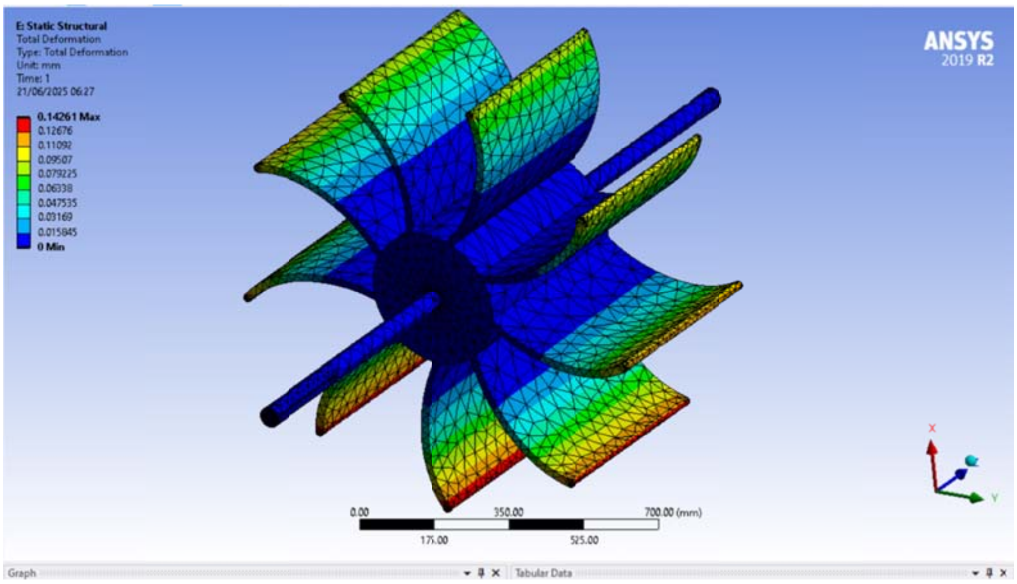

Figure 11: Total Deformation Contour Plot of Turbine 1 from Static Structural Analysis in ANSYS 2019.

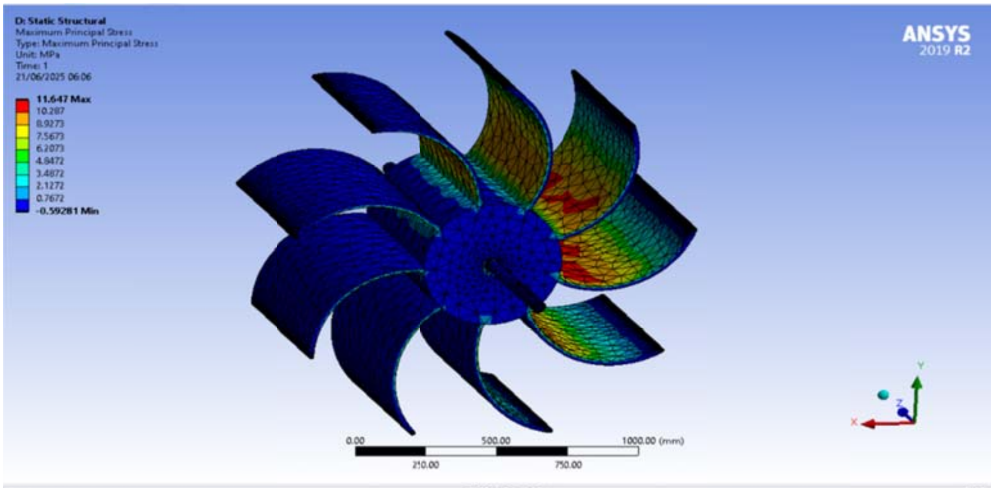

Figure 12: Total Deformation Contour Plot of Turbine 2 from Static Structural Analysis in ANSYS 2019

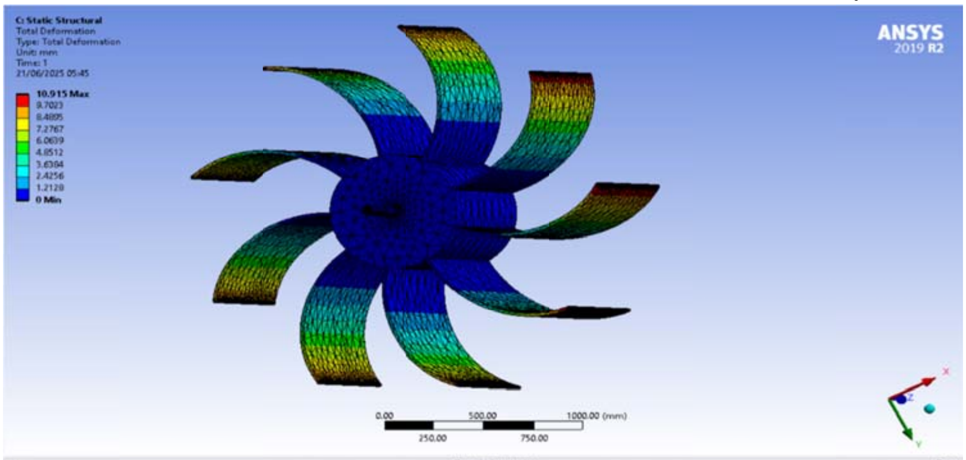

Figure 13: Total Deformation Contour Plot of Turbine 3 from Static Structural Analysis in ANSYS 2019.

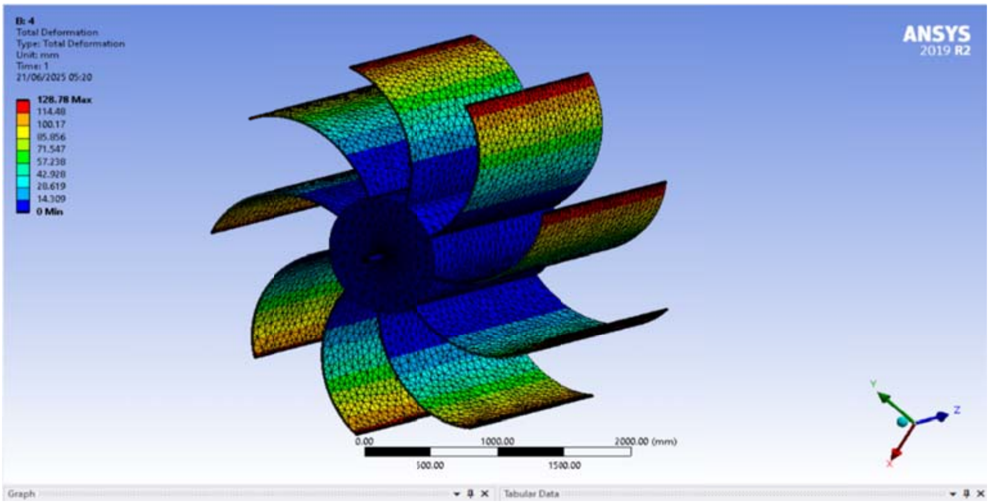

Figure 14: Total Deformation Contour Plot of Turbine 4 from Static Structural Analysis in ANSYS 2019

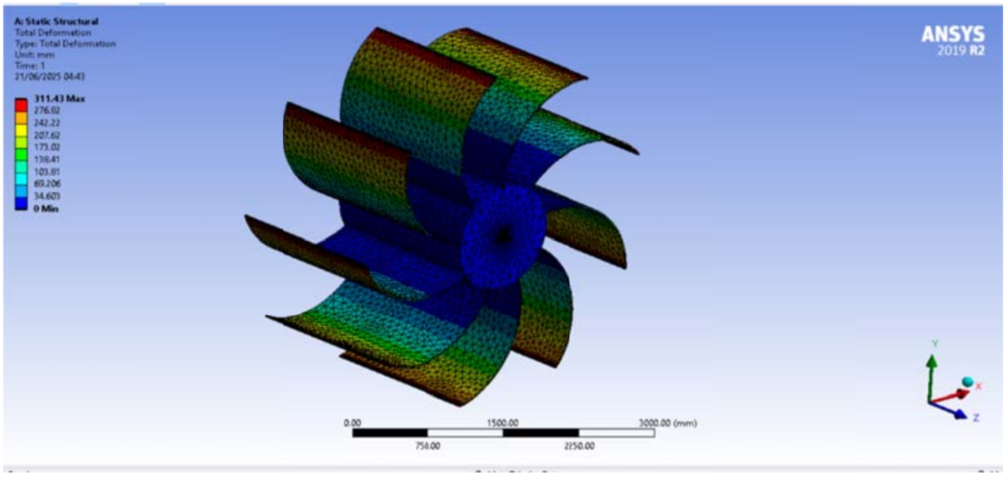

Figure 15: Total Deformation Contour Plot of Turbine 5 from Static Structural Analysis in ANSYS 2019

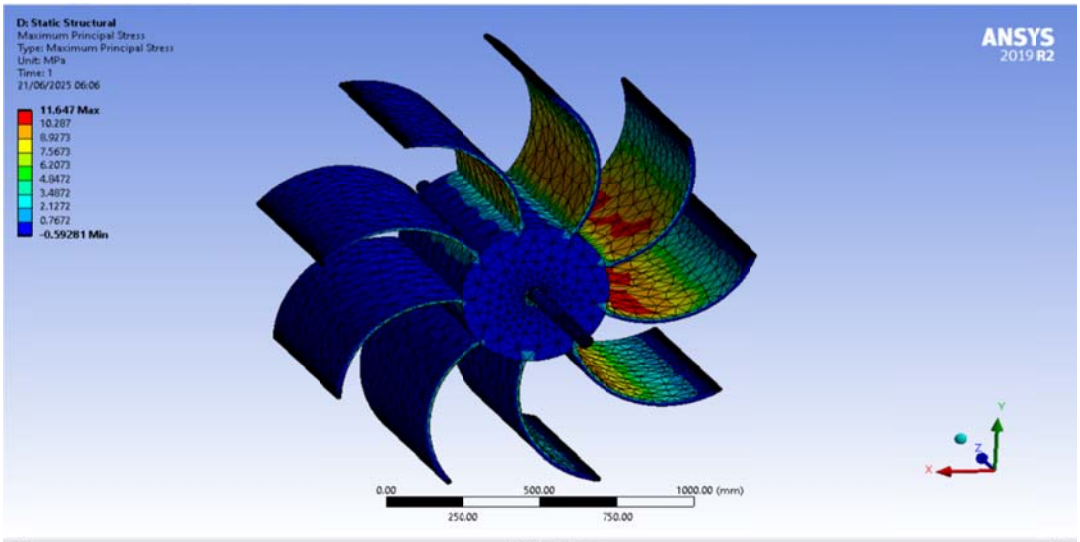

Figure 16: Total Deformation Contour Plot of Turbine 6 from Static Structural Analysis in ANSYS 2019

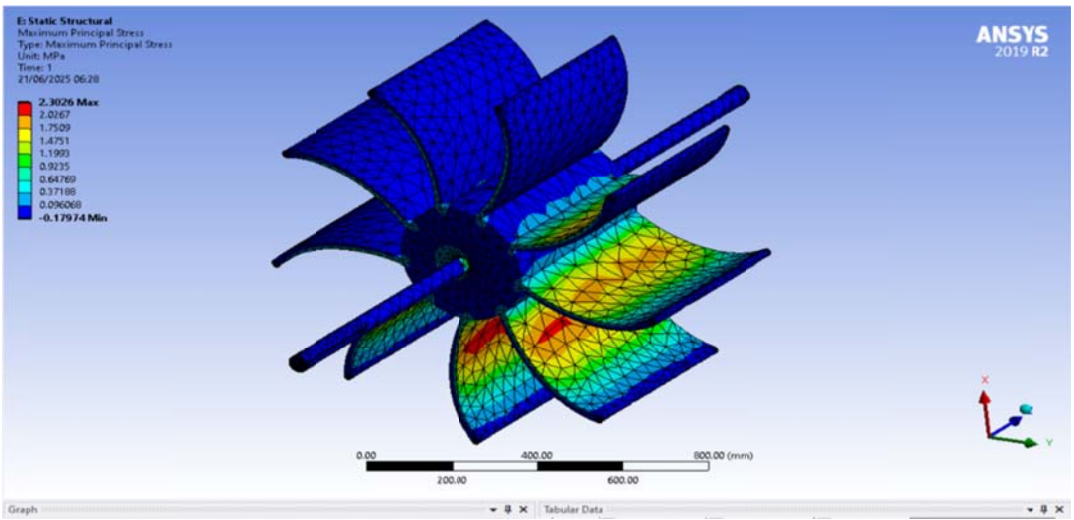

Figure 17: Maximum Principal Stress Distribution in Turbine 1 from Static Structural Analysis in ANSYS 2019.

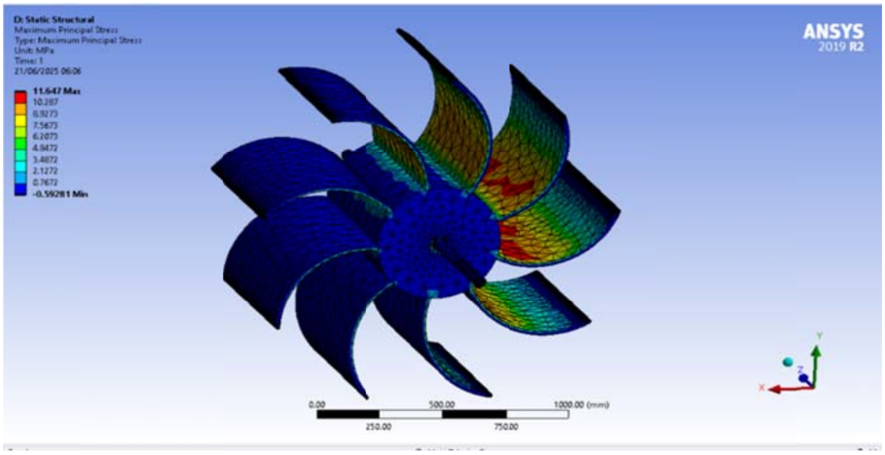

Figure 18: Maximum Principal Stress Distribution in Turbine 2 from Static Structural Analysis in ANSYS 2019

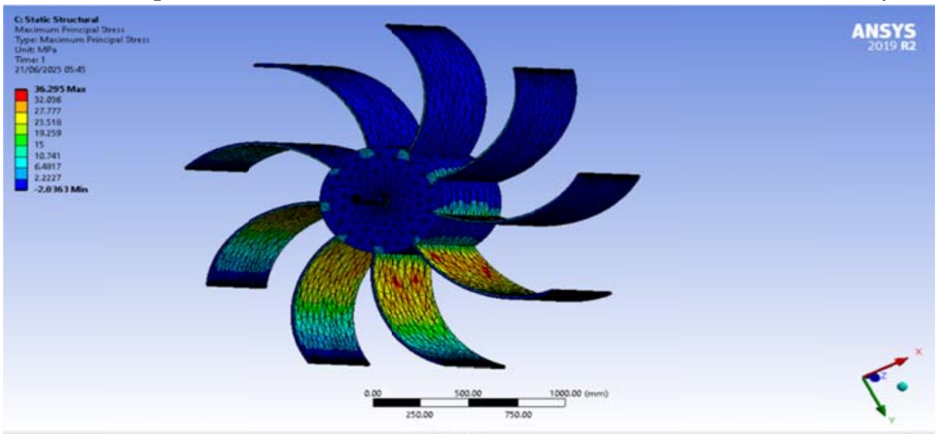

Figure 19: Maximum Principal Stress Distribution in Turbine 3 from Static Structural Analysis in ANSYS 2019

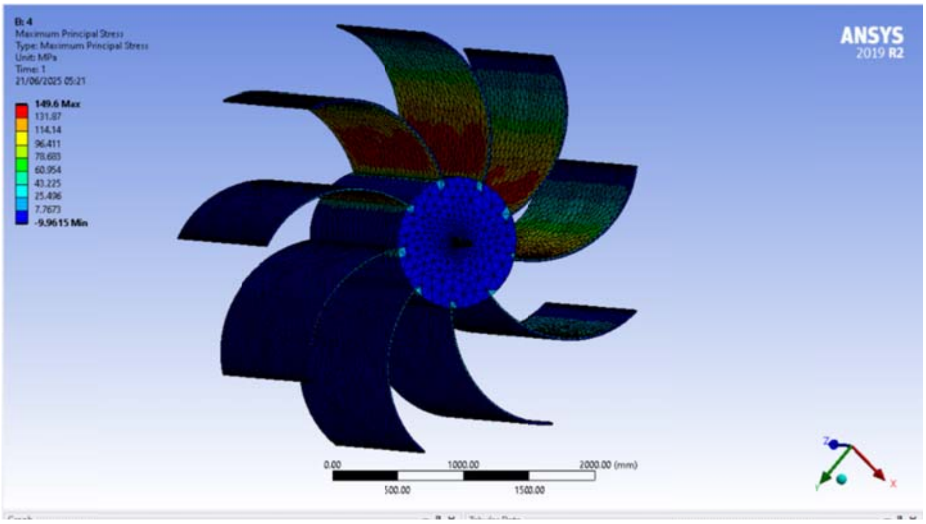

Figure 20: Maximum Principal Stress Distribution in Turbine 4 from Static Structural Analysis in ANSYS 2019

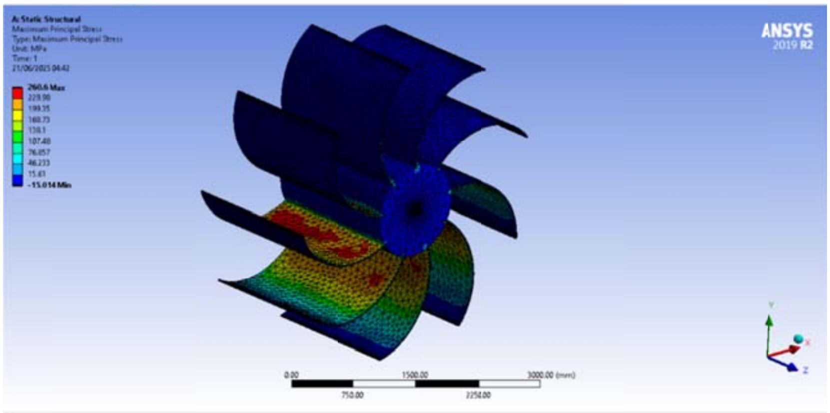

Figure 21: Maximum Principal Stress Distribution in Turbine 5 from Static Structural Analysis in ANSYS 2019

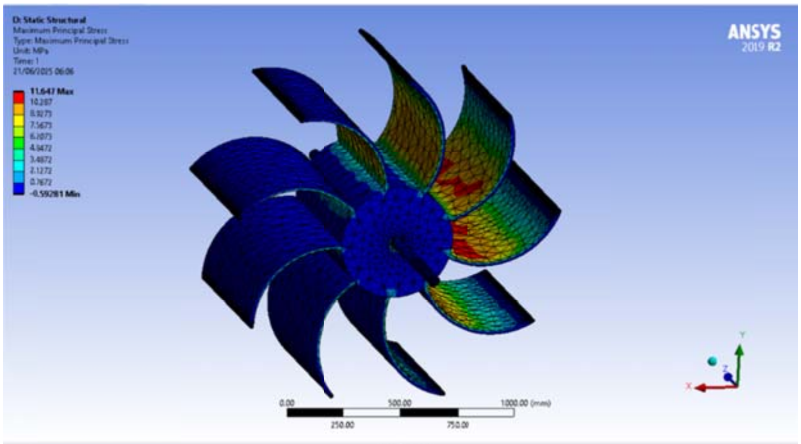

Figure 22: Maximum Principal Stress Distribution in Turbine 6 from Static Structural Analysis in ANSYS 2019

The results of the Contour Plot of Maximum Principal Stress Distribution in Turbines 1-6 from Static Structural Analysis as simulated in ANSYS 2019 are presented in Figure 23 to Figure 28.

The total deformation contour plot of water stand as obtained from the static structural analysis simulation conducted using ANSYS 2019 is presented in Figure 29. Similarly, the maximum principal stress distribution in water stand as obtained from the Static Structural Analysis simulation conducted using ANSYS 2019 is presented in Figure 30.

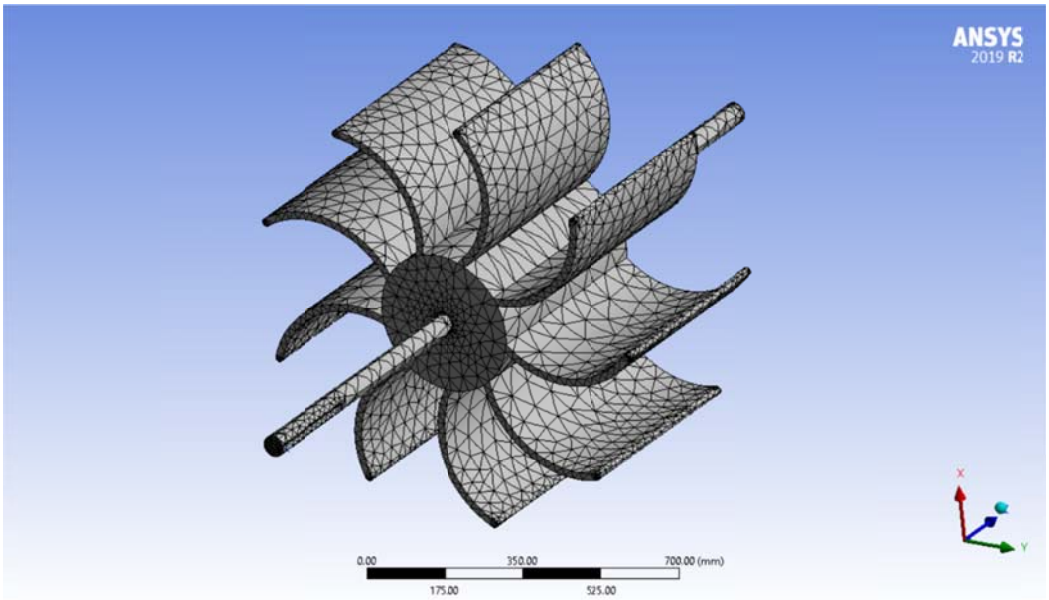

Figure 23: Contour Plot of Maximum Principal Stress Distribution in Turbine 1 from Static Structural Analysis in ANSYS 2019.

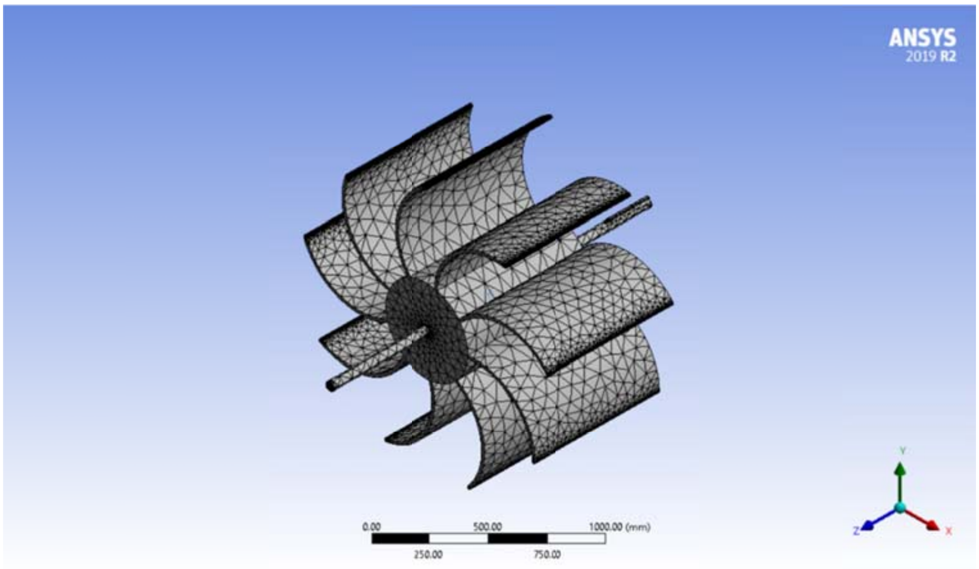

Figure 24: Contour Plot of Maximum Principal Stress Distribution in Turbine 2 from Static Structural Analysis in ANSYS 2019

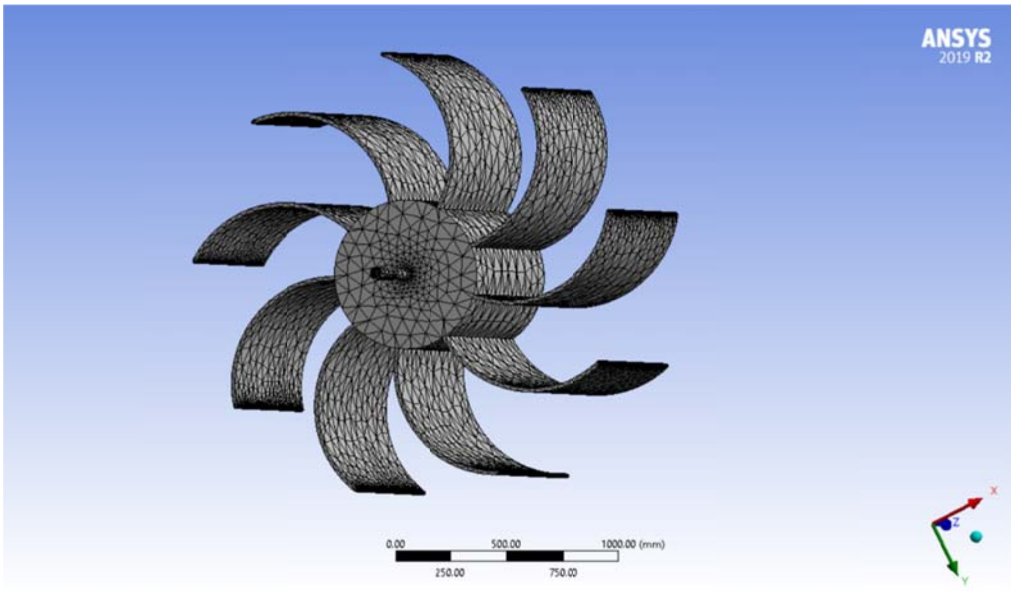

Figure 25: Contour Plot of Maximum Principal Stress Distribution in Turbine 3 from Static Structural Analysis in ANSYS 2019

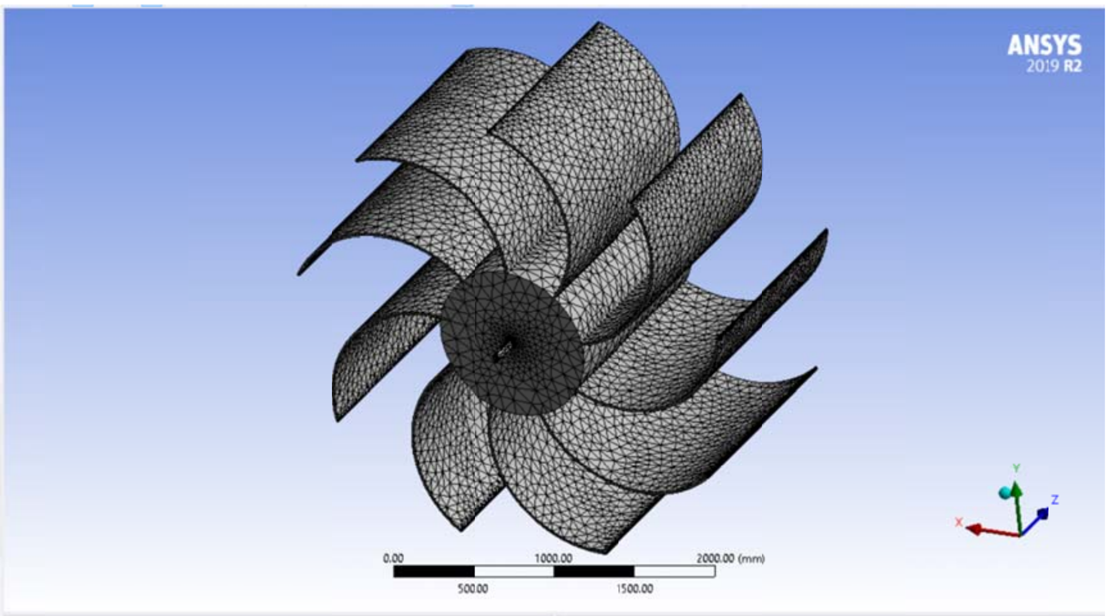

Figure 26: Contour Plot of Maximum Principal Stress Distribution in Turbine 4 from Static Structural Analysis in ANSYS 2019

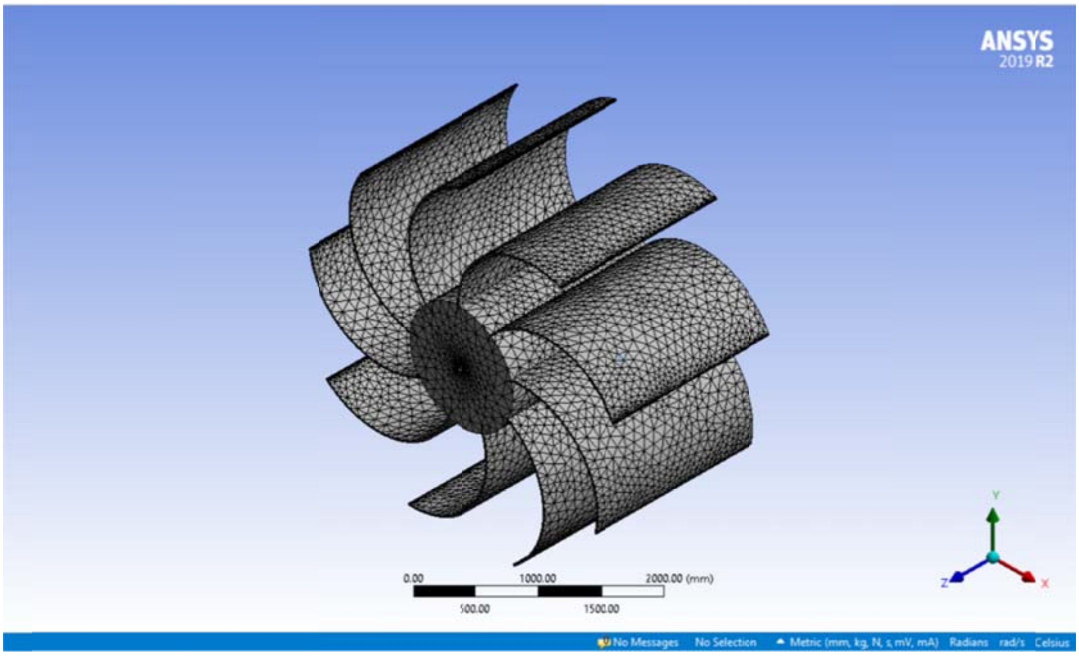

Figure 27: Contour Plot of Maximum Principal Stress Distribution in Turbine 5 from Static Structural Analysis in ANSYS 2019

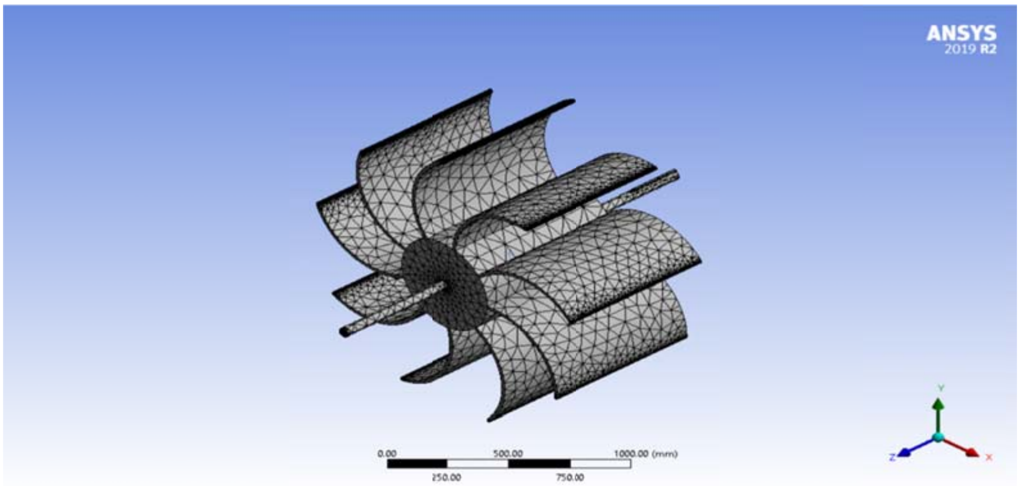

Figure 28: Contour Plot of Maximum Principal Stress Distribution in Turbine 6 from Static Structural Analysis in ANSYS 2019

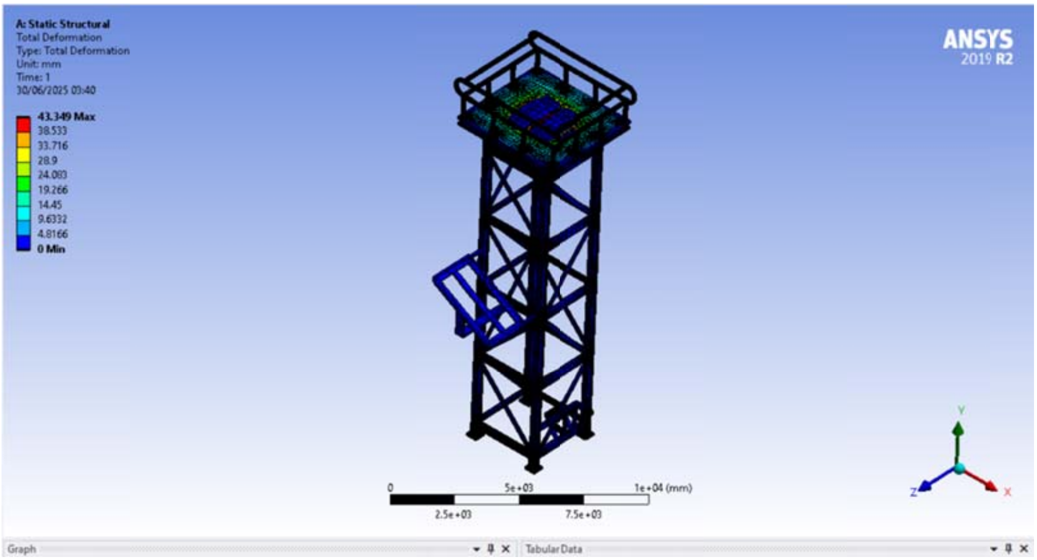

Figure 29: Total Deformation Contour Plot of Water Stand from Static Structural Analysis in ANSYS 2019.

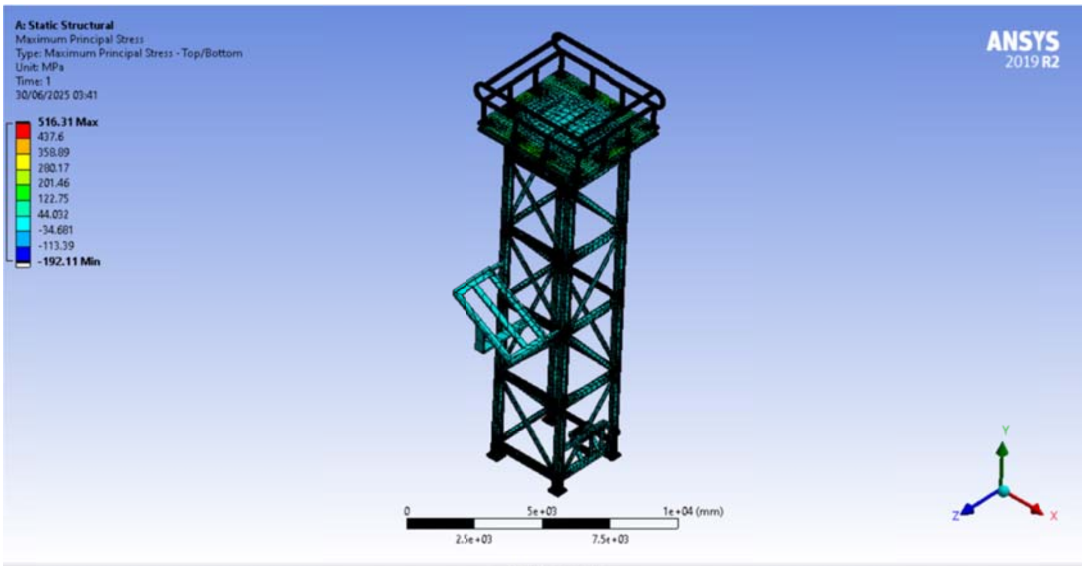

Figure 30: Maximum Principal Stress Distribution in Water Stand from Static Structural Analysis in ANSYS 2019.

These results outline the stress distribution across Turbine1 to Turbine 6 and the water stand, demonstrating the application of finite element analysis in evaluating structural performance under simulated operational conditions. This analysis is instrumental to identify potential failure zones, assessing component rigidity and displacement limits, validating design safety margins, and to optimize performance under operational loads.

The diagrams in Figure 11 to Figure 30 present results from Finite Element Analysis (FEA) simulations conducted using ANSYS 2019 on all the six turbine models and the structural stand. Each turbine underwent static structural analysis to evaluate deformation contours and principal stress distributions under simulated operational load. The deformation plots of Turbines 1 through 6 (Figure 11 to Figure 16) revealed moderate displacement concentrated near blade tips and hub edges locations typically susceptible to torsional loading. Meanwhile, the maximum principal stress plots indicated that the blade roots and shaft coupling zones experienced the highest stress intensities, as shown in Figure 17 to Figure 22. The water stand also showed moderate deformation and stress concentrations at its anchoring points (Figure 29 and Figure 30). The structural integrity, as visualized in these figures, supports the mechanical feasibility of the design, suggesting that all turbines can withstand expected hydraulic loads with minimal risk of failure under steady-state operation. However, regular monitoring and reinforcement of stress-prone zones are advisable.

4. Conclusion

The Finite Element Analysis (FEA) conducted on the turbines and water stand showed used in a Photovoltaic Pumped Hydroelectric Storage (PHES) system is presented, The FEA is conducted using ANSYS 2019. The finite element analysis (FEA) of the PHES turbines and water stands was performed to assess structural deformation and stress distribution. The results of the Finite Element Analysis (FEA) on the turbines and water stand showed that structural stress and deformation were within allowable limits, validating mechanical design safety.

References

1. Barbour, E., Wilson, I. G., Radcliffe, J., Ding, Y., & Li, Y. (2016). A review of pumped hydro energy storage development in significant international electricity markets. *Renewable and sustainable energy reviews*, *61*, 421-432.
2. Das, P., Das, B. K., Mustafi, N. N., & Sakir, M. T. (2021). A review on pump‐hydro storage for renewable and hybrid energy systems applications. *Energy Storage*, *3*(4), e223.
3. Klumpp, F. (2016). Comparison of pumped hydro, hydrogen storage and compressed air energy storage for integrating high shares of renewable energies—Potential, cost-comparison and ranking. *Journal of Energy storage*, *8*, 119-128.
4. ILghami, H., & Hadidi, A. (2021). Thermodynamic analysis of an energy storage system based on pumped hydro combined with compressed gas for use in a solar powerplant. *Journal of Energy Storage*, *33*, 102048.
5. Hadidi, A. (2025). Proposing a modified system based on recovery of preset pressurization energy in the integrated pumped-hydro and compressed gas energy storage system. *Results in Engineering*, 106118.
6. Zhang, H., Chen, D., Xu, B., Patelli, E., & Tolo, S. (2018). Dynamic analysis of a pumped-storage hydropower plant with random power load. *Mechanical Systems and Signal Processing*, *100*, 524-533.
7. Stutz, H. H., Norlyk, P., Sørensen, K., Andersen, L. V., Sørensen, K. K., & Clausen, J. (2020). Finite element modelling of an energy-geomembrane underground pumped hydroelectric energy storage system. In *E3S Web of Conferences* (Vol. 205, p. 07001). EDP Sciences.
8. El-Shaikh, M. (2016). Experimental investigation and finite element analysis for solving pumps resonance problem. *International Journal of Engineering Research & Technology (IJERT)*, *5*(4), IJERTV5IS040663.
9. Mohanpurkar, M., Ouroua, A., Hovsapian, R., Luo, Y., Singh, M., Muljadi, E., ... & Donalek, P. (2018). Real-time co-simulation of adjustable-speed pumped storage hydro for transient stability analysis. *Electric Power Systems Research*, *154*, 276-286.
10. Kim, N. H., Sankar, B. V., & Kumar, A. V. (2018). *Introduction to finite element analysis and design*. John Wiley & Sons.
11. Koutromanos, I. (2018). *Fundamentals of finite element analysis: Linear finite element analysis*. John Wiley & Sons.
12. Kurowski, P. M. (2022). *Finite element analysis for design engineers*. Sae international.
13. Lee, H. H. (2019). *Finite element simulations with ANSYS workbench 2019*. SDC publications.
14. Muhammad, A. I. S. H. A., Ali, M. A. H., & Shanono, I. H. (2020). Finite Element Analysis of a connecting rod in ANSYS: An overview.

In *IOP Conference Series: Materials Science and Engineering* (Vol. 736, No. 2, p. 022119). IOP Publishing.

15. Lisle, T. J., Shaw, B. A., & Frazer, R. C. (2017). External spur gear root bending stress: A comparison of ISO 6336: 2006, AGMA 2101-D04, ANSYS finite element analysis and strain gauge techniques. *Mechanism and Machine Theory*, *111*, 1-9.

16. Neto, M. A., Amaro, A., Roseiro, L., Cirne, J., & Leal, R. (2015). *Engineering computation of structures: the finite element method* (pp. 1-310). Cham, Switzerland: Springer International Publishing.