www.ijprems.com
editor@ijprems.com
INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 04, Issue 11, November 2024, pp : 389-391
e-ISSN : 2583-1062
Impact Factor : 7.001

STRUCTURAL ANALYSIS OF A CHAIR UNDER STATIC LOAD CONDITIONS

Ravikant[1], Hardial Singh[2]

[1,2]Assistant Professor, Department of Mechanical Engineering, Amity University Haryana, India

DOI: https://www.doi.org/10.58257/IJPREMS36675

ABSTRACT

Chairs are among the most commonly used items in daily life, primarily for seating purposes. Numerous varieties are available in the market, tailored to general and specialized needs. In this study, a CAD model of a chair was created in CATIA and imported into ANSYS Workbench. Static analysis was performed on a simple chair model using two different materials through finite element analysis (FEA).

The results were compared for equivalent stress, strain, and total deformation

Keywords: Chair, FEA, Catia, Ansys

1. INTRODUCTION

Concurrent engineering practices offer substantial benefits, including increased design flexibility, enhanced productivity, and cost reduction. The design of new furniture should satisfy three main criteria: it must be aesthetically pleasing, functional, durable, and economically feasible [1]. The introduction of FEM has greatly simplified the validation of furniture designs under various conditions [2].

To maximize these advantages, finite element analysis (FEA) tools should be introduced earlier in the product development cycle, enabling non-analyst designers to use them effectively [3-7].

Without integrating FEA early on, many components are only validated during prototype testing, resulting in costly and time-consuming redesigns if issues are found. A common workaround for non-analyzed parts is over-designing to compensate for unknown factors, often leading to suboptimal products.

FEA technology is widely used in aerospace and automotive industries and is increasingly adopted by furniture manufacturers. Chairs are an essential part of daily life, with plastic chairs commonly used for general purposes, steel-framed chairs with fiber seats popular in offices, and wooden chairs manufactured to suit specific needs.

Despite their widespread use, chairs are rarely analyzed for structural integrity and performance. Conducting FEA on chair designs can significantly reduce manufacturing costs and minimize design flaws, contributing to safer, more efficient products.

Jessica Song from JSJ Corporation applied ANSYS software in chair analysis and simulation, following the American National Standard for Office Furnishings established by the Business & Institutional Furniture Manufacturer's Association (BIFMA).

ANSYS was used to simulate BIFMA tests, identifying potential issues and providing insights into chair behavior to optimize the design for regulatory compliance. In another study, Song demonstrated ANSYS's role in optimizing the structural design of the Clara Table by refining leg wall thickness.

The study employed FEA to simulate the table leg assembly under loads per BIFMA standards, with contacts between parts representing real force transitions. Geometrical simplifications were necessary for efficient model size management. This nonlinear FEA approach accurately predicted structural responses, and structural modifications in ANSYS reduced leg deflection by about 20%, keeping stresses within permissible limits.

2. MATERIAL PROPERTIES

The materials used in this study for the chair model were Structural Steel and Wood. The properties of these materials are presented.

Table 1.

Materials	Young's Modulus (MPa)	Poisson's Ratio	Density (Kg/m^3)
Wood	9761	0.4	412
Structural Steel	21000	0.3	7800

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 04, Issue 11, November 2024, pp : 389-391
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

3. MODELING AND ANALYSIS

The CAD model of the chair was created using CATIA CAE software. This model was exported in .iges format and imported into SolidWorks, then saved in Parasolid format for subsequent import into ANSYS Workbench.

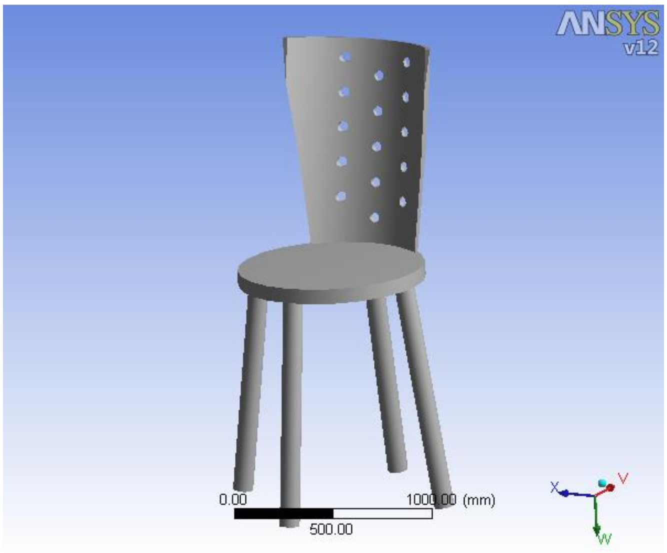

Figure 1- Cad model imported in Ansys Workbench

3.1 MESHING

Meshing of the chair model was performed in ANSYS Workbench using tetrahedral elements. The mesh comprised ,6288 nodes and 2735 elements.

As element size decreases, the number of elements increases proportionally. This increase continuously affects stress and other mechanical parameters, and significantly extends the time required to solve the discretized problems. Determining the optimal element size and count for analysis can therefore be challenging. To address this, a grid independence test was conducted to identify the optimal element size and number [8-14]

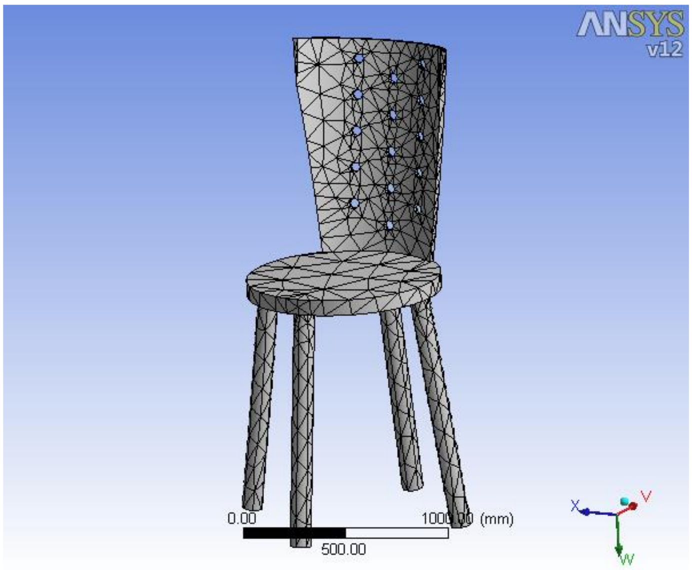

Figure 2- Meshed model of Chair

3.2 BOUNDARY CONDITIONS

After meshing, boundary conditions were applied to the model. Fixed support was assigned to the four legs of the chair, and a force of 960 N, representing the approximate weight of a 100 kg person, was applied

3.3 FEA RESULTS

The comparison table of the FEA results is shown in Table 2

Parameters	Wood	Structural Steel
Eq. Von-Mises Stress (MPa)	0.070013	0.072324
Total Deformation (mm)	0.023554	0.0011996

4. CONCLUSION

The FEA results indicate that chairs made from Structural Steel experience higher stress but less deformation compared to wooden chairs. Based on stress criteria, the wooden chair performs better, while the steel chair is preferable if minimizing deformation is prioritized.

5. REFERENCES

[1] Ceylan, Erkan, Ersan Güray, and Ali Kasal. "Structural analyses of wooden chairs by finite element method (FEM) and assessment of the cyclic loading performance in comparison with allowable design loads." Maderas. Ciencia y tecnología 23 (2021).

[2] Gawande, Mr DG, and Dr RE Thombre. "Stress Analysis on Chair Frame." International Research Journal of Engineering and Technology (IRJET) (2017): 1685-1689.

[3] Gupta, A., Sharma, A., Singh, H., & Raghav, A. K. (2018). Experimental and numerical investigation of taper helical and spiral tube thermal performance. International Journal of Energy Research, 42(14), 4417-4428.

[4] Singh, H. (2020). Flow performance comparison along the centerline in straight and s-shaped diffuser. Journal of Thermal Engineering, 6(1), 58-71.

[5] Singh, H., & Kumar, D. (2020). Effect of face width of spur gear on bending stress using AGMA and ANSYS. International Journal for Simulation and Multidisciplinary Design Optimization, 11, 23.

[6] Singh, H., & Kumar, D. (2020). Effect of face width of spur gear on bending stress using AGMA and ANSYS. International Journal for Simulation and Multidisciplinary Design Optimization, 11, 23.

[7] Verma, A., & Singh, H. (2024). CFD and artificial neural network-based modeling approach for the annual performance assessment of single slope single basin solar still. Heat Transfer.

[8] Joshi, Tanuj, et al. "Comparative investigation and analysis of hip prosthesis for different bio-compatible alloys." Materials Today: Proceedings 43 (2021): 105-111.

[9] Joshi, Tanuj, et al. "Dynamic analysis of hip prosthesis using different biocompatible alloys." ASME Open Journal of Engineering 1 (2022).

[10] Sharma, Ravikant, Vinod Kumar Mittal, and Vikas Gupta. "Homogeneous modelling and analysis of hip prosthesis using FEA." Journal of Physics: Conference Series. Vol. 1240. No. 1. IOP Publishing, 2019.

[11] Joshi, Tanuj, et al. "Dynamic fatigue behavior of hip joint under patient specific loadings." International Journal of Automotive and Mechanical Engineering 19.3 (2022): 10014-10027.

[12] Ravikant, Vinod Kumar Mittal, and Vikas Gupta. "Homogeneous and heterogeneous modeling of patient-specific hip implant under static and dynamic loading condition using finite element analysis." Journal of The Institution of Engineers (India): Series D 105.1 (2024): 1-20.

[13] Joshi, Tanuj, et al. "Design and analysis of metal-to-metal contact bolted flange joint using FEA tool." Advances in Engineering Design: Select Proceedings of FLAME 2020. Springer Singapore, 2021.

[14] Mittal, Vinod Kumar, and Vikas Gupta. "Dry and wet lubrication analysis for multi-material hip assembly." International Journal of Automotive and Mechanical Engineering 19.1 (2022): 9606-9622.