Journal of Advanced Research in Mechanical Engineering and Technology
Volume 10, Issue 3&4 - 2023, Pg. No. 18-27
Peer Reviewed Journal

Research Article

Finite Element Analysis of Mechanical Chair

Nitesh Pandey[1], Manish kumar[1], Amit Kumar Srivastava[1], Pankaj Gupta[1]

Jaypee University of Engineering and Technology, Guna (M.P.), India.

INFO

Corresponding Author:
Pandey N, Jaypee University of Engineering and Technology, Guna (M.P.), India.
E-mail Id:
niteshpandey46974@gmail.com
Orcid Id:
https://orcid.org/0009-0006-2878-075X
How to cite this article:
Pandey N, kumar M, Kumar A, Gupta P. Finite Element Analysis of Mechanical Chair. *J Adv Res Mech Engi Tech* 2023; 10(3&4): 18-27.

Date of Submission: 2023-11-20
Date of Acceptance: 2023-12-20

ABSTRACT

This research explores the design of a steel chair addressing the crucial need for adaptability in modern living spaces. Advanced engineering techniques, specifically Finite Element Analysis (FEA), are employed to meticulously analyze and enhance the structural performance of this versatile furniture piece. The study focuses on simulating various scenarios using FEA to ensure that the chair meets stringent ergonomic standards, providing user safety and comfort in both seating and table configurations. The integration of transformative features, plays a pivotal role in refining the chair's form through iterative processes based on predefined constraints. The paper systematically presents FEA findings, emphasizing stress distribution, displacement, and mass considerations within the chair's structure. Simultaneously, visual insights into the chair's design evolution through Generative Design offer a comprehensive understanding of its transformative journey.

Keywords: Stress analysis, finite element analysis (FEA), Autodesk Fusion 360, designing, CAD/CAM – computer aided design and computer-Aided manufacturing

Introduction

This study explores the application of Finite Element Analysis (FEA) in the optimization and analysis of a chair's structural performance, utilizing Autodesk Fusion 360 as the chosen platform. Fusion 360 is a robust CAD/CAM software known for its parametric modeling capabilities, integrated CAM features, and versatility in synchronously incorporating advanced technologies like FEA.[2,4,9] The choice of Fusion 360 is grounded in its collaborative workflows, cost-effectiveness, and efficient design iteration, offering an ideal environment for a comprehensive examination of the chair's mechanical behavior. Finite Element Analysis (FEA) serves as a cornerstone in engineering disciplines, allowing for a detailed understanding of structural responses under various conditions. In the context of chair design, FEA enables precise simulations of stress distribution, displacement, and other critical factors, providing invaluable insights into the chair's performance. The paper focuses on elucidating the procedural aspects of FEA within Fusion 360, emphasizing its role in validating and enhancing the structural integrity of the chair.[1,4] Unlike generative design, FEA directly simulates the impact of loads and constraints on the chair's components, providing engineers and designers with quantitative data to make informed decisions about the design's robustness and safety.[13] Key elements of the FEA analysis process include defining materials, specifying constraints, applying loads, and conducting linear structural analyses. Autodesk Fusion 360 streamlines this process by providing a user-friendly interface for preprocessing, solving mathematical equations, and interpreting results using tools like Hypermesh, LS-Dyna, and Hyperview. The main objectives of this research paper include:

- Learn how to use Finite Element Analysis (FEA) in Autodesk Fusion 360, understanding each step-in detail.
- Record the step-by-step process from drawing in 2D and 3D to creating realistic images, testing in simulations, and figuring out the cost – all within Fusion 360.
- Show and explain the outcomes of the FEA process using different tools like pictures, tables, and graphs for a better understanding.
- Examine the results from Finite Element Analysis closely, paying attention to things like how stress is spread, how much things move (displacement), and other important factors for how the chair holds up.

- Use the detailed FEA results to make sense of how the chair performs under different situations and restrictions.
- Take useful lessons from the FEA analysis that can help design not just this chair but also other furniture and structures, contributing to better ways of making things strong and safe.
- These goals work together to show the use of FEA in Fusion 360 to make sure a chair is strong and reliable. The hope is that will not only help with understanding this chair but also lead to better ways of designing furniture and structures in general.

Methodology

Procedure for Finite Element Analysis (FEA) and Design Optimization

The methodology begins with defining the problem and ideation, progressing through idea conceptualization, visualization, and subsequent verification. Once conceptually solidified, chair design is digitally articulated using software.[3] The generated prototype then undergoes Finite Element Analysis (FEA), a critical step in ensuring structural integrity and optimal performance.[1] The design process initiates with identifying a problem or gap, leading to idea generation, conceptualization, visualization, and conceptual verification.[10]

Following the FEA, various outcomes are obtained, including stress distribution, mass, displacement, and strength of the chair. These outcomes serve as crucial metrics for evaluating the chair's structural performance under different conditions. The design optimization process is categorized into early, medieval, and end stages, encompassing imagination, idea generation, conceptualization, sketching, 3D modelling, and final outcome selection.[11,12]

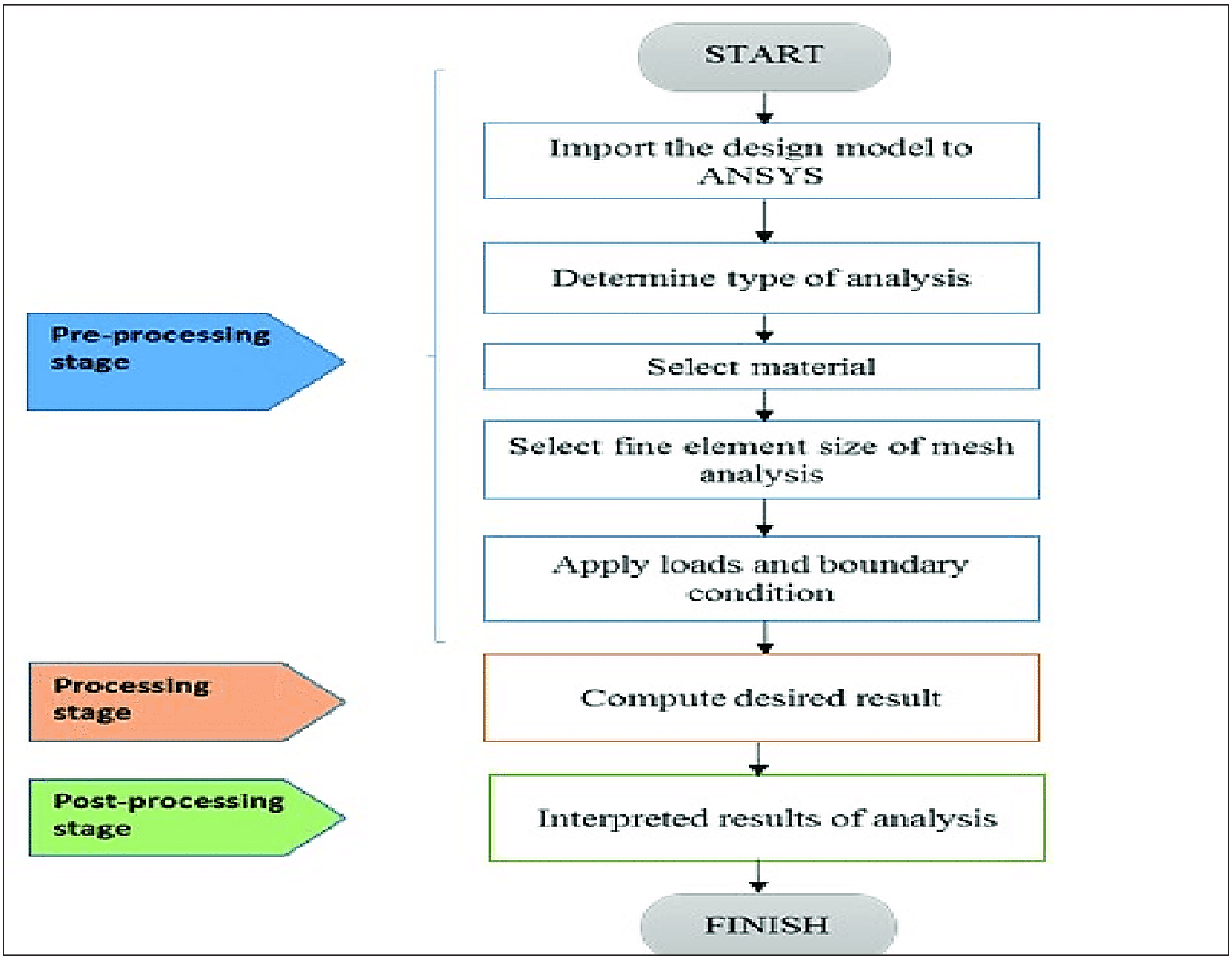

Figure 1.Procedure in toolbar of Finite Element Analysis (FEA)

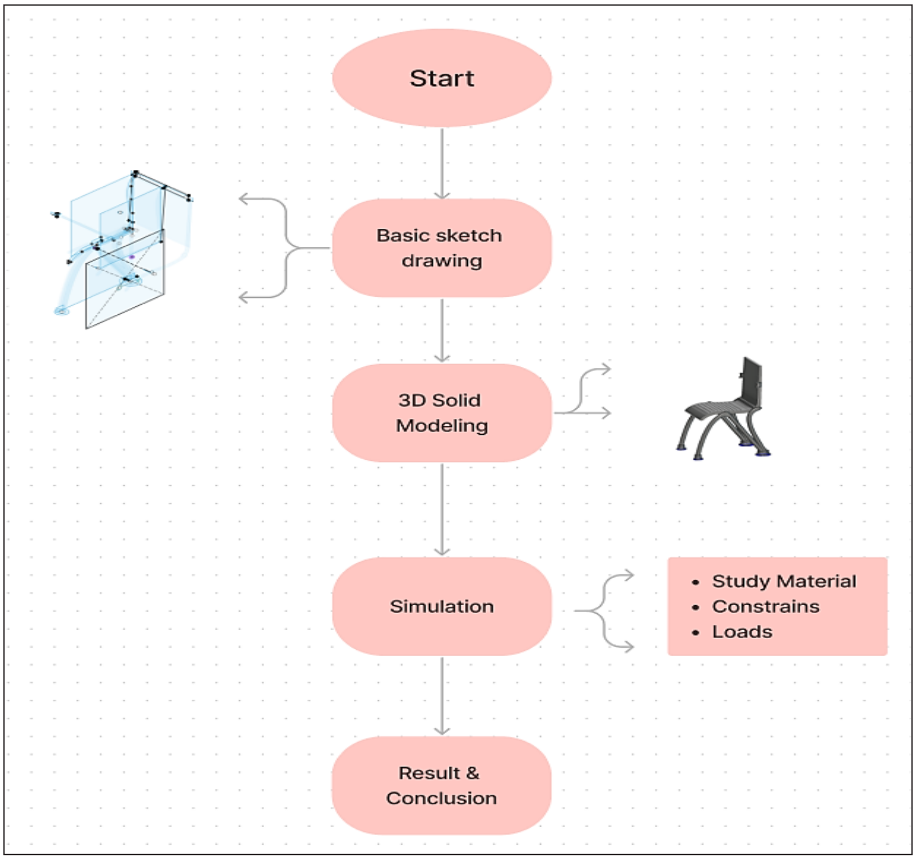

Figure 2.Design Approach

Applying Finite Element Analysis in chair

The specific chair component under consideration is introduced, and the initial design as seen in Fig. 3 is presented for FEA. The goal is to optimize the design for enhanced structural performance and usability.

Figure 3. Design of the chair

Finite Element Analysis Defining Procedure

To initiate the Finite Element Analysis (FEA) process in Fusion 360, begin by launching the software and opening the designated design file. Once within the Fusion 360 environment, seamlessly navigate to the Simulation workspace, where the analytical procedures will take place. In the Simulation workspace, commence the FEA by selecting “New Study” or “New Material Study.” Specify the analysis type as “Static Stress” to assess the structural behaviour under static loads.

Next, designate the material for the study, opting for steel in the case of a chair design. Ensure accurate representation by assigning appropriate material properties like Young’s Modulus and Poisson’s Ratio. Simulate real-world conditions by applying constraints, fixing the positions of the chair’s bases using structural constraints. Introduce external forces through the application of loads; for instance, apply a vertical structural load on the chair’s seating area. Specify load magnitude by converting weight to Newtons, considering gravitational acceleration.[5]

Include the effects of gravity by applying a downward load based on the gravitational acceleration of 9.8 m/s^2 in Fig 4. Conduct a thorough review of applied loads and constraints to validate their fidelity to the intended physical scenario. Subsequently, initiate the analysis by clicking on the “Solve” button, prompting Fusion 360 to employ the

Finite Element Method for calculating the chair structure's response under the specified conditions.

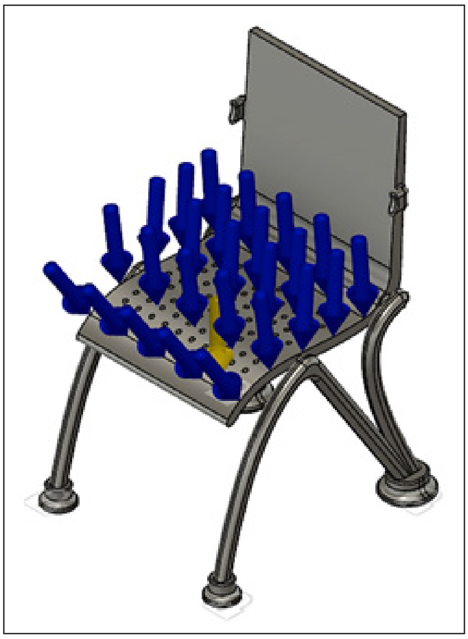

Figure 4.Structural load

Throughout the analysis process, diligently monitor Fusion 360 for any warning messages or errors. Once the analysis is complete, review the results, examining stress distribution, displacement, and other pertinent data. Interpret these results within the context of the design, identifying areas of concern such as high stress, deformation, or potential failure. If necessary, implement design iterations based on the analysis findings to enhance structural performance.

Figure 5.Structural constraints

Thoroughly document each step of the FEA process, including material properties, applied loads, constraints, and results. Enhance the documentation with screenshots and visualizations that effectively communicate key aspects of the analysis. Finally, compile a comprehensive report summarizing the FEA analysis, emphasizing key findings, and providing recommendations for design improvements if warranted. Following this systematic approach ensures a detailed and well-documented record of the Finite Element Analysis conducted in Fusion 360.

Result and Discussion

Stress Analysis

The Fusion 360 analysis for the static stress of the chair design gave us useful insights into how the structure behaves under specific conditions. The study, which focused on a 1:1 simulation model, employed steel as the material. Steel has a density of 7.850E-06 kg/mm³, a Young's Modulus of 210,000.00 MPa, a Poisson's Ratio of 0.30, and a yield strength of 207.00 MPa. These material properties help us understand how the chair responds to different forces and loads.[6,7]

Mesh

Configuring the mesh parameters in Fusion 360 is crucial for gaining a comprehensive understanding of how the chair responds to stress. The average element size, designated as 10% of the overall model size, plays a key role in determining the analysis's level of detail and precision. Maintaining a consistent mesh size for all parts of the chair through the scale mesh size per part ensures uniformity across the entire model. The utilization of parabolic elements contributes to a more accurate representation of deformations.

Table 1.Brief description of mesh outcome

Average Element Size (% of model size)	
Solids	10
Scale Mesh Size Per Part	No
Average Element Size (absolute value)	-
Element Order	Parabolic
Create Curved Mesh Elements	Yes
Max. Turn Angle on Curves (Deg.)	60
Max. Adjacent Mesh Size Ratio	1.5
Max. Aspect Ratio	10
Minimum Element Size (% of average size)	20

The inclusion of curved mesh elements becomes integral for capturing the intricate curves and complex shapes of the chair, enhancing the realism of the simulation. Allowing curves to bend up to a maximum of 60 degrees during mesh creation adds flexibility, especially in areas with pronounced curves. The control of the difference in size between neighbouring mesh elements (max adjacent mesh size ratio) at 1.5 facilitates a smooth transition between various mesh densities. Limits on the aspect ratio, capped at a maximum of 10, prevent elements from stretching or distorting excessively. In Fig. 10 the minimum element size, established at 20% of the average size, prevents the creation of overly small elements that could compromise analysis accuracy. In this analysis, adaptive mesh refinement is not utilized, as indicated by zero refinement steps.

Table 2. Adaptive Mesh Refinement

Number of Refinement Steps	0
Results Convergence Tolerance (%)	20
Portion of Elements to Refine (%)	10
Results for Baseline Accuracy	von Mises Stress

The convergence tolerance for results is set at 20%, ensuring the analysis concludes when results are within this percentage of convergence. In a scenario involving mesh refinement, 10% of the elements would undergo refinement based on the results. Prioritizing von Mises stress as the baseline accuracy parameter acts as a primary indicator of potential material failure under applied loads.

Material

The material specifications utilized in the Fusion 360 analysis, particularly for the chair design, play a crucial role in understanding how the structural response to stress.

The unique properties of the selected steel material include a density of 7.850E-06 kg/mm³, Young's Modulus of 210,000.00 MPa, Poisson's Ratio of 0.30, yield strength of 207.00 MPa, ultimate tensile strength of 345.00 MPa, thermal conductivity of 0.056 W / (mm°C), thermal expansion coefficient of 1.200E-05 / °C, and specific heat of 480.00 J / (kg°C).

Table 3. Steel properties

Density	7.850E-06 kg / mm^3
Young's Modulus	210000.00 MPa
Poisson's Ratio	0.30
Yield Strength	207.00 MPa
Ultimate Tensile Strength	345.00 MPa
Thermal Conductivity	0.056 W / (mm C)
Thermal Expansion Coefficient	1.200E-05 / C
Specific Heat	480.00 J / (kg C)

These characteristics are essential inputs for the Finite Element Analysis (FEA), defining how the steel behaves under diverse conditions. They impact mass, elasticity, deformation under stress, resistance to permanent deformation, maximum stress endurance, and the material's response to temperature changes. Ensuring accuracy in these material properties enhances the precision of the simulation, offering valuable insights into the chair's structural integrity and performance when subjected to static stress conditions.

Contacts

Bonded

Within the Fusion 360 analysis of the chair design, the inclusion of bonded connections stands as a fundamental aspect, serving to facilitate the simulation of interactions among distinct structural elements. Specifically denoted as [S] Bonded117, [S] Bonded118, and [S] Bonded119 in the context of Simulation Model 1:1 and Component1:1, these labels uniquely identify and characterize the bonded connections, emphasizing a cohesive structural linkage between the specified components.

Table 4. Brief description of Bonds

Name
[S] Bonded117 [Simulation Model 1:1||Component1:1]
[S] Bonded118 [Simulation Model 1:1||Component1:1]
[S] Bonded119 [Simulation Model 1:1||Component1:1]

The term "bonded" conveys a state in which these components collaboratively share loads and resist deformation, ensuring a synchronized response to externally applied forces. These bonded connections assume a pivotal role in faithfully representing the physical integrity of the chair throughout the Finite Element Analysis (FEA). Their significance lies in offering valuable insights into the inter-component relationships and their collective impact on the overall structural behaviour when subjected to static stress conditions.

Mesh

The meshing process utilized an average element size of 10% of the model size for solids, resulting in 173,929 nodes and 105,916 elements. The study, set for static stress analysis, incorporated contact tolerances of 0.10 mm and gravitational effects with a magnitude of 9.807 m/s² in the downward direction.

Load Case I

Constrains

In Fusion 360 analysis, Load Case1 involves fixed constraints denoted as Fixed1, aiming to replicate immobility within structural elements. These constraints, set as Fixed in the X, Y, and Z directions.

The imposition of these constraints in the Finite Element Analysis (FEA) ensures an accurate portrayal of static stress conditions, enabling a realistic simulation of the chair's response to applied loads. This methodology enhances analytical precision, facilitating a comprehensive exploration of structural integrity and potential stress points inherent in the design.

Loads

Gravity

In Fig. 10 Gravity load of 9.807 m/s² is applied to simulate the impact of gravitational forces on the chair structure. Specified as acting in the negative Z-direction, this load type accurately represents the acceleration due to gravity.

Figure 6.Selected entity for gravity

Table 5.Brief description of gravitational load

Type	Gravity
Magnitude	9.807 m / s^2
X Value	0.00 m / s^2
Y Value	0.00 m / s^2
Z Value	-9.807 m / s^2

This inclusion in the Finite Element Analysis (FEA) ensures a realistic assessment of the chair's response to external forces, contributing to the precision of the simulation under static stress conditions. By mimicking real-world gravitational effects, this approach enhances the overall accuracy of the analysis, providing valuable insights into the structural behaviour of the chair when subjected to varying loads.

Force1

In the Fusion 360 analysis, a Force1 load is implemented to simulate a vertical force on the chair's sitting area as seen in Table. 6 With a magnitude of 1176.80 N and directed in the negative Z-axis, this force mimics an external load acting downward.[2]

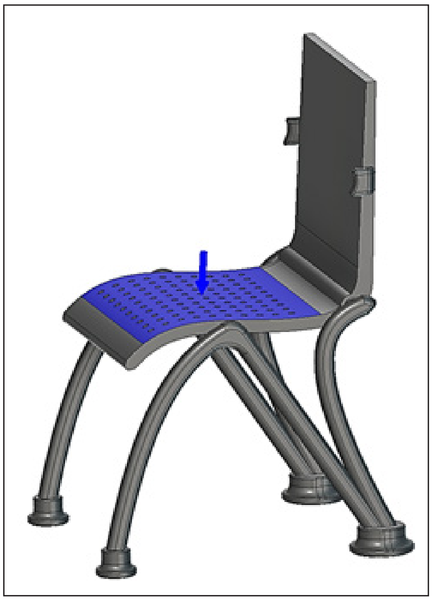

Figure 7.Selected entity of applied force

Table 6.Brief description of applied force

Type	Force1
Magnitude	1176.80 N
X Value	0.00 N
Y Value	-87.02 N
Z Value	-1173.578 N
Force Per Entity	No

The X and Y values are set to zero, signifying no force in those directions. The Force Per Entity parameter is configured as No, indicating that the entire force is applied as a single unit rather than distributed. This Force1 input in the Finite Element Analysis (FEA) ensures a precise assessment of the chair's structural response to applied loads, enriching the simulation's accuracy under static stress conditions.

Result Summary

The results obtained from the Fusion 360 analysis provide comprehensive insights into the structural performance of the chair under static stress conditions. In Fig. 8 safety factor, critical indicator of structural robustness, ranges from 9.756 to 15.00, ensuring a satisfactory margin of safety.[8]

In Fig 9 Stress analysis, represented by von Mises stress and principal stresses, unveils the distribution of forces within the chair's structure. In Fig. 9, blue denotes minimal stress, red signifies high pressure (up to 21.219 MPa). The seat area displays a distinct cyan mix, indicating a moderate stress level. This visual representation highlights stress patterns for effective analysis.[8]

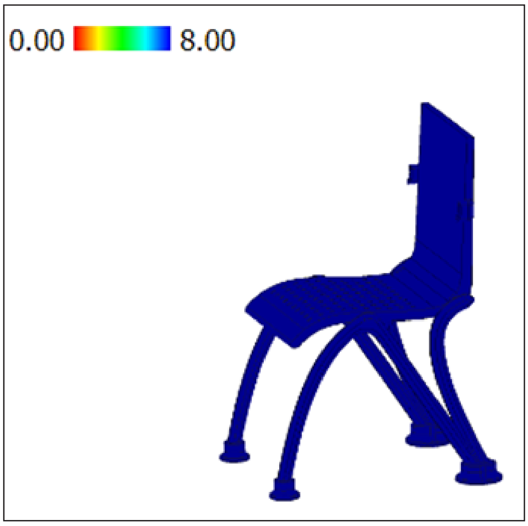

Figure 8.Safety Factor (Per body)

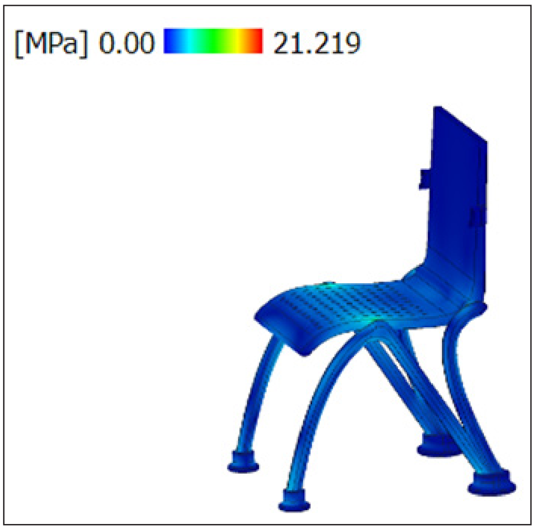

Figure 9.Von stress

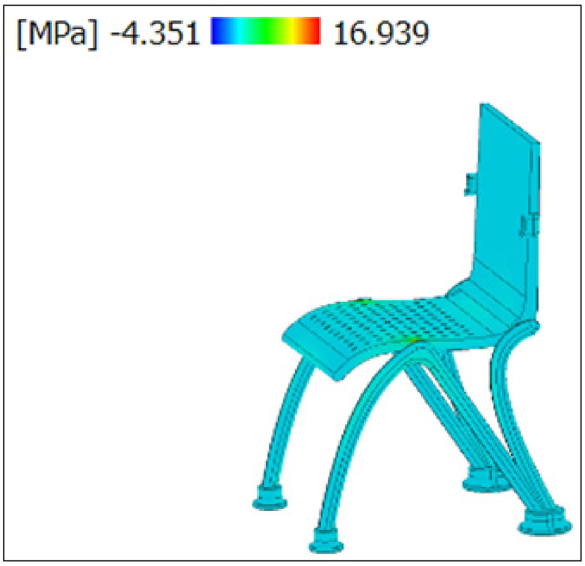

Figure 10.1st Principal

Fig. 10 illustrates 1st Principal stress ranging from -4.351 to 16.939 MPa. The chair's cyan colour signifies overall stress, while green in the seat area indicates heightened stress concentration in that specific region.

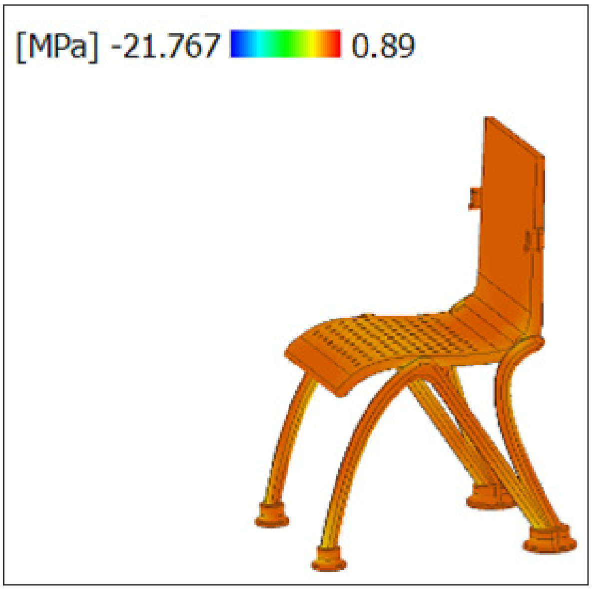

Figure 11.3rd Principal

In Fig. 12 Displacement values indicate the degree of deformation, with total displacement ranging from 0.00 mm to 0.034 mm in various directions.

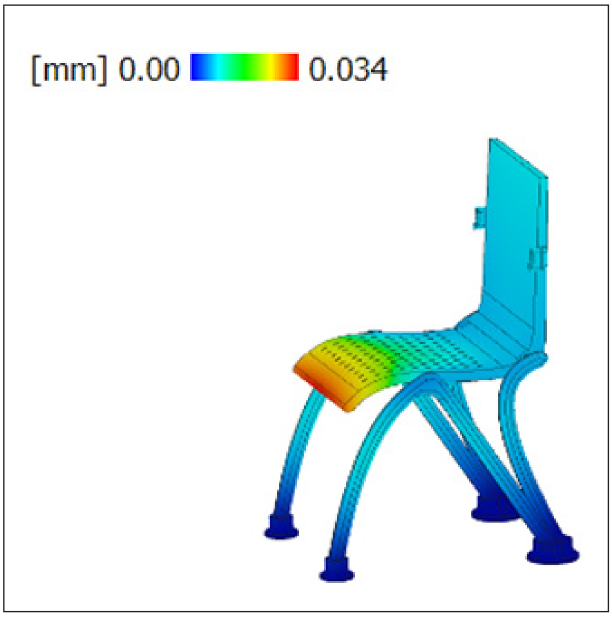

Figure 12.Displacement

In Fig. 12, the front part of the seat area is marked by red, indicating maximum displacement. As moving inward, the colour transitions to yellow, green, and blue, signifying progressively less displacement.

Table 7.Brief description of result outcome

Name	Minimum	Maximum
Safety Factor		
Safety Factor (Per Body)	9.756	15.00
Stress		
von Mises	4.426E-05 MPa	21.219 MPa
1st Principal	-4.351 MPa	16.939 MPa
3rd Principal	-21.767 MPa	0.89 MPa
Normal XX	-7.769 MPa	5.42 MPa
Normal YY	-20.627 MPa	13.214 MPa
Normal ZZ	-8.172 MPa	5.759 MPa
Shear XY	-7.002 MPa	6.741 MPa
Shear YZ	-9.393 MPa	8.604 MPa
Shear ZX	-8.15 MPa	7.072 MPa
Displacement		
Total	0.00 mm	0.034 mm
X	-0.005 mm	0.005 mm
Y	-0.007 mm	0.005 mm
Z	-0.033 mm	0.002 mm
Reaction Force		
Total	0.00 N	7.569 N
X	-2.075 N	1.904 N
Y	-1.173 N	3.521 N
Z	-4.178 N	7.005 N
Strain		
Equivalent	3.270E-10	1.611E-04
1st Principal	2.020E-10	1.138E-04
3rd Principal	-1.608E-04	1.429E-08
Normal XX	-1.664E-05	2.373E-05
Normal YY	-8.976E-05	5.027E-05
Normal ZZ	-3.375E-05	4.176E-05
Shear XY	-8.669E-05	8.346E-05
Shear YZ	-1.163E-04	1.065E-04
Shear ZX	-1.009E-04	8.756E-05
Contact Pressure		
Total	0.00 MPa	8.127 MPa
X	-3.073 MPa	2.125 MPa
Y	-4.819 MPa	7.252 MPa
Z	-6.741 MPa	5.199 MPa
Contact Force		
Total	0.00 N	337.673 N
X	-66.967 N	52.155 N
Y	-130.458 N	85.223 N
Z	-335.646 N	221.364 N

These results, depicted in detail in Table. 7, provides a thorough overview of the chair's Finite Element Analysis (FEA) results, giving us valuable insights into how it handles different loads. The safety factor, ranging from 9.756 to 15.00, shows how well the design can handle applied loads, ensuring there's a safety margin. A higher safety factor means a stronger design with more safety against potential issues. The von Mises stress, ranging from 4.426E-05 MPa to 21.219 MPa, is crucial for understanding material strength. Values for 1st and 3rd principal stresses, along with normal and shear stresses, explain how stress is distributed within the chair. Negative principal stresses indicate areas under compression. Displacement values, from 0.00 mm to 0.034 mm, tell us how much the chair deforms under the applied loads. Detailed data along X, Y, and Z axes helps us understand the direction and number of deformations. Reaction forces at different points of the chair, along the X, Y, and Z axes, show how the chair responds to external loads. These values give insights into the forces exerted on the chair's structure. Strain, showing material deformation, is presented as equivalent, principal, and normal strains. These values help assess the extent of material deformation. Contact pressure, ranging from 0.00 MPa to 8.127 MPa, reveals how pressure is distributed at contact points. Contact forces along X, Y, and Z axes help understand interactions between different parts, crucial for spotting stress concentration and potential failure points. This detailed numerical data gives a clear picture of how the chair reacts to external forces. The safety factor ensures a safety margin, stress distribution identifies critical points, displacement reveals deformations, reaction forces show load-bearing capacity, strain assesses material deformation, and contact pressure and force unveil crucial contact points. These results empower designers to enhance the chair's design for better structural performance and reliability.

Deformation

Analysing deformation in chair design resulted in five images: "Actual," in Fig 13(a) "Adjusted 0.5x,'' in Fig 13 (b) "Adjusted," in Fig 13 (c) "Adjusted 2x," in Fig 13 (d) and "Adjusted 5x." in Fig 13 (e). These visuals illustrate how the chair responds to different levels of deformation. In Figures 13(b), (d), and (e), as deformation increases from 0.5X to 5X, noticeable changes emerge. The chair exhibits a downward bend, indicating its response to different deformation levels. These visualizations offer a crucial understanding of how the chair structure behaves under varying conditions, informing design decisions, and highlighting potential areas for reinforcement or optimization to ensure stability and structural integrity.[7]

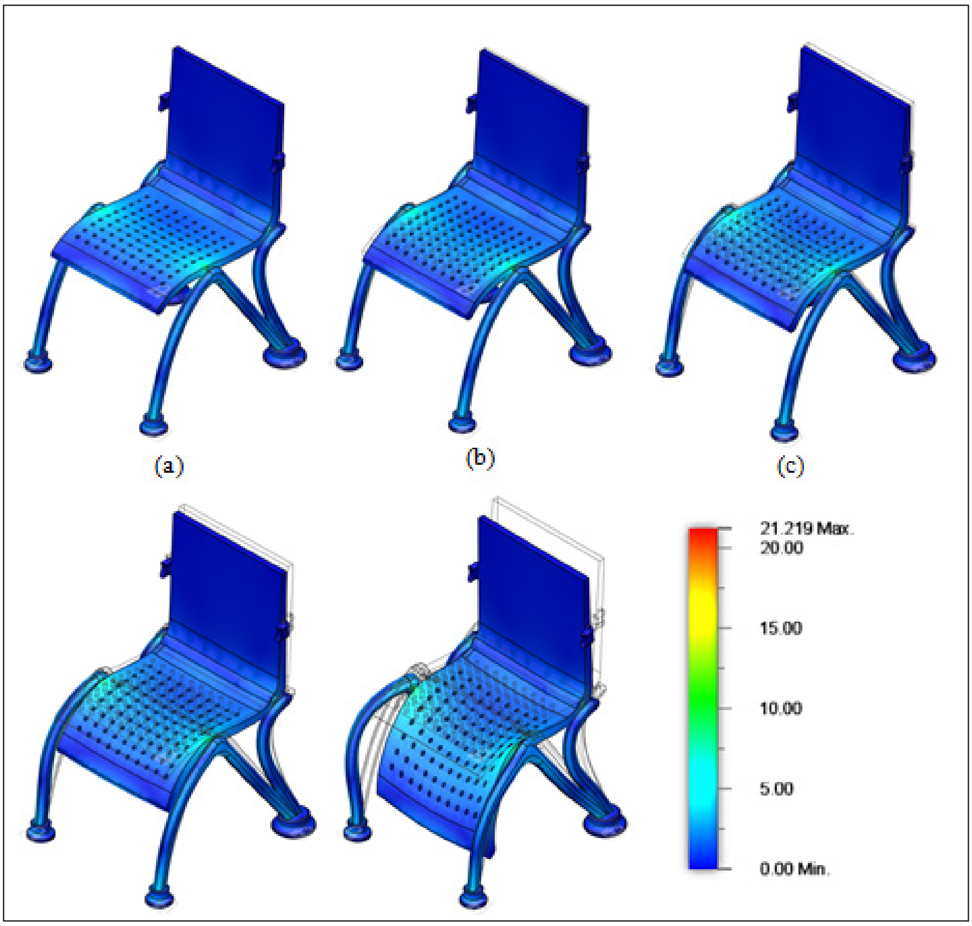

Table 7.Brief description of result outcome

Conclusion

- Successfully applied Finite Element Analysis (FEA) in Autodesk Fusion 360 to optimize the structural performance of a steel chair.
- Achieved safety factors ranging from 9.756 to 15.00, affirming the structural robustness and providing a considerable margin of safety.
- Von Mises stress ranged from 4.426E-05 MPa to 21.219 MPa, unveiling forces within the structure and aiding in pinpointing areas of stress concentration.
- Documented reaction forces from 0.00 N to 7.569 N and strains ranging from 3.270E-10 to 1.611E-04, providing a comprehensive understanding of the chair's response to external loads.
- Established Load Case1 with fixed constraints (Fixed1) and applied Gravity and Force1 loads to simulate real-world scenarios and assess the chair's response.
- Configured meshing parameters, including average element size, parabolic elements, and curved mesh elements, ensuring precise stress distribution analysis.
- The research analysed the chair changes when pushed or pulled. As the pushing or pulling increases from 0.5x to 5x, the chair visibly bends downward. These observations help to make important decisions about how to improve the design, focusing on areas that might need strengthening.
- The parameters of the selected material, steel, play a critical role in understanding the chair's structural behaviour under various conditions. By accurately simulating these factors in FEA, it makes smart design choices to make the chair strong and reliable.
- FEA analysis in Fusion 360 yielded quantitative insights into safety factors, stress distribution, displacement, and component interactions, informing an iterative design process.
- The numerical outcomes serve as a foundation for precise design improvements, contributing to the creation of structurally sound and adaptable furniture for modern living spaces.

Acknowledgements

Funding: The present work has received no funds in any manner from any organization.

Conflicts of interest: The authors declare that they have no involvement in any organization or entity with any financial interest.

References

1. Çolakoğlu, M. H., & Apay, A. C. (2012). Finite element analysis of wooden chair strength in free drop.
2. Himarosa, R. A. (2020, February). Design, Frame Analysis and Manufacture of Handcycle Prototype. In Journal of Physics: Conference Series (Vol. 1471, No. 1, p. 012058). IOP Publishing.
3. Wu, D., Terpenny, J., & Schaefer, D. Additional services for Artificial Intelligence for Engineering Design, Analysis and Manufacturing.
4. Mahto, H. K., Pandey, N., Mahalik, V., Shrivastava, A., Pandey, A., Tewari, P., ... & Gupta, P. DESIGN AND ANALYSIS OF THE COMPONENT OF GLASS CLEANING ROBOT USING GENERATIVE DESIGNING MODULE.
5. Yılmaz, T., & Güntekin, E. (2012). Finite element modeling of chair frames. Turkish Journal of Forestry, 13(2), 134-139.
6. Ceylan, E., Güray, E., & Kasal, A. (2021). 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.
7. Koua, H., Zhangb, X., Chenc, X., Liud, J., & Diaoe, D. (2022). The Mechanical Analysis of University Chairs based on Body Pressure Distribution. International Core Journal of Engineering, 8(11), 84-100.
8. Oladapo, B. I., Vincent, B. A., Oke, A. O., & Agbor, E. A. (2015). Design and finite element analysis on car seat height screw adjuster using autodesk inventor. Int. J. Sci. Res. Eng. Stud.(IJSRES), 2(8).
9. Bijalwan, A., & Misra, A. (2016). Design and structural analysis of flexible wearable chair using finite element method. Open Journal of Applied Sciences, 6(7), 465-477.
10. Wilis, P. K., & Hyunsoo, L. (2018). Simulation Testing On Improved Design Of College Chair Using Finite Element Method (Fem). eProceedings of Engineering, 5(1).
11. Kasal, A. (2006). Determination of the strength of various sofa frames with finite element analysis. Gazi University Journal of Science, 19(4), 191-203.
12. Li, S., & Hu, W. (2023). Study on Mechanical Strength of Cantilever Handrail Joints for Chair. Bioresources, 18(1).
13. Bhad, P. U., & Buktar, R. B. (2022). Multiple novel generative design solutions for various mechanical engineering related products using Autodesk Fusion 360 software. International Journal of Design Engineering, 11(1), 1-26.