

Topology Optimization of an 3d Printed Prosthetic Limb

**Pankaj Gaigole¹, Zulfikar Jasdanwala², Krishna Vyas³, Chetan Thakre⁴,
Aarya Valenkar⁵, Shreya Navle⁶**

^{1,2,3,4,5}Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, India

⁶Department of Mechanical Engineering, Humanities, Vishwakarma Institute of Technology, Pune, India

ABSTRACT

This Project Presents The Design And Optimization Of A Prosthetic Limb Using Topology Optimization Techniques In Ansys, With The Primary Objective Of Minimizing Mass While Maintaining Structural Integrity. The Initial Design Underwent A Detailed Finite Element Analysis (Fea), Followed By Topology Optimization To Retain Only 30% Of The Original Mass. The Optimization Process Strategically Removed Non-Load-Bearing Material, Resulting In A Lightweight Yet Structurally Efficient Geometry. The Optimized Design Was Then Fabricated Using Fused Deposition Modeling (Fdm) With Acrylonitrile Butadiene Styrene (Abs) As The Printing Material. Post-Printing, The Physical Prototype Was Evaluated To Validate Its Structural Viability. The Outcome Demonstrates The Potential Of Topology Optimization In Improving The Performance And Manufacturability Of Prosthetic Components, Paving The Way For More Sustainable And Cost-Effective Medical Solutions.

INTRODUCTION:

Prosthetic limbs play a crucial role in enhancing the quality of life for individuals with limb loss by restoring functionality and enabling greater independence. However, the design of prosthetic components presents unique challenges, particularly in achieving an optimal balance between strength, weight, and user comfort. Excessive weight in prosthetic limbs can lead to discomfort, fatigue, and even long-term musculoskeletal issues. Therefore, reducing the mass of prosthetic components without compromising their structural integrity is a key objective in the field of biomedical engineering and prosthetics design. In recent years, advancements in simulation-based engineering, material science, and additive manufacturing have paved the way for innovative design methodologies such as topology optimization. Topology optimization is a mathematical approach that determines the most efficient material distribution within a given design space, subject to defined loading and boundary conditions. By eliminating unnecessary material in regions with low stress contribution, this technique facilitates the creation of lightweight, structurally efficient components that would be difficult or impossible to design using traditional methods.

This project explores the application of topology optimization in the design of a prosthetic limb using ANSYS software. The primary goal was to reduce the mass of the limb while preserving its mechanical performance, with the optimization target set to retain only 30% of the original material volume. The process began with the creation of a baseline model, followed by structural analysis to understand stress distribution. This data served as the input for the topology optimization module, which generated an opti-

mized geometry reflecting the most efficient material layout.

To validate the practical applicability of the optimized design, the final model was fabricated using 3D printing technology. Acrylonitrile Butadiene Styrene (ABS), a commonly used thermoplastic polymer in additive manufacturing, was selected due to its good balance of strength, toughness, and printability. The use of ABS also ensured that the prototype was cost-effective and lightweight, making it a suitable candidate for real-world prosthetic applications.

This report details each stage of the project—from modeling and simulation to optimization and fabrication—providing insights into the benefits and challenges associated with implementing topology optimization in prosthetic limb design. It also demonstrates the powerful synergy between computer-aided engineering (CAE) tools and modern manufacturing techniques, highlighting their potential to revolutionize the development of customized, performance-driven biomedical devices.

METHODOLOGY

Methodology

The methodology followed for this project consisted of four main stages: geometry creation, finite element analysis (FEA), topology optimization, and 3D printing. The primary focus was on achieving significant weight reduction while ensuring structural performance under anticipated loading conditions. The entire simulation and optimization workflow was carried out using **ANSYS Workbench**, leveraging its built-in **Topology Optimization module** within the **Structural Analysis environment**.

1. Geometry Creation and Preprocessing

The initial geometry of the prosthetic limb was designed to resemble a typical lower-limb component, incorporating features necessary for load-bearing and user fitment. This geometry was imported into ANSYS SpaceClaim for simplification and defeaturing, ensuring compatibility with meshing and analysis.

Material Assignment:

The material chosen for both simulation and final fabrication was **ABS (Acrylonitrile Butadiene Styrene)**.

The mechanical properties of ABS used in the simulation were:

- Young's Modulus: ~2 GPa
- Density: ~1040 kg/m³
- Yield Strength: ~40 MPa



Fig1. CAD Model

Material Assignment:

The material chosen for both simulation and final fabrication was **ABS (Acrylonitrile Butadiene Styrene)**.

The mechanical properties of ABS used in the simulation were:

- Young’s Modulus: ~2 GPa o Poisson’s Ratio: ~0.35
- Density: ~1040 kg/m³
- Yield Strength: ~40 MPa

2. Finite Element Analysis (FEA)

Before optimization, a static structural analysis was conducted to evaluate the stress distribution and deformation behavior of the unoptimized geometry.

Steps:

The model was discretized using tetrahedral elements of element size 7mm, ensuring sufficient mesh refinement in stress-concentrated regions. Boundary conditions were applied to simulate realistic loading: 1) One end of the limb (upper face) was fixed to represent attachment to the socket or upper assembly. A distributed force was applied on the foot-end to mimic body weight and walking loads, the image of how force applied is shown below.^[1]

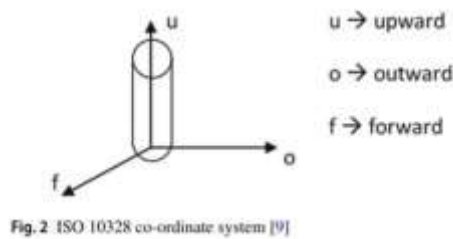


Fig.2 ISO 10328 co-ordinate system [9]

Table 2 Force components as per ISO-10328

Force Components	Heel-Strike (N)	Toe-Off (N)
x-component (outward)—o	-607.58	-112.42
y-component (upward)—u	3240.44	2997.99
z-component (forward)—f	648.09	337.27

Fig2. Force application

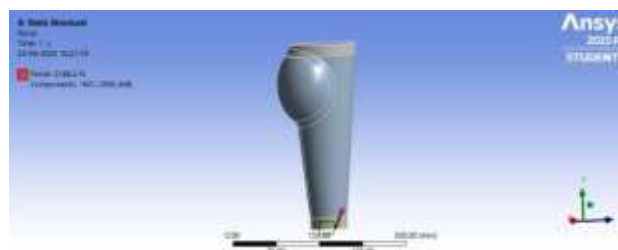


Fig 3. Boundary conditions

Solver settings were adjusted to obtain accurate results in terms of stress, displacement, and strain.

The output of the static structural is shown below, it was observed that a maximum of 3.007MPa of von mises stresses were induced in the limb.

Optimization Setup:

Design Region: The limb was divided into three regions as seen in the CAD model of the limb, the upper and lower regions are preserved and the body in between is given as the body on which the optimization process will take place.

Objective Function: The goal was to minimize compliance (i.e., maximize stiffness), a typical criterion for achieving structurally efficient designs.

Constraint: The only major input required was setting the **mass retention** constraint to 30%, meaning the solver was instructed to retain only 30% of the original material volume.

Manufacturing Constraints: A symmetry constraint was enabled to maintain structural and a minimum member size was specified to ensure the design would be suitable for 3D printing. The optimization process automatically removed material from regions with low stress while preserving critical load paths, resulting in an organic, structurally optimized geometry. Below shows the result of the optimization process, the result converged after 24 iteration done by the algorithm.

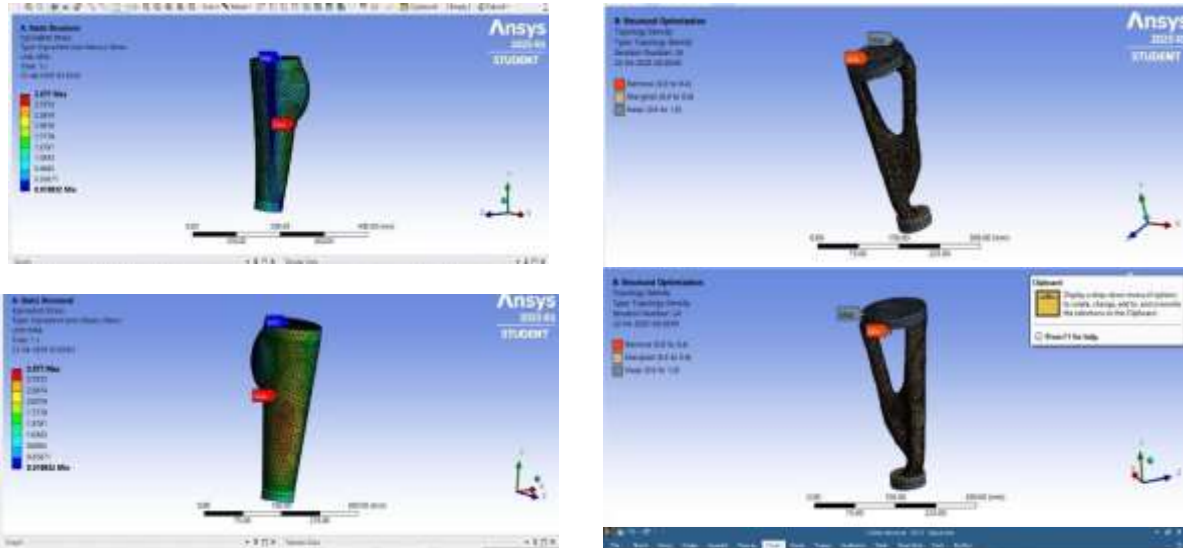


Fig4. StaticStructuralAnalysisof Limb

3. Topology Optimization in ANSYS

In ANSYS, the *Structural Optimization* module was directly integrated with the existing *Static Structural Analysis* setup. By linking the two, most of the boundary conditions, load and analysis settings were automatically carried over, eliminating the need to redefine them.

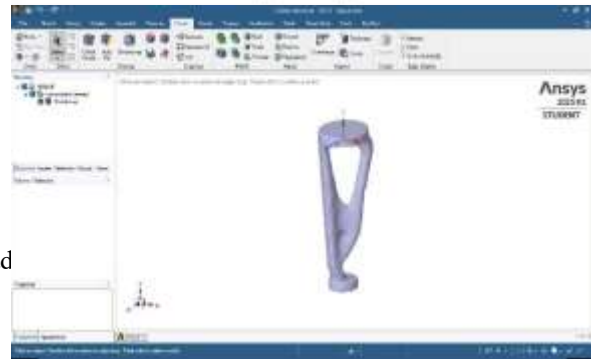


Fig 5. Topolgy Optimized Geometry

4. Additive Manufacturing

The final, optimized CAD model was exported in **STL format** and prepared for 3D printing using Orca slicing software. ABS filament was used in an **FDM (Fused Deposition Modeling)** 3D printer.

Print settings:

- Layer height: 0.2 mm
- Infill density: As per structural needs, 100% for load-bearing areas
- Supports: Enabled, especially for overhangs
- Nozzle temperature: ~230°C
- Bed temperature: ~100°C

The printed component was examined for dimensional accuracy and structural integrity

RESULT

Results and Discussion

The objective of reducing the mass of the prosthetic limb to 30% of its original volume was successfully

achieved through the topology optimization process in ANSYS. The optimized geometry displayed a structurally efficient, organic layout that strategically retained material along critical load paths while eliminating mass from low-stress regions. This resulted in a lightweight, manufacturable design with minimal compromise to its mechanical strength. **1. Topology Optimization Outcome**

The final topology-optimized model retained **30% of the original material volume**, as per the design constraint. The structure exhibited typical characteristics of a stress-adaptive form—such as lattice-like features, contoured surfaces, and hollowed-out regions—particularly in non-load-bearing zones. A comparison of the **baseline model** and the **optimized model** showed:

Mass Reduction: From 100% to ~30%

Maximum Stress: Slightly increased in the optimized design, but remained well within the yield strength of ABS.

Parameter	Baseline Design	Optimized Design
Mass (kg)	2.31	0.634
Max Stress (MPa)	3.007	16
Safety Factor	12	2.5

2. FEA Validation of Optimized Design

A post-optimization FEA simulation was conducted to validate the load-bearing capacity and deformation response of the new design. The stress contours confirmed that the material was retained in areas of high stress concentration. No critical regions exceeded the ABS yield strength, ensuring mechanical reliability under the applied loading conditions.

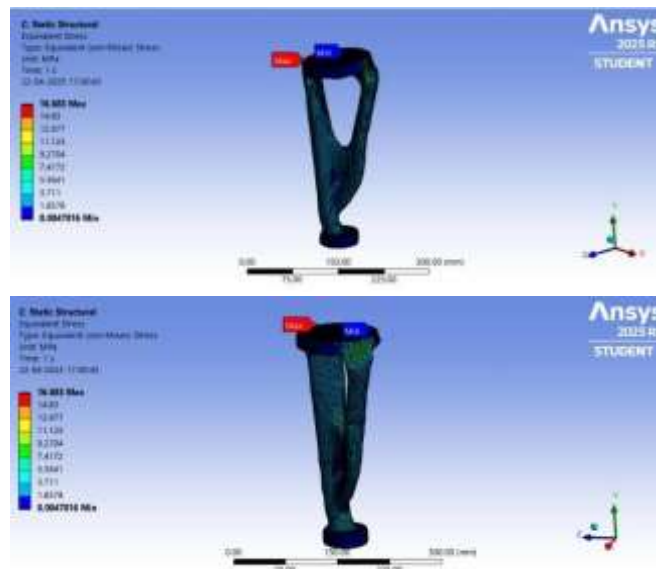


Fig 5. Static Structural Analysis Topolgy Optimized Geometry

3. Fabrication and Physical Evaluation

The optimized geometry was exported as an STL file and successfully printed using an FDM 3D printer with **ABS filament**. The printed part exhibited good dimensional accuracy, minimal warping, and an aesthetically futuristic appearance due to the optimization’s organic structure.

The final printed prototype was inspected for:

- **Dimensional conformity**
- **Surface finish**

Structural stability under manual loading

While no destructive testing was performed, the prototype handled moderate mechanical loads without visible deformation or failure, suggesting strong correlation with simulation results.

4. Images of Printed Product



Fig 6. Image of Printed Geometry

This section effectively bridges the gap between simulation and real-world feasibility, demonstrating that the topology optimized limb not only meets theoretical design goals but is also practically manufacturable and functional using modern additive manufacturing techniques.

Conclusion

This project successfully demonstrated the application of topology optimization in the design of a lightweight prosthetic limb using ANSYS. By retaining only 30% of the original material volume, the optimized design achieved significant weight reduction while maintaining structural integrity. The use of

ABS material and FDM 3D printing validated the manufacturability of the final design, confirming its practical applicability. This approach highlights the potential of simulation-driven design and additive manufacturing to develop efficient, cost-effective, and customizable biomedical solutions.

ACKNOWLEDGEMENT

We would like to express our heartfelt gratitude to our project advisor and mentors for their invaluable guidance, insightful feedback, and continuous support throughout the design and development of the prosthetic leg using topology optimization and 3D printing. Their technical expertise and encouragement played a pivotal role in helping us navigate complex design decisions and refine our methodology to meet both performance and manufacturability goals.

We are also deeply thankful to our academic institution for providing access to essential resources such as ANSYS simulation software, 3D printing facilities, and laboratory space, all of which were instrumental in validating our optimized design and translating it into a physical prototype.

Our sincere appreciation goes to our peers and team members for their dedication, collaboration, and shared commitment to the success of this project. The collective effort and team spirit were fundamental in overcoming challenges and ensuring timely completion of each development phase.

Finally, we would like to acknowledge the foundational contributions of researchers and authors in the fields of prosthetic design, additive manufacturing, and structural optimization. Their published work served as both inspiration and a critical reference, significantly enriching our understanding and approach to the problem.

Thank you to everyone who contributed to this endeavour, directly or indirectly.

REFERENCES

1. Jansari, Tapan & Deiab, Ibrahim. (2019). Comparative study of a topologically optimized lower limb prosthesis. *International Journal on Interactive Design and Manufacturing (IJIDeM)*. 13. 10.1007/s12008-019-00540-3.
2. Price, M. A., Beckerle, P., & Sup, F. C. (2019). Design optimization in lower limb prostheses: A review. *IEEE Transactions on Neural Systems and Rehabilitation Engineering*, 27(8), 1574–1588. <https://doi.org/10.1109/TNSRE.2019.2927094>
3. Ceddia M, Trentadue B, De Giosa G, Solarino G. Topology Optimization of a Femoral Stem in Titanium and Carbon to Reduce Stress Shielding with the FEM Method.
4. *Journal of Composites Science*. 2023; 7(7):298. <https://doi.org/10.3390/jcs7070298>
5. Faustini, M.C., et al., An experimental and theoretical framework for manufacturing prosthetic sockets for transtibial amputees. *IEEE Trans Neural Syst Rehabil Eng*, 2006. 14(3): p. 304-10.
6. Rajput, S., Burde, H., Singh, U. S., Kajaria, H., & Bhagchandani, R. K. (2021). Optimization of prosthetic leg using generative design and compliant mechanism. *Materials Today: Proceedings*. <https://doi.org/10.1016/j.matpr.2021.04.026>
7. Singh, M., Saif, A., Reddy, C. S. S., Nath, A., Akhil, M., Raj, R., & Ganesan, D. P. (2024). Design optimization of prosthetic leg with consideration of different materials for various gait patterns. *AIP Conference Proceedings*, 2986, 030002. <https://doi.org/10.1063/5.0192762>
8. Rajput, S., Burde, H., Singh, U. S., Kajaria, H., & Bhagchandani, R. K. (2021). Optimization of prosthetic leg using generative design and compliant mechanism. *Materials Today: Proceedings*, 46(1), 8708–8715. <https://doi.org/10.1016/j.matpr.2021.04.026>