Journal Computational Simulation 9[20]1-10: e40920110
Article

Nodal analysis of transtibial prostheses using the finite element method with Matlab

Análisis nodal de prótesis transtibial por el Método de Elementos Finitos mediante el uso de Matlab

Cortez-Solis, Reynaldo*[a], Fuentes-Castañeda, Pilar[b], Betanzos-Castillo, Francisco[c], Jaramillo-Rodriguez, Eduardo[d]

[a] Tecnológico Nacional de México – TES Valle de Bravo • KUD-2900-2024 • 0000-0001-7519-1815 • 1113392
[b] Tecnológico Nacional de México – TES Valle de Bravo • KUD-2889-2024 • 0000-0001-6567-9614 • 428699
[c] Tecnológico Nacional de México – TES Valle de Bravo • AIE-1532-2022 • 0000-0002-7245-703X • 206209
[d] Tecnológico Nacional de México – TES Valle de Bravo • OIU-7288-2025 • 0009-0002-6448-7383 • 2139852

SECIHTI classification:

Area: Engineering.
Field: Engineering.
Discipline: Mechanical Engineering
Subdiscipline: Mechanical design

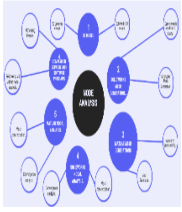
https://doi.org/10.35429/JCS.2025.9.20.4.1.10
Article History:
Received: June 30, 2025
Accepted: December 15, 2025

* ✉ [reynaldo.cs@vbravo.tecnm.mx]

Abstract

CAE analysis involves different solutions and methods for all 2D and 3D CAD models. Different methods are used to perform this type of analysis. One of the main methods used in this project was finite element analysis using software. The main focus was to analyze the meshing of a socket for a transtibial prosthesis, verifying the connection and discretization of the mesh throughout the model. We began with 3D modeling in Solidworks mechanical design software and mesh analysis in Matlab computational numerical analysis software. Different configurations and parameters were evaluated to discretize and verify the convergence of the mesh throughout the three-dimensional or wireframe model of the part.

Nodal analysis of transtibial prostheses using the finite element method with Matlab

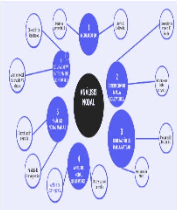
Objetives	Methodology	Contribution
-3D modeling. -Mesh generation in Solidworks software. -Mesh generation in Matlab computational numerical software. -Nodal analysis.		Mesh analysis and discretization using the finite element method with software, verifying convergence between nodal joints.

Mesh, nodal analysis, and 3D modeling

Resumen

El análisis CAE implica diferentes soluciones y métodos para todos los modelos CAD en 2D y 3D. Se utilizan diferentes métodos para realizar este tipo de análisis. Uno de los principales métodos utilizados en este proyecto fue el análisis de elementos finitos mediante software. El objetivo principal era analizar el mallado de un encaje para una prótesis transtibial, verificando la conexión y la discretización de la malla en todo el modelo. Se comenzó con el modelado 3D en el software de diseño mecánico Solidworks y el análisis de mallas en el software de análisis numérico computacional Matlab. Se evaluaron diferentes configuraciones y parámetros para discretizar y verificar la convergencia de la malla en todo el modelo tridimensional o de alambre de la pieza.

Análisis nodal de prótesis transtibial por el Método de Elementos Finitos mediante el uso de Matlab

Objetivos	Metodología	Contribución
-Modelado 3D. -Mallado en Software Solidworks. -Mallado en Software numérico computacional Matlab. -Análisis Nodal		Análisis de malla y discretización de esta por el método de elementos finitos mediante software, verificar convergencia entre la unión nodal.

Malla, análisis nodal y Modelado 3D

Area: Promotion of frontier research and basic science in all fields of knowledge

Citation: Cortez-Solis, Reynaldo, Fuentes-Castañeda, Pilar, Betanzos-Castillo, Francisco, Jaramillo-Rodriguez, Eduardo. [2025]. Nodal analysis of transtibial prostheses using the finite element method with Matlab. Journal Computational Simulation. 9[20] 1-10: e40920110.

RENIECYT
Registro Nacional de Instituciones y Empresas Científicas y Tecnológicas
1702902 SECIHTI

Introduction

Structural analysis of mechanical components is fundamental in engineering design and validation, where obtaining accurate results based on load behavior is critical to ensuring safety, functionality, and durability. Traditional analytical methods are insufficient for complex geometries or boundary and contour conditions. In addition to this, the Finite Element Method (FEM) has been chosen as the predominant numerical technique for these purposes. This type of FEM method focuses on the discretization of the continuous domain of a part into a mesh of simpler subdomains (finite elements), interconnected at discrete points known as nodes. It is in this context that nodal analysis emerges as the fundamental stage, since the overall behavior of the structure is determined from the calculation of the displacements, stresses, and deformations at each of these nodes.

Mesh generation and analysis are therefore fundamental elements that directly define the accuracy and efficiency of the simulation. For all these analyses, there is powerful software available for finite element analysis (CAE). Therefore, for this article, numerical algorithms were implemented in environments such as MATLAB, as it offers flexibility and depth of understanding compared to other software. MATLAB, as computational domain software with all its specialized toolboxes and its capacity for intensive matrix handling, is ideal for developing and validating nodal analysis methodologies, allowing control over each phase of the process: from mesh generation and the application of boundary conditions to the assembly of the global stiffness matrix and the resolution of the system of equations.

Likewise, a methodology is presented for the nodal analysis of mechanical part meshing using MATLAB computational software and 3D modeling in Solidworks CAD software, comparing the meshes in both software programs and thus verifying the discretization and nodal union in each socket element. The main objective is to develop an analysis that allows the systematic evaluation of the influence of meshing parameters (such as element type, density, and regularity) on the results of the nodal analysis.

It also aims to highlight MATLAB's ability to perform sensitivity analysis and mesh optimization, key aspects for reducing discretization error without incurring prohibitive computational costs.

Antecedentes

In 3D modeling, according to (Bermejo, 2020), all the elements that make up a 3D scene are formed by polygons. These polygons have to be processed by the hardware in order to be displayed as we want them to be. The greater the number of polygons, the higher the resolution and the greater the calculation process. This can cause slowdowns in the display of views and even program crashes. In rendering, it is more of the same, including the textures and lighting created in the scene.

The analysis of mechanical parts using computer control software and the transfer of CAD models in SolidWorks in the CAE (Computer-Aided Engineering) section allow us to identify the development that technology has undergone. Therefore, a CAD model is taken as a reference in order to create an STL file. These three-dimensional files are first designed by a geometric modeler, which follows a construction algorithm to create a representation of simple boundaries that cover the surface of the solid using triangles. This triangle mesh is stored in an STL format, which is used to define the actual geometry of the solid with a wide variety of industrial applications, as well as rapid prototyping and manufacturing. (Gutiérrez Madrigal, 2018).

Finite element analysis (FEA), also known as the finite element method, is a numerical technique for solving field problems described by a set of partial differential equations. The finite element method is commonly used in many engineering disciplines, such as machine design, acoustics, electromagnetism, soil mechanics, fluid dynamics, among others. In mechanical engineering, FEA is widely used for structural, vibration, and thermal problems. However, it is not the only tool available for numerical analysis. Other numerical methods used in engineering include: the finite difference method, the limit state method, and the finite volume method.

Nevertheless, due to its versatility and high numerical efficiency, FEA has come to dominate the market for engineering analysis software, while other methods have been relegated to niche applications. (González Woge, 2020).

Finite element (FE) methods are widely used in biomechanics and bioengineering due to their ability to simulate and analyze the behavior of complex systems under various loading conditions. In the context of biomechanics, FE analysis can provide valuable information about the distribution of stresses and strains within biological structures such as bones, muscles, ligaments, and tendons.

Numerous studies have been conducted in this field. The first model was presented by Zhang, who investigated the effect of friction and slippage at the interface between the stump and the socket. They found that the friction factor has a significant impact on pressures, shear loads, slippage, and bone movement. Jia investigated how inertial loading affected shear stress and interface pressure. L. Zhang used contact analysis to simulate friction conditions between the skin and the socket. Meng used finite element models to examine stress in the stump of transfemoral amputees with compression/release stabilised sockets (CRS). Sofia C. performed a finite element analysis (FEA) to estimate the interaction between the stump and the socket during the gait cycle of fourteen transfemoral amputees. (Messaad, 2025).

Mesh Generation

There are two distinct ways of working with three-dimensional objects in a CAD system: through solid modeling or surface modeling. In the first case, you work with a library of simple solids (sphere, cone, box, cylinder, etc.) and can perform all kinds of Boolean operations with them. Although the vast majority of mechanical components can be constructed in this way, there are cases where this methodology cannot be applied (aerodynamic vehicles, for example). The main advantages of this method are the simplicity with which the geometry is defined and the ease of automating the interface with the mesh generation system.

On the other hand, surface modeling allows for complete generality in the geometry to be constructed, but requires a precise description of each piece (patch) of the surface, which in many cases can be difficult to achieve. On the other hand, surface modeling allows for complete generality in the geometry to be constructed, but requires a precise description of each piece (patch) of the surface, which in many cases can be very cumbersome. (Vénere, 1996).

Methodology

The following shows the development and application of the methodology of this project as a starting point for this research. It should be noted that there is various literature on the web that discusses the generation of meshes and analysis of mechanical parts in SolidWorks and MATLAB, However, in this project presentation, nodal analysis was performed in both software programs, verifying the connection and polygonal behavior of the mesh, since each part of the discretization is essential for observing the structure of the transtibial prosthesis. Previously, CAE analysis of the same prosthesis was performed with two CAD design and modeling software programs. The main point is to verify the convergence of the nodes by transferring a 3D digital model and generating a code that allows the wireframe or mesh model of the prosthesis to be developed. A diagram showing the steps to follow for nodal analysis and obtaining results is shown.

Box 1

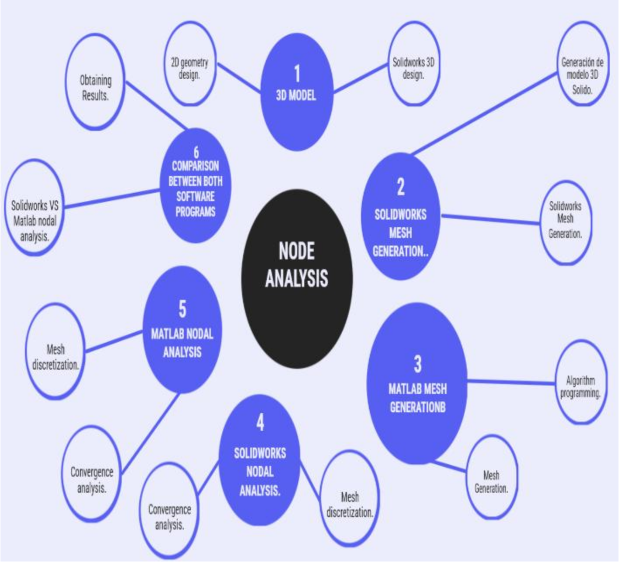


Figure 1
Stages of development and nodal analysis
Source: Own Elaboration

1. 2D design and 3D modeling in software:

At this stage of the design process, a two-dimensional or three-dimensional digital model is used as a reference, depending on the case. This model must contain structured geometry so that it can be analyzed in CAD design software. Once the geometry or 2D model is available, this phase also focuses on the design of the model in two dimensions (2D) or three dimensions (3D) and its subsequent preparation for the generation of a file in STL (Standard Tessellation Language or Stereolithography) format, which is the universal standard for the interpretation of geometries by meshing and analysis software.

2. Solidworks Mesh Generation:

Once the discrete representation of the geometry in STL format has been obtained, the next critical step in the analysis is the generation of the finite element mesh. This step consists of discretizing the 2D and 3D geometric domain into a set of simpler and more regular subdomains called interconnected elements at discrete points or nodes. For this study, a fine mesh has been generated, characterized by a high density of elements, with the main objective of minimizing discretization error and capturing stress and strain gradients more accurately, especially in areas of stress concentration.

3. Matlab Mesh Generation:

Once the STL file has been generated, it is imported into the MATLAB environment for processing, as MATLAB has specific functions (such as stlread or importGeometry from the PDE Toolbox) for reading these files and storing the information on vertices and triangular faces in matrix arrays. This numerical representation of the geometry, typically using two matrices: Vertices (Nx3) and Faces (Mx3), is the basis on which the nodal analysis and finite element mesh generation algorithms will be implemented in the subsequent stages of this research.

It should be noted that the transition from an initial mesh to a fine mesh is controlled by defining specific refinement criteria. These criteria, implemented through functions and parameters in MATLAB, include:

Maximum Element Size (Hmax): Global parameter that defines the largest edge length allowed for any element in the mesh. For a fine mesh, this value is set significantly lower than for a coarse mesh.

Minimum Element Size (Hmin): Limits excessive refinement in regions where it is not necessary, controlling computational cost.

Geometry Gradient Refinement: The algorithm can automatically refine regions with high curvature (defined in the original STL) to better approximate the geometry.

Local Refinement Using Density Functions: Functions can be defined that specify a desired element size at specific spatial coordinates, allowing greater control to refine areas of particular interest, such as holes, corners, or areas where loads are applied.

4. Solidworks nodal analysis:

In order to validate the methodology implemented in MATLAB and the accuracy of the results obtained from the nodal analysis of the generated fine mesh, a comparative analysis was carried out using the SolidWorks Simulation module. This Finite Element Analysis (FEA) application software demonstrates the verification process as a fundamental part of ensuring the reliability of the code implemented in MATLAB and enabling a comparison of mesh generation to be made, taking as a reference: an identical and perfectly defined geometric representation, material properties (isotropic elastic linear model), including the Modulus of Elasticity (Young's Modulus), Poisson's ratio, and density. Identical constraints were also applied as boundary conditions (fixations, supports, inputs, outputs, forces) in the same geometric locations. All these configurations were designed to generate a finite element mesh of the same type as MATLAB, i.e., with a global element size comparable to that of the fine mesh used for the MATLAB analysis.

5. Matlab Nodal Analysis:

The implementation of nodal analysis in MATLAB was carried out explicitly considering the critical influence of discretization, complemented by a mesh convergence study to ensure the robustness and accuracy of the results.

Cortez-Solis, Reynaldo, Fuentes-Castañeda, Pilar, Betanzos-Castillo, Francisco, Jaramillo-Rodriguez, Eduardo. [2025]. Nodal analysis of transtibial prostheses using the finite element method with Matlab. Journal Computational Simulation. 9[20] 1-10: e40920110.
https://doi.org/10.35429/JCS.2025.9.20.4.1.10

This systematic approach allowed us to determine an optimal mesh density that balances numerical accuracy with computational efficiency.

For the section on mesh convergence analysis, reference was made to to (Autodesk, 2023) states that this is an essential methodology for quantifying the influence of discretization and establishing objective criteria for selecting a sufficiently fine mesh. In this study, a Key Interest Variable was selected, such as maximum displacement in a critical direction or maximum von Mises stress.

6. Comparison between both software programs:

In this section, the main objective is to conduct a comprehensive comparison between the MATLAB and SolidWorks Simulation platforms, evaluating their distinctive features in the context of nodal analysis of mechanical parts. This comparison focuses not only on numerical accuracy, but also on operational aspects, control, flexibility, and applicability, which are crucial for selecting the appropriate tool depending on the application and method used.

Mathematical Modeling FEM

In order to perform the finite element analysis, different systems of linear equations were used as a reference, as well as differential equations that allowed the convergence of each of the nodes and the geometries of each 2D polygon to be visualized.

On the other hand, mesh discretization equations were used, as well as the dominant stiffness matrix and the application of boundary conditions.

Nodal matrix

Node Matrix (p or Nodes): A matrix of size n×dn×d, where nn is the total number of nodes and dd is the dimensionality (2 or 3). Each row contains the coordinates (x,y)(x,y) or (x,y,z)(x,y,z) of a node.

$$p=\begin{bmatrix} x1 & y1 & z1 \\ x2 & y2 & z2 \\ \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots \\ xn & yn & zn \end{bmatrix} \qquad [1]$$

Node connectivity matrix.

$$t=\begin{bmatrix} x1,1 & y1,2 & z1,3 \\ x2,1 & y2,2 & z2,3 \\ \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots \\ xm,1 & ym,2 & zm,3 \end{bmatrix} \qquad [2]$$

Where: an $m \times k$ matrix, where m is the number of elements and k is the number of nodes per element. Each row contains the indices of the nodes that make up the element.

Next, the Global Stiffness Matrix (K) assembly is analyzed for each element, and its elemental stiffness matrix [k]e is calculated based on the material properties and element geometry. These elemental matrices are then assembled into the global stiffness matrix K using a direct summation process based on nodal connectivity.

The following steps are also carried out:

Application of Boundary Conditions: The rows and columns of K corresponding to the restricted degrees of freedom (known displacements) are modified, imposing Dirichlet conditions.

- Assembly of the Nodal Force Vector (f): The applied loads are distributed to the corresponding nodes, forming the global force vector f.
- Solution of the System of Equations: The linear system of equations is solved to obtain the vector of unknown nodal displacements (U).

Where: stresses and strains in a structural element can be calculated using the following equation:

$$(K)\{U\} = \{F\} \qquad [3]$$

Likewise, all of the analysis performed is based on the theory of the Finite Element Method (FEM) for linear elasticity. The mathematical fundamentals that govern the mechanical behavior of solids and their numerical discretization are also addressed.

The starting point is the theory of isotropic linear elasticity. For a continuous domain Ω, the boundary value problem is defined by [1, 2]:

Cortez-Solis, Reynaldo, Fuentes-Castañeda, Pilar, Betanzos-Castillo, Francisco, Jaramillo-Rodriguez, Eduardo. [2025]. Nodal analysis of transtibial prostheses using the finite element method with Matlab. Journal Computational Simulation. 9[20] 1-10: e40920110.
https://doi.org/10.35429/JCS.2025.9.20.4.1.10

$$\nabla \cdot \sigma + f = 0 \; in \; \Omega \qquad [4]$$

where σ is the Cauchy stress tensor and f is the body force vector.

Kinematic relationships between displacement and deformation.

Where:
ε is the tensor of infinitesimal deformations and
$u = [\, u \,,\, v \,,\, w \,]^{T}$ is the displacement vector.

Normal strains: Each normal component (εxx, εyy, εzz) corresponds to the derivative of the displacement in its own direction:

$$\mathcal{E}_{xx} = \frac{\partial u}{\partial x} \qquad [5]$$

$$\mathcal{E}_{yy} = \frac{\partial v}{\partial y} \qquad [6]$$

$$\mathcal{E}_{zz} = \frac{\partial w}{\partial z} \qquad [7]$$

A constitutive relationship can also be defined Constitutive Relationship (Generalized Hooke's Law):

$$\sigma = D\varepsilon \qquad [8]$$

where D is the elastic constitutive matrix. For an isotropic material, it depends on the modulus of elasticity E and Poisson's ratio . For a plane stress state:

$$D = \frac{E}{1-v^2}\begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{E}{1-v^2} \end{bmatrix} \qquad [9]$$

Boundary Conditions:
Essential (Dirichlet):
$u = \bar{u}$ en Γu [10]

Natural (Neumann):
$\sigma \cdot n = \bar{t}$ en Γt [11]

Finally, a nodal analysis can be performed using the finite element discretization method for each of the elements and nodes:

$$u^{(e)}(x,y,z) \approx \sum_{i=1}^{nn} Ni^{(e)}(x,y,z)u_i = N^{(e)}u^{(e)} \qquad [12]$$

where:
n_n is the number of nodes per element.
$Ni^{(e)}$ are the shape functions of the element.
u_i are the nodal displacements (unknowns).
$N^{(e)}$ is the shape function matrix.
$u^{(e)}$ is the vector of nodal displacements of the element. (Zienkiewicz, 2013)

Results

The model was designed in SolidWorks software to perform computational domain analysis, then the mesh type was created, using a fine mesh as a reference, in order to verify the polygons and the connection between them, as shown in Figure 2.

Box 2

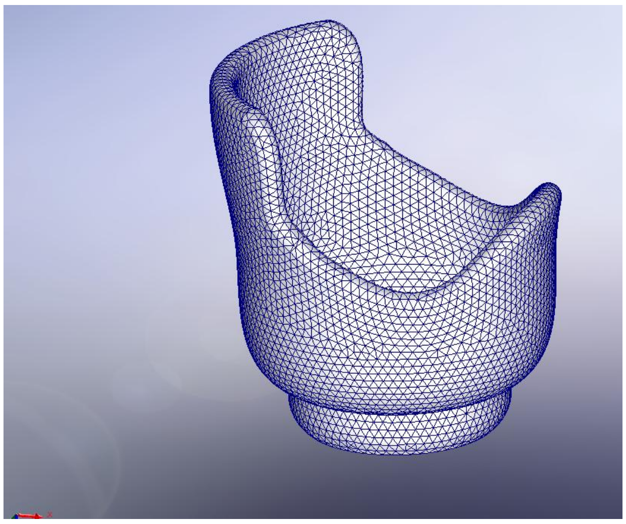


Figure 2
3D model and computer model.
Source: Own Elaboration.

Box 3

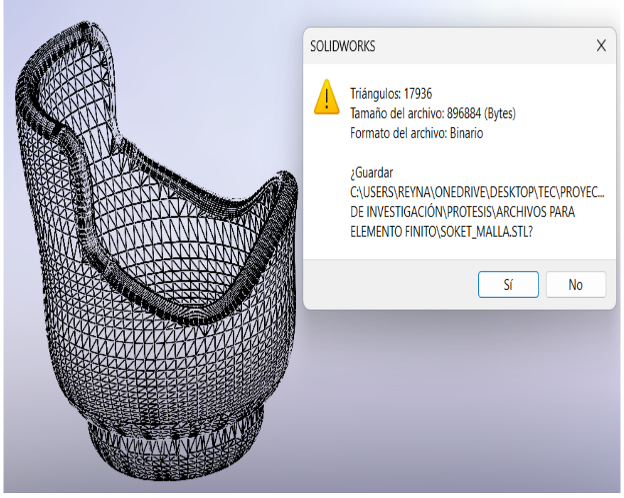
Property	Value	Units
elastic model	230-250	GPA
Poisson's ratio	0.20-0.30	N/D
shear modulus	5.0-6.0	GPA
mass density	1780	Kg/m^3
thermal conductivity	5 - 15	W/m·K

Source: Own Elaboration

Nodal analysis in SolidWorks and Matlab

Once the model was created in SolidWorks, a nodal analysis was performed, using 17,936 nodes as a reference. The file was then converted to an STL (stereolithography) extension, as shown in Figure 3.

Box 3


SOLIDWORKS

Triángulos: 17936
Tamaño del archivo: 896884 (Bytes)
Formato del archivo: Binario

¿Guardar
C:\USERS\REYNA\ONEDRIVE\DESKTOP\TEC\PROYEC...
DE INVESTIGACIÓN\PROTESIS\ARCHIVOS PARA
ELEMENTO FINITO\SOKET_MALLA.STL?

Figure 3
conversion of the STL file..
Source: Own elaboration.

For the mesh configuration, the element size values of 0.7982757cm, a minimum element size of 0.65684442cm, and a polygonal value of 8 were used, as shown in Figure 4.

Box 4

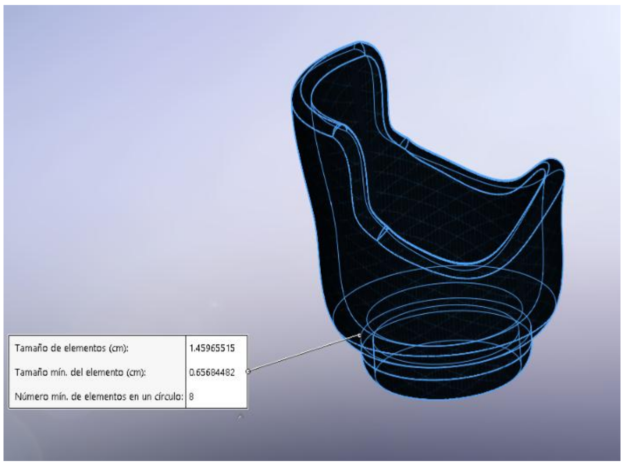

Tamaño de elementos (cm): 1.45965515
Tamaño mín. del elemento (cm): 0.65684482
Número mín. de elementos en un círculo: 8

Figure 3
mesh element configuration
Source: Own Elaboration

Likewise, computational numerical domain was performed in Matalab with the same configurations of the material and the mechanical properties of the material (carbon fiber), where the command with the following nomenclature “stlread” was used (Mathwords, 2020).

This command allowed us to read the STL file generated by SolidWorks and view the mesh model in MATLAB with the same number of nodes, as shown in Figure 4.

Box 5

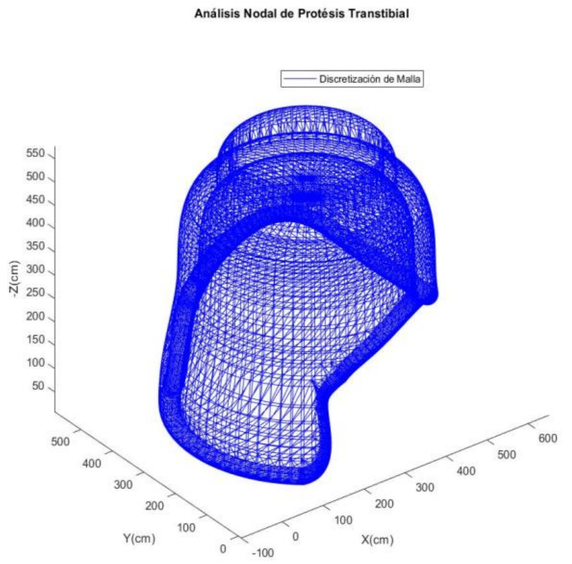


Figure 4
3D model and computer model.
Source: Own elaboration.

The graph was generated using a 3D coordinate system with X, Y, and Z axes. In the center of the graph, you can see the prosthesis model, consisting of a dense network of blue lines representing the elements of the mesh. The model was discretized using a total of 17,936 nodes, allowing for a detailed and accurate representation of the complex geometry of the prosthesis.

The mesh generated is a surface mesh with triangular elements, typical of an STL file, which facilitates its subsequent structural or dynamic analysis in MATLAB.

Therefore, a 2D analysis of the model was performed in MATLAB against SolidWorks, from which it was possible to obtain results showing that the mesh presents stable convergence in both nodes. the first in SolidWorks with that discretization of lines into polygons. Convergence is also shown in MATLAB, taking as a reference the union of each of the nodes and that there is no breakage of the polygons themselves, as shown in Figures 5 and 6.

Cortez-Solis, Reynaldo, Fuentes-Castañeda, Pilar, Betanzos-Castillo, Francisco, Jaramillo-Rodriguez, Eduardo. [2025]. Nodal analysis of transtibial prostheses using the finite element method with Matlab. Journal Computational Simulation. 9[20] 1-10: e40920110.
https://doi.org/10.35429/JCS.2025.9.20.4.1.10

Box 6

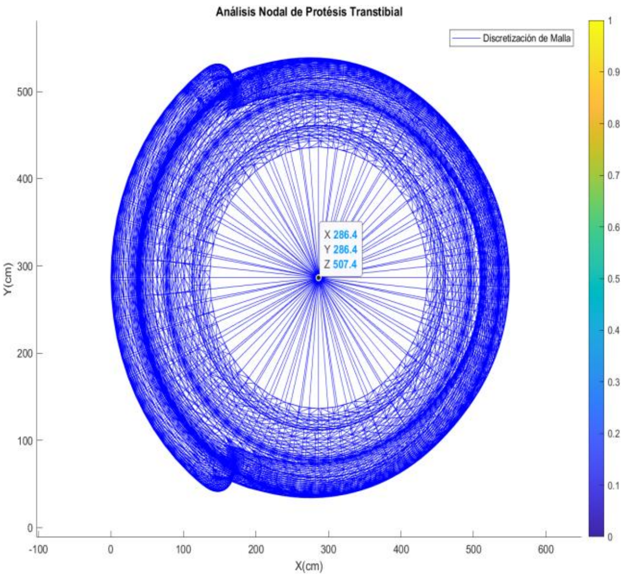


Figure 5

References for joining nodes in MATLAB with x, y, z coordinates.

Source: Own Elaboration

Box 7

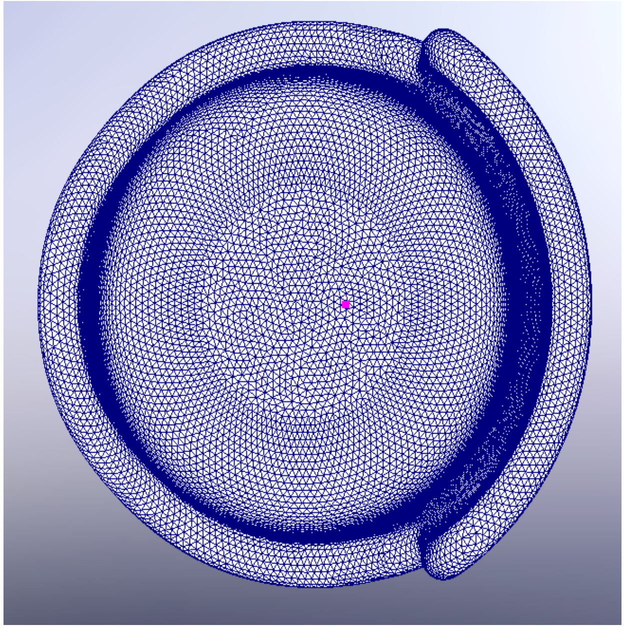


Figure 6

References for joining nodes in SolidWorks with x, y, z coordinates.

Source: Own elaboration.

Based on these results, it can be determined that a nodal analysis of a transtibial prosthesis was performed using two different approaches for the discretization of the fine mesh:

Model 2. Figure 5: Mesh generated in SolidWorks, exported as an STL file, and imported into MATLAB.

Model 1. Figure 6: Mesh generated directly in MATLAB, including a color map to visualize some nodal property (probably deformation, displacement, or distribution of modal analysis values).

Box 8

Table 2

comparison between both software programs

appearance	**model type**	***justification***
Geometric accuracy	Model 1.	By using the original STL faithfully to the CAD design.
Nodal visualization.	Model 2.	Thanks to the color mapping and nodal values displayed.
Ease of analysis.	Model 2.	Already integrated into MATLAB with analysis tools.
Preparation.	Model 1.	Requires more steps (export, import, mesh conversion).
Modal interpretation.	Model 2.	llows you to visually see the most critical node.

Discussion of results

A nodal analysis of the mesh in a mechanical part (socket for transtibial prosthesis) was performed. Based on this, the transtibial prosthesis model was 3D modeled in Solidworks CAD software, taking the mechanical properties of carbon fiber as a reference. The finite element method was also used for this type of analysis, as it is one of the methods that allows the behavior of the nodes to be verified and the mesh to be visualized in wireframe mode, which allows convergence to be verified, and therefore the union of each node.

Two software programs were used as references: the first was Solidworks for 3D modeling and fine mesh generation, and the second was Matlab to transfer the stereolithography computational model to Matlab's stlread.

In order to transfer the model to Matlab, it was necessary to program a pseudocode, which first required reading the file and then generating matrix arrays in the three dimensions x, y, and z.

Finally, the union of each node and the convergence of each line at each of the union points had to be checked, taking three nodes as a reference, in accordance with the triangular polygonal mesh type.

However, this analysis takes as a reference (Sadiq, 2025), where a study was conducted on a socket for transfemoral prostheses, whose materials (Perlon, carbon, glass, and laminating resin) can be used for future analysis in this work, since these authors have demonstrated the manufacture of different physical models, thus obtaining good results in terms of support and deformation both in software and in reality. Therefore, analyzing both results in the type of socket in this work, it can be seen that the use of these materials can be supported when physically manufacturing the proposed socket models.

Conclusions

For detailed structural analysis and geometric fidelity, the model generated from SolidWorks (Model 1) is more convenient, as it ensures that the geometry corresponds exactly to the prosthesis design. For the interpretation of nodal results and preliminary studies, the model generated directly in MATLAB (Model 2) is superior thanks to its integration with advanced visualizations, such as the color map and selection of critical nodes.

Declarations

Conflict of interest

The authors declare no interest conflict. They have no known competing financial interests or personal relationships that could have appeared to influence the article reported in this article.

Author contribution

The contribution of each of the participants in this project was significant and fundamental to the completion of this article. According to the authors, the contributions of each of them are listed below:

Cortez-Solis, Reynaldo: The contribution was the idea generated from the need of a person with limited resources to develop a transtibial prosthesis model, with the aim of creating a model for personal use with less expensive and more durable materials, 3D model design, and computational domain programming in Matlab.

Fuentes-Castañeda, Pilar: Her contribution was the analysis of the mechanical properties of the materials, as well as the mechanical analysis of the prosthesis, evaluation of different materials as a computational model, with the aim of manufacturing the physical prototype in the future and testing it correctly on the person who inspired this project.

Betanzos-Castillo, Francisco: My contribution was in the area of mathematical modeling, as well as the research and analysis of formulas related to the analysis of the mesh and behavior of each node in a matrix format, addressing a 2D computational domain.

Jaramillo-Rodriguez, Eduardo: The colleague modified and verified the 3D model.

Availability of data and materials

No data available.

Funding

The research does not currently receive funding from any external institution, as it has been supported by the National Technological Institute of Mexico, TES-Valle de Bravo, as well as personal acquisitions to carry out the project.

Acknowledgements

Special thanks to the National Technological Institute of Mexico TES-Valle de Bravo for allowing us to develop this project and for giving us access to its facilities when necessary to carry out tests or analyses related to the project.

Abbreviations

CAD	Computer-Aided Design.
CAE	Computer-Aided Engineering.
STL	Stereolithography.
MATLAB	MATLAB: Matrix Laboratory.

References

Basics

Autodesk. (2023, 10 8). Como realizar un Estudio de Convergencia de malla.

Bermejo, P. J. (2020). Docta Ucm.

González Woge, O. G. (2020). Introducción al método del elemento finito: Solidworks y Matlab. Ideas en Ciencias de la Ingeniería, 1(1), 27-47.

Gutiérrez Madrigal, I. &. (2018, 11 26). Jóvenes en la Ciencia.

Supports

Mathwords. (20 de abril de 2020). *stlread returns different numbers of vertices/points*.

Zienkiewicz, O. C. (2013). *The Finite Element Method: Its Basis and Fundamentals*. Butterworth-Heinemann.

Vénere, M. J. (1996). Procedimientos para la generación de mallas tridimensionales de elementos finito. *Revista internacional de Métodos Numéricos para cálculo y diseno en ingenieria, 12*(1), 3-16.

Messaad, T. B. (2025). Biomechanical analysis of prosthetic liners in transfemoral amputees using finite element. *Journal of the Brazilian Society of Mechanical Sciences and Engineering., 47*(2), 59.

Discussions

Sadiq, G. S. (2025). Optimal Mechanical Properties of Composite Material and Pressure Socket Analysis for Through-Knee Amputation. ELSEVIER, 28(107525).