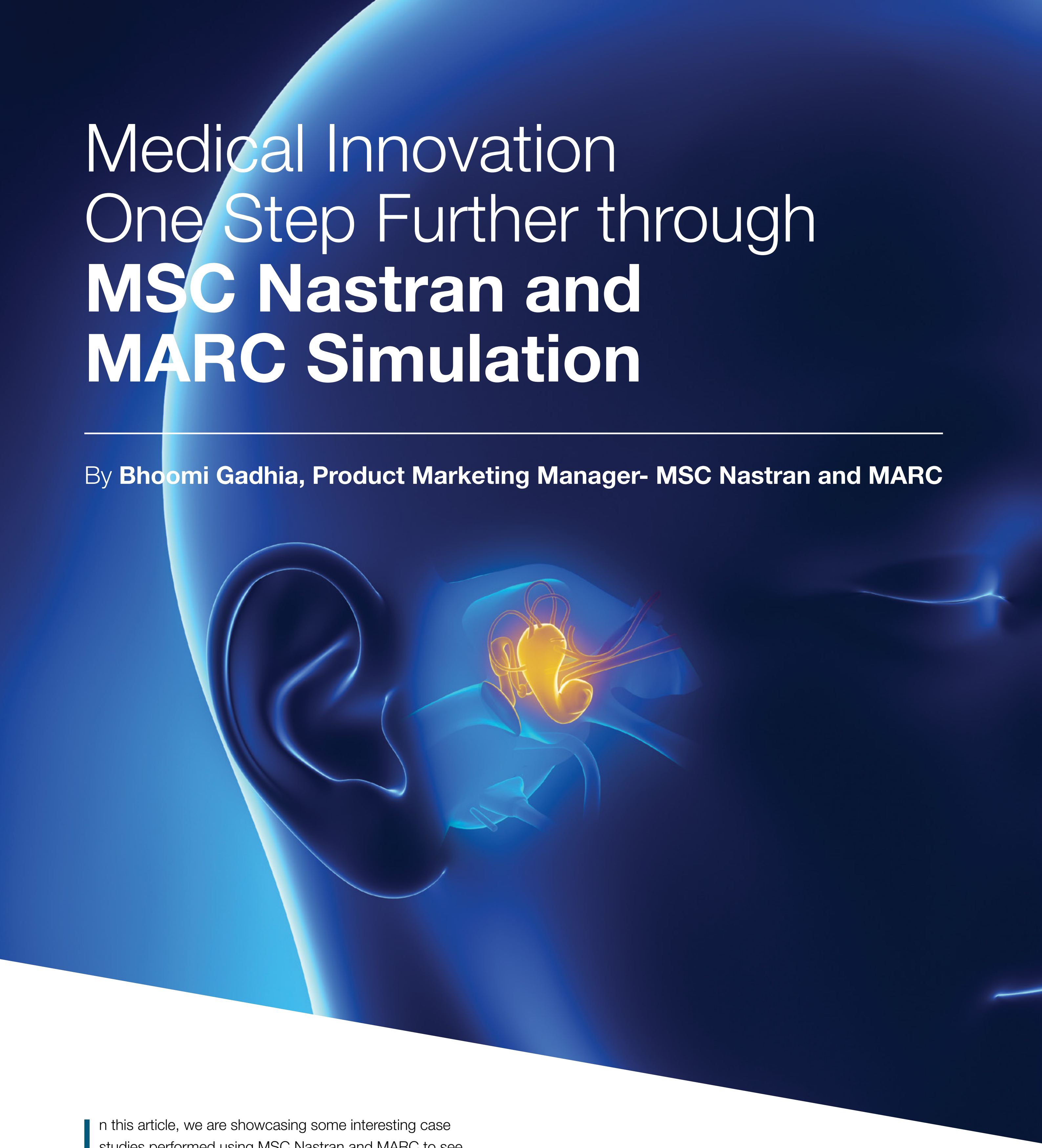
Medical Innovation One Step Further through **MSC Nastran and MARC Simulation**

By **Bhoomi Gadhia, Product Marketing Manager- MSC Nastran and MARC**



In this article, we are showcasing some interesting case studies performed using MSC Nastran and MARC to see how the finite element analysis is leveraged in the medical industry and taken innovation one step further using simulation.

Hearing is one of the five sensations critical to the perception of the surrounding and communication. The overall function of the ear is to convert physical vibrations to nervous impulses. In other words, the vibration from sound is transduced to electric signals in the ear, where they are processed by the central auditory system in the brain. Structural analysis is a crucial part of any design, implementation, and maintenance process in vehicles, machine tools, spacecraft, buildings and even medical devices.

Bone-Anchored Hearing Aid, which is commonly referred to as BAHA provides patients with a higher level of user satisfaction compared to other hearing aids.

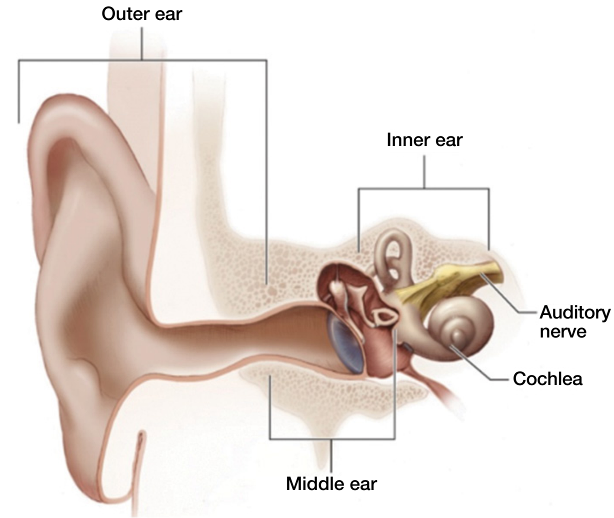


Figure 1 Structure of ear: Outer, middle and inner ear

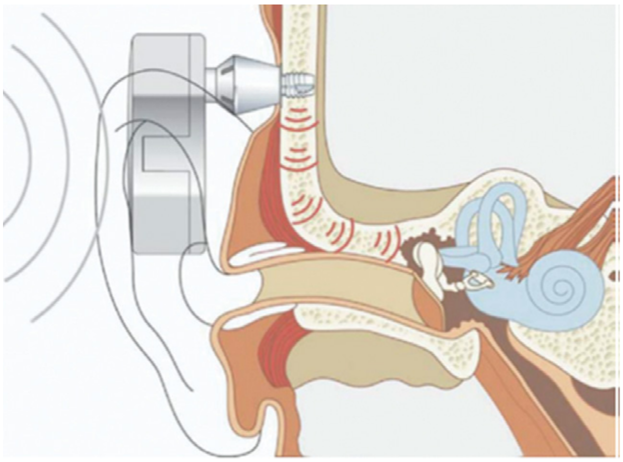


Figure 2Schematic overview of BAHA implant in function

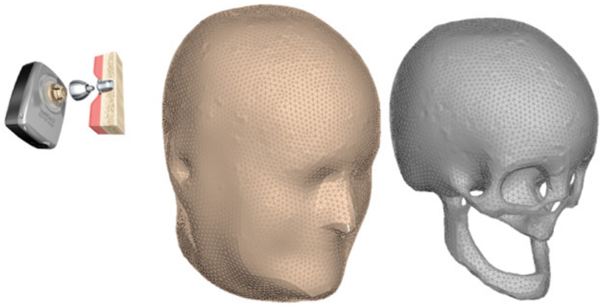


Figure 3 BAHA device, meshed anatomical data of skin and 3D mesh of the skull

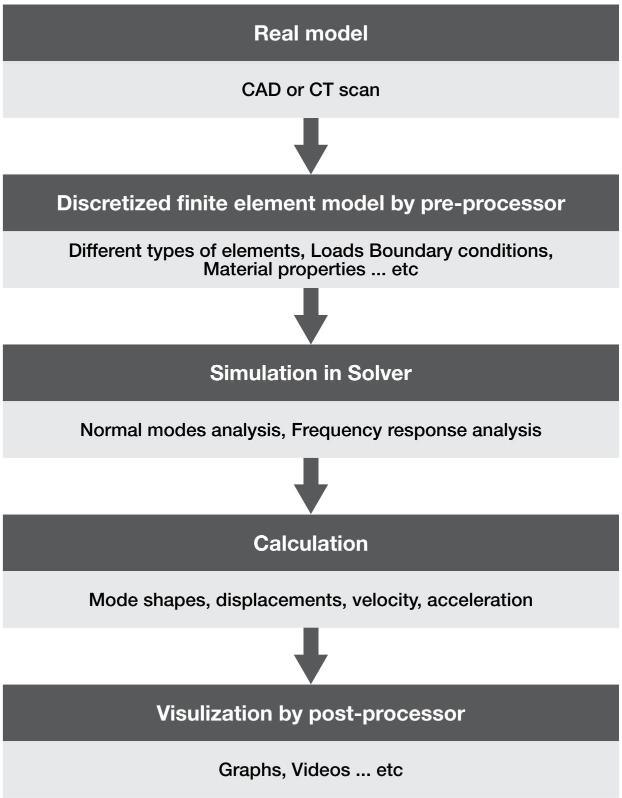


Figure 4 Finite element process

With the help of computational mechanics such as the Finite-Element Method (FEM), the performance of BAHA can be improved before actual costly physical models are built.

Lena Kim, PhD candidate at Chalmers University of Technology, built a 3D FEM model of Human Head to perform a study which leverages MSC Nastran’s FEA results to correlate with the physical tests which in-turn helps the cost-reduction of the Bone-Anchored Hearing Aid.

In this study a valid 3D FE model of the human head is developed, which is used to investigate and simulate the vibration pattern of bone-conducted sound using MSC Nastran. FEA model enable us to investigate the factors that affect the bone conduction pathway, find the correct position to produce the vibration level for the hearing sensation, and further optimize the device in a patient-specific way.

Structural analysis- specifically modal analysis is the first step and plays an important role in the sound and vibration analysis. By performing modal analysis, one can find the system’s natural frequency and mode shapes (shape of vibration) without external force and damping. The results of modal analysis characterize the basic dynamic behavior of the

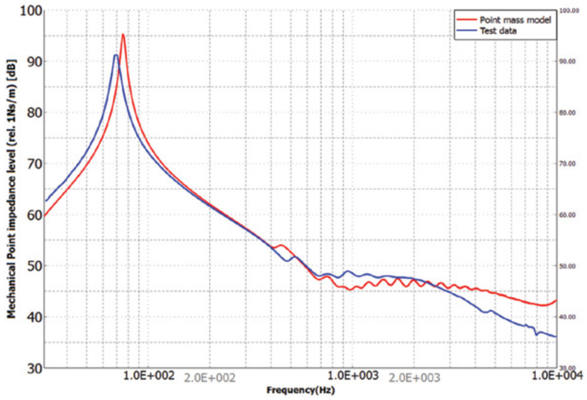


Figure 5 Mechanical point impedance magnitude at BAHA position in NSM model. The red line represents the simulation result with NSM model and the blue line indicates the experimental data from Cochlear.

structure and indicate how the structure will respond under dynamic loading.

FE analysis approximates the geometry or original model in several different types of elements. The behavior of the structure of interest is obtained by analyzing the overall behavior of the elements. The schematic in Figure 4 illustrates the process from the real model to visualization of result. Each of these procedures requires several sets of commercial software which includes MSC Nastran for structural analysis, Beta CAE pre and post-processor ANSA and Meta-post.

The dynamic frequency response was simulated using a commercially available structural analysis software, MSC Nastran. For this model, loads, frequency range, analysis output, and damping coefficients were assigned in the form of Nastran codes. Two types of analysis were performed in this study: normal mode and frequency response analysis

The response of the skull surface of the surrogate was simulated and the output was the velocity for mechanical point impedance. The test data was compared against results using point mass method using MSC Nastran and the results show good agreement with test data.

To conclude, using MSC Nastran frequency response analysis of head simulator model was performed at mechanical point impedance (MPI). The results were remarkably consistent with physical test data in both Non-strutural mass (NSM) and Fluid-Structure (FS) models with anti- resonance frequency at approximately 75-90 Hz; there was only a 5% difference in magnitude level. This study proved that FEA results are close to the physical test and hence they result in cost-reduction of the BAHA device.

Prosthetic Feet using Carbon Fiber Composites: Design, Simulation, & Testing

Prosthetic limbs represent an area of advanced biomedical device technology where artificial limbs meet with advanced aerospace-grade composite materials. The development of prosthetic feet using carbon fiber composite materials has paved the way for many amputees to resume an active and athletic life-style. By combining advanced materials, an understanding of design and stiffness tailoring of composite materials, and aerospace manufacturing techniques, the end result is prosthetic limbs with realistic flexure, “springiness”, and strength. These lifelike prosthetics are a far cry from the “wooden leg” or “stump” associated with injured pirates of old! The technology has advanced so far that that runners with composite feet and legs are qualified to run in the Olympics!

The active users demand for “springy” but, strong prosthetic legs. Major challenges to create these type of Prosthetic legs include Fatigue Durability, balance between strength, stiffness

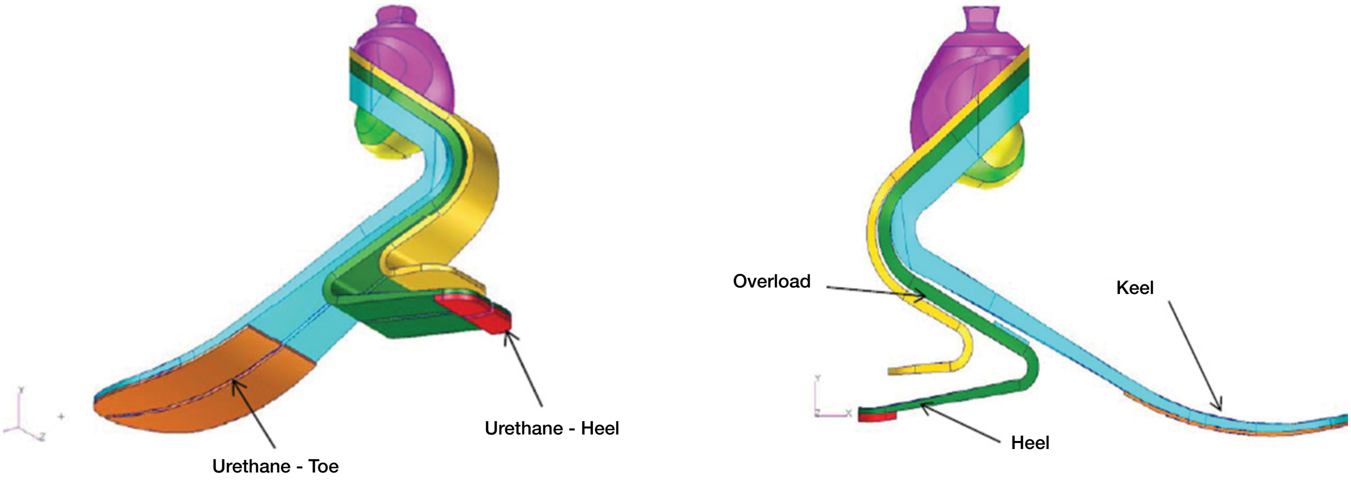


Figure 6 Carbon Fiber Prosthetic Foot - Components: Heel, Keel, Overload, Urethan (Toe and Heel)

> **“Dr. T. Kim Parnell from Parnell Engineering and Consulting used MSC Marc to perform Finite Element Analysis on the heel design, test different material properties and come up with an optimal design.”**

and weight amongst many others. The demand for a prosthetic foot that is suitable for all terrain, with lightweight, exceptional shock absorption and great energy return is very much needed. Composites play a huge role for lightweighting. Dr. T. Kim Parnell from Parnell Engineering and Consulting used MSC Marc to perform Finite Element Analysis on the heel design, test different material properties and come up with an optimal design.

Model description was done using MSC Marc/Mentat, where the contact bodies are defined such that Heel, Urethane are flexible contact bodies and Overload, Keel are assumed rigid and materials are defined as composites with Tsai-Wu failure criteria and the results of 2 Urethan Configurations are compared in order to come up with the final design.

After performing analysis using Marc, it was concluded that, Short Urethan Heel, 1/8” thick causes increased heel bending and it is more flexible whereas, Long Urethane Heel, 1/16” thick, has reduced heel bending due to more uniform heel contact, stiffer response due to earlier contact with Keel.

After comparing different results of both heel types, it was concluded that the Proposed 1/16” Long Heel is stiffer due to earlier secondary contact with Heel and earlier contact with Keel. 1/16” Long heel gets progressive stiffening and the stress results for heel improved even for same displacement and thus higher applied load.

To conclude, simulation using MSC MARC/Mentat with physical testing help to better understand the delamination failure mode.

See More Co-Simulation Applications:
www.mscsoftware.com/cosim

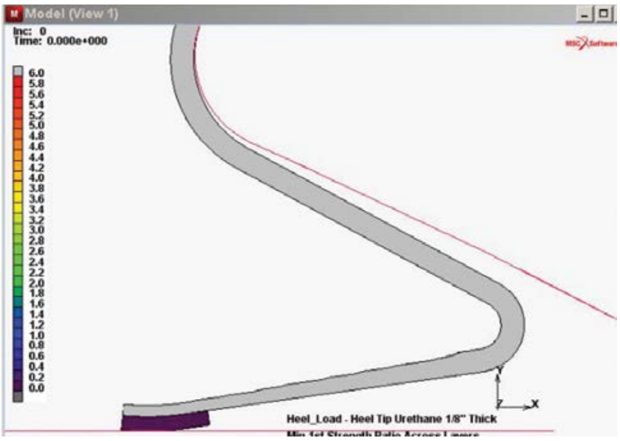


Figure 7Proposed 1/16” Long

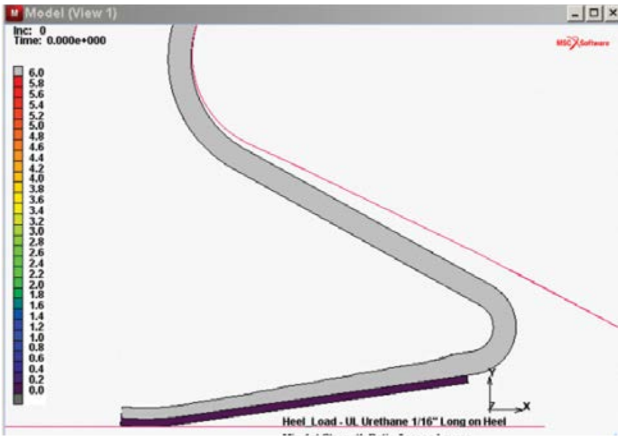


Figure 8 Initial 1/8” short