

Optimization in the Vibro-Acoustic Design of Hearing Instruments

Acknowledgement

SIMULIA would like to thank Morten B. Søndergaard of GN ReSound A/S for his significant contributions to this Technology Brief.

Summary

In the design of hearing instruments it is important to achieve the highest possible gain without introducing feedback between the microphone and loudspeaker. With more gain, a larger hearing loss can be accommodated and a greater number of users benefit.

Maximizing gain while minimizing the possibility of feedback requires an optimal choice of design parameters. In this Technology Brief, we outline how Abaqus/Standard and Isight can be combined in a process to optimize the vibro-acoustic characteristics of hearing instruments.

Background

The trend for the design of hearing instruments may be simply stated as 'smaller is better.' A smaller device will be more comfortable and less visually conspicuous. As the size of a given instrument reduces however, the proximity of the loudspeaker and microphone increases; consequently, it becomes more difficult to achieve high gain without feedback.

A more traditional design approach will involve a combination of experience and 'trial and error,' where different approaches for minimizing feedback are manually tested. Because the feedback path for a hearing instrument is very complex and involves many variables, this method can be time consuming.

While the coupled structural-acoustic capabilities of Abaqus/Standard can be directly applied to the development of a hearing instrument, cost and uncertainty in the design process can be further reduced by additionally using Isight to determine an optimal configuration in an automated fashion.

Finite element modeling approach

A finite element model of a behind-the-ear (BTE) instrument is built in Abaqus/Standard. A coupled structural-acoustic analysis is then carried out, and the results are compared to experimental data to validate the model. This model is then incorporated into an Isight process for an automatic, parameterized optimization.



Key Features and Benefits

- Automatic optimization process with Isight
- Coupled structural-acoustic analysis capabilities in Abaqus/Standard
- Pro/ENGINEER associative interface with Abaqus/CAE

A GN ReSound Azure71 hearing instrument, connected to a PVC tube and a 2cc coupler, is modeled as a fully coupled structural-acoustics system in Abaqus/Standard. In addition to the structure itself, the model includes the air inside the tube and surrounding the device. The finite element mesh of the entire model is shown in Figure 1.

The structure is modeled with continuum elements and the air is modeled with acoustic elements. The acoustic-

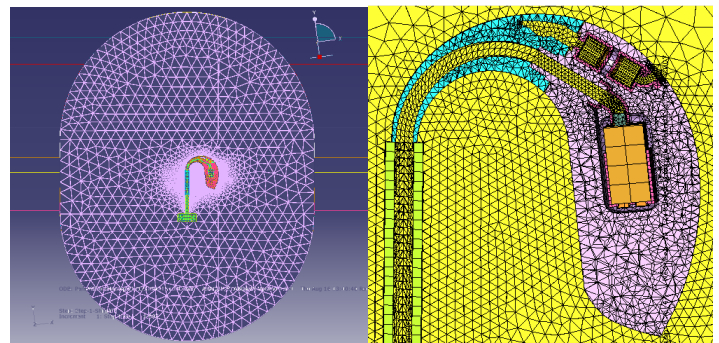


Figure 1: Finite element mesh

structural coupling is realized with surface-based tie constraints. With this approach, the air and the structure can have dissimilar meshes at the interface, making meshing more convenient.

Three major analysis steps are performed:

1. A static shrink-fit step to compute the stress in a rubber suspension that results from inserting it between the loudspeaker (receiver) and the device housing
2. A frequency extraction step to compute the coupled structural-acoustic modes
3. A steady state dynamic step in which the structure is sinusoidally excited at the receiver

The boundary condition on the perimeter of the exterior acoustic region is made to simulate an infinite expanse of air.

Validation of the finite element model

To validate the simulation, real physical prototypes were made from the actual model geometry. Figure 2 shows the comparison between the measured and simulated velocity on the device surface at the location noted in the figure. The peaks and valleys of the response curves show sufficient agreement to use the Abaqus/Standard model with confidence.

Optimization approach

Isight is used for the optimization process. In choosing the variables to optimize, only properties that can be changed in practice were considered. In this study a total of 5 parameters were selected. Two correspond to different types of materials and the remainder allow for variation in density and stiffness.

To randomly spread the parameters within the design space and cover the different possible configurations in an optimal way, the Optimal Latin Hyper Cube technique is used. This is done to minimize the number of analyses and is defined in the Design Of Experiments (DOE) component of Isight. In this case the best practice number of runs for 5 parameters is $(n+1)(n+2) = 42$.

The optimization loop in Isight is shown in Figure 3. The Data Exchanger components are used to rewrite the Abaqus input file when different parameters are analyzed. The Abaqus component is used to vary two density values, one stiffness value, and submit the input files for analysis.

A Python script is then used to evaluate the Abaqus results by comparing them to a target curve. The target curve is defined as the maximum gain in the model before the optimization. If the optimization process predicts a higher possible gain, then the given set of parameters shows a reduced feedback configuration.

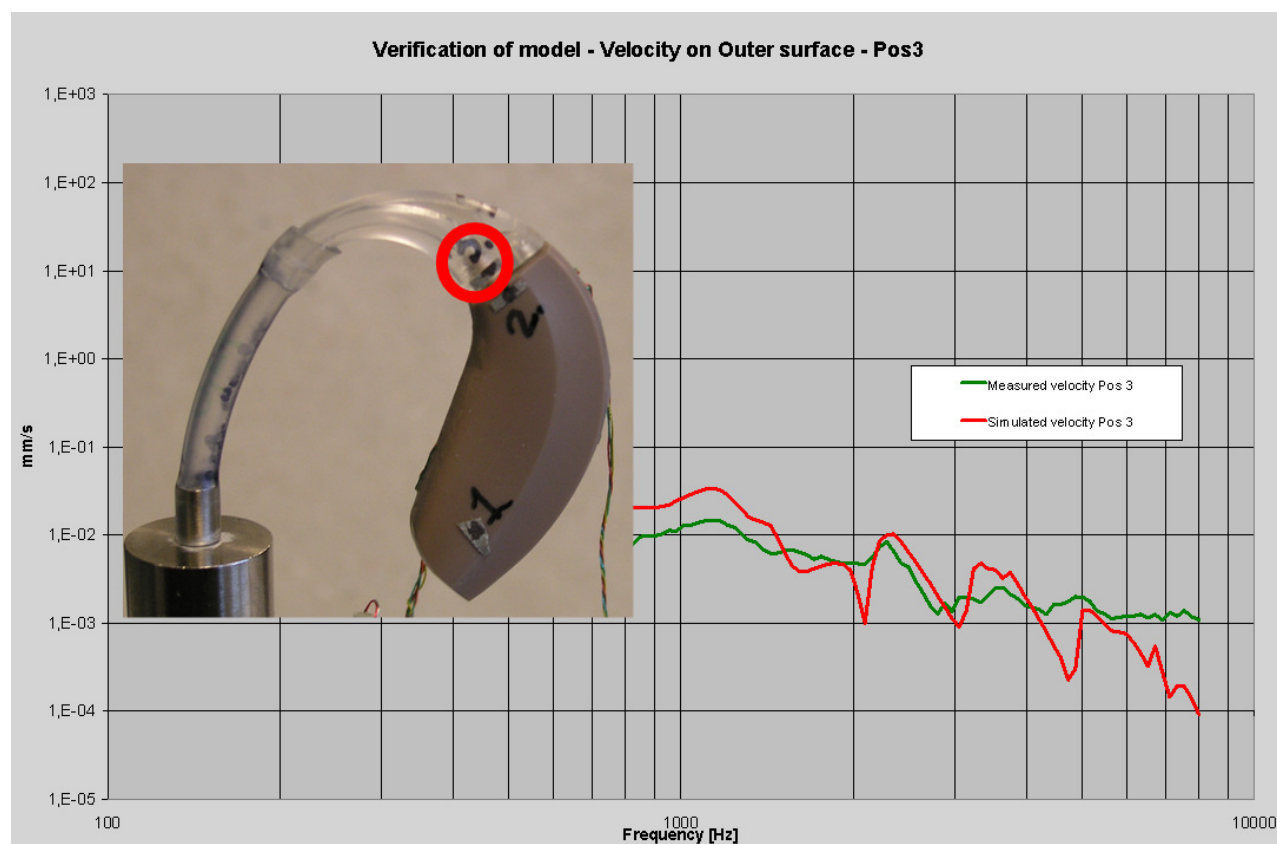


Figure 2: Validation of the finite element model with velocity outputs

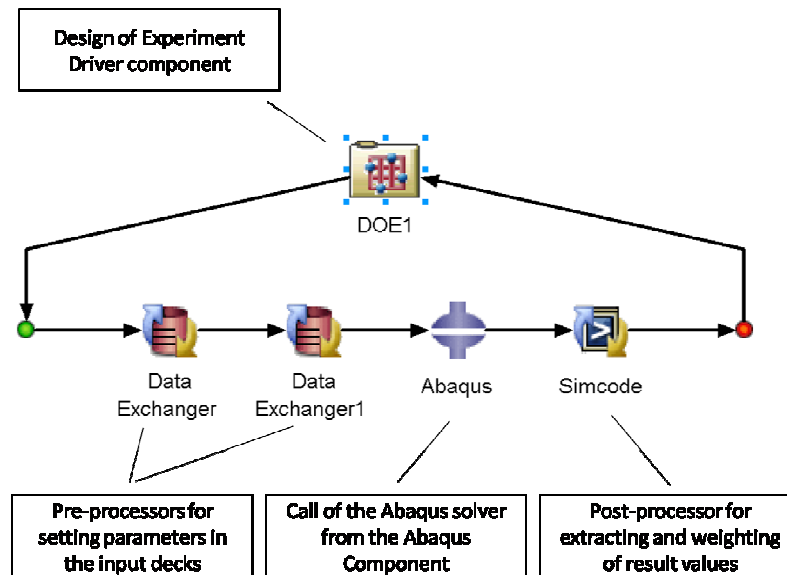


Figure 3: Optimization process in Isight

Two weighting functions are used to maximize the targets and make them more visible in the data. The first function weighs the solutions that have a gain greater than 5 dB above the target value. The second weighting function then weighs the solutions that perform better than the target in a specified frequency range.

Outcome of optimization process

Among the five design parameters, the optimization process finds two that dominate the output, namely the density and the stiffness. Figure 4 shows the response surface of the output as a function of these two parameters. The red area is where the optimal values reside.

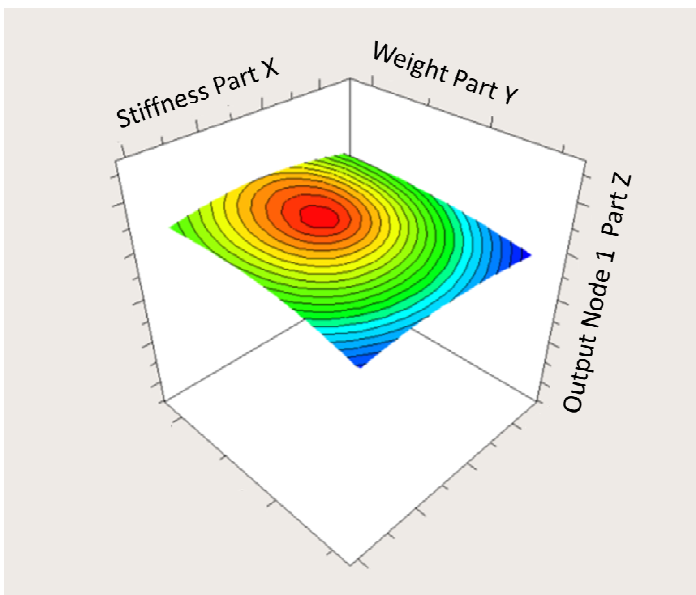


Figure 4: Response surface for the output

With Isight, it is possible to achieve an improvement in gain of as much as 6 dB in the desired frequency range. This is clearly better than the typical 2-3 dB improvement achieved with the traditional “trial and error” approach.

Conclusion

An optimization process in Isight that employs an Abaqus/Standard coupled structural-acoustic finite element model is successfully carried out to optimize the gain in a hearing instrument. Of the chosen design parameters, the two that dominate the device performance are found, and the feedback path between receiver and microphones is reduced by approximately 6 dB. The benefits of such an automatic optimization process can be summarized as follows:

- large reduction in manual work
- systematic evaluation of design parameter combinations
- postprocessing tools to efficiently analyze the output data

The automatic optimization process with Isight allows a much quicker investigation over the design space than a traditional manual approach and greatly reduces prototyping time.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus Version 6.13 documentation references:

- Analysis User's Guide
 - Acoustic and shock analysis, Section 6.10
 - Natural frequency extraction, Section 6.3.5

Isight References

- Isight User's Guide

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Vélizy, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include Abaqus, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright Dassault Systèmes, 2009

