Paper 195

Proceedings of the Eleventh International Conference
on Computational Structures Technology,
B.H.V. Topping, (Editor),
Civil-Comp Press, Stirlingshire, Scotland

Accuracy of Dynamic and Acoustic Analysis of Lightweight Panel Structures: A Comparison of ABAQUS and ANSYS

P.H. Kirkegaard[1], K.A. Dickow[1], L. Andersen[1] and A. Sjöström[2]
[1]Department of Civil Engineering, Aalborg University, Denmark
[2]Division of Structural Mechanics, Lund University, Sweden

Abstract

Currently there is an increasing focus on the transmission of low frequency sound in buildings, as sources such as road and air traffic or even home theatre subwoofers become part of everyday life for many people. Hence, development of efficient and accurate methods for prediction of sound in such buildings is important. In the low-frequency range, prediction of sound and vibration in building structures may be achieved by finite-element analysis (FEA). The aim of this paper is to compare the two commercial codes ABAQUS and ANSYS for FEA of an acoustic-structural coupling in a timber lightweight panel structure. For this purpose, modal analyses are carried out employing a fully coupled model of sound waves within an acoustic medium and vibrations in the structural part. The study concerns the frequency range 50–250 Hz.

Keywords: structure-borne sound, building acoustics, finite-element analysis, ABAQUS, ANSYS.

1 Introduction

Low-frequency sound is a potential nuisance to inhabitants in lightweight building structures. Hence, development of efficient and accurate methods for prediction of sound in such buildings is important. For a variety of simple structures, analytical solutions have been established [1], and statistical energy analysis (SEA) [2, 3, 4] has in general been found to provide a reliable prediction of sound transmission. However, SEA has limited validity for lightweight structures such as wooden floor and wall panels. Statistical energy analysis (SEA) and the European standard EN 12354 [5] have been found to provide a reliable prediction of sound transmission in heavy structures, e.g. concrete buildings. However, predictions of sound transmission in lightweight structures using energy-based methods may be imprecise

[6]. Instead, recent research indicates that the finite-element analysis (FEA) can be used for prediction of sound transmission in the low-frequency range through such lightweight building structures [6, 7, 8]. The present paper continues along this line by employment of a fully coupled three-dimensional model of the acoustic field and a lightweight structure. The commercial FEA codes ABAQUS [9] and ANSYS [10] have capabilities for fully coupled analysis of structures interacting with acoustic media, thus allowing the analysis of structure-borne and air-borne sound. The aim of the paper is to compare the two commercial codes regarding accuracy and ease of modelling with focus on the coupling between the acoustic field and the structure. Modal analyses are carried out in the frequency domain up to 250 Hz. Section 2 describes the computational model applied for the analyses. The results are presented and discussed in Section 3 and the main conclusions are given in Section 4.

2 Computational Models

Lightweight wall and floor constructions are often made from panel structures with plates on stud (or joist) frames. In such a panel, the frames may be either single or double-stud frames with or without different porous and elastic layers included to reduce the transmission of sound and vibration through the panel. In the case of single-stud frames there is direct structural coupling from one side of the panel to the other. In the present analyses, a single room consisting of three panels is considered. The room with the internal dimensions $3.7 \times 3.7 \times 2.7$ m^3 (length × width × height) is modeled by the commercial finite-element analysis (FEA) software packages ABAQUS v. 6.11-1 [9] and ANSYS v. 14.0 [10], respectively. In all the FEA models, three walls are considered as flexible lightweight panels with plates mounted on studs. The cavities inside the panels are assumed as illustrated in Figure 1.

The structure consists of timber and the plates have a thickness of 0.02 m whereas the studs have a cross-sectional area of 0.1×0.1 m^2. The frame of each panel has the same cross-sectional dimensions. Hence, the cavities between the studs have a width of 0.5 m. In the case of standard unidirectional studs, the cavities are 2.5 m high. Between adjacent wall panels, and between the walls, columns and beams are placed with a cross-sectional area of 0.14×0.14 m^2.

Homogeneous and isotropic linear materials are assumed for the panels. The material properties are:

- Timber (plates, studs) Young's modulus E= 14 GPa, Poisson ratio $\upsilon = 0.35$, density $\rho = 500$ kg/m3.
- Air: Bulk modulus $K_a = 141360$ Pa, density $\rho_a = 1.2$ kg/m3.

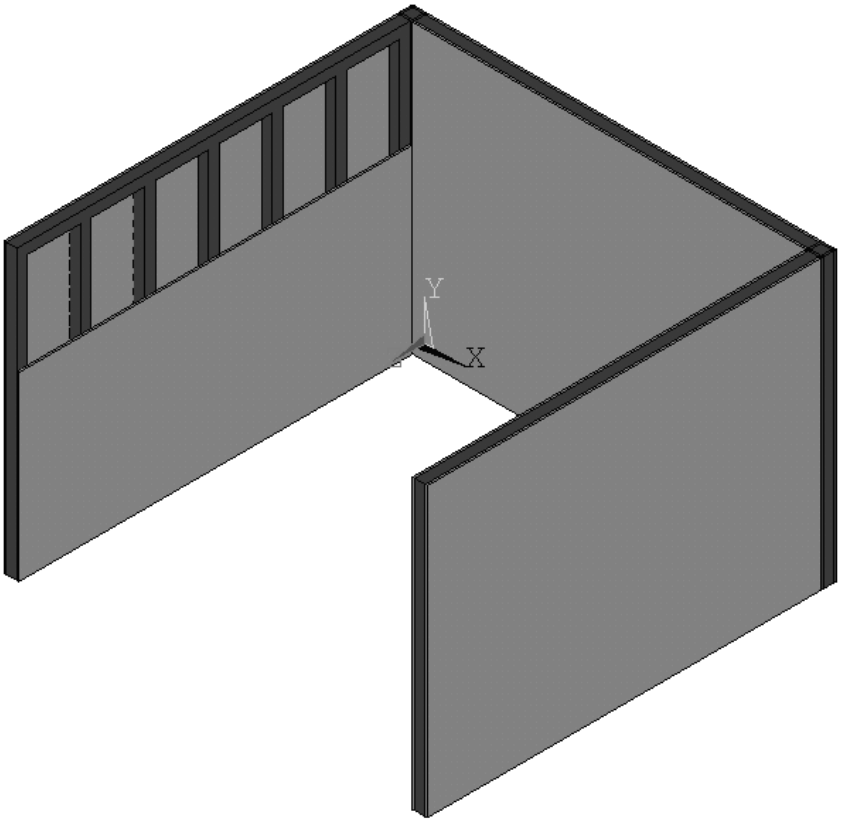

Figure 1: Computational model including the three opposing walls. A part of a plate has been removed to reveal the underlying studs.

The computational mesh for the ABAQUS model with three walls is obtained using solid 3-D stress finite hexahedral elements (C3D20) with 20 nodes and quadratic spatial interpolation of the displacement field for the structure, and acoustic hexahedral elements (AC3D20) with 20 nodes are utilized for the air. Full integration with 3×3×3 Gauss points is adopted. Based on the wave lengths expected to occur in the model at the higher frequency of 250 Hz, the mesh size (maximum element side length) has been put to 0.1 m in all parts of the model, see Figure 2 and 3. A single element over the plate thickness has been assumed to provide an adequate representation of the bending stiffness. The frames and plates are fully fixed along the outer edge and free otherwise, i.e. the frames and plates are fixed on the surfaces with outward normal in the positive or negative Y directions. At all internal interfaces between structural elements, the frames and columns (or beams) are rigidly connected, employing surface-to-surface based tie constraints in ABAQUS. Similar constraints are applied between the air (pressure) and the structure (displacement). Finally, it is noted that the walls, floors and ceilings that are not included in the model are all assumed to behave as rigid, hard surfaces, leading to full reflection of the acoustic waves at these boundaries.

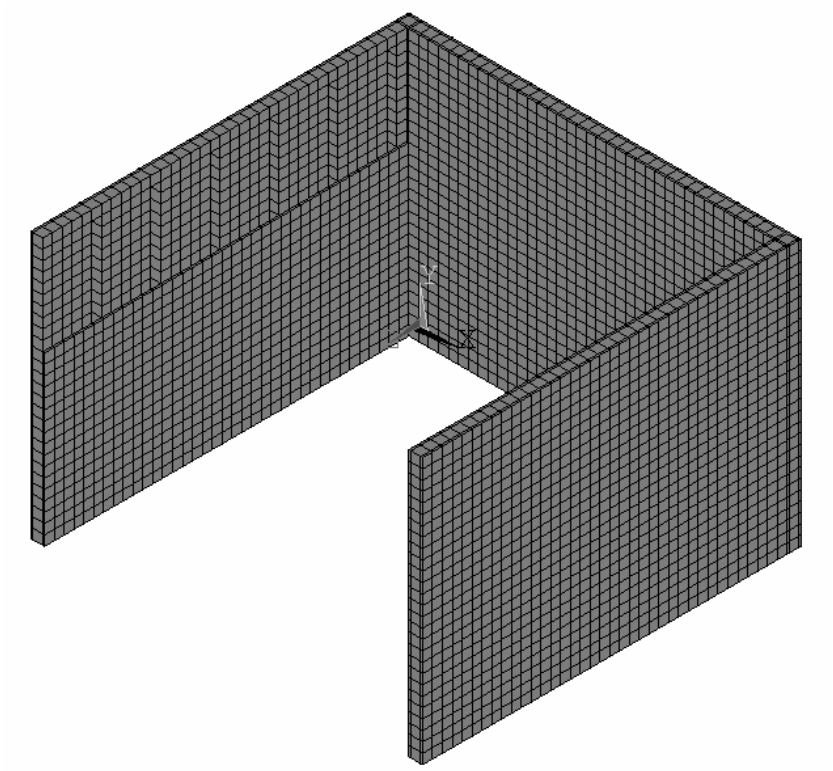

Figure 2: Mesh of structural parts of the model with three walls.

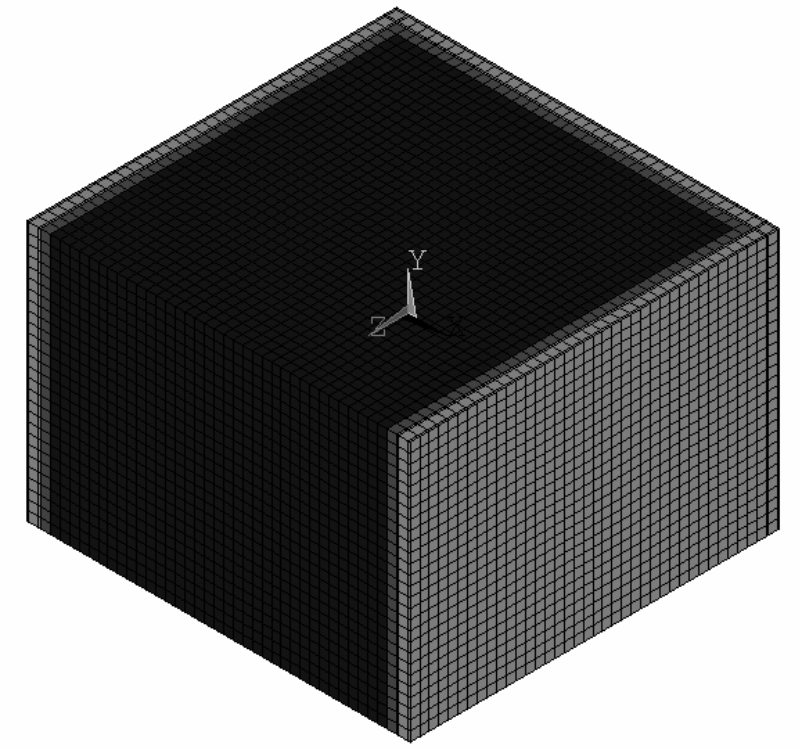

Figure 3: Mesh of (ANSYS) model with three walls and air.

The ANSYS model has been modelled using solid 3-D stress finite hexahedral elements (SOLID186) with 20 nodes and quadratic spatial interpolation of the displacement field for the structure, and acoustic hexahedral elements (FLUID220) are utilized for the air. FLUID220 is a higher order 3-D 20-node solid element that exhibits quadratic pressure behavior. It is used for modeling the fluid medium and the interface in fluid/structure interaction problems. The element node has four degrees of freedom per node: translations in the nodal UX, UY and UZ directions, and pressure. The translations are only applicable at nodes that are on the interface. The mesh size is chosen as 0.1 m in all parts of the model and boundary conditions are applied as described for the ABAQUS model. According to [9] the SOLID186 element corresponds to C3D20 if KEYOPT(2) = 2, and KEYOPT(6) = 0 are chosen for the SOLID186 element, i.e. full integration and pure displacement formulation are assumed. For the ANSYS model two different coupling approaches between the structure and the air have been considered. First, for acoustic elements (FLUID220) that are in contact with a solid KEYOPT(2) = 0 is used which allows for fluid-structure interaction. This results in un-symmetric element matrices with UX, UY, UZ, and PRES (pressure) as the degrees of freedom. For all other acoustic elements in the fluid domain KEYOPT(2) = 1 results in symmetric element matrices with the PRES degree of freedom. This approach for the coupling between the structure and air is illustrated in figure 3, where one layer of FLUID220 elements with KEYOPT(2) = 0 is used and the interior part is model using KEYOPT(2) = 1. The second approach is using KEYOPT(2) = 2 for all FLUID220 elements symmetric element matrices are obtained for all element in the fluid domain.

3 Results and discussion

The following section presents results from a modal analysis of the models described in section 2. The undamped eigen-frequencies of the model have been calculated for the frequency range 1–250 Hz. It is noted that the computational model produces a number of modes at the frequency 0 Hz, corresponding to a quasi-static change of the pressure inside the air-filled cavities within the panels. This explains why the lower frequency limit has been put to 1 Hz. The eigen-frequenices have been estimated using the Lanczos algorithm implemented in ABAQUS for a symmetric problem which is obtained using surface-to-surface based tie constraints in ABAQUS. By using ANSYS two solution strategies have be investigated according to the two different coupling approaches between the structure and the air.

Prior to the analyses the ABAQUS and ANSYS models have been compared with respect to following quantities to ensure equal models:

- Physical mass
- Center of mass
- Moment of inertia about origin
- Moment of inertia abut center of mass
- Number of nodes and elements
- Number couplings

3.1 Modal Analysis

Figure 4 shows estimated undamped eigen-frequencies when only the timber structural elements are considered. The estimated structural modes are distributed with the first eigen-frequency occurring from about 108 Hz to about 220 Hz. As expected the results show no deviations. The results obtained for the models where the air is included do also show a good agreement, see Figure 5. However for estimated frequencies above 200 Hz. small deviations are seen, see Figure 6. These modes are identified as higher coupled modes between the panels and the air. It is seen that the results obtained in this range with ABAQUS model indicate a lower stiffness compared to the ANSYS model. This deviation could be related to the coupling approaches used. Figure 7 shows the estimated eigen-frequencies obtained with ANSYS using the two different coupling described in section 2. These results have a good agreement with respect to the number of eigen-modes but also with respect to the size of the frequencies. Only above 230 Hz fewer modes have been estimated using coupling approach 1. This could be due to use of the ANSYS unsymmetric solver. It is know that such a solver gives acceptable results for structure-acoustic modal analysis problems. However, the solver is sensitive to the so-called “shift” (the starting frequency). Especially, if some eigen-frequencies are clustered, the results depend on the first step at which a hidden eigen-frequency is expected.

4 Conclusions

This paper has considered a comparison of the two commercial codes ABAQUS and ANSYS for FEA of an acoustic-structural coupling in a timber lightweight panel structure. For this purpose, modal analyses are carried out employing a fully coupled model of sound waves within an acoustic medium and vibrations in the structural part. The study concerned the frequency range 50–250 Hz. In general the two codes deliver result with a good agreement, however for higher eigen-modes it was found that the ANSYS model was a little bit stiffer. This deviation could be because of the different coupling approaches used in the study. In the future analysis of the acoustic performance of lightweight building this issue will be subject to further research by the authors.

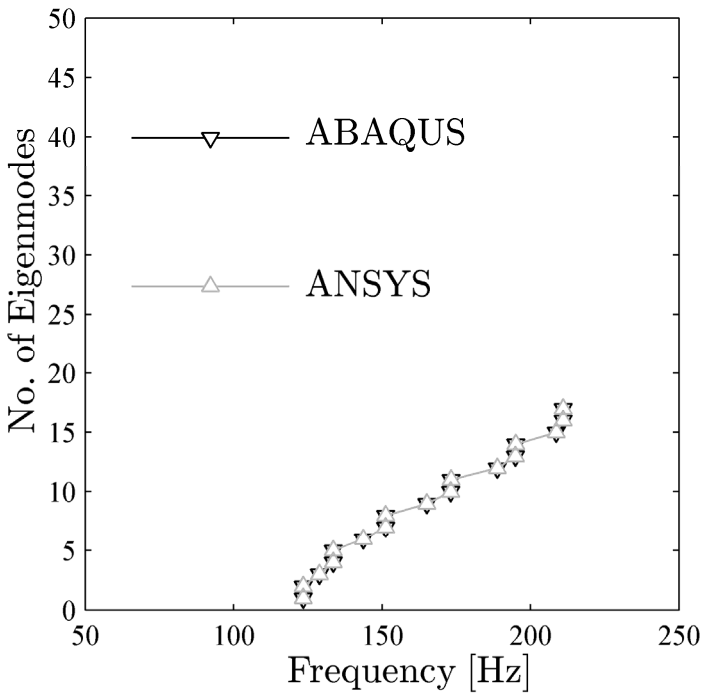

Figure 4: Eigen-frequencies - with no air.

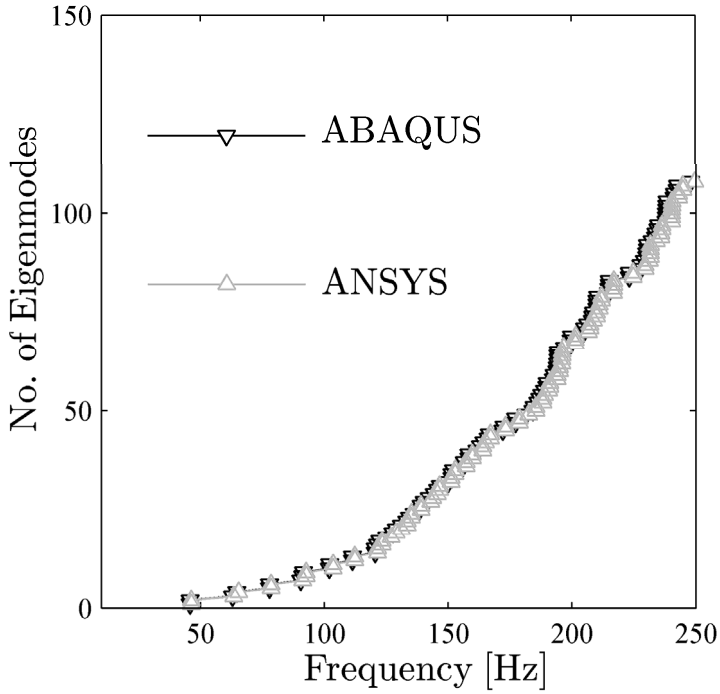

Figure 5: Eigen-frequencies - with air.

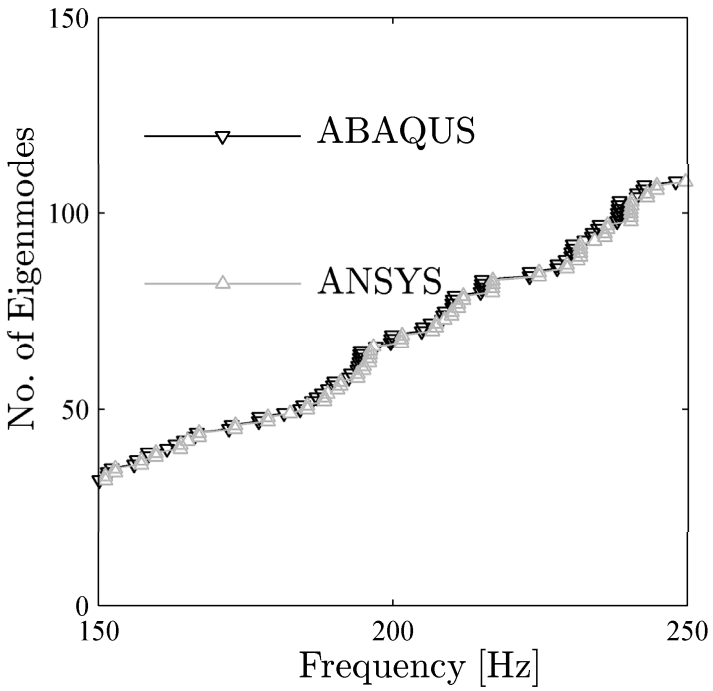

Figure 6: Eigen-frequencies - with air (zoom).

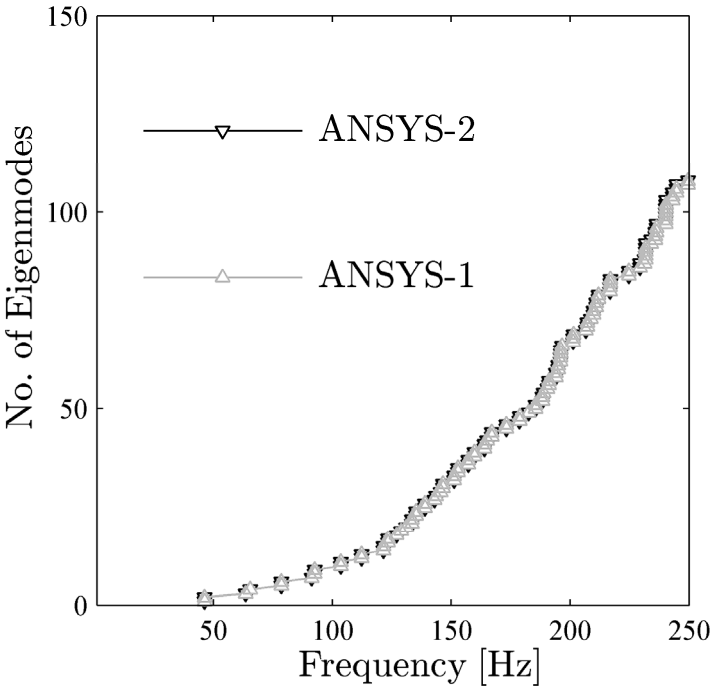

Figure 7: Eigen-frequencies - with air.

Acknowledgement

The present research is part of the InterReg project "Silent Spaces", funded by the European Union. The authors highly appreciate the financial support.

References

[1] L. Cremer, M. Heckl, and B.A.T. Petersson. Structure-Borne Sound. Springer-Verlag, 3rd edition, 2005.

[2] R. J. M. Craik: Sound Transmission Through Buildings using Statistical Energy Analysis. Gower, 1996.

[2] C. B. Burroughs, R. W. Fischer, F. R. Kern: An Introduction to Statistical Energy Analysis. The Journal of the Acoustical Society of America 101,pp. 1779–1789, 1997.

[3] F. Fahy, P. Gardonio: Sound and Structural Vibration. Elsevier, 2nd edition, 2009.

[4] EN 12354-1:2000 Building Acoustics. Estimation of Acoustic Performance in Buildings from the Performance of Elements. Airborne Sound Insulation between Rooms. European Committee for Standardization, 3rd edition, 2000.

[5] J. Mahn: Prediction of Flanking Noise Transmission in Lightweight Building Constructions: A Theoretical and Experimental Evaluation of the Application of EN12354-1. Technical report, University of Canterbury, Acoustics Research Group, 2007.

[6] J. Brunskog, P. Davidsson: Sound Transmission of Structures. A Finite Element Approach with Simplified Room Description. Acta Acustica united with Acustica 90, pp.847–857, 2004.

[7] P. Davidsson: Structure-acoustic analysis, finite element modelling and reduction methods,. Ph.D. thesis, Lund University, Department of Structural Mechanics, 2004.

[8] J. S. D. Bard, G. Sandberg: Prediction model of the flanking transmission through a light-weight building construction utilizing fluid-structure interaction procedures. In: Proceedings of Internoise 2008. 2008 .

[9] Dassault Systèmes Simulia Corp., Providence, RI, USA: ABAQUS Analysis. User's Manual, Version 6.11.

[10] ANSYS, Inc., Southpointe, Canonsburg, PA, USA: ANSYS Mechanical ADPL User's Manual, Version 14.0