Finite Elements in Analysis and Design 149 (2018) 45–56

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

journal homepage: www.elsevier.com/locate/finel

Numerical verification of an efficient coupled SAFE-3D FE analysis for guided wave ultrasound excitation

C.S. Long *, P.W. Loveday, D.A. Ramatlo, E.V. Andhavarapu

CSIR Material Science and Manufacturing, P.O. Box 395, Pretoria, 0001, South Africa

ARTICLE INFO

Keywords:
Semi-analytical finite element (SAFE)
Piezoelectric transducer model
Cut-off frequency
Modal amplitude
Abaqus/Explicit
Train rail

ABSTRACT

Numerical verification of a method to simulate piezoelectric transducers exciting infinite elastic waveguides is presented. The method, referred to as SAFE-3D, combines a 3D finite element (FE) model of a transducer with a 2D semi-analytical finite element (SAFE) model of the waveguide and accounts for the dynamics of the transducer. An interpolation procedure is employed to transfer forces and displacements between the SAFE and 3D FE models, and therefore nodes at the interface between the two models are not required to be coincident. An Abaqus/Explicit analysis, employing a thermal equivalent piezoelectric model and absorbing boundary conditions to prevent end reflections, is used to verify the accuracy of the SAFE-3D model. A piezoelectric transducer attached to the web of a rail and driven with frequency content which excites a mode cut-off is considered. A driving signal which does not contain cut-off frequencies is used for comparison. Time domain displacement results computed using Abaqus/Explicit and SAFE-3D are compared directly. Several methods to alleviate the numerical difficulties encountered by the SAFE-3D method, when transforming frequency domain displacements to the time domain, close to cut-off frequencies are evaluated. It is shown that post-processing methods have a similar effect to adding damping, but are less numerically expensive if iterative tuning of parameters is required. A SAFE-based method to extract modal amplitudes from Abaqus/Explicit time domain results is used to evaluate the accuracy of SAFE-3D in the frequency domain. Good agreement between the SAFE-3D method and results computed using Abaqus/Explicit is achieved, despite the Abaqus/Explicit and SAFE-3D models predicting slightly different cut-off frequencies.

1. Introduction

Guided wave ultrasound (GWU) is well suited for inspection and monitoring applications of elongated structures such as plates, rods, pipes and rails [1–3]. By controlling which propagating modes are excited, and with knowledge of the propagation characteristics, systems can be designed so that propagating energy can be distributed across the entire cross-section of the waveguide or concentrated in specific locations, or in geometrical features, depending on what damage is being sought. Guided waves can propagate long distance, especially when compared to conventional ultrasonic inspection (up to kilometers in some cases [4]). Furthermore, GWU is known to propagate in structures that are covered, submerged or buried reducing preparation efforts and cost [5–7]. These properties make GWU very attractive for monitoring and inspection applications since long distance inspections can be carried out from a single stationary source.

In order to design a GWU-based non-destructive evaluation (NDE) system, it is necessary to understand how guided waves are excited, how they propagate (dispersion, attenuation, etc.), how they interact with discontinuities and damage (scattering) and finally how they are sensed (transduction). A conventional time-domain finite element analysis can be carried out to analyse the excitation, propagation, scattering and sensing. However, this type of analysis is generally very numerically expensive (if it is possible at all) especially at higher frequencies and over significant propagation distance, due to the fine spacial and temporal discretisation required. Furthermore, since the analysis is carried out in the time domain, modal information is not obtained directly and has to be extracted in some way. Due to these drawbacks, the semi-analytical finite element (SAFE) method [8–10] has become a popular analysis and design tool in the GWU community. The SAFE method naturally computes results based on their modal contributions and responses at significant distances can be estimated efficiently since

* Corresponding author.
E-mail address: clong@csir.co.za (C.S. Long).

https://doi.org/10.1016/j.finel.2018.05.001
Received 19 December 2017; Received in revised form 7 March 2018; Accepted 4 May 2018
Available online XXX

the propagation direction (in which the structure is elongated) is treated analytically.

The focus of this paper is on the analysis of guided wave excitation. An efficient implementation of a method previously proposed by one of the authors [11,12] is presented, which allows several design iterations to be computed without having to solve the SAFE eigenvalue problem multiple times, and does not require transducer nodes to be coincident with the waveguide nodes. We also consider the performance of this method when exciting the waveguide at frequencies where modes cut-off on the frequency axis. These frequencies have previously been avoided [13].

Previous authors have considered the analysis of guided wave excitation. Willberg et al. [1] present an overview of relevant work, including a brief discussion of adhesive material, which we neglect in this study (but which could be included as a thin soft layer of elements between the transducer and the waveguide).

Hybrid models are popular in scattering studies, and are capable of modelling wave excitation, propagation and scattering using different discretisations or even different models entirely [14,15]. Although it may be possible for hybrid methods to include a detailed transducer model (especially when the procedure employs commercial finite element software, e.g. Ref. [15]), excitation sources are usually simulated as prescribed forces or displacements.

Lowe et al. [16] and Fateri et al. [17] consider an aluminium rod with a large (relative to the waveguide) transducer attached. They demonstrate the importance of including the transducer in the numerical model (as opposed to simply modelling the transducer as a distributed force). Reflections and mode conversion from a coupled piezoelectric transducer are considered. A full 3D Abaqus model of the waveguide and transducer is used for comparison with a single point excitation. At the excitation frequency considered in their work, there are only three possible propagating modes, L(0,1), T(0,1) and F(1,1), and the torsional mode is neglected. The comparison was performed in the time domain with modes separated based on Time of Arrival (ToA).

Kalkowski et al. [18] propose a technique based on the SAFE method for modelling waveguides with piezoelectric transducers attached. A piezoelectric SAFE element is presented and discrete piezoelectric elements are incorporated by computing scattering matrices at locations where geometry changes discretely. The proposed method is well suited to prismatic transducers (with regular shape in the propagation direction) such as simple rectangular patch and sandwich transducers, but may present difficulties when transducers have complex shape. Their proposed method is verified numerically using a simple beam model and validated experimentally with a short beam with anechoic terminations. The paper also presents a summary of some other relevant works.

Jezzine et al. [19] consider the case of a transducer fixed to a free end of a waveguide (i.e. on the arbitrary cross-section) using techniques similar to those employed for scattering from free ends and discontinuities [20]. They present comparison with analytical and previously published experimental results.

One of the authors of the current work previously proposed a method to couple a SAFE model of the waveguide with a full 3D model of a piezoelectric transducer [11,12]. The method involves computing the effective stiffness of the infinite waveguide, and then solving the transducer dynamics with the appropriate boundary condition, and then finally using the reaction forces from this analysis to compute the forced response of the waveguide. This approach properly accounts for the dynamics of the transducer and has been validated by comparison with experimental results away from cut-off frequencies [12,13]. This method is generalised in this current work, so that the interface nodes between the SAFE and 3D meshes are not required to be coincident. This is accomplished by using a simple interpolation strategy. Furthermore, the resonance-like behaviour encountered when exciting a mode of propagation close to its cut-off frequency is studied and addressed. The procedure is compared with results from a time domain solution computed using the commercial finite element package Abaqus/Explicit. This comparison represents a verification that the method correctly solves the idealised mathematical problem. Validation would require a very carefully controlled experiment to isolate the effects of damping and transducer adhesion to the waveguide.

2. Problem formulation and implementation

The presentation in this section will focus on the coupling of the 3D transducer FE model and the 2D waveguide SAFE model. More detail regarding the conventional SAFE formulation can be found in for example [8–10].

For the presentation, we will explicitly differentiate between displacements computed in the physical 3D domain and transformed displacements in the SAFE domain which are introduced in Section 2.2. Displacements in the 3D FE domain (which are assumed to be harmonic) are written as:

$$u_x(x,y,z,t) = u_x(x,y,z)e^{j\omega t} \tag{1}$$

$$u_y(x,y,z,t) = u_y(x,y,z)e^{j\omega t} \tag{2}$$

$$u_z(x,y,z,t) = u_z(x,y,z)e^{j\omega t} \tag{3}$$

where x, y and z are the global Cartesian coordinates, u_x, u_y and u_z are displacements in the x, y and z directions, respectively and ω is the angular frequency in time t, and j is the imaginary unit.

2.1. Piezoelectric finite element formulation

Piezoelectric transducers are often used to excite guided waves due to their ability to drive high frequencies. The formulation of conventional 3D finite elements is well know and will therefore not be presented here. Instead, only salient aspects of the piezoelectric implementation are presented. The standard piezoelectric finite element implementation is employed, as originally proposed by Allik et al. [21].

The coupled constitutive piezoelectric relations can be written as:

$$\begin{aligned} \boldsymbol{\sigma}_u &= \boldsymbol{c}_E \boldsymbol{\epsilon}_u - \boldsymbol{e}^T \boldsymbol{\epsilon}_\phi, \\ \boldsymbol{\sigma}_\phi &= \boldsymbol{e} \boldsymbol{\epsilon}_u + \boldsymbol{p}_S \boldsymbol{\epsilon}_\phi, \end{aligned} \tag{4}$$

where $\boldsymbol{\sigma}_u$ represents the mechanical stress tensor while $\boldsymbol{\sigma}_\phi$ is the electric flux density, which is the electrical equivalent of stress. The strain is given by $\boldsymbol{\epsilon}_u$ while the electrical equivalent of strain is the electrical field $\boldsymbol{\epsilon}_\phi$ which is computed as the negative of the potential spacial gradient. The third order piezoelectric coupling tensor relating displacements u and potentials ϕ is denoted $\boldsymbol{e}$. The mechanical elasticity and dielectric constitutive matrices are represented by $\boldsymbol{c}_E$ and $\boldsymbol{p}_S$ respectively.

The harmonic response is computed by solving the linear system of equations which results from the finite element formulation, written as:

$$\begin{bmatrix} \boldsymbol{D}_t & \boldsymbol{K}_{u\phi} \\ \boldsymbol{K}_{u\phi}^T & \boldsymbol{K}_{\phi\phi} \end{bmatrix} \begin{Bmatrix} \boldsymbol{U} \\ \boldsymbol{\Phi} \end{Bmatrix} = \begin{Bmatrix} \boldsymbol{F} \\ \boldsymbol{Q} \end{Bmatrix} \tag{5}$$

where $\boldsymbol{U}$ and $\boldsymbol{\Phi}$ are the assembled nodal displacements and electrical potentials respectively and $\boldsymbol{F}$ and $\boldsymbol{Q}$ represent assembled forces and charges respectively. The stiffness matrix is made up of terms relating only to electrical properties $\boldsymbol{K}_{\phi\phi}$, those coupling electrical and mechanical properties $\boldsymbol{K}_{u\phi}$ and the frequency dependant dynamic stiffness of the transducer relating only to mechanical properties:

$$\boldsymbol{D}_t = \boldsymbol{K}_{uu} - \omega^2 \boldsymbol{M}. \tag{6}$$

These equations are partitioned into known and unknown degrees of freedom in order to solve unknown displacements and potentials as well as reaction forces and charges. If the model is of a transducer consisting of elastic and piezoelectric parts, electric potentials of elastic parts are simply prescribed to be zero.