Finite Elements in Analysis and Design 89 (2014) 1–7

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

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

A robust spot weld model for structural vibration analysis

R.O. De Alba Alvarez [a,b,*], N.S. Ferguson [a], B.R. Mace [a,c]

[a] Institute of Sound and Vibration Research, University of Southampton, Highfield, Southampton SO17 1BJ, UK
[b] ESI North America, 32605 W 12 Mile Road, Suite 350, Farmington Hills, MI, 48334, USA
[c] Department of Mechanical Engineering, University of Auckland, New Zealand

ARTICLE INFO

Article history:
Received 31 January 2013
Received in revised form
16 April 2014
Accepted 18 April 2014
Available online 19 June 2014

Keywords:
Finite element analysis
Spot weld
Sensitivity
Structural dynamics
Robustness

ABSTRACT

A finite element spot weld is proposed. The model is only weakly sensitive to element size, in contrast to some existing models, for which predictions of the static and dynamic responses can be strongly sensitive to the size of the elements in the substructures to which the spot weld is connected, to such an extent that numerical results may not converge. The proposed model comprises a number of multipoint constraint connections to the attached substructures, so that they may have incompatible meshes. It involves stiffness elements distributed around the perimeter of the spot weld. The case of two plates connected by three spot welds is considered. Numerical results are presented and compared with those of CWELD models and with experimental measurements. The results from the proposed spot weld model show good accuracy, low sensitivity to the element dimensions and good convergence properties.

1. Introduction

The spot weld is one of the most important structural joints in the automotive industry; a vehicle body typically contains thousands of spot-welds. The finite element (FE) method can be used to analyze spot welded structures and several models have been proposed in the literature. However, there are still issues in the application of these models.

The detailed representation of the spot weld using solid elements has often been used as a benchmark in static analysis [1–3]. In this way, a smooth and reliable stress field is predicted. However the model may poorly estimate the interaction forces between the spot weld and the structure. Therefore, when they are used in dynamic analyses, these elements can result in differences between experimental results, for example when using brick element representations of different dimensions which produce non-physical sensitivities in the dynamic characteristics such as the natural frequencies [4–5]. Furthermore, such a model of a single spot weld involves many degrees of freedom (DOFs), so that to model each of the spot welds in detail in a large structure leads to a major increase in model size.

On the other hand, in simplified approaches the spot weld is modeled using an elastic component that is attached to the substructure DOFs in general in one of two ways: (1) by directly connecting the joint nodes to nodes in the substructures (node-to-node connection) and (2) using interpolation elements or multipoint constraints (MPCs) to connect the joint nodes to the substructures. Here, the parameters of the connecting element represent the stiffness characteristics of the joint and therefore their influence on the rest of the structure.

The node-to-node connection requires coincident meshes; if the location of the joint changes, then the mesh of both surfaces needs to be modified. Moreover, the stiffness is generally underestimated [5–6] leading to inaccurate results. In contrast, when interpolation elements or MPCs are used to connect the elastic component to the substructures (solid, beam or springs) [7–8], the connection can be placed at any location using the existing surface meshes. This latter feature offers a great advantage, since it is then possible to assemble components with different mesh characteristics or to assemble components with complex geometries, for which it is very difficult to have coincident nodes.

Unfortunately, when these elements are used, it has been shown that they have a high sensitivity to the element size [9–10]. Moreover, Palmonella et al. [9] identified the element area as a parameter that can be updated in order to reduce the error in the prediction of dynamic properties in a FE model when compared to experimental measurements. It has also been found that for dynamic predictions, some of the lowest natural frequencies do not converge even if the element size is much smaller than the wavelength [6,11].

In this paper the cause of the large sensitivity to element size is discussed and an alternative spot weld model is proposed. The sensitivity, which often leads to poor convergence or, indeed,

* Corresponding author.

http://dx.doi.org/10.1016/j.finel.2014.04.010
0168-874X/

failure to converge at all, is a result of singularities (particularly associated with the rotational DOFs) that occur when point loading is applied, as opposed to loading distributed over an area. Point loading implies infinite stresses at that point and possibly infinite response, even in continuous models of the system. The FE model proposed overcomes this issue by modeling the spot weld as an array of springs. It provides a good physical representation of the spot weld and the forces at the connections are distributed over an area, imposing a surface to surface link between the connected substructures. The model is robust to changes in the mesh size and coincident meshes are not required.

In the following section the sensitivity of spot weld models to element size is discussed and demonstrated by application to an example of two simply supported plates with a single connection. In Section 3 a spot weld model robust to element size is proposed. In Section 4 the application of the new spot weld model is demonstrated in a model of two simply supported plates with three point connections. The numerical results from the proposed spot weld model can then be compared to the numerical evaluation of a coupled analytical benchmark model of two simply supported plates connected by point springs formulated using an assumed modes and mobility approach. In order to evaluate the performance of the proposed element, mesh sensitivity and convergence are evaluated. The resulting natural frequencies are compared to experimental measurements. Finally, conclusions are given in Section 5.

2. Sensitivity to element size

2.1. Plate stiffness matrix formulation

In this section the dependence of the diagonal terms in the stiffness matrix of a Heterosis plate element on element size is discussed. When the out-of-plane behavior is studied, it is seen that the terms associated with the rotational DOFs are highly sensitive to the element size.

When two plates are connected using a conventional spot weld model, constant stiffness values are added to the diagonal terms of the DOFs, both translational and rotational, involved in the connection. The magnitudes of the stiffnesses associated with rotational plate DOFs depend on the element size, while the added rotational stiffness due to the spot weld does not, resulting in natural frequencies and/or dynamic or static solutions that are sensitive to element size. This is a problem that results from the stress singularities that arise from truly point loading, as opposed to loading that is distributed over a finite area. Point loading implies infinite stresses at the point of application of the load and infinite stress gradients. Note that these singularities may be weak in the sense that the response – displacements and rotations – need not be infinite. For example, consider an infinite thin Kirchhoff plate excited by a time harmonic point force. The shear stress at a distance ε from the excitation point is proportional to ε^{-1}, while the response is bounded, the input mobility (velocity per unit force) being $1/8\sqrt{D\sigma}$ ([12], p. 255), where D and σ are the bending stiffness and mass per unit area respectively. For moment excitation, on the other hand, the rotational input mobility (rotational velocity per unit moment) is $\omega(1-i\Gamma)/16D$, where Γ depends on the nature of the moment. For two point forces separated by a distance $2a$, with $a\to 0$, then $\Gamma = 4\ln(\gamma ka/2)/\pi$, $\gamma = 1.781\ldots$ and thus involves a logarithmic singularity ([12], p. 275). The same issues arise in discrete, FE models of systems, which furthermore suffer from the difficulty in approximating the stress field around the excitation point using (typically) polynomial approximations.

To illustrate the sensitivity to element size, consider the Heterosis plate element [13]. This is a plate element derived from the Mindlin–Reissner plate theory, used to describe the behavior of thick plates. However, as reduced order integration is used to evaluate the shear stiffness matrix, this element does not suffer from shear locking, possesses correct rank and can be applied to both thick and thin plates.

This element has 9 nodes: 4 corner, 4 mid-side and one central node (see Fig. 1). The central node has two rotational DOFs and each other node has 5 DOFs which describe in-plane and out-of-plane motion. Drilling DOFs (i.e. rotation about plate normal axis) are not included. There are 42 DOFs in total.

The displacement field within the element is interpolated using serendipity basis functions, whilst the rotations in the x and y directions are interpolated using Lagrange basis functions. The out-of-plane co-ordinates (w, θ_x, θ_y) of a point within the element are given by

$$w(x,y) = \sum_{i=1}^{8} N_i(x,y) w_i; \quad \theta_x(x,y) = \sum_{i=1}^{9} P_i(x,y)\theta_{x,i};$$

$$\theta_y(x,y) = \sum_{i=1}^{9} P_i(x,y)\theta_{y,i} \tag{1}$$

where the subscript i indicates the node number and P_i and N_i are Lagrange and serendipity basis functions. These basis functions are described in terms of the normalized co-ordinates

$$\xi = \frac{2x - s_x}{s_x} \text{ and } \eta = \frac{2y - s_y}{s_y} \tag{2}$$

where s_x and s_y are the lengths of the sides of the element in the x and y directions respectively. Eq. (1) can be written in matrix

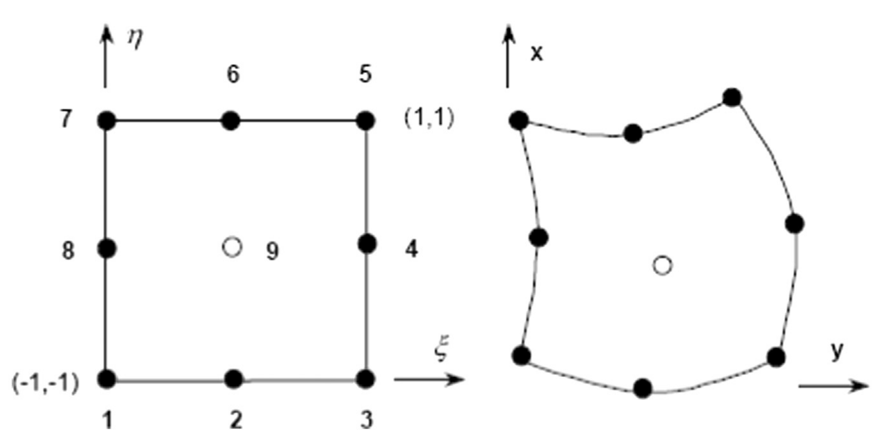

Fig. 1. A single Heterosis finite element.