

Finite Element Model for Spot Welds Using Multi-Point Constraints and its Dynamic Characteristics

2011-01-1697

Published
05/17/2011

Fumiyasu Kuratani, Kazuhei Matsubara and Takashi Yamauchi
University of Fukui

Copyright © 2011 SAE International

doi:10.4271/2011-01-1697

ABSTRACT

In this paper, we focus on the ACM2 (Area Contact Model 2) model as a simplified spot weld model suitable for vibration analysis and reveal its dynamic characteristics. First, the theoretical background of the multi-point constraint (MPC) used in the ACM2 model is explained. Next, we examine in detail the effect of the mesh pattern surrounding a spot weld on the modal properties (natural frequency and mode shape) of a spot welded structure. Finally, an appropriate mesh size in the area of the spot weld for the ACM2 model is presented. As an example, we used two steel plates joined by three spot welds.

The results show that the configuration and size of the patch (group of shell elements in the ACM2 model) significantly affect the modal properties of the model. When the centers of the patch and the solid element that represent the spot weld are coincident, the natural frequencies monotonically increase with the patch size. When the centers of the patch and the solid element are not coincident and the shell element size is large, the natural frequencies vary widely. When the patch is composed of shell elements whose size is the same or smaller than the solid element size determined by the spot weld diameter, the variation in the natural frequency is small.

INTRODUCTION

Spot welds are widely used in the automotive industry to join thin sheet metal. Since a typical automotive body contains several thousand spot welds, its structural behavior is strongly affected by the characteristics of spot welded joints. Appropriate modeling of spot welded joints is essential when analyzing structural behavior using finite element analysis (FEA). For the stress analysis of spot welded joints, a very detailed model is necessary and various finite element (FE)

models have been proposed [1,2]. For the vibration analysis of a large scale model such as an automotive body, the use of a detailed model is impractical because a model with large degrees of freedom (DOF) leads to excessive computational effort. Thus the major requirements of spot weld models for vibration analysis are to represent the stiffness of spot welds with small DOF and to predict accurately their influence on the rest of the structure. In addition, it is important that spot weld models are easy to generate and the locations of spot welds are easily modified. That is, the joined metal plates are meshed independently of the spot weld locations and the spot welds can be located anywhere in the mesh of the plates.

For the above requirements, Heiserer et al. [3] proposed the model known as ACM2 (Area Contact Model 2) and Jonscher et al. [4] proposed the CWELD model. The ACM2 model was developed using the finite element program NASTRAN and is the most commonly used by Japanese automotive manufacturers. For the ACM2 model, a spot weld is modeled with a single solid element and the plates surrounding the spot weld are modeled with shell elements. The solid element node and the shell element nodes are connected using multi-point constraints (MPC). This enables the spot welds to be located anywhere in the mesh of the plates. Some spot weld models, including the ACM2 model, were examined for their effect on the vibration properties of a spot welded structure [5,6,7]. However, the dynamic characteristics of the ACM2 model remain to be fully elucidated.

In this paper, we focus on the ACM2 model as a simplified spot weld model for vibration analysis and reveal its dynamic characteristics. First, the theoretical background of the MPC used in the ACM2 model is explained. Next, we examine in detail the effect of the mesh pattern surrounding a spot weld on the modal properties (natural frequency and mode shape)

of a spot welded structure. This determines the factors of the MPC equations. Finally, an appropriate mesh size in the area of the spot weld is presented. In this paper, using the finite element program ANSYS we create a finite element model for spot welds with the same features as the ACM2 model. As an example, we used two steel plates joined by three spot welds.

SPOT WELD MODEL

In the vibration analysis of a large scale spot welded structure, a simplified model for spot welds is needed. However, a model using a single beam element or a single solid element to represent a spot weld requires the matching of meshes of different plates at the spot weld connection points. This results in excessive modeling effort. In addition, a single beam model underestimates the stiffness of a real spot welded joint due to the point-to-point joint [6]. Thus the major requirements of spot weld models are to represent the stiffness of spot welds with small DOF and to predict their influence on the rest of the structure accurately. In addition, the joined metal plates are meshed independently of the spot weld locations and the spot welds can be located anywhere in the mesh of the plates.

The CWELD and the ACM2 models meet the above requirements. The CWELD model is provided as the CWELD element in NASTRAN while the ACM2 model is generated using several types of elements in NASTRAN. The ACM2 model is the most commonly used by Japanese automotive manufacturers.

ACM2 MODEL

In this paper, we focus on the ACM2 model as a spot weld model as shown in Figure 1. For the ACM2 model, a spot weld is modeled with a single solid element and the upper and lower plates are modeled with shell elements. The solid element node and the shell element nodes are connected using RBE3 (Rigid Beam Element 3) elements. This enables spot welds to be located anywhere independently of the mesh of the plates. The RBE3 element is available in NASTRAN and generates MPC equations that define the relationship between the solid element node and the shell element nodes. The factors of MPC equations are determined on the basis that the translational force applied to the solid element node is distributed to the shell element nodes. The group of shell elements in the ACM2 model is called a patch.

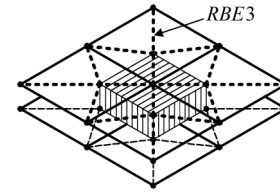


Figure 1. ACM2 model.

MPC EQUATION USING THE RBE3 ELEMENT

The MPC equation defines a linear relationship between the displacement of nodes within a FE model. Figure 2 shows the connections between one solid element node and four shell element nodes with four RBE3 elements. Since the five nodes in the ACM2 model lie in the same plane, the translations in the x , y and z directions at the solid element node are represented by the translations in the x , y and z directions at the four shell element nodes using MPC equations, that is

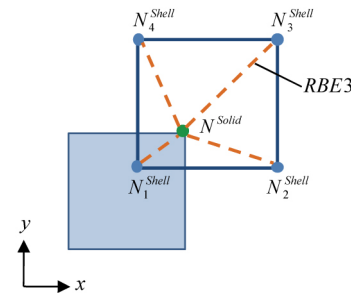


Figure 2. Solid element node and shell element nodes connected with RBE3 elements.

$$\begin{aligned} u_x^{Solid} &= \sum_{i=1}^4 \alpha_{xi} u_{xi}^{Shell} + \sum_{i=1}^4 \alpha_{yi} u_{yi}^{Shell} \\ u_y^{Solid} &= \sum_{i=1}^4 \beta_{xi} u_{xi}^{Shell} + \sum_{i=1}^4 \beta_{yi} u_{yi}^{Shell} \\ u_z^{Solid} &= \sum_{i=1}^4 \gamma_{zi} u_{zi}^{Shell} \end{aligned}$$

(1)

where u_x^{Solid} , u_y^{Solid} , u_z^{Solid} are the translations of the solid element node. u_x^{Shell} , u_y^{Shell} , u_z^{Shell} are the translations of i th shell element node. α_{xi} , α_{yi} , β_{xi} , β_{yi} , γ_{zi} are the factors of the MPC equations. The mathematical specification for the MPC equation generated by the RBE3 element is undocumented. We present a method for determining the factors of the MPC equations based on Nastran Reference Manual [8].

The MPC equations in Equation (1) are defined in the three translational directions. The case of α_{xi} and α_{yi} is demonstrated here. An in-plane force in the x direction is applied to the solid element node as shown in Figure 2.

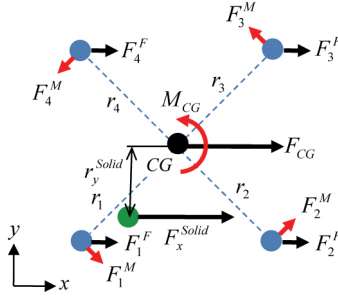


Figure 3. Concept of force distribution.

The force applied to the solid element node is transferred to the center of gravity (CG) of the four nodes of the shell element. This transfer produces the equivalent force F_{CG} and the equivalent moment M_{CG} at the CG, that is

$$\begin{aligned} F_{CG} &= F_x^{Solid} \\ M_{CG} &= r_y^{Solid} F_x^{Solid} \end{aligned} \quad (2)$$

where r_y^{Solid} denotes the y component of the distance between the CG and the solid element node F_{CG} is divided and applied to each shell element node. The distributed force F_i^F at the i th node is given by

$$F_i^F = \left(\frac{w_i}{\sum w_k} \right) F_{CG} = \left(\frac{w_i}{\sum w_k} \right) F_x^{Solid} \quad (3)$$

where w_i is a compensation factor and usually set to 1. M_{CG} is divided and applied as a force to each shell element node.

The distributed force F_i^M at the i th node is given by

$$F_i^M = \left(\frac{w_i r_i}{\sum w_k r_k^2} \right) M_{CG} = r_y^{Solid} \left(\frac{w_i r_i}{\sum w_k r_k^2} \right) F_x^{Solid} \quad (4)$$

where r_i is the distance from the CG to the i th node. The total force distributed to each shell element node is obtained by computing the sum of the F_i^F and F_i^M vectors. Dividing the resultant force into the x and y components, we obtain

$$\begin{Bmatrix} F_x^{Shell} \\ F_y^{Shell} \end{Bmatrix} = \begin{Bmatrix} \alpha_x \\ \alpha_y \end{Bmatrix} F_x^{Solid} \quad (5)$$

where

$$\begin{aligned} \{F_x^{Shell}\} &= \{F_{x1}^{Shell} \ F_{x2}^{Shell} \ F_{x3}^{Shell} \ F_{x4}^{Shell}\}^T \\ \{F_y^{Shell}\} &= \{F_{y1}^{Shell} \ F_{y2}^{Shell} \ F_{y3}^{Shell} \ F_{y4}^{Shell}\}^T \\ \{\alpha_x\} &= \{\alpha_{x1} \ \alpha_{x2} \ \alpha_{x3} \ \alpha_{x4}\}^T \\ \{\alpha_y\} &= \{\alpha_{y1} \ \alpha_{y2} \ \alpha_{y3} \ \alpha_{y4}\}^T \end{aligned} \quad (6)$$

with

$$\begin{aligned} \alpha_{xi} &= \left(\frac{w_i}{\sum w_k} \right) \pm r_y^{Solid} \left(\frac{w_i r_i}{\sum w_k r_k^2} \right)_x \\ \alpha_{yi} &= \pm r_y^{Solid} \left(\frac{w_i r_i}{\sum w_k r_k^2} \right)_y \end{aligned} \quad (7)$$

The second term of α_{xi} is for the x component of F_i^M and α_y is for the y component of F_i^M . The sign is determined by the relative position of the i th node with respect to the CG.

The force distributed to each shell element node has a distinct quality as follows: In [Figure 3](#), for the shell element nodes on the same side as the solid element node with respect to the CG (lower side), a large force is distributed in the x direction because F_i^F and the x component of F_i^M are both positive. For the shell element nodes on the upper side, a small force is distributed because F_i^F and the x component of F_i^M have opposite signs. This indicates that a large force in the x direction is distributed to the shell element node close to the solid element node. For the distributed force in the y direction, the magnitude of the force is approximately the same at the four shell element nodes, but the forces at the nodes on the left and right with respect to the CG, have opposite signs.

α_{xi} and α_{yi} in [Equation \(7\)](#) are used as the factors of the MPC equation. For other cases where a force is applied in the y or z direction to the solid element node, the factors can be determined in the same way.

EQUATION OF MOTION

For a whole spot welded structure, the equations of motion can be written as

$$[M]\{\ddot{u}\} + [K]\{u\} = \{0\} \quad (8)$$

where $[M]$ and $[K]$ are the mass and stiffness matrices and $\{u\}$ is the nodal displacement vector for the whole structure. Using [Equation \(1\)](#), we write the relationship between $\{u\}$ and $\{u^A\}$ as

$$\{u\} = \begin{Bmatrix} u^A \\ u^O \end{Bmatrix} = \begin{Bmatrix} I \\ C_O \end{Bmatrix} \{u^A\} = [C]\{u^A\} \quad (9)$$

where $\{u^O\}$ are the translations of the solid element nodes and $\{u^A\}$ are the translations and rotations excluding $\{u^O\}$. $[I]$ is the identity matrix and $[C_O]$ is the matrix that consists