이종재료 금속조인트의 굽힘에 의한 잔류응력 해석

백태현*(군산대학교 기계공학부)
정걸, 박태근(군산대학교 대학원 기계공학과)

Residual Stress Analysis in Bi-material Metal Joint under Bending Moment by Finite Element Method

Tae Hyun BAEK (Mech. Eng. Dept., KNU)
Girl Jung, Tae Geun Park (Mech. Eng. Dept., Grad. School, KNU)

ABSTRACT

It was observed that after unloading or removal of the load from the specimen subjected to bending stress, partial or full elastic spring back occurred and considerable stresses have resulted while plastic deformation was considered. ABAQUS is a suite of powerful engineering simulation programs, based on the finite element method. In this paper, it was used as the main tool to analyze elastic and plastic deformations of bi-material metal joint. In the case of elastic deformations, the results were comparable to the theoretical data. Plastic deformations and residual stresses of bi-material metal joint under bending moment were obtained by ABAQUS; where the theory needs to be studied and improved further to verify the results.

Key Words: Finite element method (유한요소법), Bi-material metal joint (이종재료 금속조인트), Elastic deformation (탄성변형), Plastic deformation (소성변형), Residual stress (잔류응력).

1. INTRODUCTION

Bending is probably the most common type of deformation encountered in engineering structures. Partial or full elastic spring back may occur and considerable stresses may result in bi-material metal joint after unloading the moment in the case of plastic deformations. ABAQUS is a suite of powerful engineering simulation programs, based on the finite element method. It was used as the main tool to analyze elastic and plastic deformations of bi-material metal joint. To develop an understanding of the mechanics in bi-material metal joint under bending moment, a simple case has been treated in this paper. The residual stresses of bi-material metal joint under bending moments after elastic deformations were simulated by ABAQUS and the results were comparable to the theoretical data. The analytical theory on plastic deformations of bi-material metal joint under bending moment and the residual stresses after unloading is still not as perfect as that of homogeneous material. Therefore, in this paper, ABAQUS was introduced to analyze the mentioned problem, where the theory needs to be studied and improved further to verify the results. It was observed that the points where the residual stresses were zero weren't on the neutral surface but appeared to move up exceeding the geometric central surface of the bi-material metal joint.

2. THEORETICAL FORMULATION

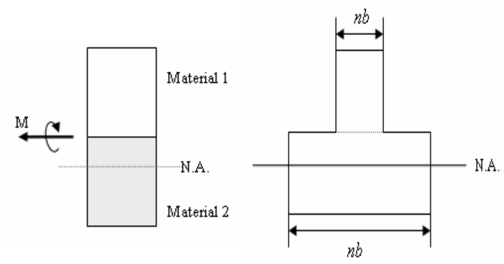
2.1 Stresses in elastic range of bi-material metal joint

A cross section of bi-material metal joint is shown in Fig. 1. The normal strain ε_x varies linearly with the distance *c* from the neutral axis of the section as shown in Fig. 2a. Here, ρ is the curvature measured from the mid-plane, and E_1 , E_2 are moduli of elasticity of material 1 and material 2, respectively. Also, the resistance to bending of the member would remain the same if both element portions were made of the first material by transforming the cross section as shown in Fig. 1b where *n* is equal to E_2 / E_1, and then Eq. (1) could be used to determine the normal stresses, here *c* is the distance from the neutral surface, and *I* is the moment of inertia of the transformed section with respect to its centroidal axis.

$$\sigma_x = -\frac{Mc}{I} \tag{1}$$

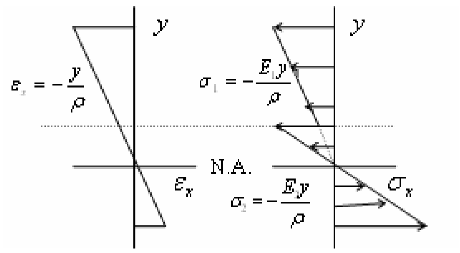
To obtain the stress σ_1 at a point located in the upper portion of the cross section of the original composite bar, the stress σ_x at the corresponding point of the transformed section can be computed by using Eq. (1). However, to obtain the stress σ_2 at a point in the lower portion of the cross section, Eq. (1) should be multiplied by n to compute he stresses at the corresponding point of transformed section as shown in Fig. 2b. Thus, as shown in Eq. (2).

$$\sigma_2 = n\sigma_x \tag{2}$$



(a) (b)

Fig. 1 Cross section of bi-material member



(a) (b)

Fig. 2 Strain and stress distribution in bi-material member

2.2 Stresses in plastic range of homogenous material member

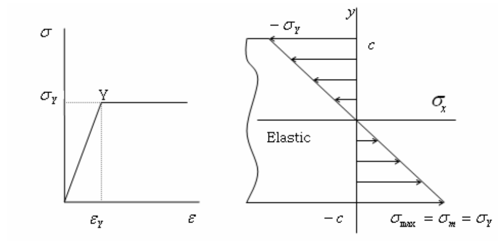
As the bending moment increases to M_y, which is referred to as the maximum elastic moment, as shown in Fig. 3, the value of the bending moment M_y at the onset of yield is:

$$M_Y = \frac{I}{c}\sigma_Y \tag{3}$$

As the bending moment further increases, plastic zones develop in the member, as shown in Fig. 4a, the stress uniformly equal to the $-\sigma_Y$ in the upper zone and to $+\sigma_Y$ in the lower zone. In this case, the moment is

$$M = -b\int_{-c}^{c} y\sigma_y dy \tag{4}$$

Here b is the width of the member. As M increases, the plastic zones expand until, at the limit, the deformation is fully plastic.

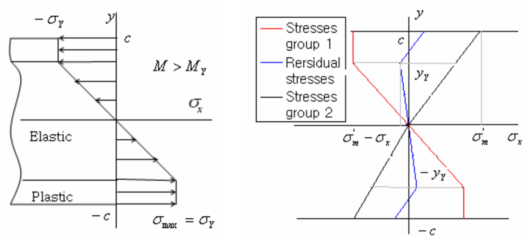


(a) (b)

Fig. 3 Stress-strain diagram and onset of maximum elastic moment

2.3 Residual stresses of homogenous material member after unloading bending moment

On one hand, the stresses due to the application of the given bending moment M and, on the other, the reverse stresses due to the equal and opposite bending moment $-M$ which is applied to unload the member. The first group of stresses reflects the elastoplastic behavior of the material during the loading phase and the second group the linear behavior of the same material during the unloading phase. The distribution of residual stresses in the member can be obtained by adding two groups of stresses as shown Fig. 4b [1].



(a) (b)

Fig. 4 Normal and residual stresses distribution in the member with plastic zones

3. COMPUTER SIMULATION

ABAQUS is a suite of powerful engineering simulation programs, based on the finite element method that can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations [2]. The stated problems were simulated by ABAQUS in this paper. Materials properties involved in this paper are listed in Table 1.

Table 1 Properties of involved materials

Material	Modulus of Elasticity	Yield Strength
Steel	210 GPa	250 MPa
Rigid plate	210E+15 Pa	$+\infty$
Aluminum	70 GPa	230 MPa

To avoid the misleading convergence and nonlinearity during analysis in ABAQUS, two rigid plates having the same cross section as the model with 0.02 *m* thickness were attached to model's both ends. Pressure was applied on the upper surface of the right end rigid plate while the left end plate was fixed at the cross section to simulate the bending moment. It should be noted that the color ranges which indicate the magnitudes of stresses in individual image such as in Figs. 6, 7, 9, 10, 12 and 13, have no relationship with other images in this paper.

3.1 Stresses in elastic range of bi-material metal joint

The model's dimensions are shown in Fig. 5. Pressure was 21.667 *MPa*. This load could not cause the plastic deformations. Results are shown below.

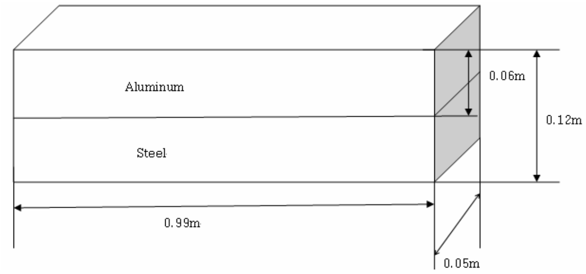


Fig. 5 Dimensions of bi-material metal joint

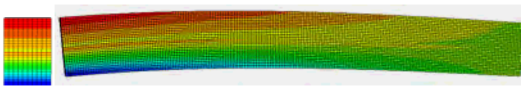


Fig. 6 Contour plot of bending stresses under load in elastic range of bi-material metal joint (deformation scale factor: 11.2824)

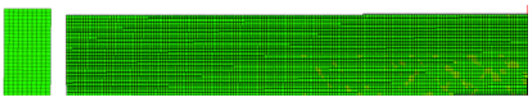


Fig. 7 Contour plot of residual stresses in elastic range of bi-material metal joint (deformation scale factor: 1)

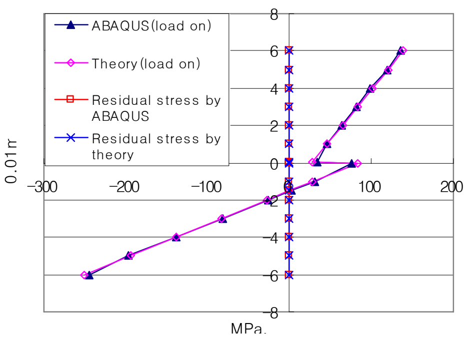


Fig. 8 Stress distributions in elastic range of bi-material metal joint

3.2 Stresses in bi-material metal joint with one material in elastic range and the other in plastic range

Pressure was 33.025 *MPa*. In this case, the maximum stress in steel part exceeds the yield stress while the aluminum part's still in the range of elastic deformations. Results obtained by ABAQUS are shown below.

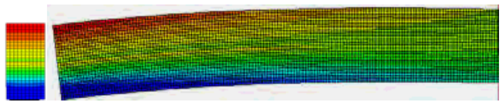


Fig. 9 Contour plot of bending stresses under load in bi-material metal joint with one material in elastic range and the other in plastic range (deformation scale factor: 7.12569)

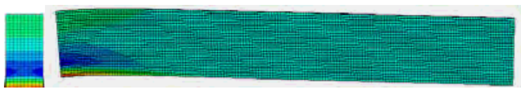


Fig. 10 Contour plot of residual stresses in bi-material metal joint with one material in elastic range and the other in plastic range (deformation scale factor: 190.804)

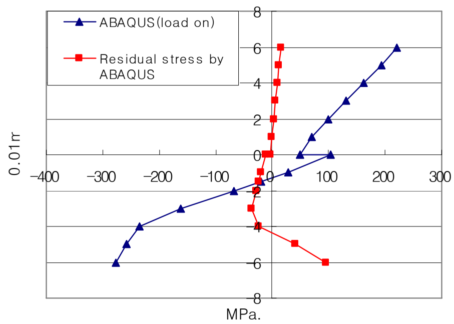


Fig. 11 Stress distributions in bi-material metal joint with one material in elastic range and the other in plastic range

3.3 Stresses in bi-material metal joint with both materials in plastic range

Pressure was 40.2725 *MPa*. In this case, both materials have plastic deformations. Results obtained by ABAQUS are shown below.

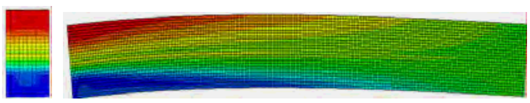


Fig. 12 Contour plot of bending stresses under load in bi-material metal joint with both materials in plastic range (deformation scale factor: 5.22755)

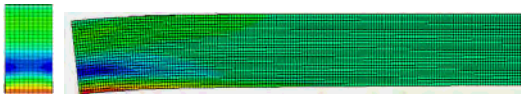


Fig. 13 Contour plot of bending stresses under load in bi-material metal joint with both materials in plastic range (deformation scale factor: 37.6644)

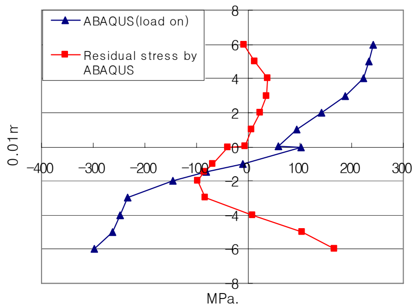


Fig. 14 Stress distributions in bi-material metal joint with both materials in plastic range (deformation scale factor: 37.6644)

3.4 Stresses in bi-material metal joint along different bending moment

From the plots of stresses in Figs. 15 and 16, as *M* increases, it is observed that the normal stresses due to the application of the given bending moment *M* will increase until it reach the yield stresses, and then, they will be in uniform to the yield stresses in the plastic deformation zones. Also, the residual stresses increase with the increase of *M*, and the points where the residual stresses are zero are not on the neutral surface but observed to move up exceeding the geometric central surface, this needs further theoretical study and research to verify the result.

4. CONCLUSIONS

From the presented analysis, combined with the theoretical results and computer simulations, the conclusions below were drawn.

Finite element method and the software based on it can be used to analyze the residual stresses in bi-material joint under bending moment. From the sections 3.1, the results obtained by ABAQUS were comparable to the theoretical data, while in cases 3.2 and 3.3 still need further theoretical study to verify its integrity. Because it was observed that the transformation section method introduced in section 2.1 wasn't valid to bi-material metal joint if plastic deformations zone developed in either or both materials. The residual stresses increase with increase of *M*, and the points where the residual stresses were zero weren't on the neutral surface but appeared to move up exceeding the geometric central surface as shown in Fig. 16.

Moreover, it was found that although the materials have been assumed to be idealized elastoplastic materials, this means they have the stress-strain relationships like Fig. 3a, the S.S1 of plastic deformation zones obtained by ABAQUS still had relative low slowly increments, this misleading results occurred because of the algorithm that ABAQUS/CAE used to create contour plots for element variables, such as stress. The contouring algorithm requires data at the nodes; however, ABAQUS/CAE calculates element variables at the integration points. ABAQUS/CAE calculates values of element variables by extrapolating the data from the integration points to the nodes. If the differences in Mises stress values fall within the specified averaging threshold, nodal averaged Mises stresses are calculated from each surrounding element's invariant stress value. Invariant values exceeding the elastic limit can be produced by extrapolation process. Only the Mises stress is required to have a magnitude less than or equal to the value of the current yield stress while the individual stress components may have magnitudes that exceed the value of the current yield stress [2].

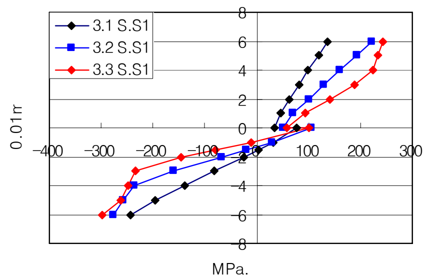


Fig. 15 Bending stress distributions by different bending moment

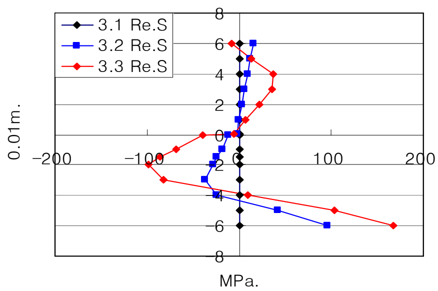


Fig. 16 Residual stress distributions by different bending moment

ACKNOWLEDGEMENT

This work was supported by grant from 2005 Sanhak Research Consortium of the Small and Medium Business Administration of the Republic of Korea.

Reference

[1]. Ferdinand P. Beer and E, Russell Johnston, JR. (1992) "Mechanics of materials," McGraw-Hill Inc., pp. 183-228, 1992.

[2]. "Getting Started with ABAQUS/Standard," Karlsson & Sorensen. Inc., 2002