International Journal of Automotive Technology, Vol. 7, No. 3, pp. 329−336 (2006)
Copyright © 2006 KSAE
1229−9138/2006/027−14

EVALUATION OF THE FINITE ELEMENT MODELING OF A SPOT WELDED REGION FOR CRASH ANALYSIS

J. H. SONG[1)], H. HUH[1)*], H. G. KIM[2)] and S. H. PARK[2)]

[1)]Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology, Daejeon 305-701, Korea
[2)]Automotive Steel Research Center, POSCO, Gwangyang, Jeonnam 545-090, Korea

(Received 25 October 2005; Revised 22 December 2005)

ABSTRACT−The resistance spot-welded region in most current finite element crash models is characterized as a rigid beam at the location of the welded spot. The region is modeled to fail with a failure criterion which is a function of the axial and shear load at the rigid beam. The calculation of the load acting on the rigid beam is important to evaluate the failure of the spot-weld. In this paper, numerical simulation is carried out to evaluate the calculation of the load at the rigid beam. At first, the load on the spot-welded region is calculated with the precise finite element model considering the residual stress due to the thermal history during the spot welding procedure. And then, the load is compared with the one obtained from the model used in the crash analysis with respect to the element size, the element shape and the number of imposed constraints. Analysis results demonstrate that the load acting on the spot-welded element is correctly calculated by the change of the element shape around the welded region and the location of welded constrains. The results provide a guideline for an accurate finite element modeling of the spot-welded region in the crash analysis of vehicles.

KEY WORDS : Resistance spot weld, Failure load, Electro-thermal analysis, Crashworthiness, Finite element modeling

1. INTRODUCTION

Simulation of vehicle crashes using the finite element method has made remarkable advances in the last several years. Most automotive companies now utilize CAE to perform virtual crash tests on computers. Engineers can now quickly adjust the structural performance before performing a real crash test. The virtual simulation is so less costly and faster than real crash tests that development time and manufacturing costs could be dramatically reduced. Spot welding is widely applied to join the sheet components in auto-motive industries. A typical vehicle may have over 7000 spot welds. The spot welds contribute a lot for the strength and fatigue life of the whole structure (Deng *et al.*, 2000; Kang, 2005; Lee *et al.*, 2005). The welded spot fails when the load first satisfies one of the failure conditions as the load increases (Zuniga and Sheppard, 1995; Lee *et al.*, 1998). Since spot welds in structural components often fail under the combined loads during vehicle crashes, an accurate calculation of the load action on the spot-welded region and a failure criterion of spot welds are helpful for the crash analysis in the early automotive design stage and it can be implemented into finite element codes for accurate simulation of the crush of spot-welded structural members (LSTC, 2003).

An auto-body is generally expected to contain several thousands of spot welds. As major contributors of a vehicle's structural stiffness and strength, spot-welded joints must be properly represented in structural simulations so that engineering simulation such as crash analysis can be reliably performed. However, due to the existence of a large number of spot welds in a vehicle structure, it is often impractical to model each spot-welded joint in detail in a full scale crash analysis using the finite element method. The resistance spot-welded region in most current finite element crash models is characterized as a rigid beam at the location of the welded spot (Xu and Deng, 2003; Yancey, 2004). The role of this rigid beam is simply to transfer the load across the welded component. A spot-welded element is designed to be failed by some failure criterion which is a function of the axial and shear load action on the rigid beam (Lin *et al.*, 2002; Lin *et al.*, 2003; Langrand and Combescure, 2004). Figure 1 shows the general modeling of the spot-welded region in crash analysis. Since the failure of a spot-welded element is directly governed with the load, the calculation of the load at the

**Corresponding author.* e-mail: hhuh@kaist.ac.kr

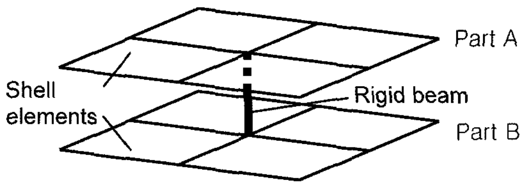


Figure 1. General modeling of the spot welded region in crash analysis.

rigid beam is important to evaluate the failure of the spot weld and, in addition, to obtain the accurate simulation result of vehicle crashes.

In this paper, numerical simulation is carried out in order to evaluate the calculation of the load at the spot-weld region modeled with the rigid beam used in crash analysis. The load calculated from the precise finite element model of the spot-welded region considering the residual stress due to the thermal history during the spot welding process is regarded as the reference value and the value of load is compared with the one obtained from the spot-welded model using the rigid beam with respect to the element size, element shape and number of imposed constraints. First, the effect of mesh size on the load at the rigid beam is examined. And then, the effect of the element shape on the welded components is investigated. Lastly, the location and the number of welded constraints are changed for the correct prediction of the load on the spot-welded element. Analysis results demonstrate that the correct prediction of the load on the spot-welded element is achieved by the change of the element shape and the location of welded constrains.

2. NUMERICAL ANALYIS OF SPOT WELDING PROCESS

The load calculated from a precise finite element model of the spot welding region considering the residual stress due to the thermal history during the spot welding procedure is regarded as the reference value in order to evaluate the load characteristics of spot-welded region modeled with a single rigid beam in crash analysis. In the precise model, electo-thermal finite element analysis is carried out in order to examine the thermal history during the spot welding process and thermo-mechanical analysis is, then, carried out with the temperature distribution obtained from the thermo-electrical analysis in order to calculate the residual stress in the spot-welded region.

2.1. Calculation of Thermal History with Elecro-Thermal Analysis

Electro-thermal finite element analysis is carried out in order to calculate the thermal history during the spot welding process. Since the heat generated for electric resistance spot welding is obtained by charging a large electric current through workpieces which have electric resistance in the domain and contact surface, the amount of the heat generation per unit volume is calculated by the electric potential in the domain and then it is applied to the heat transfer equation to calculate the temperature distribution in the electrodes and workpieces.

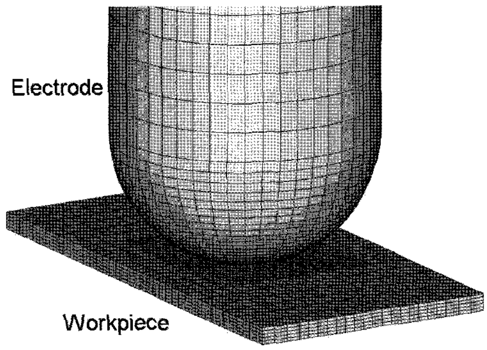
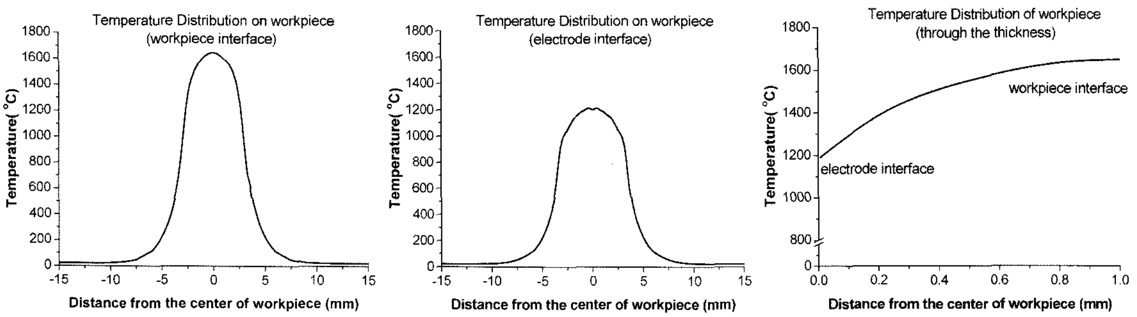


Figure 2. Finite element modeling for the electo-thermal analysis of a spot welding process.

Table 1. Conditions of a spot welding process.

Condition	Two layer sheets (1.0 t)
Nugget diameter	6.0 mm
Current	12000 A
Welding time	15 cycle
Pressure	2.5 kN

Figure 2 shows the finite element model of an electrode and a workpiece used in the analysis. For precise description of temperature and residual stress in spot-welded nugget, fine finite elements are used within and around the nugget and larger elements are used in the outer region. To consider the electric contact resistance, an artificial interface element is used on each contact surface. The finite element mesh is composed of 7319 eight-node brick elements in an electrode and 10432 elements in a workpiece. The electric and thermal properties such as electrical resistance, thermal conductivity, and specific heat are applied with the variation of the temperature in the analysis (Huh and Kang, 1997). The workpiece is assigned to be SPRC 35R and its initial thickness is 1.0 mm. The welding condition for two layer sheets with the thickness of 1.0 mm is explained in Table 1. A constant current of 12000 amperes passes through the electrodes and workpieces during the time of 15 cycles while the holding force of 2.5 kN is imposed on the electrode. The analysis is carried out with the help of a commercial finite element code, ABAQUS/Standard, implicit solving scheme (HSK, 2004).



(a) Workpiece interface (b) Electrode interface (c) Through the thickness

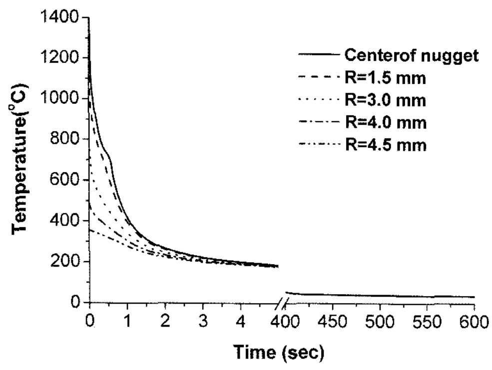
Figure 3. Temperature distribution on a workpiece at the end of the weld process.

Figure 3 shows the temperature distribution of the workpieces at the end of weld. The temperature at the interface between workpieces is higher than that at the interface between an electrode and a workpiece because the electric contact resistance at the interface between the workpieces is larger than that between an electrode and a workpiece. The temperature at the interface between workpieces rises up to 1600°C so that the nugget starts to be formed from the center of workpiece interface.

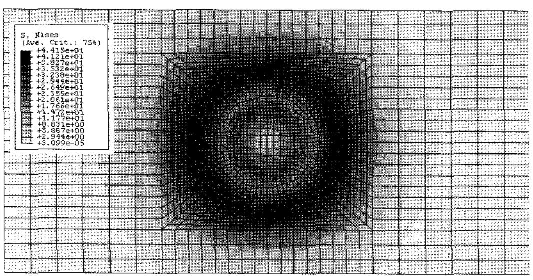
2.2. Calculation of Residual Stress with Thermo-Mechanical Analysis

With the calculated temperature distribution in a workpiece, thermo-mechanical analysis is performed in order to calculate the residual stress in spot-welded region. The temperature at the end of the spot welding process is applied as the initial condition and convectional heat transfer is considered as the boundary condition. The mechanical properties such as the Young's modulus and the Poisson's ratio with the variation of the temperature are applied in the analysis (ASM, 1985).

Figure 4 shows the temperature variation and distribution of von-Mises stress. As time goes on, heat transfer such as conduction and convection proceeds and the temperature of a workpiece decreases to the room temperature. A workpiece has localized high temperature gradient at the center of the nugget. Due to this localized high temperature gradient, the residual stress after the spot welding process is concentrated around the spot-welded nugget. Figure 5 shows the thickness distribution after the welding procedure. The thickness of the workpiece near the nugget is reduced due to the load applied from the electrode during the spot welding process. The thickness is 0.92 mm compared to its initial value of 1.0 mm at the center of the nugget.



(a)Temperature distribution



(b)Residual stress in a workpiece(von-Mises stress)

Figure 4. Results of thermo-mechanical analysis.

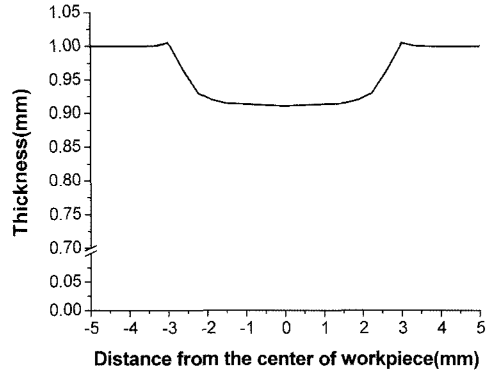


Figure 5. Thickness distribution in a workpiece after the spot welding process.

3. CALCULATION OF LOAD ON THE SPOT-WELDED REGION IN THE PRECISE MODEL

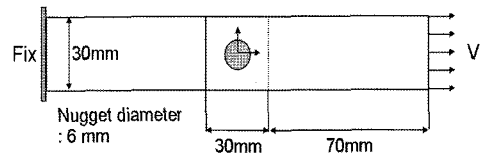
This paper is concerned with the evaluation of the load at the spot-weld region modeled with the rigid beam used in crash analysis. The load calculated from the precise finite element model of the spot-welded region is regarded as the reference value and the value of load is compared with the one obtained from the spot-welded model using the rigid beam. The load on the spot weld is calculated with the precise finite element model of the spot-welded region considering the residual stress due to the thermal history during the spot welding procedure. Since the failure of a spot weld is governed with shear and normal loads, two kinds of specimens with different loading type shown in Figure 6 is selected as the analysis model. The analysis is carried out with a commercial finite element code LS-DYNA3D explicitly. The residual stress and thickness distribution induced in the welding process is considered in the analysis. The material is SPRC35R and its dynamic behavior is described with the Johnson-Cook model (Johnson and Cook, 1983) as shown in equation (1).

$$\bar{\sigma}=[264.8+641.6\bar{\varepsilon}^{0.59}]\left[1+0.105\ln\frac{\dot{\bar{\varepsilon}}}{\dot{\bar{\varepsilon}}_0}\right][1-T^{*0.23}]$$

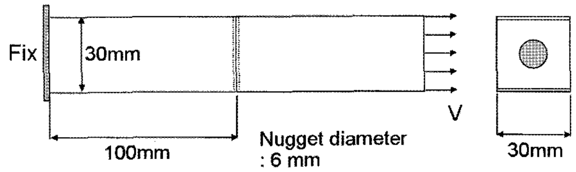
where T^* is the homologous temperature represented by equation (2).

$$T^*=\frac{T-T_{room}}{T_{melt}-T_{room}},\ \dot{\bar{\varepsilon}}_0=1/\mathrm{sec}$$

where T is the current temperature and T_{melt} is the melting temperature of the specimen. The first tem bracketed in the equation (1) is the strain hardening term; the second is the strain-rate hardening term and the third is the thermal softening term. In the Johnson-Cook model, the stresses at the strain-rate of 1/sec and higher levels are linearly interpolated in the logarithmic scale. Coefficients of the Johnson-Cook model is obtained from experiments using the universal testing machine, the high speed tensile test machine and tension split Hopkinson bar (Kang and Huh, 2000; Huh *et al.*, 2002; Huh *et al.*, 2003; Huh *et al.*, 2004).

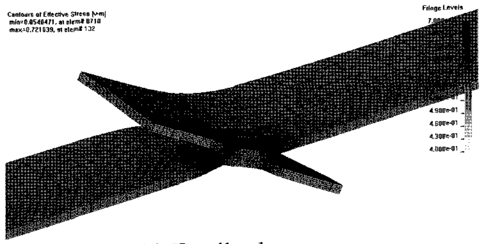


(a) Tensile-shear type

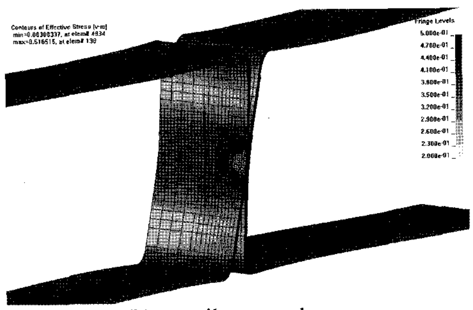


(b) Tensile-normal type

Figure 6. Schematic description of the analysis model.



(a) Tensile-shear type



(b) Tensile-normal type

Figure 7. Distribution of the von-Mises stress.

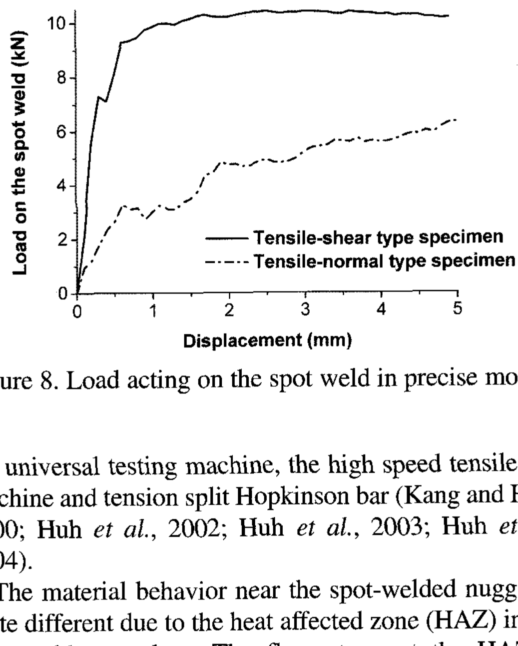
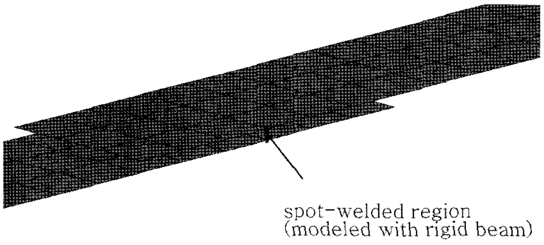
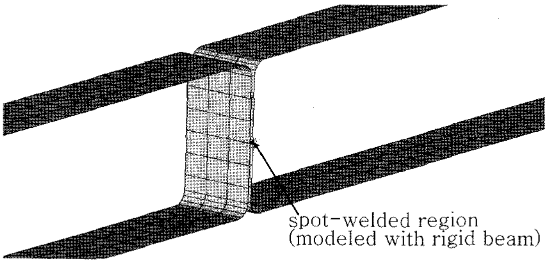


Figure 8. Load acting on the spot weld in precise model.

The material behavior near the spot-welded nugget is quite different due to the heat affected zone (HAZ) in the spot weld procedure. The flow stress at the HAZ is assumed to be 1.7 times larger than that in the adjacent region (Zuniga and Sheppard, 1995; Yancey, 2004). The specimen is elongated with the constant velocity of 1.0 m/s while the failure of the spot welds is not considered in the analysis. The distribution of the von-Mises stress is shown in Figure 7. Figure 8 shows the load acting on the spot weld. For both kinds of specimens, the maximum load on the specimen is regarded as the reference value in order to evaluate the load characteristics of the spot-welded region modeled using a single rigid beam in crash analysis.



(a) Tensile-shear type



(b) Tensile-normal type

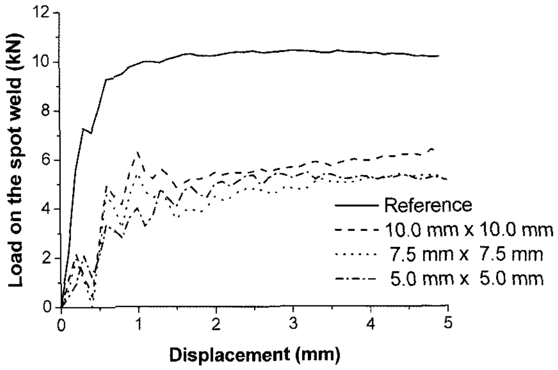
Figure 9. Finite element model of specimen which the spot welded region is model with the rigid beam.

4. EVALUATION OF THE FINITE ELEMENT MODELING OF SPOT-WELDED REGION

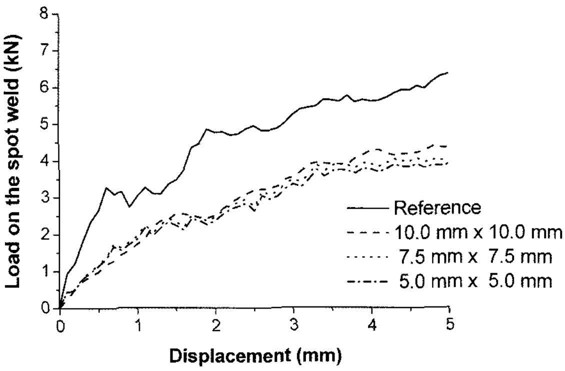
In the previous section, the load on the spot weld is calculated with the precise finite element model of the spot weld region. However, due to the existence of a large number of spot welds in a vehicle structure, it is often impractical to model each and every spot-welded joint in detail in a full scale crash analysis. The resistance spot welds in most current finite element crash models are characterized as a rigid beam at the location of the spot weld. Since the failure of a spot welded element is directly governed with the applied load on the spot weld, the calculation of the load at the rigid beam is important to obtain the accurate simulation result of vehicle crashes. Figure 9 shows the finite element model of a tensile-shear type specimen and a tensile-normal type one. The specimen is elongated with the constant velocity of 1.0 m/s while the failure of the spot welds is not considered in the analysis. The material is SPRC35R and its dynamic behavior is considered with the Johnson-Cook model. For the consistency of crash analysis, the effect of HAZ and residual stress of the spot-welded region are ignored in this model. The spot weld is characterized as a rigid beam. The load on the rigid beam is evaluated with respect to the element size, element shape and number of imposed constraints by the comparison of the one obtained from the precise model.

4.1. Effect of Element Size

The load on the spot weld is examined with respect to the element size for both the tensile-shear type and the tensile-normal type specimen. Since the element size of the model for crash analysis varies from 10.0 mm to 5.0 mm, three kinds of the rectangular elements with 10.0 mm × 10.0 mm, 7.5 mm × 7.5 mm and 5.0 mm × 5.0 mm are used in the analysis (Du Bois, 2001). Whole regions of the specimen are modeled with the rectangular shell element.



(a) Tensile-shear type



(b) Tensile-normal type

Figure 10. Comparison of the load on the spot-welded region with respect to the element size.

Figure 10 shows the load on the spot-welded region modeled with a rigid beam. The figure explains that the calculated load acting on the spot weld is inaccurate when the spot-welded region is modeled with a single rigid beam and the difference with the reference model is increased as the element size becomes smaller. It is because the constrained region where the single rigid beam covers is narrow compared to the actual nugget size of 6.0 mm. This difference with the reference model gives the inaccurate results about the failure of spot-welded region and moreover, inaccurate failure of spot-welded region degrades the reliability of crash analysis.

4.2. Effect of Element Shape

The load on the spot weld is calculated with respect to the shape of finite element models used in modeling of the

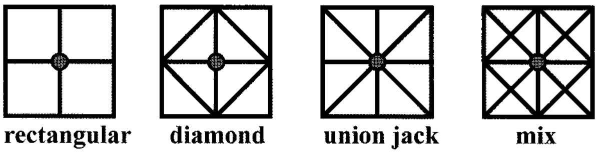
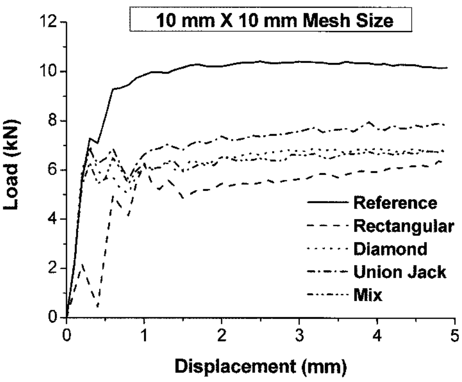
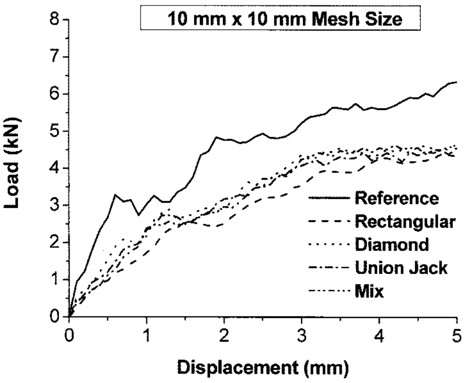


Figure 11. Four types of element used in the modeling of spot-welded region.



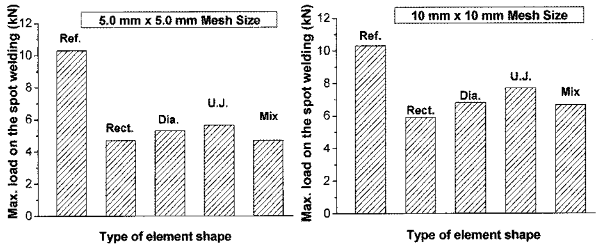
(a) Tensile-shear type



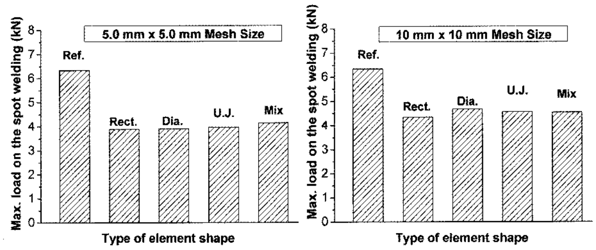
(b) Tensile-normal type

Figure 12. Comparison of the load on the spot-welded region with respect to the element shape.

spot-welded region. Four types of spot-welded models such as a rectangular shape, a diamond shape, a union jack shape and a mixed shape are used in the finite element modeling of the spot-welded region. One rigid beam is applied for the constraint of a spot weld and the outer region of the spot weld is modeled with the rectangular element. Figure 11 shows the four types of elements. The element size is assigned to be 10.0 mm × 10.0 mm, 7.5 mm × 7.5 mm and 5.0 mm × 5.0 mm respectively. Figure 12 shows the load on the spot-welded region with different element type when the element size of 10.0 mm × 10.0 mm is used. Figure 13 illustrates the comparison of the load on the spot weld with respect to the shape of finite element models. The comparison represents that the difference between the



(a) Tensile-shear type



(b) Tensile-normal type

Figure 13. Comparison of the maximum load on the spot-welded region with respect to the element shape.

precise model and simplified one is still remarkable when both the rectangular element and the triangular element are used in the modeling of the spot-welded region.

4.3. Effect of Rigid Constraints

The previous results show that calculated load on the spot weld modeled with a single rigid beam is inaccurate. In order to consider the actual nugget size of 6.0 mm accurately in the modeling of the spot-welded region, additional constraints are imposed around the center of the nugget with a rhombic shape. Figure 14 shows the shape of the imposed rigid constraints around the spot-welded region. The element size and the shape of finite element model are same as the previous analysis.

Figure 15 shows the load on the spot-welded region with different element type when the element size of 10.0 mm × 10.0 mm is used. Figure 16 explains that the comparison of the maximum load on the spot weld with respect to the element size and the shape of finite element models. The comparison explains that the difference between the reference model and simplified one is drastically decreased with both the tensile-shear type and the tensile-normal type specimen. The analyses results indicate that it is adequate from the viewpoint of the load

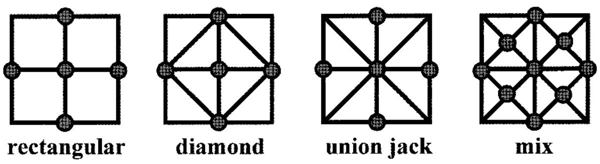
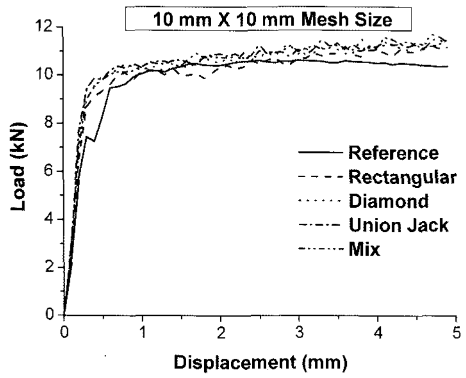
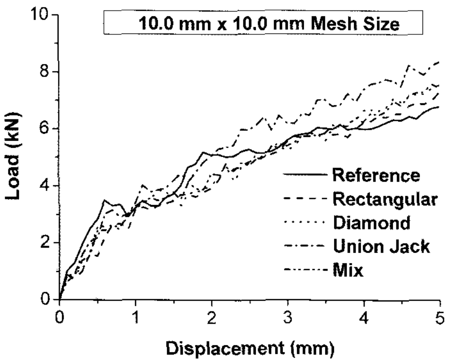


Figure 14. Shape of imposed constraint of each element.

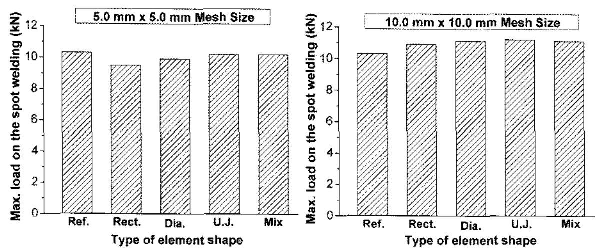


(a) Tensile-shear type

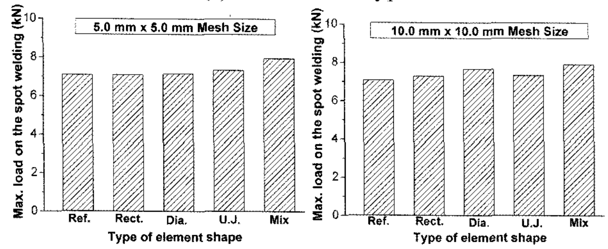


(b) Tensile normal type

Figure 15. Comparison of the load on the spot welded region with respect to rigid constraints.



(a) Tensile-shear type



(b) Tensile normal type

Figure 16. Comparison of the maximum load on the spot welded region with respect to rigid constraints.

calculation when the spot-welded region is modeled in the diamond shaped finite element with the rhombic rigid constraints and the mesh size of 5.0 mm in the crash model. The figure also explains that the rectangular shaped finite element with the rhombic shaped rigid constraints is useful for the modeling of the spot-welded region when the mesh size of 10.0 mm is utilized in the crash analysis. Analysis results fully demonstrate that the correct calculation of the load on the spot weld element is achieved by the change of the element shape and welded constrains. The accurate calculation of the load on the spot weld is helpful to predict the accurate failure model of the spot weld.

5. CONCLUSION

This paper is concerned with the evaluation of the load characteristics of the spot-welded region modeled with the rigid beam used in crash analysis. The load calculated from the precise finite element model of the spot-welded region considering the thermal history during the spot welding procedure is regarded as the reference value and the value is compared with the one obtained from the spot-welded model using the rigid beam with respect to the element size, shape of the modeled element and rigid constraints in spot weld. Analyses results indicate that the shape of imposed rigid constraints has critical influence on the evaluation of the strength of the spot weld. The results also demonstrate that the correct prediction of the load on the spot-weld element is achieved by the change of the element shape and the location of welded constrains. Present simulation results provide a guideline for an accurate finite element modeling of spot welds in the crash analysis of vehicles.

REFERENCES

ASM Handbook Committee (1985). *Metals Handbook 9th Edn., American Society for Metals.* Metal Park. Ohio.

Deng, X., Chen, W. and Shi, G. (2000). Three dimensional finite element analysis of the mechanical behavior of spot welds. *Finite Elem. Anal. Des.*, **35**, 17–39.

Du Bois, P. A. (2001). *Crashworthiness Engineering with LS-DYNA*. Livermore Software Technology Co.. Livermore. California.

Huh, H. and Kang, W. J. (1997). Electrothermal analysis of electric resistance spot welding procedures by a 3-D finite element method. *J. Mater. Process. Techno.*, **63**, 672–677.

Huh, H., Kang, W. J. and Han, S. S. (2002). A tension split Hopkinson bar for investigating the dynamic behavior of sheet metals. *Exp. Mech.* **42**, **1**, 8–17.

Huh, H. and Kang, W. J. (2002). Crash-worthiness assessment of thin-walled structure with the high-strength steel sheet. *Int. J. Vehicle Design* **30**, **1/2**, 1–21.

Huh, H., Lim, J. H., Song, J. H., Lee, K.-S., Lee, Y.-W.

and Han, S. S. (2003). Crashworthiness assessment of side impact of an auto-body with 60 trip steel for side member. *Int. J. Automotive Technology* **4**, **3**, 149–156.

Huh, H., Lim, J. H., Kim, S. B., Han, S. S. and Park, S. H. (2004). Formability of the steel sheet at the intermediate strain rate. *Key Eng. Mater.* **274-276**, 403–408.

HSK (2004). *ABAQUS/Standard User's Manual.* Hibbit, Karlsson & Sorensen Inc.. Pawtucket. Rhode Island.

Johnson, G. R. and Cook, W. H. (1983). A constitutive model and data for metals subjected to large strain, high strain rates and high temperature. *Proc. 7th Int. Symp. Ballistics*, Hague, Netherlands, 115–120.

Kang, W. J. and Huh, H. (2000). Crash analysis of auto-body structures considering the strain-rate hardening effect. *Int. J. Automotive Technology* **1**, **1**, 35–41.

Kang, H. T. (2005). Fatigue damage parameter of spot welded joints under proportional loading. *Int. J. Automotive Technology* **6**, **3**, 285–291.

Langrand, B. and Combescure, A. (2004). Non-linear and failure behavior of spotwelds: a global finite element and experiments in pure and mixed modes I/II. *Int. J. Solids Struct.,* **41**, 6631–6646.

Lee, H. Y., Kim, N. H. and Lee, T. S. (2005). Overload failure curve and fatigue behavior of spot-welded specimens. *Eng. Fract. Mech.,* **72**, 1203–1221.

Lee, Y., Wehner, T., Lu, M., Morrissett, T. and Pakalnins, E. (1998). Ultimate strength of resistance spot welds subjected to combined tension and shear. *J. Test. Eval.,* **26**, **3**, 213–219.

Lin, S.-H., Pan, J., Wu, S.-R., Tyan, T. and Wung, P. (2002). Failure loads of spot welds under combined opening and shear static loading conditions. *Int. J. Solids Struct.,* **39**, 19–39.

Lin, S.-H., Pan, J., Tyan, T. and Prasad, P. (2003). A general failure criterion for spot welds under combined loading conditions. *Int. J. Solids Struct.,* **40**, 5539–5564.

LSTC (1999). *LS-DYNA Keyword User's Manual.* Non-linear dynamic analysis of structures, Livermore Software Technology Co.. Livermore. California.

Xu, S. and Deng, X. (2003). An evaluation of simplified finite element models for spot welded joints. *Finite Elem. Anal. Des.,* **40**, 1175–1194.

Yancey, R. N. (2004). Impact modeling of spot welds. *Proc. 8th Int.l Conf. Numerical Methods in Industrial Forming Processes*, Columbus, Ohio, U.S.A., 1215–1218.

Zuniga, S. M. and Sheppard, S. D. (1995) Determining the constitutive properties of the heat-affected zone in a resistance spot weld. *Modeling Simul. Mater. Sci. Eng.,* **3**, 391–416.