International Journal of Numerical Methods and Applications

Published Online: December 2013
Available online at http://pphmj.com/journals/ijnma.htm
Volume 10, Number 2, 2013, Pages 91-97

STRESS CONCENTRATION CHARACTERISTICS OF WELDING CONNECTIONS BASED ON NUMERICAL ANALYSIS

Kyu-Nam Cho

College of Science and Technology
Hongik University
Sejong, 339-701, Korea
e-mail: kncho@hongik.ac.kr

Abstract

Structural members were constructed through welding and were subjected to various loads. In this paper, stress concentration of typical welding connections due to variable loads has been studied using numerical approach. Fillet welding, penetration welding and partial penetration welding characteristics are investigated by changing parameters such as penetration length, welding leg length, size and penetration angle. To verify proper structural configuration, various steel welding connections are chosen to be objective structures and relevant structural analysis is carried out using Finite Element Methods combined with ANSYS code. Stress concentration phenomenon and structural behavior have been the main outcomes of interest. It is found that in certain cases, a typical welding connection has lower stress concentration factors with well chosen welding parameters.

Received: July 26, 2013; Accepted: August 27, 2013

2010 Mathematics Subject Classification: 49K21.

Keywords and phrases: welding parameters, stress concentration, Finite Element Methods, welding connections.

This work is supported by 2011 Hong-Ik University Research Fund.

1. Introduction

Among various steel welding connections, asymmetric perpendicular cross type section of welding connection was the subject structures. Structural analysis was carried out by changing the welding length of the edges and connecting angles of the members. Finite Element analysis combined with ANSYS code was carried out [1, 2]. Stress concentration factors of each different welding connection were obtained.

2. Finite Element Modeling of Asymmetric Welding Connections

4 typical asymmetric welding connections with homogeneous materials were chosen as subjects and modeled by using 2-dimensional plane-stress element. Parameters were based on thickness of the member, welding penetration length, angle of the attached welding parts, and size of the welding section. ANSYS Mechanical APDL V12.0 was used to carry out modeling process and the following finite element analysis [3]. Fine meshed modeling techniques were adopted where abrupt stress changes are anticipated. Plane 82 element with multi nodal points was used. Young's modulus was 206,000 N/mm^2, Poisson's ratio 0.3 [4].

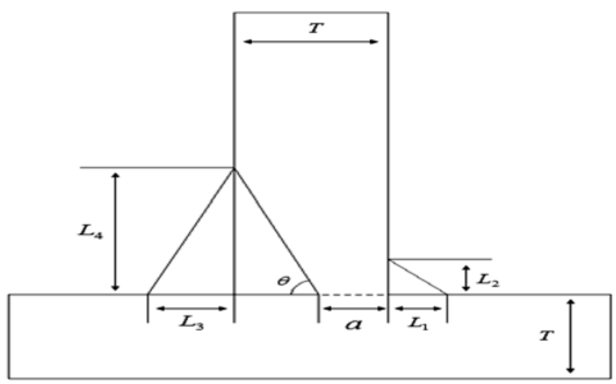

Figure 1. Asymmetric cross section type of the welding connection.

Thickness of the member T, welding penetration length a, and angle of the attached welding part to the base material θ of different models are listed in Table 1. The models had welding penetration length of 0 mm [5, 6].

Table 1. Asymmetric cross section types of the welding connection (unit : mm, deg.)

Dimensions							
Model	T	a	θ	L_1	L_2	L_3	L_4
U_{a1}	18	0	15.5	5	5	5	5
U_{a2}	18	0	29	5	5	5	10
U_{a3}	18	0	39.8	5	5	5	15
U_{a4}	18	0	48	5	5	5	20

First model had thickness of 18 mm, length of L_1 to be 5 mm, that of L_4 to be 5 mm and corresponding angle of the attached welding part to the base material was 15.5 degree. They were modeled as shown in Figure 2. The hot areas were modeled by fine meshes.

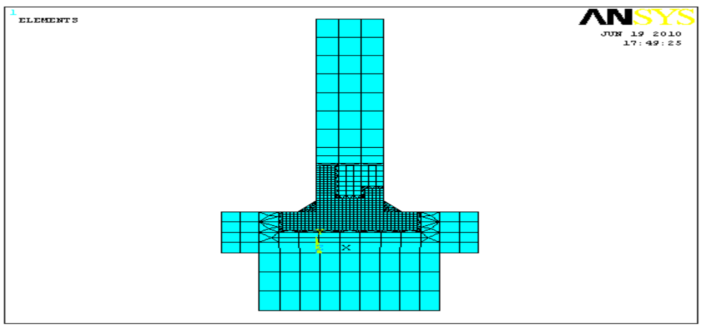

Figure 2(a). Mesh shape of model 1.

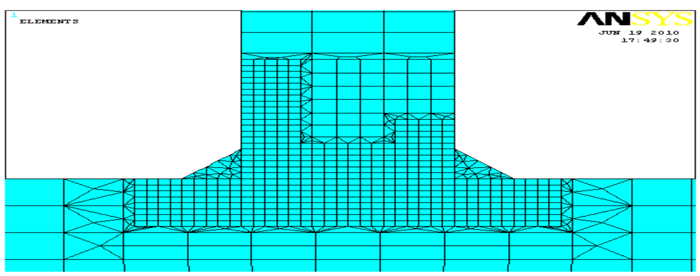

Figure 2(b). Mesh detail of model 1.

Second model had thickness of 18 mm, length of L_1 to be 5 mm, that of L_4 to be 10 mm and corresponding angle of the attached welding part to the base material 29 degree. It was modeled as shown in Figure 3. The hot areas were modeled by fine meshes.

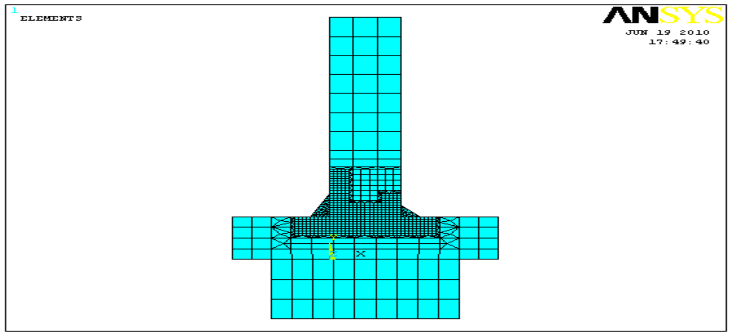

Figure 3(a). Mesh shape of model 2.

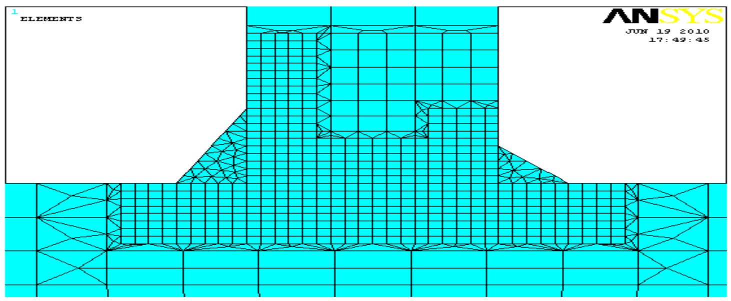

Figure 3(b). Mesh detail of model 2.

The third model had thickness of 18 mm, length of L_1 to be 5 mm, that of L_4 to be 15 mm and corresponding angle of the attached welding part to the base material 39.8 degree. Fourth model had thickness of 18 mm, length of L_1 to be 5 mm, that of L_4 to be 20 mm and corresponding angle of the attached welding part to the base material was 48 degree. Like the first and the second model, the third model's hot spots were modeled by fine meshes.

3. Finite Element Modeling Analysis using ANSYS

Finite element analysis was carried out with proper boundary conditions and external load conditions. Stress concentration phenomenon was examined by nodal solution contour plot obtained.

Lower part of the model was fixed and the upper part of the model was constrained properly to remove the corresponding rigid body motions. In order to figure out the stress concentration phenomenon of the structural element, unit loads were applied at nodes of upper part of the model.

4 models were analysed in the view point of stress concentration with pre-mentioned boundary conditions and applied loads. Stress concentration phenomenon of each model was obtained. 4 models with thickness of 18 mm, length of L_1 5 mm, that of L_4 5 mm to 20 mm and corresponding angle of the attached welding part to the base material 15.5 degree to 48 degree were analysed using ANSYS code. Figure 4 shows the analysis result of model 1. Nodal solution with von Mises stress of the first model was represented in Figure 4(a) and (b). It was observed that the high stress occurs at toe area of the model. Stress concentration factors at toe area of every 4 model was calculated and compared with each other. Figure 5(a) and (b) show the analysis results of the second model.

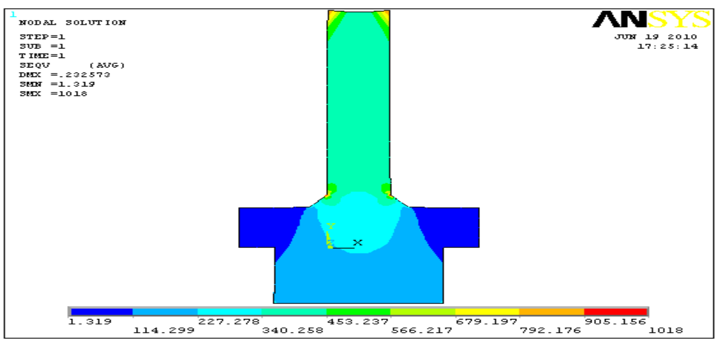

Figure 4(a). von Mises stress of model 1 in Nodal solution.

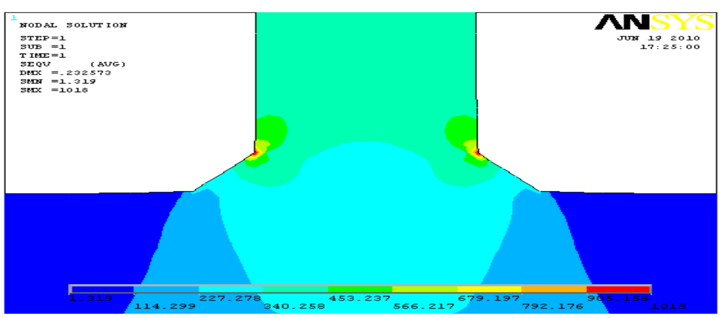

Figure 4(b). Stress detail of model 1 in Nodal solution.

It was also observed that the high stress occurs at toe area. Structural analysis of the third and the fourth models also shows how stress is concentrated at the toe area as expected.

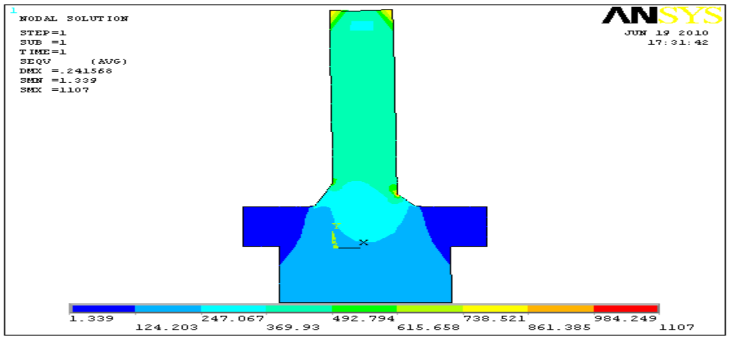

Figure 5(a). von Mises stress of model 2 in Nodal solution.

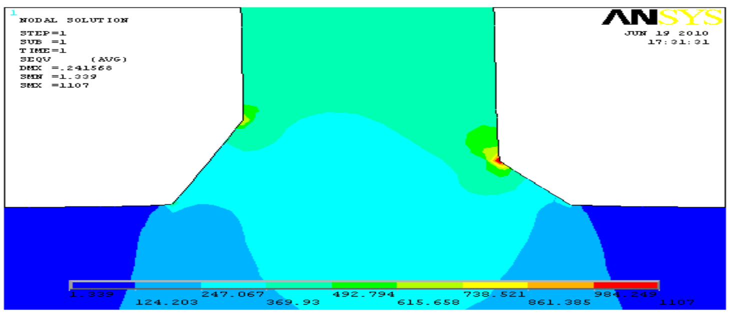

Figure 5(b). Stress detail of model 2 in Nodal solution.

4. Analysis Results and Conclusions

4 models were analysed in view point of stress concentration. Stress concentration phenomenon of each model was obtained. 4 models with thickness of 18 mm, length of L_1 5 mm, of L_4 5 mm to 20 mm and corresponding angle of the attached welding part to the base material 15.5 degree to 48 degree were analysed by using ANSYS code. Table 2 shows the result of the analysis.

Table 2. SCF of asymmetric welding joint group model

Group	Model	Stress concentration factor		Hot spot
		Toe	Root	
U_a	U_{a1}	3.13	.	toe
	U_{a2}	3.43	.	toe
	U_{a3}	3.64	.	toe
	U_{a4}	3.80	.	toe

Table 2 shows that increment of stress of each model compared with the first one was 9.49%, 16.32%, and 21.30%, respectively. Conclusions of this study are summarized as follows. Stress concentration phenomenon is highly dependent on the welding size and the angle of the attached welding part to the base material. A symmetric welding configuration is thought to be optimal regardless of welding connection types and size. An asymmetric welding configuration has disadvantages to the symmetric one in the view point of stress concentration. It is also observed that high stress occurs at the toe area.

References

[1] ANSYS User’s Manual, 2012.

[2] J. Y. Ko, ANSYS and Finite Element Analysis, Sigma Press, 2009.

[3] Saeed Moaveni, Finite Element Analysis with ANSYS, ITC, 2008.

[4] N. W. Paik and K. S. Jang, Railroad Technologies, Academic Press, 2004.

[5] K. N. Cho and W. I. Ha, A study on welding connection’s fatigue analysis through numerical and experimental approaches, J. Computational Structural Analysis (1993), 113-123.

[6] T. M. Padilla, T. P. Quinn, D. R. Munoz and R. A. L. Rorrer, A mathematical model of wire feeding mechanisms in GMAW, Welding J. 82 (2003), 100-109.