FEM analysis with shell element model in order to design support structure for PV system

Yasuhiro MAEDA*, Yoshihito OZAWA**, Wataru TAKANO***, Naoto TAKEHANA***
and Yuta ONODE***

* Graduate School of Fukushima University
1 Kanayagawa, Fukushima 960-1296, Japan

E-mail: y_maeda@okuji.co.jp

** Fukushima University
1 Kanayagawa, Fukushima 960-1296, Japan

*** Okuji Kensei Co., Ltd
4-1-8 Minamihonmachi, Chuo-ku, Osaka 541-0054, Japan

Received: 1 April 2019; Revised: 5 August 2019; Accepted: 7 January 2020

Abstract

In recent years, accidents have frequently occurred under the rapid spread of the photovoltaic power generation system in Japan. In order to make the design procedure for the support structure safety, in the previous studies, the authors have compared the experimental results with the FEM analytical results by using the beam element model. The results show that it can evaluate the deformation of the support structure qualitatively, determine the buckling member, and calculate the buckling load. However, it was not easy to quantitatively evaluate the deformation of the support structure in the analysis with the beam element model. In this study, in order to do quantitative evaluation for the deformation of the support structure, FEM analysis was performed by using the shell element model and through comparing experimental results with analytical results. From these results, it is found that the buckling member was determined and its buckling load could be calculated precisely. Furthermore, in order to examine the deformation and fracture process of the angle members, we conducted elastoplastic FEM analysis for the angle member. We will clarify the deformation behavior of the angle steel member until the buckling, and show new knowledge in the destruction of the whole of the support structure.

Keywords : FEM analysis, Photovoltaic system, Buckling member of support structure, Shell element model

1. Introduction

In recent years, the use of renewable energy has dramatically increased, especially, the solar power generation which has been rapidly increasing since 2013, and it is expected to become one of the key power sources in Japan. However, the members of the support structure were damaged by strong wind and heavy snow, and then the photovoltaic system (PV system) was destroyed suddenly. When such an accident occurs, the credibility of the PV system would be lost, and the development in use would be delayed (Japan association for wind engineering., 2017). In the previous study, we proposed an analytical model by using three dimensional elastic beam elements, in order to obtain the principal understanding for the safety design procedure and guidelines (MAEDA et al., 2019). This analytical model consisted of an eccentricity model which can consider the joint gap of members at the bolt joint part.

In these analytical results by using the beam element model, it was clarified that the deformation evaluation of the whole of the support structure, and the angle pillar member of support structure buckled, and then the whole structure was suddenly destroyed, and we succeeded in qualitative analysis, however the clearance of the bolt-hole in the experiment, the deformation of thread and the error of deformation due to the beam element model are not taken into consideration. Therefore, it was suggested that more precise and quantitative evaluation for deformation behavior of the

support structure is required (MAEDA et al., 2019). In order to establish a safe and efficient design method for the support structure of the PV system, which is designed individually under different environmental conditions, the multi-scale model analysis method was proposed in this research. In the FEM analysis for the whole system of the support structure with the macro scale model, in which the deformation of the support structure and the large stress concentration in a member of the structure could be examined from the results of FEM analysis. In the elastoplastic FEM analysis on the buckled member with the micro scale model, the deformation behavior of the member could be examined precisely, and the buckling load of the member up to the onset of the destruction of the support structure could be clarified.

2. Loading test for the support structure of the PV system

The support structure equipped with the solar cell module aligned at four lines and two columns was treated as one unit ground installation of the PV system. Since the snow load and wind pressure are dominant as the external environmental forces on the PV systems, we performed the experiments by simulating snow load and wind pressure. As a specimen in the experiment for the snow load, half of the units were arranged, and one unit was used as the other specimen for the wind pressure. The schematic view of the PV system is shown in Fig. 1. Table 1 shows the geometrical dimensional data of the support structure members used in the experiments. In the snow load experiment, the loading procedure with step of 240 N on the specimen is shown in Fig. 2 (a), and the pressurizing procedure with step of 250 N on the specimen is shown in Fig. 2 (b) in the wind pressure experiment. The snow load was assumed as the surface load acting on the solar cell module and was described as pressure. Pressure p was applied until the specimen was destroyed, and the evaluation of deformation and damage was conducted for the specimen. The detail of the experiments are given in the following references. (MAEDA et al., 2019)

Table 1 Geometrical dimensional data of members used for support

No.	Cross-section shape [mm]	Length [mm]		No.	Cross-section shape [mm]	Length [mm]	
		Snow load experiment	Wind load experiment			Snow load experiment	Wind load experiment
1	10×45×66.2×11.6×1.6	3986	3986	6	L-45×25×1.6	1759	1759
2	L-45×45×1.6	1051	679	7	L-45×25×1.6	754	1166
3	L-45×45×1.6	1425	1290	8	L-45×25×1.6	1126	2027
4	L-45×45×1.6	1440	1419	9	L-45×25×1.6	1533	2428
5	L-45×45×1.6	1513	1671	10	L-85×45×4.5	135	135

Fig. 1 Support structure of PV system. Members of specimen and displacement measurement positions are denoted by numbers and colored circles, respectively.

Fig. 2 Loading and pressurizing procedure to specimen.

Fig. 3 FEM model of support structure using shell elements.

3. FEM analysis for the support structure by using shell element

In order to conduct more precise quantitative evaluation for the deformation of the support structure, the FEM analysis is performed by using the commercial advanced nonlinear simulation software “Marc” (MSC Software Co. Ltd.) with the aid of the pre and post processor “Mentat”. The FEM analytical model of the support structure is constructed by using shell element. The shell element is described as a neutral plane with no thickness, and its cross-sectional shape is set by inputting the numerical parameter of each member for the plate thickness of the element. In the analytical result by using the shell element model, it is possible to examine the change of the cross-sectional shape which cannot be evaluated in the analysis with the beam element model, and the load transmission position can be precisely examined in the improved analysis. Since the support structure is consisted in thin and long members, the shear thickness deformation in the direction is much smaller than the bending deformation in the length direction. So the bilinear 4-node thin-shell element, in which the shear deformation cannot be considered, was used for the element of analysis, and the FEM analysis was conducted by using numerical algorithms including geometric non-linearity. Load direction in the case of snow load is different from the direction of wind pressure. When snow load is simulated, it acts on the member No. 1, however it acts in the vertical direction to the horizon. For simulating wind pressure, the force acts in the perpendicular direction to the surface of member No. 1. The point load was set to the bolt fastening part of the support structure, because the load and pressure were transmitted from the solar cell module to the support structure through bolts and bolted parts. The member of the support structure comes into contact with another adjacent member due to an increase in deformation which affects the deformation of another member. In the case that of such a phenomenon, then the member does not pass through another member.

The analytical model is shown in Fig. 3. The analytical model for snow load consist of 8,000 nodes and 7,024 elements, and the analytical model for wind pressure has 65,451 nodes and 59,932 elements. Since, only the steel structure is modeled in this analysis, the connecting conditions at the joint between member and base is set to the fixed end as in the experiment. Each connecting element at the joint between two members is set as rigid bar element (RBE) and the local θ_z is not constrained. The material properties are Young's modulus 205 GPa and Poisson's ratio 0.3 in the analysis. The analytical model for the wind pressure is larger than that for the snow load due to the specimen size of the experiment, and then the numbers of nodes and elements in the case of the wind pressure is larger at almost twice that of the snow load. Additionally, the loading condition caused in the case of wind pressure is the different from that in the case of snow load. The snow load is applied in the vertical z direction. However, the wind pressure is applied on the direction perpendicular to the inclined surface of the PV module. Therefore, the numbers of nodes and elements has been increased in order to perform FEM analysis precisely with better load transfer conditions through many connecting parts in the model.

4. The analytical results for the support structure using the shell element model and discussion

Let us focus on member No.5 of the support structure where the displacement was measured in the experiment. From experimental results, FEM analytical results with the shell element model and that with the beam element model with eccentricity, the relationships between the pressure and the displacement of member No.5 are shown in Fig. 4 for the snow load, and Fig.5 for the wind pressure. From Fig. 4 and Fig. 5, it can be seen that the displacement δ_y of the pillar member became larger as the pressure p increases. It is considered that this behavior in Figs.4 and 5 to be mainly

Table 2 Comparison of buckling load P_{cr} and compressive force P_C of FEM results at element No. 2-4

(a) Snow load

	Member No. 2	Member No. 3	Member No. 4	Member No. 5
Buckling load P_{cr} [kN]	20.3	11.1	10.8	9.8
Compressive load P_C at 5100 Pa [kN]	8.4	-6.8	5.0	9.6

(b) Wind pressure

	Member No. 2	Member No. 3	Member No. 4	Member No. 5
Buckling load P_{cr} [kN]	48.6	13.5	11.2	9.1
Compressive load P_C at 4180 Pa [kN]	8.5	-3.6	11.1	8.5

(a) Element No. 5 at left

(b) Element No. 5 at right

Fig. 4 Relationship between pressure p and displacement δ_y in snow load

(a) Element No. 5 at left

(b) Element No. 5 at right

Fig. 5 Relationship between pressure p and displacement δ_y in wind pressure.

caused by the difference in the cross-sectional rigidity between the beam element and the shell element, and the reduction of the moment of inertia of the cross-section due to the cross-sectional shape change of the member. In the shell element model, since it takes into account the shape of the member, the geometrical rigidity of the cross-section and the deformation of the cross-sectional shape could be precisely expressed. On the other hand, in the beam element model, since the moment of inertia of cross-section for the members were set to a constant value in both of x and y axes, the geometrical rigidity of the cross-section was higher than that with the shell element model. Therefore, with the beam element model, the deformation of member No.5 does not shows good agreement with the experimental results, however, by using the shell element model, it is seen that the analytical results of the deformation show good agreement with the experimental results.

From the analytical results, we should examine whether the determination of buckling member and the evaluation

Table 3 Comparison of pressure at buckling stress between experimental results and theoretical results by FEM.

Load type	Pressure at buckling [Pa]		
	Experimental results	Shell element model	Beam element model
Snow load	5038 - 5278	5180	5330
Wind load	4200 - 4400	4200	4577

Fig. 6 Relationship between compressive load P_c and pressure p . Red line shows Euler buckling load P_{cr} of pillar member and its buckling occurs at intersection of the red line to the blue line from analytical result.

of buckling load could be obtained or not. Table 2 shows the comparison of the buckling load P_{cr} with compressive force P_c of theoretical results at member No. 2-4. The compressive force P_c is calculated from Eq. (1) with the compressive stress σ_z acting in the middle part of each pillar member.

$$P_c = \sigma_z \times A \quad (1)$$

where A is sectional area of the member [mm^2]. The pillar member No. 5 in the experiment for the snow load and the pillar member No. 4 for the wind pressure buckled. In both cases, it can be seen from Table 2 that the compressive force of the member in the analytical result is almost the same value as Euler's buckling load for the buckling member in each experiment. Therefore, the buckling member can be determined from the FEM analysis.

In order to evaluate the buckling load P_{cr} of the pillar members where the onset of damage occurs in the case of snow load and wind pressure, the evaluation procedure is shown as follows;

The relationship between the compressive load P_c of the pillar member and the experimental pressure p is shown in Fig. 5 (a) and (b).

The buckling loads P_{cr} for the members No. 4 and No. 5 are calculated by using the Eq. (2);

$$P_{cr} = \pi^2 EI / l^2 \quad (2)$$

where E is Young's modulus [MPa], I is the moment of inertia of cross-section [mm^4] and l is the length of the pillar member [mm]. The buckling loads P_{cr} for the pillar members of No. 4 and No. 5 are also indicated by the red line in Fig. 6 (a) and (b), respectively. It is seen that the buckling occurs at the intersection of the red lines to the blue lines for the analytical results. Table 3 shows the pressure when the support structure is destroyed in the experiments and the buckling load of the pillar member is calculated from the analytical results with both FEM models. It is seen that the FEM analytical results by the shell element model shows good agreement with the experimental results because the load transmission and the deformation behavior in the cross section could be precisely simulated.

5. FEM analysis for the angle pillar member by using the solid element model

From the experiments, it was found that the support structure of the PV system was destroyed at the moment when the buckling of a member occurred at the onset of the fracture process of the whole structure. Therefore, let us focus on the member in which the large stress concentration is observed, by conducting the precise FEM analysis for the member with the solid element model, and then clarify the deformation of the member, the shape change and its buckling. From.

Table 4 Mechanical property of SS400 for member.

	Young's modulus [MPa]	Poisson's ratio	Yield stress [MPa]	Tensile strength [MPa]	Maximum elongation [%]
Analysis model	205	0.3	235	534	25%

Fig. 7 Analytical model using solid elements and its boundary condition.

the FEM analysis results, this knowledge can effectively prevent the destruction of support structure

For an FEM analysis model, a 20-node solid element of hexahedron secondary element was used. The number of nodes is 42,500, and the number of elements 7,908. The middle part of the L shape member was highly deformed, the dimension of an element in these parts are set to 2.5 mm in the longitudinal direction of the member, 11 mm in the transverse direction to the length, and 0.5 mm in the width direction of the cross section. Since it has been found in the experiment that at near both ends part of the member was not highly deformed by bending moment, the coarse elements are introduced in these parts and the length of an element is set to 15 mm in the longitudinal direction of the member. Boundary conditions of the analytical model are shown in Fig. 7. The points C and D indicate the edge points of the L shape cross section for the referenced cross section at the center of longitudinal direction, and the point B is set at its corner. Point A is at the intersection of the load action line to the line segment BC of the cross section, and the distance d_{CD} is described by the length between the point C and D. In this elastoplastic analysis, we examined the displacement in y direction δ_y at point A and the distance d_{CD} for the deformation behavior of the angle member. In setting the boundary condition of the analytical model, since the angle member of the support structure is fastened with a bolt at each end, the constraint conditions at the end of the member were set as the pin support loaded condition, and the displacement in length direction was set fixed at one end. The displacement at the nodes of the bolt-hole circle at the loaded end was constrained in x and y directions, and that at the nodes of fixed end was constrained in x , y and z directions. For the load condition, the compressive load F in the length direction is applied to the bolt-hole position at the loaded end of the member. The material properties used in the elastoplastic analysis are shown in Table 4. (Honda et al., 2015; MSC Marc, Element Library, Vol.B 2018; Nonlinear CAE association 2006)

6. The analytical results for the buckled angle pillar member and discussion

We conducted the elastoplastic analysis to examine the fracture process when the compressive load was applied to the angle member. The analytical result of the deformation behavior in the member will be precisely described below.

The relationship between the load F applied to the bolt-hole circle and the displacement δ_y at the point A is shown in Fig. 8. It can be seen that the displacement δ_y become larger as the load F increases. At the middle stage of loading, the

Fig. 8 Relationship between load F applied to bolt-hole position and displacement δ_y at point A in FEM analysis with solid element model.

Fig. 9 Contour of stress σ_z at load level $F=7,360$ N .

Fig. 10 Relationship between load applied to bolt-hole circle and displacement d_{CD} in analytical result. Black dotted line shows elastic limit.

Fig. 11 The change of deflection and cross-sectional shape at the middle part of the analytical model at applied compressive load.

nonlinear behavior is observed in the $F - \delta_y$ curve and the rigidity is largely reduced. Elastoplastic analysis is continued up to the maximum load of 7,542 N. In order to examine whether the non-linear behavior in Fig. 8 is caused by plastic deformation of the member, the stress distribution of σ_z in the angle member is precisely examined in the analytical result. The stress contour σ_z is shown in Fig. 9 at the load level $F=7,360$ N just before maximum load. As load is gradually increased, the plastic deformation is first observed on the edge of the BD side in the referenced cross section at the center part of longitudinal direction at 7,360 N. In Fig. 8, the nonlinear behavior is observed from the middle stage of loading where the almost part of the member is found to be in the elastic deformation zone until 7,360 N in Fig. 9.

Since the cross-sectional shape of the member was highly deformed in the inspection of the support structure after the destruction in the experiment, by observing the change of the cross-sectional shape of the angle member, we examined the nonlinear deformation behavior within at the elastic deformation zone before 7,360 N. The relationship between the load F and the distance d_{CD} is shown in Fig. 10, and at that position the change of deflection and cross-sectional shape in the middle is shown in Fig.11. The black dotted line indicates the elastic limit of 7,360 N. From Fig. 10, as the load F increases, the d_{CD} starts to increase so that the L shape of the cross section is beginning to be deformed and opened into a flat shape. It can also be observed from Fig. 11 that the cross section of L shape is highly deformed and rotated around the axis of longitudinal direction of the member. This nonlinear behavior should be caused by the bending deflection and the reduction in the geometrical rigidity due to the cross-sectional shape changes. Therefore, the opening of the cross-sectional shape makes the reduction of the moment of inertia of cross-section, and the nonlinear deformation behavior

occurs within the elastic deformation zone. In further load increase, the small scale of plastic deformation appears in a part of the member, and then the angle member suddenly buckles just over the load level 7,360 N. It is necessary to understand the deformation behavior and fracture process of the angle member, and the examination of these phenomena lead to the important knowledge of the maintenance for the support structure.

By examining them by using the multi-scale model analysis combining the analysis for support structure in the macro-scale with that for the member in the micro-scale, we could obtain the efficient analytical procedure to establish the safety design for the support structure. In the macro scale, it was possible to find the member with a large stress concentration in the whole of the support structure. In the micro scale, the buckling load and the fracture process until the buckling occurs were examined. By using the multi-scale model analysis method, the load capacity could be clarified to prevent the destruction of the whole of the support structure, and the design of an efficient and safe support structure would be improved.

7. Conclusion

In this study, we aimed to precisely evaluate the deformation of the support structure from the result of FEM analysis with the shell element model. Furthermore, in order to observe the deformation and the fracture process of the angle member, we conducted the elastoplastic analysis for its member. From the results of FEM elastoplastic analysis, the opening of the cross-sectional shape reduces the moment of inertia of the cross-section, and we could observe the nonlinear bending deformation behavior of the angle member even in the elastic deformation region of the material. After that, it is found that the plastic deformation appear in a part of the member and then the angle member suddenly buckled. The obtained results in this study are summarized as follows.

- i. It was clarified by using the shell element model that the FEM analysis can evaluate the deformation of the whole of the support structure precisely.
- ii. It was found that the buckling pillar members could be determined due to the compressive force derived from FEM analysis with the shell element model, and we could also estimate the snow load and wind pressure when the support structure is destroyed.
- iii. In the case of the compressive load acting on the bolt-hole of the angle member, the cross section of the pillar member was changed and the moment of inertia of the cross-section was reduced. Therefore, it was found that the relationship between the load F and the displacement δ_y for the angle member is shown as a nonlinear behavior even within the elastic deformation zone, and the member is suddenly buckled after plastic deformation appeared.

The bending deformation of the member and the opening of the cross-sectional shape for the angle member obtained in this study can be used as a reference value for maintenance of the support structure. In our future work, deep understanding with the multi-scale model analysis could give us the fundamental knowledge in order to establish safety design procedures and guidelines for support structures.

References

- Aihara, T., Terasaki, H. and Nakamura, R., Characteristics of wind force acting on photovoltaic array on the ground, Proceedings of National Symposium on Wind Engineering, 21st (2010), pp.43–45 (in Japanese).
- AIJ Design, Standard for Steel Structures Based on Allowable Stress Concept (2017).
- Honda A., Kumagai T and Takagi H., Buckling Loads of Axially Compressed Member Stiffened by Eccentrically Loaded Compression Members, Summaries of technical papers of annual meeting, Architectural Institute of Japan (2015) (in Japanese)
- Ikeda, H., Matsuda, K., Kato, K., Ootake, K., Nakamuta, K. and Ikeda, K., The influence of photovoltaic array arrangement placed parallel to the predominant wind direction on wind force coefficients, Journal of Structural Engineering, Vol.60A (2014), pp.436–445 (in Japanese).
- Isozaki A., Inaba Y and Kuwamura H., Euler Buckling of High-Strength Thin-Walled Stainless Steel Members, Proceeding of the architectural research meetings, Kanto Chapter, Architectural Institute of Japan (2000) (in Japanese)

Japanese).

- Kawai, T., Zakutsumondaikaiseki (konpyutaniyorukozokogakukoza II-6-B) (1974), pp.105–137 (in Japanese).
- Kozokaisekinotamenoyugenosohojissenhandobukku, Nonlinear CAE association (2006), p.44 (in Japanese).
- Maeda, Y., Ozawa, Y., Takehana, N., Takano, W. and Onode, Y., Evaluation of FEM analysis model on solar module steel structure design, 13th World Congress on Computational Mechanics / 2nd Pan American Congress on Computational Mechanics (2018), pp.1948–1957.
- Maeda, Y., Ozawa, Y., Takehana, N., Takano, W. and Onode, Y., Proposal of FEM analysis model with three-dimensional elastic beam element in order to design the support structure for ground installation type of photovoltaic power generation system, Transactions of the JSME (in Japanese), Vol.85, No.871 (2019), DOI:10.1299/transjsme.18-00397.
- Maeda, Y., Ozawa, Y., Takehana, N., Takano, W. and Onode, Y., Strength evaluation of solar module steel structure and FEM analysis on beam element model, Proceedings of the annual meeting of the JSME/MMD (2017), pp.1299–1303 (in Japanese).
- Manual of wind resistant design for photovoltaic system, Japan association for wind engineering (2017) (in Japanese).
- Matsuura, M., Hatanaka, A. and Yamaguchi, T., Study on load carrying capacity of a small power transmission tower made of angle steel members, Journal of Structural Engineering, Vol.58A (2012), pp.50–61.
- Mohri, F., Azrar, L. and Potier-Ferry, M. Lateral post-buckling analysis of thin-walled open section beams, Thin-Walled Struct Vol. 40 (2002), pp.1013–36.
- Mohri, F., Bouzerirab, C. and Potier-Ferry, M. Lateral buckling of thin-walled beam-column elements under combined axial and bending loads, Thin-Walled Struct Vol. 46 (2008), pp.290–302.
- MSC Marc, Element Library, Vol.B (2018).
- Nakazawa, Y., Tamura, K., Kusaka, T. and Hojo, M., Effects of Cross Sectional Shape on Plastic Buckling Behavior of Thin-Walled Polygonal Shell Members, Transactions of the JSME (in Japanese), Vol.73A, No.727, (2007).
- Takahashi, K., Yonago, A. and Mizuno, M., Distortion of Curved Thin-Walled Open -Cross-Section Members, Proceedings of the Japan Society of Civil Engineers (1982) (in Japanese).
- Sato A., Mitsui K., Okada H and Ono T., Flexural buckling stress of built-up compression member composed with light gage members -Part.4 Derivation of effective slenderness ratio-, Summaries of technical papers of annual meeting, Architectural Institute of Japan (2014) (in Japanese)
- Somekawa, D., Koizumi, T., Taniguchi, T. and Taniike, Y., Wind loads acting on the photovoltaic panels arrayed near ground, Proceedings of National Symposium on Wind Engineering, 22nd (2012), pp.157–160.
- Somekawa, D., Koizumi, T., Taniguchi, T. and Taniike, Y., Wind loads acting on pv panels and support structures with various layouts, The Eighth Asia-Pacific Conference on Wind Engineering (2013), pp.235–242.
- Takamori, K., Nakagawa, N. and Nishimura, H., Design wind force coefficients for ground mounted photovoltaic panels : Part 1. Outline of wind tunnel test and wind force coefficients of single panel, Summaries of technical papers of annual meeting (2017) (in Japanese).
- Takamori, K., Somekawa, D., Okuchi, M. and Uematsu, Y., Wind resistance test of a photovoltaic system installed on metal roof, Proceedings of National Symposium on Wind Engineering, 24nd (2016), pp. 319–324 (in Japanese).