Transactions of the TSME (2016) Vol. 4, No. 1, 25-34
Journal of Research and Applications in Mechanical Engineering

ISSN 2229-2152 print
DOI: 10.14456/jrame.2016.3

Research Article

STUDY OF SAFETY EVALUATION FOR ASSEMBLED MECHANICAL STRUCTURE

Y. Ohbuchi[1,*]
M. Nishi[1]
K. Tsaukamoto[2]
H. Sakamoto[1]
[1] *Advanced Mechanical System Department, Graduate School of Science and Technology, Kumamoto University, 2-39-1, Chuo-ku, Kumamoto, 860-8555, Japan*
[2] *Department of Mechanical Engineering, National Institute of Technology, Kagoshima College, 1460-1, Shinkou Hayato-cyou, Kirisyhima, 899-5193, Japan*

ABSTRACT:
Recently, there are a lot of products and parts, which can make mechanical assembling easy. The jointing methods of these products and parts are many and various. Nowadays, strength evaluation of structure using CAD modeling and CAE analysis has become popular and widely used. It became very easy to evaluate the strength and safety factor of structure using them, but sometimes validity of obtained result is not satisfied due to incorrect boundary conditions or wrong model of joint. However, the modeling and analyzing in detail for various parts and its jointing methods are difficult in consideration of time and cost. In this study, the suitable analysis condition for evaluation of various jointing methods is examined utilizing these products and parts. A products for structure material made of aluminum alloy and series of parts for fastening are used as the objects for this study. Analysis is carried out for each joint, and finally the assembled whole structure is evaluated. When we can select suitable analysis condition for the various fastening method, we can analyze easily and evaluate safety of designed structure. Finally, we proposed a simplifying model which represents complex shape and jointing conditions of those products and it works well and can reduce calculating time as well as modeling time.

Keywords: *Assembled structure, CAD modeling, CAE analysis, Safety design*

1. INTRODUCTION

There are a lot of products and parts, which can make mechanical assembling easily. The jointing methods of these products and parts are many and various, like welding, riveting, pin connecting, bolt fastening, etc. In design of structure using these jointing, working stress and safety factor evaluation are necessary.

Nowadays, many structures have been designed by CAD modeling and analyzed by CAE. However, it is difficult to determine the suitable boundary condition and to evaluate safety factor for various jointing method in real products. For the same structure, since analytical result of stress is influenced by jointing condition in assembling, the safety factor changes greatly. In the case of structure analysis, improper joint condition causes incorrect evaluation result and influences the safety of structure [1]. These are the important factors for safety design of products. However, the modeling and analyzing in detail for various parts and its jointing methods are difficult in consideration of time and cost [2].

In this study, suitable boundary conditions for various jointing of assembled structure in CAD modeling and CAE analyses are studied systematically to evaluate strength and safety easily and accurately.

* Corresponding author: Y. Ohbuchi
E-mail address: yohbuchi@mech.kumamoto-u.ac.jp

2. JOINTING METHOD

The jointing methods of these products and parts are many and various, like welding, riveting, pin connecting, bolt fastening, etc. In this study, the jointing method using commercial product parts for frame structure was evaluated. Specifically, the frame of vehicle performance testing equipment was evaluated. Figure 1 shows an example of assembled structure. The equipment was designed by authors using the principle of the existing chassis dynamometer [3]. It was designed for performance evaluation of electric vehicle [4] which designed by our University team for solar car race [5]. It was assembled very easily using commercial product parts for frame structure. Fig. 1 shows the frame of the structure, and Fig. 2 shows its CAD model. For the frames, ALFA FRAME SYSTEM™ (NIC Autotec, Inc. [6]) which is aluminum alloy structural material with exclusive brackets was used. It can be easily assembled. The equipment has twin roller [7] for ensuring safety, not to fall off the driving wheel.

Fig. 1. Flame structure.

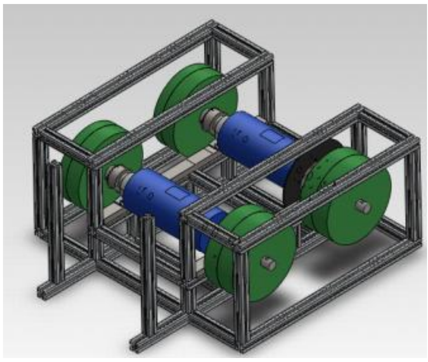


Fig. 2. CAD model.

3. ANALYSIS MODEL

Many kinds of jointing methods are modeled by CAD and analyzed by CAE. Here, the analysis was executed by SolidWorks-Simulation™. Firstly, a simple T-shaped assembling model is used. We compared the difference of stress in members and displacement of structure caused by jointing condition and assembling method. The analysis conditions are describes below. The structural parts made of aluminum-alloy A6063-T6 of square bar (W*H*L=10*10*100) was used. The scheme model of the analysis is shown in Fig. 3.

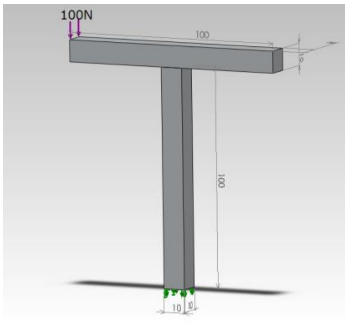


Fig. 3. T-shaped assembling model.

Two bars are assembled in T-shaped, and load of 100N is applied vertically on the end of assembled T-shaped model. The stress and displacement are calculated on each condition. The six kinds of joint conditions are used as follows [8].

1. Fixed at contact surface.
2. Fixed at contact edges.
3. Welded at one side of the contact surface.
4. Welded at two sides of the contact surface.
5. Welded at circumference of the contact surface.
6. Fastened by bolt-nut.

The analysis results of stress and displacement are shown in Table 1, and stress distribution is shown in Fig. 4.

(a) Surface fixing (b) Bolt-nut fixing

Fig. 4. Stress distribution.

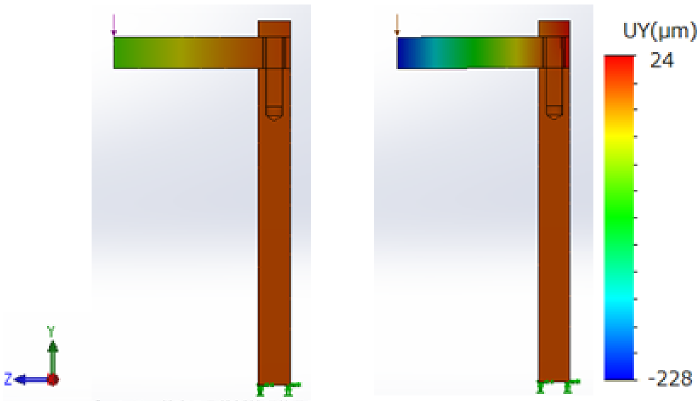
Table 1: Maximum stress and displacement

	Stress [MPa]	Displacement [μm]
Surface	39.4	740
Edges	40.0	740
Welding1	39.1	740
Welding2	39.1	740
Welding4	39.1	740
Bolt-nut	72.3	750

On the CAD model, each model was assembled by each jointing method, but the result was hardly a difference except for bolt-nut fixing. From this result, it is considered that these models were recognized as same joint condition in the analysis. In other words, it is too difficult to evaluate real structure correctly without understanding of the internal processing on the analysis, how the jointing and contact conditions are treated in CAD modeling and CAE analysis.

4. Difference Caused by Contact Condition

From the foregoing analysis results, it was found that the mating condition on the assembling in CAD modeling did not influence the analysis results. Next, we analyzed L shaped bolt fastened model in the case of suitable contact condition. We compared vertical direction displacement of horizontal part for two conditions, one is contact condition and the other is bonded condition between the part and vertical parts. Fig. 5 and Table 2 show each result. The maximum vertical direction displacement for contact condition was 10 times as large, and the minimum value was 2.7 times as large as bonded condition. It seems that these results caused by the movement for the clearance between the bolt and bolt hole. Because of the contact condition, the horizontal part became easy to move. Based on this result, we analyzed a part of the flame structure we used.



(a) Bond condition (b) Contact condition

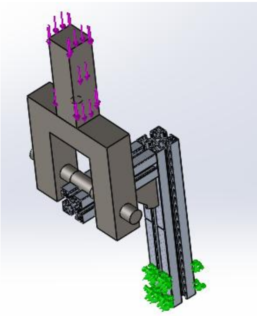
Fig. 5. Vertical direction displacement.

Table 2: Vertical direction displacement

	Displacement [μm]	
	Max.	Min.
Bond	2.2	-82.1
Contact	24.0	-228.2

5. Analysis for Assembled Structure

We analyzed a part of our previously designed assembled structure shown in Fig. 1 which was made using commercial product parts for frame structure (Alfa Frame System™; NIC Autotec, Inc.). Many products are manufactured after the CAD modeling and CAE analyze in recently design process. This assembled structure is manufactured in this process. The CAD modeling is one part of structure assembled by special bracket (Fig. 6). The contact condition is not bonding, i.e. not whole body. All of the contact conditions are set one by one. And the analysis model includes jig model for force applying, to compare with the analysis result and experimental result. The analysis model is loaded 400N vertical down direction by the jig. The load condition is vertical down direction and distributed load. The supporting part of this structure is fixed. (Here, this loading condition is incorrect and should be avoided in suitable use. This is only for experiment.) From the analytical result in Fig. 7, the maximum stress appeared in fixed bolt located between the bracket and the frame. When the load comes up to 400N, the maximum stress is 359MPa and the safety factor is 0.75, therefore the bolt will yield.



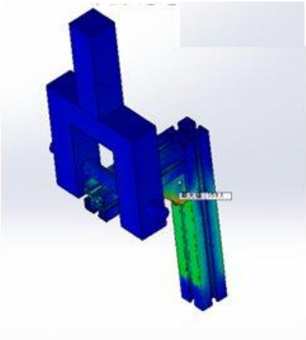


Fig. 6. Model for analysis. **Fig. 7.** Analytical result.

6. Experimental Result and Analysis

6.1 Experimental result

An experiment was performed to examine the validity of analytical result. The fixing condition is same as analysis model. The outlines of experimental conditions are below. The test equipment is universal material test machine (SHIMADZU : AG-25TB), structure frame parts made of aluminum alloy and special connecting bracket of Alfa Flame System™. Test mode is compression, testing velocity is 20mm/min, and load is applied at arm. The setup of experiment and the view of fractured state after loading are shown in Figs. 8 and 9 respectively.

Fig. 8. Setting of experiment.

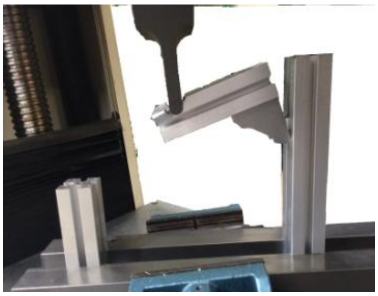


Fig. 9. Fractured status.

According to the analytical result, the fracture will occur as breaking of bolt, actually but in the experiment, frame was broken at the part of fixed nut inner side of frame, as shown in Fig. 9. The changes of load obtained are shown in Fig. 10. From this graph, yielding occurred about 1kN loading. At the position of fracture in this experiment, safety factor was 2.1 for load of 400N in the analytical result in Fig. 7. Therefore stress evaluation is suitable, but safety evaluation for structure is not sufficient because the fracture occurred at different part.

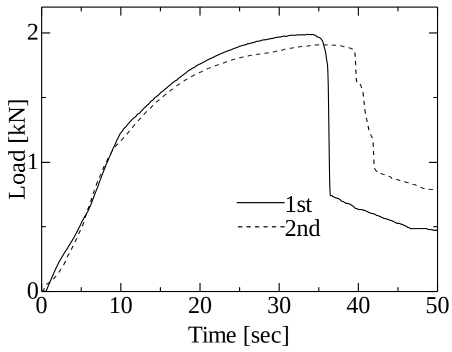


Fig. 10. Loading test result.

In the test result, different breaking point and breaking load are observed. Probable causes are as follows. There are some surfaces of different contact condition between analysis model and actual structure. In the analysis model, friction contact surfaces in actual assembly are treated as bonded surface (whole solid) because calculation becomes too complex. Therefore, it is necessary to confirm the analysis is possible using same contact condition to actual assembly, and to confirm validity of analytical result for other contact conditions. In addition, pre-load by the fastening torque was not considered in this analysis on the bolt fastening point. It is necessary to consider the pre-load by bolt fastening in the analysis, because the load will influence to the critical load of yielding.

6.2 Pre-load by bolt fastening

It is necessary to represent the stress caused by fastening torque on the analysis correctly to evaluate the fastening by bolt correctly. Therefore, we tried to apply pre-load for the bolt in the analysis by using thermal stress. Thermal contraction is caused by lowering the temperature of the bolt, and it can be used as an axial force. The relations of fastening torque and axial force and relation of thermal stress are obtained in the following expressions.

$$T = k\,d\,F \tag{1}$$

T : Fastening torque [N・m]
F : Axial force [N]
d : Nominal diameter [m]
k : Torque coefficient

$$\sigma = E\,\Delta T\,\alpha \tag{2}$$

σ : Thermal stress [Pa]
E : Elastic modulus [Pa]
ΔT : Temperature difference [K]
α : coefficient of thermal expansion [K^{-1}]

The standard fastening torque is 5.2[N・m] for M6 size bolt used here, the axial force becomes 4300[N]. Then, the temperature difference required is 56.1[K], and the thermal stress becomes 153.3 [MPa] by calculation. A heat transfer analysis was performed by lowering the temperature of the shaft of bolt, and the analysis result was used to a static analysis. The heat transfer analysis result and the static analysis result are shown in Figs.11 and 12. These results show that the using of thermal stress represents axial force of bolt. Thereby, we analyzed the above L shaped model with pre-load by using this analysis method. The vertical direction displacement results are shown in Fig. 13 and Table 3.

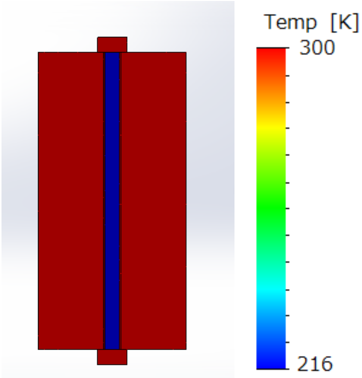


Fig. 11. Thermal distribution.

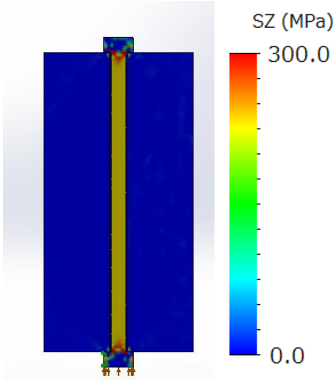
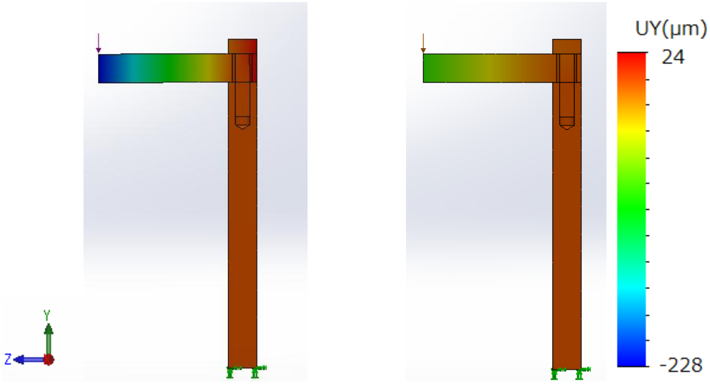


Fig. 12. Thermal stress analysis result.



(a) Without thermal stress (b) With thermal stress

Fig. 13. Vertical direction displacement without thermal stress.

Table 3: Displacement with and without thermal stress

	Displacement [μm]			
	Vertical direction		Synthetic	
	Max.	Min.	Max.	Min.
Without T.S.	24	-228	231	0
With T.S.	0.65	-89.8	90.6	0

In the same way as above mentioned analysis, we analyzed the assembled structure model with and without the bolt pre-load or not. These results are shown in Figs. 14, 15 and Table 4.

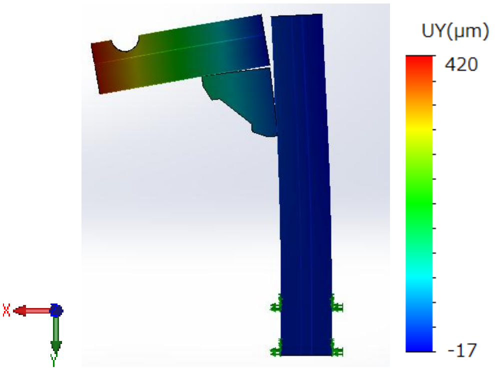


Fig. 14. Vertical direction displacement without pre-load.

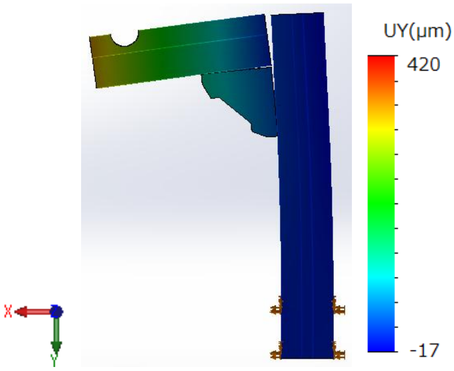


Fig. 15. Vertical direction displacement with pre-load.

Table 4: Displacement with and without thermal stress

	Displacement [μm]			
	Vertical direction		Synthetic	
	Max.	Min.	Max.	Min.
Without T.S.	420	-17	510	0
With T.S.	350	-17	420	0

From the result, the maximum displacement with pre-load was 20% smaller than that without pre-load. The minimum safety factor point appeared on the contact surface between lower frame and nut, and it was 1.42, as shown in Fig. 16. And then, the minimum safety factor of the nut was 3.67, the bolt was 2.34. At the previous analysis result, the minimum safety factor point appeared on the lower bolt, and the bolt will yield first. But this analysis result shows that the lower frame will yield first. It well agrees with the experimental result. Therefore, we come to analyze more practically by using thermal stress analysis.

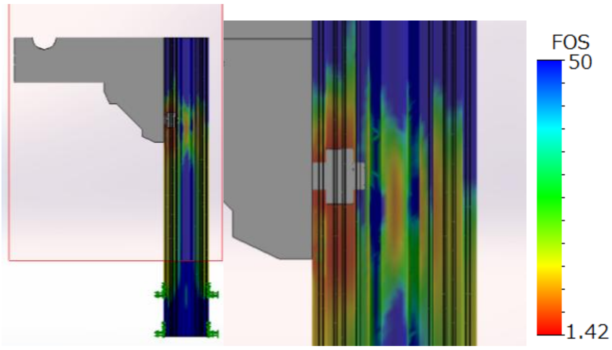


Fig. 16. Safety factor of the flame.

We analyzed this new analysis model with 1kN loading (experimental yielding load). But displacement and stress result are too higher than experimental result. It seems that the result was influenced by the contact condition between nut and frame. So, it is necessary to improve the contact condition and to make the correct analysis model.

6.3 Elasto-plastic analysis

In the analysis described above, the fracture behavior is agree with experimental result, same part of frame yielded first. However, yielding lord was not agreed and was very smaller than experimental value. Therefore, it will be required to be representing more detail of material behavior and contact condition.

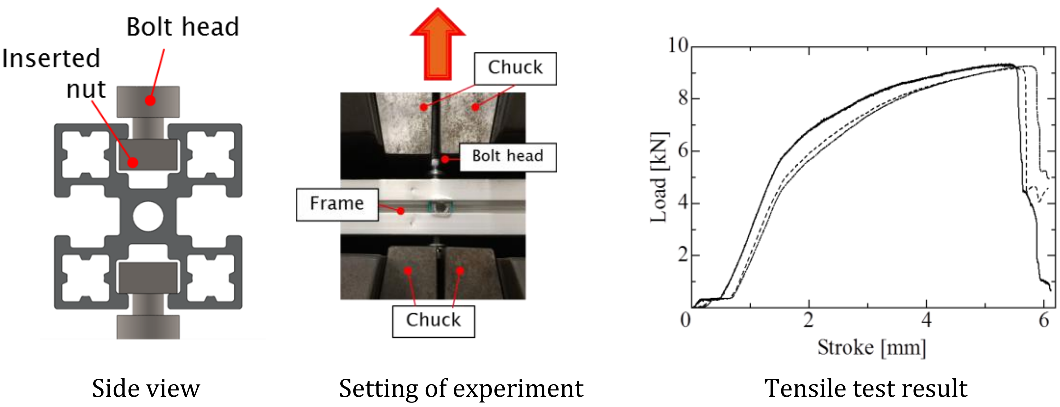
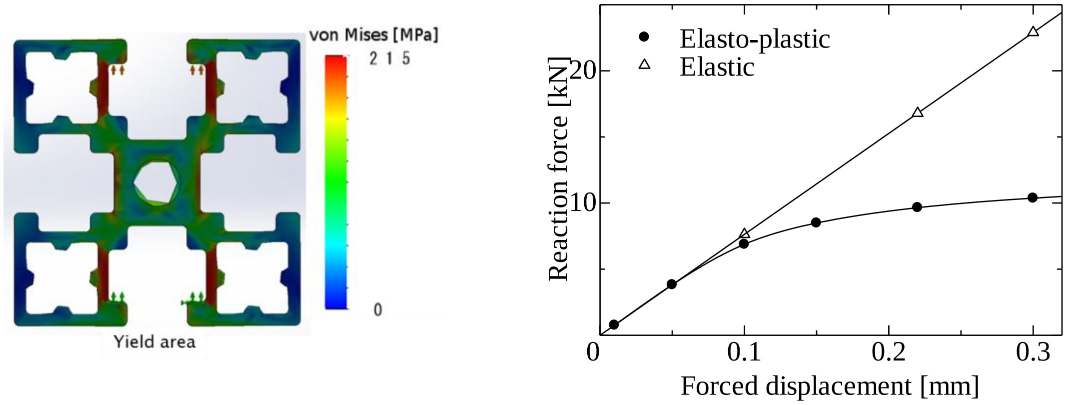


Fig. 17. Loading test of frame with bolt.

Fig. 17 shows fracture test of frame with bolt. In this test, nuts were inserted into frame groove, and bolts which screwed into nuts were pulled by chuck of loading test machine. In this test, the frame was yielded at 5.2 kN of load. From this experiment, the yielding stress of the frame was 286.4 MPa.



Stress distribution

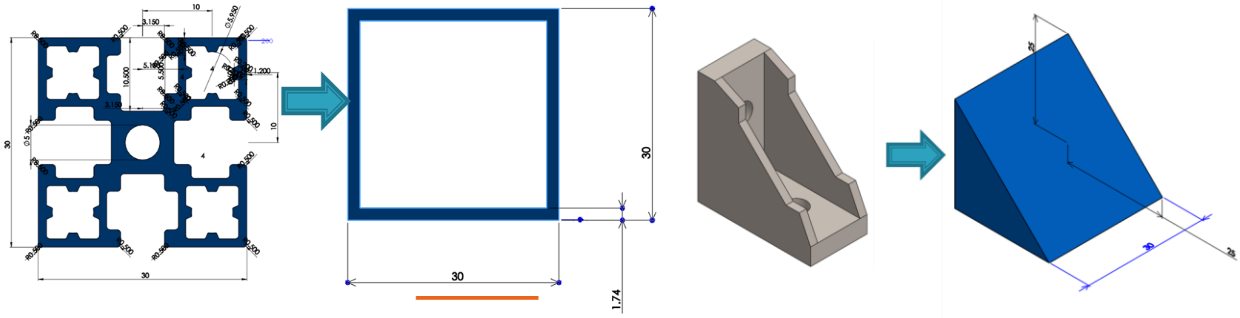
Analysis result of reaction force

Fig. 18. The result of elasto-plastic analysis.

For more correct analysis, frame and nuts contact region was applied constrained displacement and calculated reaction force. The analytical results are shown in Fig. 18. Applying elasto-plastic analysis, reaction force shows nonlinear near 5kN. Here, the accuracy of calculation is within 4% comparing with experiment. This well agreed with experimental yielding load. So, it is shown that the correct modeling leads correct result. However, this elasto-plastic analysis is very time consuming, so it was needed about over 500 seconds for one condition and it is 87 times of elastic analysis. Therefore it is not efficient applying this correct condition for all connecting parts.

6.4 Proposal about simplify model for analysis

As mentioned above, correct connection and contact model with elasto-plastic analysis can well express actual results, but there are many joints and contact area in mechanical structure. So it is not actual and reasonable to apply correct condition for all joint. Therefore, we propose simplifying model to express actual joints easily and to satisfy sufficient accuracy. For frame, this has same dimension of outer square and same geometrical moment of inertia. For bracket, that shape is simple triangle pole without bolt and nut, as shown in Fig. 19. Here, in this simplifying model, the length of one side and the geometrical moment of inertia are decided to have same values as original frame material, but shape of cross section is square. Bracket is also represented by triangle pole.

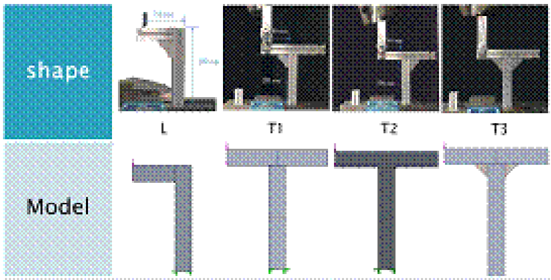


Frame

Bracket

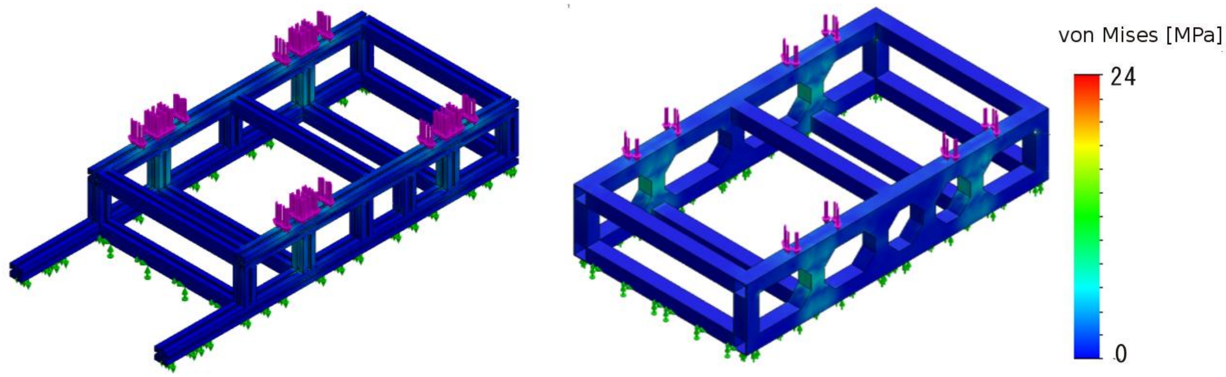
Fig. 19. Simplifying of frame and bracket.

Figure 20 shows that experience of four different shapes of joint connection, and its simplifying model shown with its yielding load, bracket size and stress of frame. In this figure, bracket sizes were decided that the calculated stress become almost same value, comparing with actual yielding stress. In the experiment, yielding stress of the frame was 286.4 MPa. Therefore, the accuracy of calculations is within 5% comparing with experiments.



	L	T1	T2	T3
Yielding Load [N]	1200	800	2100	3000
Bracket size [mm]	2.8	1.6	4.5	25
Stress of frame [MPa]	273.4	289.8	270.6	301.2

Fig. 20. Decided bracket size in simplify model.



Analysis at first design   Analysis using simplified model

Fig. 21. The result of elasto-plastic analysis.

Figure 21 shows an example of application of simplify model. This is a part of frame structure shown in Figs. 1 and 2. Left figure is the model of first design. In this model, joints are all bonded as default setting of CAD software. Right figure is the model using simplify joint with sized bracket to express actual joint of frame.
The maximum stress value of frame was 24.0 MPa at first design, and the calculation time was 11 seconds. In the simplify model, the maximum stress value is 10.5 MPa, and calculation time is 1 second, therefore it has been reduced 90%. Then, using simplify model, we can get 60% lower value of stress, so the first design was overestimate of the stress and over-strength. From this result we can obtain more suitable and accurate stress estimation using simplify model with short calculating time.

6.5 Discussions

From the results of analysis by using simple model shown in Fig. 3 for various jointing conditions, different jointing condition in CAD modeling is treated as same condition in analytical model. Therefore, it is important for evaluation of strength and safety of assembled structure to know certain information of internal treatment of CAE analysis model. It is very dangerous to use CAE analysis as black box in safety design and strength evaluation of assembled structure with jointing parts. Even for the simple assembled structure, different breaking mode and different value of breaking load are observed. Though almost same yielding load is obtained for the same breaking mode, different breaking mode is obtained by CAE analysis for assembled structure. It is insufficient for evaluation of safety. From this analysis result, the bolt yielded first. On the other hand, the frame yielded by the nut on the experiment. The axial force caused by fastening torque was not considered in the analysis, but it affects practically. Thus, by using model considering the axial force, the analysis result showed the same yielding part with practically result. The analysis model approached more actual structure. However, the stress analysis result was too higher than experimental result, it is necessary to improve the contact condition and to obtain the correct analysis model.

To make good use of CAE effectively for safety evaluation, it is necessary to examine the model, data sheet, contact condition, loading condition, etc. used in the analysis. And suitable modeling method and procedure which can well reproduce experimental result is expected. Also, easy model which can evaluate accurately each contact area is needed. The goal of this study is to propose the method and procedure which can easily evaluate safety and strength of assembled structure with many joints and various jointing conditions. By applying the method, safety evaluation of actual assembled structural product will become possible. Here, we could propose an example of simplify model, and its shows suitable estimation of strength with short calculating time.

7. Conclusions

In this study, the suitable analysis condition for evaluation of various jointing methods was examined utilizing commercial products and parts for structure, which can make mechanical assembling easy. Analysis was carried out for each joint, and finally the assembled whole structure was evaluated. Also, we proposed simplifying model to express actual joints easily and to satisfy sufficient accuracy. The results obtained in this research are summarized as follows.

1. Mating conditions in CAD assembly model did not influence the analysis condition, so it is necessary to set contact conditions individually for each part. Even if the modeling is performed in detail for actual jointing parts and method, we cannot obtain correct analytical result when we cannot set suitable contact conditions or suitable acting forces.

2. Shaft of bolt has pre-load caused by fastening torque in practical use, and in the analysis of this study, the pre-load was represented by using of thermal stress caused by the temperature difference between shaft of bolt and other parts. The analysis became practical by using the bolt pre-load, and revising contact conditions, and using elasto-plastic analysis, and the result well agreed with the experimental result. However, it required calculating time about 87 times the elastic analysis.

3. Based on these results, a simplified analysis model was proposed, and approximate correct value of stress was obtained in short calculation time. The proposed simplifying model can well express the frame strength with bracket joint accurately with short calculating time.

4. The strength analysis of the product which was designed and manufactured before was performed again by using the proposed simplified analysis model, and it was confirmed that it was a frame with enough strength.

References

[1] Yokoyama, T. Fracture Risk Evaluation of Railroad Wagon Rigging Rail Fastening Part and Loosing Behavior Analysis by 3D Finite Element Method, Master's Thesis of The University of Tokyo, 2004, (in Japanese).

[2] Yoshimoto, I. Design Point of Screw Fastening, Japanese Standards Association, 1992, (in Japanese).

[3] Sakamoto, T. Electric load analysis of a hybrid electric vehicle, Journal of Mechanical Systems for Transportation and Logistics, Vol. 2(1), 2009, pp. 66-77.

[4] Nishi, M., Ohbuchi, Y., Sakamoto, H. and Inoue, H. Development of evaluation equipment of electric vehicle for student project team, Proc. Int. Conf. of Engineering Education and Research (iCEER2014), Mo5_4, 2014.

[5] Solar Car Race Suzuka, URL: http://www.suzukacircuit.jp/solarcar_s/, referred on Aug. 2016.

[6] NIC Autotec, Inc., URL: http://www.nic-inc.co.jp/alfaframe/, referred on Aug. 2016.

[7] Miyamoto, M. Performance evaluation method by test board, Production Study, Vol. 13(5), 1961, pp.146-153.

[8] Nishi, M., Ohbuchi, Y. and Sakamoto, H. Evaluation of assembled structure for safety design, Proc. 4th Asian Conference on Engineering Education (ACEE2014), 2014, pp. 223-224.