Transactions of the **14th International Conference on Structural Mechanics in Reactor Technology** (SMiRT 14), Lyon, France, August 17-22, 1997

F08/2

Optimization of the contact area for the evaluation of bolted member stiffness

Kim T.W., Kim K.S., Park K.B.
Korea Atomic Energy Research Institute, Korea

ABSTRACT

A new approach to estimate the radii of the contact area in the bolted members is proposed. Typical optimization technique is combined with the finite element method to improve the disadvantages in the existing works. The compatibility of the finite element model with physical model is enhanced with gap elements. The results show that the proposed method is less affected by the mesh patterns or changes in the magnitude of the applied loads.

INTRODUCTION

A bolted joint is widely used to connect or fix mechanical elements. When the bolt is used, an initial tensile load is applied to the bolt as a preload. The purpose of the preload is to clamp the bolted members in compression for better resistance to either static or dynamic loads and to develop the friction force in order to resist the shear loads. Since the joint should maintain proper tightness at the interface plane during design life, the exact determination of the preload is essential for integrity of the joint. When the external load F is applied to the bolted joint with initial preload of P, the resultant forces developing in the bolt and member are as follows[1].

$$F_b = \frac{K_b F}{K_b + K_m} - P \quad (1)$$

$$F_m = \frac{K_m F}{K_b + K_m} - P \quad (2)$$

Where, K_b is the stiffness of the bolt and K_m is the stiffness of member. Two equations above indicate that the member stiffness is a fundamental parameter for the design of the bolted joint.

There are lots of literature dedicated to the estimation of the member stiffness with both theoretical and experimental trial. Early works applied the theory of elasticity to develop the stress distribution occurring in members[2] and induced several types of approximate equations as well as experimental back-ups. The frequent assumption adopted to develop the equations is that the stresses induced in the members are uniform throughout a region surrounding the bolt hole with zero stress outside this region. Since the real stress is very complicated, simplified shapes are referred to develop the design purpose equations associated with the assumption of uniform stress distribution[1][2][3]. Then, the discrepancies between each design equation is rather scattered.

Many authors introduce the finite element method to provide more realistic profile of the stress distribution and factors affecting on the characteristics of the member stiffness. Most of works are devoted to develop the simple but realistic finite element model and to derive the empirical equation using finite element analysis data. The advantage of the finite element analysis is to monitor the separation of the members at interface plane as shown in Fig.2. Though the results of the finite element analysis derived so far provides reasonable agreement with experimental results[3][4][5], there are still issues pending.

One of the main issues is to improve the accuracy of the results obtained from the typical finite element analyses. Past works suggest two alternatives for the issue. The one is to neglect the influence of the separation under the assumption that the global stiffness of members are essentially same regardless of the phenomenon. Then, this technique increases the area of stress distribution because of the surplus stresses caused by restraint imposed on the separation region. Another is an iterative method, which reforms a successive mesh patterns after monitoring the stress contour at the contact area in every trial run. The introduction of finer mesh in the vicinity of the contact area or the adoption of the gap element as supports may enhance the validity of this technique. The main disadvantage of this algorithm is that the accuracy of the analysis is governed by the initial mesh patterns and the magnitude of external load. This also requires the additional trial runs with finer mesh and tedious works of checking stress distribution.

A new approach combining typical finite element method with general optimization technique is proposed to enhance the convenience and accuracy of the existing method. This method consists of typical finite element analysis and optimization routine, which performs sequential searches to estimate the contact region by monitoring the stress condition at the contact zone.

ANALYSIS

Fig.3 indicates the finite element model used in this study. Since the present study is limited to joints in which both members are of the same material and equal thickness, only the upper part of the joint is modeled based on the symmetry. Therefore, the bottom of the member could be modeled as a rigid boundary. Typical four nodes axisymmetric elements are used to model the main body of the member and gap elements are introduced to simulate the bottom plane. Since the gap elements permit the displacement caused by both separation and sliding, the compatibility with physical model could be highly enhanced. The main disadvantage of the gap element is to increase the computational expenses because of its nonlinearity. The contact stiffness of the gap element is defined as larger number in order to model the rigid support as shown in Fig.4. A fixed pattern of mesh and element size is developed after considering the total dimension of the members. Typical element size in radial direction is chosen within the range from 2% to 10% of the radius of member, while a 6% or 10% of the thickness is used for the vertical mesh size.

The design variable, objective function and constraint equations for the optimization are reversely designed by considering the optimized case. If the radius of the contact (Rc in the Fig. 3) in each trial run is kept as the design variable, the value of Rc should coincide with the radius of the real contact zone in the finial case. The proposed objective function, design variable and constraint equations are as follows.

Objective Function ;

minimize ; displacement developed at supports area

$$\text{objective function} = \sum_{r=R_b}^{R_c} |u_z(r)| \qquad (3) \qquad \text{, or}$$

minimize ; contact area

$$\text{objective function} = R_c^{\,2} \qquad (3\text{-a})$$

constraint equations ;

$$0.0 < R_c < \text{max. radius of member} \qquad (4)$$

$$\sigma_z = 0 \quad \text{at} \quad R_c \qquad (5) \qquad \text{, or}$$

$$F_z = 0 \quad \text{at} \quad R_c \qquad (5\text{-a})$$

The objective function defined through equation (3) controls the radius of contact zone by monitoring the vertical displacement in the temporary contact zone. When the solution reaches to the optimum position, the value of the objective function shall be close to a small number. Otherwise, the number shall be large due to the separation or virtual compression. Since the equation (3) is designed to trace the magnitude of separation, the stability of the solution is highly affected by the magnitude of the applied load and dimensions of the member. The other objective expressed as equation (3-a) is designed to extract the solution by means of the constraint function only. Since the optimum position is mainly controlled by the constraint criteria, efforts to prevent the local minimum is unavoidable. The constraint function defined as the equation (5) pursue the magnitude of the stress component (referred as σ_z in the Fig.2) at the Rc whether the current stress value is close to the contour of zero stress distribution. When the ambiguity of the constraint criteria because of the complexity in the stress field happens, the reaction forces expressed as equation (5-a) could be an alternative to the equation (5) in view of the quantitative approach. The criteria adopted for both constraint equations is less than 0.1% of the applied load.

The finite element analyses and optimization works are performed on a HP/Apollo workstation using ANSYS version 5.2[9]. An internal optimization module attached in ANSYS is used to solve the optimization problem. Detailed literature on the optimization technique and the finite element analysis can be found in Ref. 9.

RESULTS AND DISCUSSION

Fig.5 indicate the typical results of the optimized case and temporary one. While the temporary case shows an excessive displacement, the optimized case closely meets the presumed criteria. Then, it is verified that the proposed functions and criteria in the optimization procedure is compatible with the physical behavior of the member. The functions and criteria adopted for the proposed method are designed with consideration of the stress field in the member. Then, the stability and sensitivity of the solution is controled by the analysis conditions such as dimensions of the member, material properties and the magnitude of the applied load. The solution obtained with combination of the equation (3) and (5) reveals some numerical difficulties in the specific ranges because of the complexity of the strain field. In this case, the oscillation of stress field around the end of contact zone was monitored. And the ambiguity in tracing the contour of the displacement with stress criteria produces an oscillatory decision in optimization procedure. After reviewing the possible combination of the objective and constraint function, the set consisted of the equation (3-a) and (5-a) is concluded to be the best one. The deviation of the solution obtained by the equation (3-a) and (5-a) with fixed mesh size falls within 3% as the analysis parameters vary.

Table 1 lists the deviations obtained after applying various mesh patterns. While the mesh size is varied from 2% or 3% to 10%, a maximum 5% of deviation is found when the coarse mesh pattern is replaced with the finer one. This shows that the influence of the mesh pattern on the convergence and accuracy of the solution is effectively mitigated by introducing the proposed optimization technique. Thus, the surplus works required to generate finer mesh as well as tedious trials to monitor the contact radius could be saved by the proposed method.

Fig.6 and 7 compare the results of the present study with existing works based on the common finite element analyses and experimental investigations. The general trend of the present study represents a parabolic curve with increases in the thickness of the member while the typical finite element analyses still trace a linear one. But the deviation obtained from both cases is still acceptable. When the ratio of the T/Rw is small as shown in the Fig.6, the present study suggests smaller contact area than the results of the previous finite element analyses. Because the existing model[3][4] does not permit the separation in the model, an excessive stress caused by preventing the separation is expected. This stress is believed to increases the contact area and the influence by this stress might mitigated as the thickness of the member decreases as shown in Fig.7. The experimental data shows an agreement with the present study as shown in Fig.6. The discrepancy in criteria to locate the contact zone and the influence of the mesh pattern might be considered as additional causes. Referring to the Fig.7, the present study offers an increasing deviation as the thickness of the member becomes a large number. The main reason of this issue is due to increases in radial displacement. When the thickness of the member increases, the radial displacement tends to broaden the strain field. The boundary condition of the proposed model is designed to permit separation and radial displacement using gap elements, then an increase in the contact radius can be expected as a result. The inherent dissimilarity and other numerous uncertainties residing in each technique could also provide an additional discrepancy. Therefore, it is concluded that the compatibility with the physical model is enhanced by introducing the proposed method.

The evaluation of the stiffness is performed using the displacement developed in the area where the external load is applied [3][5]. Fig.8 compares the results of the present study with the existing works. Referring to the Fig.7 and 8, the stiffness values are inversely proportional to the radii of contact zone. But the magnitude of the deviation in the stiffness is not directly proportional to the variation of the contact radius.

CONCLUSIONS

The feasibility of the proposed technique is proven to be valid through discussions on the stability and accuracy of results. The best set in the optimization formulation is achieved by the combination of the contact area(eq'n (3-a)) and reaction forces(eq'n(5-a)). The influence of the mesh pattern on the convergence and accuracy in the solution is minimized by introducing the optimization technique. Then, the costs required to determine the best available mesh pattern could be saved by the present study. After comparing the results with existing ones, it is proved that the present study enhances the compatibility of the finite element model with the physical model by introducing the optimization technique. The result shows that the stiffness values are inversely proportional to the radii of contact zone while the magnitude of the deviation in the stiffness is not directly relevant to the contact radius.

REFERENCES

1. Shigley, J.E., and Mischke, C.R., 1989, *Mechanical Engineering Design*, 5th Edition, McGraw-Hill : New York, NY
2. Motosh, N., 1976, Determination of joint stiffness in bolted connection, ASME *Journal of Engineering for Industry*, Vol. 98, No.3 : 858-861
3. Wileman, J., Choudhury, M., and Green, I., 1991, Computation of member stiffness in bolted connections, ASME *Journal of Mechanical Design*, Vol.113 : 432-437
4. Gould, H.H., and Mikic, B.B., 1972, Areas of contact and pressure distribution in bolted joints, ASME *Journal of Engineering for Industry*, Vol.94, No.3 : 864-870
5. Tang, J., and Deng, Z., 1988, Better stress and stiffness estimates for bolted joints, Machine Design, Nov. : 114-117
6. Grosse, I.R., and Mitchell, L.D., 1990, Nonlinear axial stiffness characteristics of axisymmetric bolted joints, ASME Journal of Mechanical Design, Vol.112 : 442-449
7. Lehnhoff, T.F., Ko, K.I., and Mckay, M.L., 1994, Member stiffness and contact pressure distribution of bolted joints, ASME *Journal of Mechanical design*, Vol.116 : 550-557
8. Lehnhoff, T.F., and Wistehuff, W.E., 1996, Nonlinear effects on the stress and deformation of bolted joints, ASME *Journal of Pressure Vessel Technology*, Vol.118 : 54-58
9. S.A.S., 1995, ANSYS User's Manual Rev. 5.2, Vol. I - IV, Swanson Analysis Systems, Houston, PA.

Table 1 Deviation of the contact area caused by coarse mesh pattern

Objective Function	Area Function		Displacement Function	
Mesh Cases / T / Rw**	7%*	10%*	7%*	10%*
0.31	2.3	2.5	2.3	3.8
0.59	1.6	3.5	1.7	3.5
0.91	1.9	3.1	1.9	3.1
1.20	1.6	3.4	1.5	2.9
Mesh Cases / T / Rw***	7%**	10%**	7%**	10%**
0.83	1.9	4.9	1.0	3.7
1.67	2.7	4.1	2.7	3.9
2.50	1.8	3.5	1.6	2.7
3.33	2.1	4.2	1.3	3.5

* Reference data is obtained by 2% mesh pattern.

** Reference data is obtained by 3% mesh pattern.

*** See Fig.2

unit) %

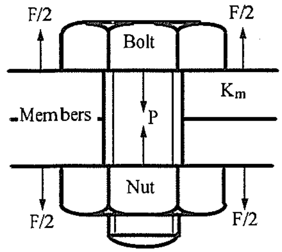

Fig.1 Configuration of a bolted joint

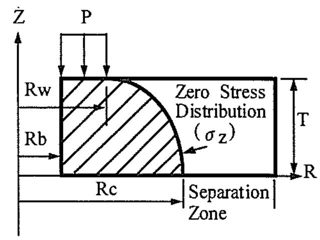

Fig.2 Assumed stress field

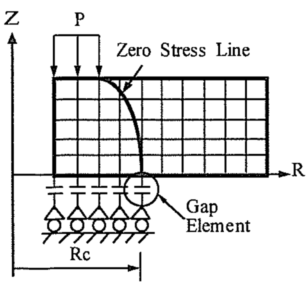

Fig.3 Finite element model with gap elements

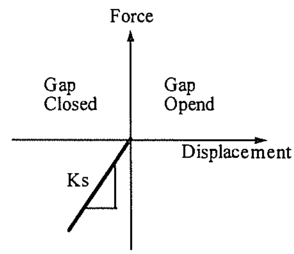

Fig.4 Bi-linear stiffness of a gap element

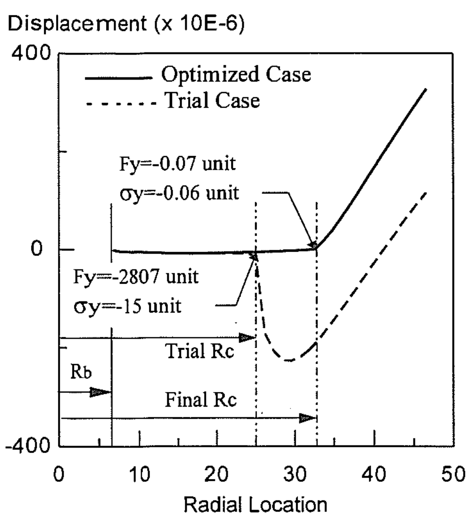

Fig.5 Results of the objective and constraint value for the optimum and trial case

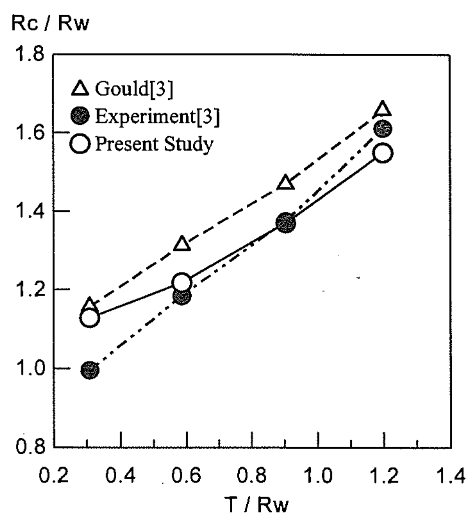

Fig.6 Variation of the contact radius with increase of the member thickness (T/Rw < 1.4)

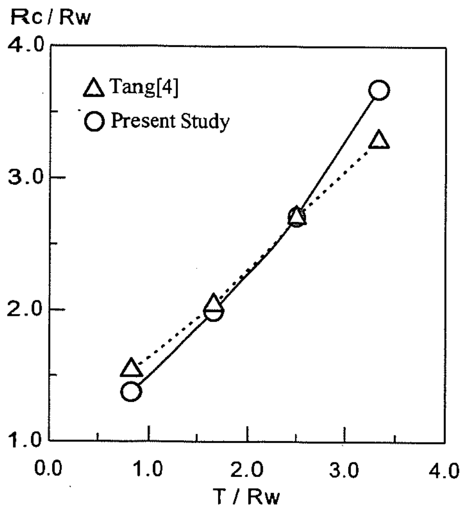

Fig.7 Variation of the contact radius with increase of the member thickness (T/Rw < 4.0)

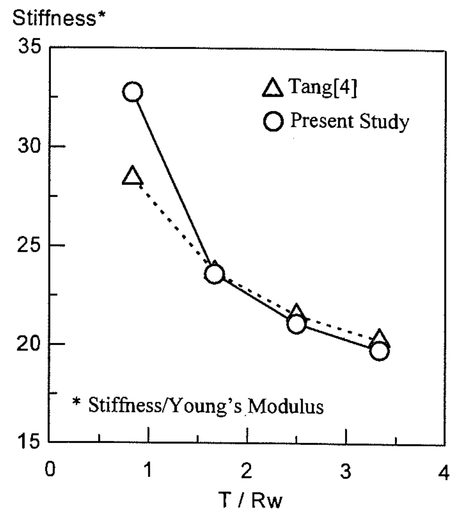

Fig.8 Variation of the member stiffness with increase of the member thickness (T/Rw < 4.0)