Estimation method of interfacial stiffness of bolted joint
in multi-material structure by inverse analysis

Yoshinao KISHIMOTO*, Yukiyoshi KOBAYASHI*, Toshihisa OHTSUKA*, Akira MATSUMOTO* and
Motoi NIIZUMA*
* Department of Mechanical Engineering, Tokyo City University
1-28-1 Tamazutsumi, Setagaya-ku, Tokyo, Japan
E-mail: ykishimo@tcu.ac.jp

Abstract
Multi-material structure is expected to be the main scheme to construct automobiles. Many methods have been
studied to properly fix or bond dissimilar materials. The fixing by the bolts and nuts is one of the primary fixing
methods, and has the advantage of easy assembly and disassembly. The interfacial stiffness of the bolted joints is
lower than the stiffness of the base materials and varied by the clamping force of the bolts and nuts, because the
micro asperities formed on the interfaces just contact each other. The contact analysis by using the surface profile
of the interfaces in microscale is one of the accurate estimations. However, the estimation method of the interfacial
stiffness in macroscale is also necessary for the in-situ evaluation. This study has developed an estimation method
of the interfacial stiffness by the inverse analysis of the clamping force and the natural frequency of the structure.
The inverse analysis algorithm introduces the mathematical model of the interfaces in which the contact of the
surfaces is assumed to be the contact of the elastic asperities whose peak heights obey the Gaussian distribution.
The hammering test was conducted by using the specimen which consisted of the steel plate and the aluminum
alloy plate joined by the bolts and nuts. Moreover, as the contact analysis, the finite element method simulated the
contact of the asperities formed on the interfaces by using the surface profile of the interfaces. The results showed
that the proposed method could estimate the interfacial stiffness which reproduced the natural frequency of the
specimen subjected to the clamping force of the bolts and nuts. The interfacial stiffness estimated by the proposed
method was comparable to that calculated by the contact analysis.

Keywords : Inverse problem, Finite element method, Multi-material structure, Bolted joint, Interfacial stiffness,
Natural frequency, Clamping force, Surface roughness

1. Introduction

Multi-material structure is expected to be the main scheme to construct automobiles. Many methods have been
studied to properly fix or bond dissimilar materials. The fixing and bonding methods seem to be classified into three
categories; welding, adhesion and mechanical fixing. Especially, the mechanical fixing techniques, such as the fixing by
the bolts and nuts, have advantages in that intermetallics are not produced and the assembling process can be simplified
by the automation. Beside the fixing by the bolts and nuts, recent studies have developed the mechanical clinching and the
riveting in order to achieve fast joining (Lambiase, 2015; Chen et al., 2016; Lambiase et al., 2016; Atia and Jain, 2017;
Hirsch et al., 2017; Lambiase and Ilio, 2018; Ma et al., 2018). Also the mechanical clinching and the riveting require the
proper clamping force. The high clamping force damages the clamped members, and the low clamping force results in
the failure of the fixing. The interfacial stiffness depends on the clamping force as with the fixing by the bolts and nuts.

This study has focused on the fixing by the bolts and nuts. The fixing by the bolts and nuts is one of the primary fixing
methods, and has the advantage of easy assembly and disassembly. For the construction of the multi-material structure,
the bolts and nuts are not only used to simply fix the dissimilar materials, but also often applied to additionally support
the polymer adhesives (Nichols et al, 2006; Catalanotti et al., 2011; Coelho and Mottram, 2015; Hammami et al., 2016).
However, the interfacial stiffness of the bolted joints is lower than that of the welding part and varied by the clamping force of the bolts and nuts. The estimation of the interfacial stiffness is one of the most important concerns for the design of the structure in that the joints are often the weakest part in the structure. Some studies have claimed that the reduction of the interfacial stiffness is caused by the fact that the micro asperities formed on the interfaces just contact each other, and the force is transferred by the contact of the micro asperities (Ito et al., 1977; Koizumi et al., 1978; Ono et al., 1991; Kishimoto and Endo, 2007; Yamamoto et al., 2008). Even if the polymer adhesives is inserted into the interfaces, the interfacial stiffness is mainly dominated by the contact of the asperities because the stiffness of the polymer is much lower than that of the base materials such as steel and aluminum alloy.

In the field of tribology, the contact mechanics and the mathematical models of the micro asperities have been discussed (Greenwood and Williamson, 1966; Greenwood and Tripp, 1970; Olofsson, 1995; Björklund, 1997; Olofsson and Hagman, 1997; Hagman and Olofsson, 1998). The general approach seems to be based on the assumption in which the peak heights of the asperities obey the Gaussian distribution and each contact of the asperities is the Hertzian contact. With the development of the computational approaches, some studies in recent years have utilized the finite element method (FEM) for the contact analysis of the interfaces in microscale (Karupannasamy et al., 2013; Hol et al., 2015; Wagner et al., 2015; Santhapuram and Nair, 2017). These studies have actually measured or preliminarily presumed the surface profile of the interfaces, and created the finite element (FE) model of the micro asperities.

Although the contact analysis by using the surface profile of the interfaces in microscale is one of the accurate estimations, the estimation method of the interfacial stiffness in macroscale is also necessary for the in-situ evaluation. This study has proposed an estimation method of the interfacial stiffness by the inverse analysis of the clamping force and the natural frequency of the structure. The inverse analysis algorithm introduces the mathematical model of the interfaces in which the contact of the surfaces is assumed to be the contact of the elastic asperities whose peak heights obey the Gaussian distribution. The hammering test was conducted by using the specimen which consisted of the steel plate and the aluminum alloy plate joined by the bolts and nuts. In order to evaluate the proposed method, the direct calculation of the mathematical model of the interfaces and the contact analysis were also conducted by using the measured surface profile because it is difficult to obtain the true value of the interfacial stiffness. In the contact analysis, the FEM simulated the contact of the asperities formed on the interfaces. This paper describes the details of the proposed estimation method and the results of the comparison of the proposed method, the contact analysis and the direct calculation.

2. Estimation method of interfacial stiffness

2.1. Mathematical model of interfaces

Figure 1 illustrates the contact model of nominally flat surfaces. As shown in the left of Fig. 1, the piece 1 and the piece 2 are joined under the compressive stress \( p_z \). \( z \)-axis is defined along the normal direction of the interfaces of the pieces. \( x \)-axis and \( y \)-axis are defined to be normal to each other as shown in Fig. 1. The contact model describes the contacts of the spherical asperities shown in the middle of Fig. 1. The orthotropic model shown in the right of Fig. 1 simulates the interfacial stiffness. The total height of the asperities is \( L \), and the gap between the mean planes of the asperity peak heights is \( d \). The elastic modulus \( E_z \), \( G_{yz} \) and \( G_{zx} \) are derived by the following process, and the elastic modulus on the other directions is set to zero.

Assuming that the asperity peak heights obey the Gaussian distribution and the contact of asperities is the Hertzian contact, the relationship between the nominal compressive stress \( p_z \) and the gap \( d \) is described by using the homogenized elastic modulus \( C \) as follows (Greenwood and Tripp, 1970).

\[
P_z = C F_0 (d)
\]

(1)

where

\[
F_0 (d) = \frac{1}{\sqrt{2 \pi } \sigma} \int_0^\infty \xi^m \exp \left[ -\frac{1}{2} \left( \xi + \frac{d}{\sigma} \right)^2 \right] d\xi
\]

(2)

\[
\sigma = \sqrt{\sigma_1^2 + \sigma_2^2}
\]

(3)

Using the subscript \( i \) as the identifier for the piece \( i \) (= 1, 2), \( \sigma_i \) is the standard deviation of the asperity peak heights of the piece \( i \). The homogenized elastic modulus \( C \) is described as follows.

\[
C = \frac{64 \pi \sigma^2 \eta_1 \eta_2}{15} \left( \frac{1}{\beta_1} + \frac{1}{\beta_2} \right)^{-2} \left( \frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \right)^{-1}
\]

(4)

© The Japan Society of Mechanical Engineers
where \( n_i \) is the number of the asperity peaks per unit area, and \( \beta_i \) is the mean radius of the curvature at the asperity peaks. \( E_i \) and \( \nu_i \) are the Young’s modulus and the Poisson’s ratio of the base material of the piece \( i \), respectively. In Eq. (3) and Eq. (4), the standard deviation \( \sigma \) and the homogenized elastic modulus \( C \) depend on the surface topography of the interfaces and the material constants of the pieces, and are independent of the stress state. The parameters \((\sigma, C)\) are collectively named as the surface texture parameters, hereinafter.

Assuming that the small displacement occurs in the vibration of the specimen, the \( z \)-direction Young’s modulus of the interfaces can be described as follows.

\[
E_z = -\frac{\partial p_z}{\partial(d/L)} = \left(\frac{F_1'(d)}{F_1(d)} + \frac{d}{\sigma} \right) \frac{p_z L}{\sigma}
\]

Similarly, Björklund (1997) derived the relationship between the shear stress and the displacement tangential to the interfaces under the compressive stress \( p_z \) based on the Mindlin’s theory (Mindlin, 1949; Mindlin et al., 1952). Assuming that the stiffness in the tangential direction to the interfaces is isotropic, the shear modulus is described as follows.

\[
G_{yz} = G_{zy} = 4 \left( \frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \right) \left( G_1 (\frac{2 - \nu_1}{G_1} + \frac{2 - \nu_2}{G_2}) \right)^{-1} E_z
\]

where \( G_{yz} = G_{zy} \) is the shear modulus at the \( y \)-surface in the \( z \)-direction, and \( G_{zy} \) is the shear modulus at the \( x \)-surface in the \( z \)-direction. \( G_i (i = 1, 2) \) is the shear modulus of the base material of the piece \( i \).

In the FEM using the above model, the finite element whose elastic modulus are given by Eq. (5) and Eq. (6) is inserted into the interfaces. The finite element is named as the interfacial element, hereinafter. The height \( L \) and the gap \( d \) in Fig. 1 are equal to the thickness and the displacement of the interfacial element, respectively. In the bolted joints, the compressive stress \( p_z \) can be calculated by the stress analysis using the FEM if the clamping force of the bolts and nuts is given. Although the elastic modulus \((E_z, G_{yz}, G_{zy})\) is undetermined in the stress analysis, the interfacial element is approximated to be the isotropic material whose Young’s modulus \( E \) and the Poisson’s ratio \( \nu \) as follows.

\[
E = \left( \frac{1}{E_1} + \frac{1}{E_2} \right)^{-1} \quad \text{and} \quad \nu = \frac{\nu_1 + \nu_2}{2}
\]

And then, the stress analysis is executed and the compressive stress \( p_z \) and the elastic modulus \((E_z, G_{yz}, G_{zy})\) are obtained.

### 2.2. Inverse analysis

Focusing on Eq. (1) and Eq. (5), the Young’s modulus \( E_z \) depends on the compressive stress \( p_z \) and the surface texture parameters \((\sigma, C)\) by solving Eq. (1) for the gap \( d \). The natural frequency of the specimen eventually depends on only the Young’s modulus \( E_z \) in the FEM because of Eq. (6). Therefore, the problem setting is that the surface texture parameters \((\sigma, C)\) are estimated from the data set of the clamping force and the natural frequency.

The specific steps are as follows. Using the subscript \( j \) as the identifier for the data set \( j (= 1, 2, \cdots, n) \), where \( n \) is the total number of the data set, the natural frequency of the \( j \)-th data set in the hammering test is denoted by \( f_j^{\text{EXP}} \) and the natural frequency of the \( j \)-th data set in the FEM is denoted by \( f_j^{\text{FEM}}(\sigma, C) \).

First, the solution space \((\sigma, C)\) is meshed by the 2D finite elements, and the natural frequency at each node \( f_j^{\text{FEM}}(\sigma, C_k) \) is calculated by the modal analysis of the FEM. The subscript \( k \) is the identifier for the \( k \)-th node of the meshed solution space. Using the superscript \( (l) \) as the identifier for the \( l \)-th element of the meshed solution space, the natural frequency \( f_j^{\text{FEM}}(\sigma, C) \) at the arbitrary \((\sigma, C)\) in the \( l \)-th element is interpolated with the 2D piecewise polynomial interpolation which is the well known interpolation method used in the FEM as follows.
\[ f_j^{\text{FEM}}(\sigma, C) = \sum_{k} N_k^0(\xi^0, \eta^0) \times f_j^{\text{FEM}}(\sigma_k, C_k) \quad (k\text{-th node } \in \text{l-th element}) \] (8)

where \( N_k^0(\xi^0, \eta^0) \) is the shape function of the l-th element. \( \xi^0 \) and \( \eta^0 \) are the local coordinates in the element. If the finite element for the solution space is isoparametric,

\[ \sigma = \sum_{k} N_k^0(\xi^0, \eta^0) \times \sigma_k \quad \text{and} \quad C = \sum_{k} N_k^0(\xi^0, \eta^0) \times C_k \quad (k\text{-th node } \in \text{l-th element}) \] (9)

Equation (9) can be solved for the local coordinates \((\xi^0, \eta^0)\) by the Newton-Raphson method.

Second, giving the natural frequency in the hammering test \( f_j^{\text{EXP}} \), the residual sum of squares at each node of the meshed solution space \( \Pi^{\text{PPI}}(\sigma_k, C_k) \) is calculated as follows.

\[ \Pi^{\text{PPI}}(\sigma_k, C_k) = \sum_{j=1}^{n} \left( f_j^{\text{EXP}} - f_j^{\text{FEM}}(\sigma_k, C_k) \right)^2 \] (10)

As with the natural frequency \( f_j^{\text{FEM}}(\sigma, C) \), the residual sum of squares \( \Pi^{\text{PPI}}(\sigma, C) \) subjected to the arbitrary \((\sigma, C)\) in the l-th element is interpolated as follows.

\[ \Pi^{\text{PPI}}(\sigma, C) = \sum_{k} N_k^0(\xi^0, \eta^0) \times \Pi^{\text{PPI}}(\sigma_k, C_k) \quad (k \in \text{l-th element}) \] (11)

Third, the coordinate of the node \((\sigma_k, C_k)\) that minimizes the residual sum of squares \( \Pi^{\text{PPI}}(\sigma_k, C_k) \) is set to the initial, and the coordinate \((\sigma, C)\) that minimizes the following residual sum of squares \( \Pi(\sigma, C) \) is searched by the downhill simplex method (Nelder and Mead, 1965).

\[ \Pi(\sigma, C) = \begin{cases} \sum_{j=1}^{n} \left( f_j^{\text{EXP}} - f_j^{\text{FEM}}(\sigma, C) \right)^2 + w \times \Pi^{\text{PPI}}(\sigma, C) & \text{(inside meshed solution space)} \\ \text{penalty} & \text{(outside meshed solution space)} \end{cases} \] (12)

where \( f_j^{\text{FEM}}(\sigma, C) \) and \( \Pi^{\text{PPI}}(\sigma, C) \) are calculated by using Eq. (8) and Eq. (11). \textit{penalty} is a large value for the penalty in the case of that the search point \((\sigma, C)\) is located outside the meshed solution space in the process of the downhill simplex method. The parameter \( w \) is the weight between the first term and the second term. The first term often gives the lower residual sum of squares than the interpolated value \( \Pi^{\text{PPI}}(\sigma, C) \), and the second term works as the stabilization term. The weight \( w \) is set to the minimum value that prevents the search point \((\sigma, C)\) from converging the edge of the meshed solution space in the downhill simplex method.

3. Experiment

3.1. Hammering test

Figure 2 shows the schematic of the specimen and the picture of the experimental setup. The specimen was composed of the attached plate and the base plate joined by the 8 sets of M6 bolts, nuts, washers and collars. The plates were made of the S50C steel or the A5052 aluminum alloy. Hereafter, the specimen for the combination of the materials is named as St-St specimen (the steel attached plate and the steel base plate), Al-Al specimen (the aluminum attached plate and the aluminum base plate) and St-Al specimen (the steel attached plate and the aluminum base plate). The surface roughness of the interfaces of the plates was given by the maximum height roughness of the surface profile \( R_z \) [\( \mu \text{m} \)]; 100 ~ 50 (rough finishing), 25 ~ 12.5 (usual finishing) or 6.3 ~ 3.2 (fine finishing). The surface profiles of the interfaces were measured by the surface roughness meter (SJ-411 made by Mitutoyo Corporation) before assembling and after disassembling the specimen. The surface profiles were not changed by the assembling. Table 1 shows the surface texture parameters derived by the surface profiles. The bolts were made of the SCM steel and their strength class was 10.9. The nuts, the washers and the collars were made of the S45C steel.

In the hammering test, the specimen was put on the soft sponges, and the acceleration pickup (NP-3211 made by Ono Sokki Co., Ltd) was attached to the edge of the specimen. The uniaxial strain gauges were attached to the collars. The clamping force of the bolts and nuts were measured by the compressive strain on the collars. After the clamping force was applied by the torque wrench, the specimen was hammered by the impulse hammer (GK-3100 made by Ono Sokki Co., Ltd) and the acceleration response was measured by the acceleration pickup. In this test, all of the sets of the bolts and nuts were tightened to apply the same clamping force. The value of the clamping force of each set was given to be from 1 kN to 8 kN that is 40 % of the tensile strength of the class 10.9 bolt. The natural frequency of the specimen was...
Fig. 2  Schematic of specimen and experimental setup. The constituent material of the plates is the S50C steel or the A5052 aluminum alloy. The surface roughness of the interface was given by the maximum height roughness of the profile $R_z = 100 \sim 50, 25 \sim 12.5$ or $6.3 \sim 3.2 \mu m$. In the hammering test, the clamping force of the bolts and nuts was measured by the compressive strain on the collars. The natural frequency of the specimen was obtained by the acceleration response after hammering the specimen.

<table>
<thead>
<tr>
<th>Member</th>
<th>Material</th>
<th>Maximum height roughness $R_z$ [μm]</th>
<th>Number of asperity peaks per unit area $n_i$ [mm²]</th>
<th>Mean radius of curvature at asperity peaks $\beta_i$ [μm]</th>
<th>Standard deviation of asperity peak heights $\sigma_i$ [μm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Attached plate</td>
<td>S50C steel</td>
<td>100 ~ 50</td>
<td>156</td>
<td>3.597</td>
<td>3.611</td>
</tr>
<tr>
<td></td>
<td></td>
<td>25 ~ 12.5</td>
<td>756</td>
<td>0.821</td>
<td>0.724</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6.3 ~ 3.2</td>
<td>16625</td>
<td>2.324</td>
<td>0.410</td>
</tr>
<tr>
<td></td>
<td>A5052 aluminum alloy</td>
<td>100 ~ 50</td>
<td>306</td>
<td>3.700</td>
<td>7.843</td>
</tr>
<tr>
<td></td>
<td></td>
<td>25 ~ 12.5</td>
<td>2756</td>
<td>3.471</td>
<td>2.195</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6.3 ~ 3.2</td>
<td>9025</td>
<td>1.195</td>
<td>0.854</td>
</tr>
<tr>
<td>Base plate</td>
<td>S50C steel</td>
<td>100 ~ 50</td>
<td>156</td>
<td>1.326</td>
<td>1.827</td>
</tr>
<tr>
<td></td>
<td></td>
<td>25 ~ 12.5</td>
<td>756</td>
<td>1.000</td>
<td>0.574</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6.3 ~ 3.2</td>
<td>36100</td>
<td>1.455</td>
<td>0.529</td>
</tr>
<tr>
<td></td>
<td>A5052 aluminum alloy</td>
<td>100 ~ 50</td>
<td>189</td>
<td>3.775</td>
<td>7.338</td>
</tr>
<tr>
<td></td>
<td></td>
<td>25 ~ 12.5</td>
<td>2256</td>
<td>2.764</td>
<td>2.632</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6.3 ~ 3.2</td>
<td>13225</td>
<td>5.886</td>
<td>0.741</td>
</tr>
</tbody>
</table>

obtained by the fast Fourier Transform (FFT) of the acceleration response.

In the inverse analysis, the FE model of the specimen and the FE meshed solution space as shown in Fig. 3 were used. Table 2 shows the material constants of the FE model of the specimen. This study selected the 1st natural frequency (the lowest natural frequency except the rigid-body mode) when the clamping force $P = 1, 2, \cdots, 8$ kN as the data set for the inverse analysis. The total number of the data set was $n = 8$. penalty in Eq. (12) was set to $10^{12}$.

The actual FE model was the $1/4$ model of the specimen whose planes of symmetry were the $XZ$ plane and the $YZ$ plane shown in Fig. 2. The FE model consisted of the 8-node hexahedral isoparametric elements. The numbers of the nodes and the elements of the FE model were 13198 and 10607 in total, respectively. Figure 4 shows the cross-section of the FE model. The interfacial elements were inserted into the interfaces of the attached plate and the base plate. The thickness of the interfacial element $L$ was set to 0.1 mm. The elements named as shrink element were inserted into the bolt axes. The thickness of the shrink elements was also 0.1 mm. In the stress analysis to obtain the compressive stress $p_z$, the shrink elements compressed and applied the clamping force on the bolt axes. In the modal analysis to obtain the natural frequency $f_{FEM}^{j}(\sigma_k, C_k)$, the shrink elements behaved as a part of the bolt axes by given the same material constants as the base material of the bolts. The bolt axes were elongated by 0.1 mm, but this influence could be ignored. The other interfaces and the screw thread of the bolts were also ignored. The interfaces in contact with the bolts, the nuts, the washers and the collars were formed by the common nodes. The bolt axes were $\phi 6$ solid shafts in the FE model.

The FE meshed solution space consisted of the 8-node square elements. The surface texture parameters $\log_{10}(\sigma [\mu m])$ and $\log_{10}(C$ [MPa]) were allocated to the axes. The mesh width was set to 1 and the search range was determined as $-3 \leq \log_{10}(\sigma [\mu m]) \leq 3$ and $-3 \leq \log_{10}(C$ [MPa]) $\leq 6$. The numbers of the nodes and the elements of the solution space were 193 and 54 in total, respectively.

© The Japan Society of Mechanical Engineers
Fig. 3  FE model of specimen and FE meshed solution space. The FE model of the specimen consisted of the 8-node hexahedral elements. The FE meshed solution space consisted of the 8-node square elements. The surface texture parameters $\log_{10}(\sigma [\mu m])$ and $\log_{10}(C [MPa])$ were allocated to the axes.

Table 2  Material constants of FE model of specimen

<table>
<thead>
<tr>
<th>Member</th>
<th>Material</th>
<th>Young’s modulus $E_i$ [GPa]</th>
<th>Poisson’s ratio $\nu_i$</th>
<th>Density $\rho$ [kg/m$^3$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Attached plate and base plate</td>
<td>S50C steel</td>
<td>210</td>
<td>0.3</td>
<td>7860</td>
</tr>
<tr>
<td>Base plate</td>
<td>A5052 aluminum alloy</td>
<td>70.6</td>
<td>0.34</td>
<td>2700</td>
</tr>
<tr>
<td>Bolt</td>
<td>SCM steel</td>
<td>210</td>
<td>0.3</td>
<td>7860</td>
</tr>
<tr>
<td>Nut, washer and collar</td>
<td>S45C steel</td>
<td>210</td>
<td>0.3</td>
<td>7860</td>
</tr>
</tbody>
</table>

Fig. 4  Cross-section view of FE model of specimen. The interfacial elements were inserted into the interfaces of the attached plate and the base plate. The shrink elements were also inserted into the bolt axes. The shrink elements compressed and applied the clamping force on the bolt axes in the stress analysis.

3.2. Contact analysis by using surface profile

Figure 5 shows the finite element models of the interfaces formed by the surface profiles. In Fig. 5, the upper block is the attached plate model, and the lower block is the base plate model. This study set the peaks to face each other at the initial alignment. The results in the case of the other initial alignment is shown in the appendix. The main body of the FE models consisted of the 8-node quadrilateral elements. The numbers of the nodes and the elements were 15502 and 5000 in total, respectively. Table 3 shows the material constants of the FE models. The plane strain state and the elastic-plastic deformation were assumed. By the assumption of the plane strain state, the 2-dimensional FE model can simulate the cross-section of the 3-dimensional object that is assumed to be on the plane of symmetry. The stress-strain curves were approximated by the two straight-lines approximation.
Fig. 5 FE models of interfaces formed by surface profiles. The body of the FE models consisted of the 8-node quadrilateral elements. The contact elements (3-node line elements) were attached to the contact surfaces. The coefficient of friction was set to 0.3. The plane strain state and the elastic-plastic deformation were assumed. The yellow lines at the right side of each figure show the level of the mean planes of the asperity peak heights.

Table 3 Material constants of FE models of interfaces formed by surface profiles

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s modulus $E_i$ [GPa]</th>
<th>Poisson’s ratio $\nu_i$</th>
<th>Yield stress $\sigma_{Y_i}$ [MPa]</th>
<th>Tangent modulus $A_i$ [GPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>S50C steel</td>
<td>210</td>
<td>0.3</td>
<td>400</td>
<td>21</td>
</tr>
<tr>
<td>A5052 aluminum</td>
<td>70.6</td>
<td>0.34</td>
<td>200</td>
<td>7.06</td>
</tr>
</tbody>
</table>

Fig. 6 Boundary condition of FE models of interfaces formed by surface profiles. The bottom edge and the left edge of the base plate model were fixed. The left edge of the attached plate model were fixed, and the uniform displacement was applied on the top edge of the attached plate model. The soft elements at the left side of the attached plate model allowed the horizontal displacement of the model. The nominal compressive stress was obtained from the reaction force divided by the area of base of the bottom edge of the base plate model.
The contact analysis was operated by the FEM software ANSYS 18.2. Figure 6 shows the boundary condition in the contact analysis. The contact elements which were the 3-node line elements were attached to the contact surfaces. The coefficient of friction was 0.3. The bottom edge and the left edge of the base plate model were fixed. The left edge of the attached plate model were fixed. The soft elements (the Young’s modulus is 1 Pa, the Poisson’s ratio is 0) were set at the left side of the attached plate model. These elements approximately allowed the rigid displacement of the attached plate model on the horizontal direction without the indefinite solution. The uniform displacement was applied on the top edge of the attached plate model. The compressive stress $p_z$ and the gap between the mean planes of the asperity peak heights $d$ were obtained. The compressive stress $p_z$ was assumed to be the nominal compressive stress obtained from the reaction force divided by the area of base of the bottom edge of the base plate model. The gap $d$ was obtained by subtracting the displacement of the top edge of the attached plate model from the initial gap shown as the yellow lines in Fig. 5.

4. Results and discussion
4.1. Hammering test

Figure 7 shows the experimental results by the hammering test and the recalculation results in the proposed method. The recalculation results were obtained by the modal analysis of the FEM in which the surface texture parameters ($\sigma, C$)

![Graphs showing natural frequency vs. clamping force for different specimens.](image)

Fig. 7 Results of hammering test. The high clamping force and the smooth contact surfaces induced the high natural frequency and seemed to achieve the high interfacial stiffness. The natural frequency recalculated by the FEM had good agreement with the experimental values. The results of the FEM for the monolithic specimens were higher than the experimental values and the recalculation values.

![Diagram showing example of deformation shape of the 1st vibration mode in FEM.](image)

Fig. 8 Example of deformation shape of the 1st vibration mode in FEM. The 1st vibration mode was the bending mode, and almost the same shape in all of the specimens.
were estimated by the inverse analysis of the hammering test. The dashed-dotted lines show the results of the FEM to simulate the monolithic specimen in which the material constants of the interfacial elements were set to those of the base materials. In the FEM for the imaginarily monolithic St-Al specimen, the material constants of the A5052 aluminum alloy were used for those of the interfacial elements. The example of the deformation shape of the 1st vibration mode in the FEM is shown in Fig. 8. The estimated surface texture parameters are shown in Fig. 12 described later.

In all of the specimens, the natural frequency increased with the increase of the clamping force. The increase rate of the natural frequency decreased with the increase of the clamping force, and the natural frequency was almost saturated when the clamping force was higher than approximately 5 kN. The surface roughness of the interfaces also affected the natural frequency. The high clamping force and the smooth contact surfaces induced the high natural frequency and seemed to achieve the high interfacial stiffness. According to the basic theory of vibration, the natural frequency of the solid plate is proportional to the ratio of the Young’s modulus and the density \( \sqrt{E_i/\rho_i} \) of the constituent material. The value of \( \sqrt{E_i/\rho_i} \) is 5169 m/s for the S50C steel plate and 5114 m/s for the A5052 aluminum alloy plate in reference to Table 2. Therefore, the natural frequency was on the same level regardless of the constituent material in this test.

The natural frequency recalculated by the FEM had good agreement with the experimental values. The deformation shape of the vibration mode in the FEM was the bending mode as shown in Fig. 8. The results of the FEM for the monolithic specimens were obviously higher than both the experimental values and the recalculation values taking account of the interfacial stiffness. The interfacial stiffness seemed be lower than the stiffness of the base materials.

### 4.2. Contact analysis by using surface profile

In the stress analysis for the hammering test, the maximum compressive stress of the interfacial elements was 32 MPa subjected to the clamping force \( P = 8 \) kN. Hence, the maximum compressive stress is 100 MPa subjected to the clamping force \( P = 24 \) kN in which even the M6 bolt with the strength class 12.9 fractures. The contact analysis was executed until the nominal compressive stress exceeded 100 MPa. Figure 9 shows the results of the contact analysis by using the surface profiles. The fitting curves were obtained by the least-square method in which the regression function was Eq. (1) and the regression coefficients were the surface texture parameters (\( \sigma, C \)).

![Results of contact analysis](image)

Fig. 9 Results of contact analysis. The compressive stress increased with the decrease of the gap between the mean planes of the asperity peak heights. The smooth interfaces provided the high increase rate of the compressive stress for the gap. The compressive stress at the same gap was high in the order of the St-St specimen, the St-Al specimen, and the Al-Al specimen because of the stiffness of the base materials.
In all of the specimens, the compressive stress increased with the decrease of the gap between the mean planes of the asperity peak heights. The smooth interfaces provided the high increase rate of the compressive stress for the gap. It means that the smooth interfaces provide the high interfacial stiffness because the gradient of the compressive stress against the gap is much lower than the stiffness of the base materials and seems to be equivalent to the interfacial stiffness. Moreover, the gap when the compressive stress initiated was small in the specimen whose interfaces were smooth. In only the St-St specimen \( R_z = 25 \sim 12.5 \), the compressive stress incidentally initiated when the gap was much smaller than the other specimens because of the variability of the surface profiles. The compressive stress at the same gap was high in the order of the St-St specimen, the St-Al specimen, the Al-Al specimen. This order was dominated by the stiffness of the base materials of the interfaces.

Figure 10 shows the deformation shape and the stress distribution in the contact analysis subjected to 100 MPa of the nominal compressive stress. As an example, the deformation process of the interfaces of the St-Al specimen \( R_z = 100 \sim 50 \) in the contact analysis is shown in Fig. 11. The contour shows the normal stress in the vertical direction where the compressive stress is negative. As shown in Fig. 10, the contact parts were limited to the concavities and the convexities that were close each other regardless of the roughness and the constituent materials. The compressive stress was approximately 100 MPa far from the contact parts, and the high compressive stress occurred near the contact parts. Also the yield parts were limited near the contact parts, and the deformation of the asperities was small in comparison with the size of the surface profiles. Therefore, it can be expected that the real contact area is smaller than the apparent contact area, and the deformation of the asperities is smaller than the size of the surface profiles in the actual situation.

Focusing on the behavior of the asperities in the contact analysis shown in Fig. 11, the mechanics of the interfaces can be explained as follows. First, the asperities of the closest pair contact and slide on their surfaces with the decrease of

![Fig. 10 Deformation shape and stress distribution in contact analysis under 100 MPa of nominal compressive stress.](image-url)
the gap. The contacting asperities induce the small compressive stress by their stiffness. With the further decrease of the gap, the asperities of the other pair begin to contact, slide on their surfaces and compress each other. Moreover, depending on the engagement of the asperities, the sliding of the asperities and the rigid displacement on the horizontal direction in some sites are restrained. The nominal compressive stress increases in accordance with the increase of the number of the contacting asperities and the stiffness of each asperity. When the gap decreases enough, the number of the contacting asperities is saturated, and the stiffness of the asperities mainly dominates the nominal compressive stress. Therefore, the interfacial stiffness is caused by the total effect of the stiffness of the asperities and the number of the contacting asperities.

Based on the above discussion, the behavior of the natural frequency as shown in Fig. 7 can be presumed as follows. The interfacial stiffness increases just after the compressive stress occurs. After the number of the contacting asperities becomes constant, the interfacial stiffness is dominated by the stiffness of the asperities. If the increase rate of the stiffness of the asperities is low, also the increase rate of the interfacial stiffness is low. This behavior of the interfaces may cause that the natural frequency increases and the increase rate of the natural frequency decreases with the increase of the clamping force. Furthermore, the compressive stress and the interfacial stiffness are high near the bolt holes, but still low far from the bolt holes. This compressive stress distribution may cause that the total interfacial stiffness and the natural frequency is saturated by the clamping force lower than the compressive stress saturates the interfacial stiffness.

4.3. Evaluation of proposed method

Figure 12 shows the estimated surface texture parameters ($\sigma, C$). “Proposed method” shows the estimation results by the data set of the clamping force and the natural frequency shown in Fig. 7. “Contact analysis” shows the regression coefficients of the fitting curves shown in Fig. 9. “Directly derived by surface profile” shows the calculation results by applying Eq. (3) and Eq. (4) on the measurement data shown in Table 1. For the standard deviation of the asperity peak heights $\sigma$, the estimation values have good agreement with each other except the Al-Al specimen $R_z = 100 \sim 50$ and the St-Al specimen $R_z = 100 \sim 50$. Even in the Al-Al specimen $R_z = 100 \sim 50$ and the St-Al specimen $R_z = 100 \sim 50$, the maximum differences among the estimation techniques are under $7 \mu m$. For the homogenized elastic modulus $C$, the estimation results are in $0.2 \sim 300$ MPa. Although the differences among the estimation techniques appear in log scale, the estimated values are obviously less than the stiffness of the base materials.

Figure 13 shows the interfacial stiffness calculated by applying Eq. (5) on the estimated surface texture parameters ($\sigma, C$) shown in Fig. 12. The interfacial stiffness was estimated to be high in the order of the smoothness of the interfaces in all of the specimens. Seeing the maximum compressive stress is $32$ MPa when the clamping force $P = 8$ kN, the interfacial stiffness estimated by the proposed method for over $32$ MPa is the extrapolation value. Only the interfacial stiffness of the St-St specimen $R_z = 100 \sim 50$ was estimated to be higher than the that of the St-St specimen $R_z = 25 \sim 12.5$ subjected to over $25$ MPa in the proposed method. This seems to be the extrapolation error caused by the gradient of the interfacial stiffness for the compressive stress under $25$ MPa. In addition, the interfacial stiffness was estimated to be high in the order of the stiffness of the base materials (St-St specimen, St-Al specimen, Al-Al specimen).

Even the interfacial stiffness directly derived by the surface profiles has the measurement error due to individual differences. However, this value is defined to be the criterion for the evaluation of the estimation methods. In comparison with the estimation results directly derived by the surface profiles, the estimation results of the proposed method has the...
Fig. 12 Estimated surface texture parameters. For the standard deviation of the asperity peak heights \( \sigma \), the estimation values have good agreement with each other. For the homogenized elastic modulus \( C \), although the difference among the estimation techniques appears in log scale, the estimation values are obviously less than the stiffness of the base materials.

![Graph showing standard deviation of asperity peak heights and homogenized elastic modulus for different materials](image)

Fig. 13 Estimated interfacial stiffness. The interfacial stiffness were calculated by applying Eq. (5) on the estimated surface texture parameters \( (\sigma, C) \) shown in Fig. 12. The three estimation techniques tended to estimate that the interfacial stiffness was high in the order of the smoothness of the interfaces and the stiffness of the base materials. The differences among the estimation techniques were comparable.

![Graph showing interfacial stiffness for different materials and compressive stress](image)

The same difference as the contact analysis in all of the specimen. For instance, the maximum difference of the estimation results which is observed in the St-St specimen \( R_z = 6.3 \sim 3.2 \) is 142 MPa in the proposed method, and 139 MPa in the contact analysis. Therefore, it is believed that the proposed method is comparable to the contact analysis in terms of the estimation of the interfacial stiffness.
5. Conclusion

This study has focused on the bolted joints in the multi-material structure, and proposed an estimation method of the interfacial stiffness by the inverse analysis of the clamping force and the natural frequency of the structure. The hammering test and the contact analysis by using the specimen made of the steel and the aluminum alloy were performed. The results showed that the proposed method could estimate the interfacial stiffness which reproduced the natural frequency of the specimen subjected to the clamping force of the bolts and nuts. The interfacial stiffness estimated by the proposed method was comparable to that calculated by the contact analysis.

This paper has shown the result of a test piece with single interfacial part and clarified that the proposed method works in the primary situation. In real engineering problem, the number of fixing parts is not always one. In order to extend the proposed method for the evaluation of real structure which has multiple fixing parts, this study suggests the following approaches.

The first approach is supporting or clamping the both ends of one of the fixing parts by jig. In the FEM, only the FE model of the target part is created, and the supporting parts by the jig are given as the boundary conditions. In the hammering test, the vibration modes in which only the target part vibrates enable to be extracted.

The second approach is that the proposed method is applied on the part itself in the assembling process or after disassembling the target parts. In the assembling process, the fixing parts are usually assembled one-by-one. After the first assembling, there is one fixing part in the structure, and the proposed method can be applied easily. After the second assembling, there are two fixing parts in the structure, but one of the fixing parts has been identified in the first assembling. Hence, only the interfacial stiffness of the other fixing part is just identified by the proposed method. In the same way, the identification can be operated one-by-one along the assembling process. This approach does not have to simultaneously identify all of the interfacial stiffnesses after assembling. Moreover, because the proposed method estimates the interfacial stiffness subjected to the clamping force of the fixing parts, the interfacial stiffness by the secular change of the clamping force can be predicted in the assembling process.

The operation of the proposed method after disassembling the target parts is limited to the case that the secular change of the surface condition of the interface critically affects the interfacial stiffness. If the target parts are extracted, the proposed method can be easily applied on them as with the case of the assembling process. On the other hand, there is the other problem such as which part can be disassembled or not. However, this problem is how the proposed method is applied on such specific structure, and it is the technical problem. The principal aim of this paper is to investigate whether the proposed method can work in essentials. Thus, the further study for the structure which has multiple fixing parts is an issue in the near future.

The proposed method has the advantage that the measurement of the surface profile is not required. From the perspective of the design of the structure, the simulated data and the target values are substitutable for the natural frequency and the clamping force of the bolted joints in the proposed method. The proposed method is expected to derive the surface texture of the interfaces and the interfacial stiffness to achieve the design. This work was supported by JKA and its promotion funds from AUTORACE, and JSPS KAKENHI Grant Number JP18K03849.
Appendix

In order to investigate the influence of the alignment of the asperities on the interfacial stiffness, the additional contact analysis in which the upper blocks were displaced horizontally was executed. Figure 14 shows another FE models of the interfaces of the specimens. The convexities were set to face the concavities in the specimens at the initial alignment.

Figure 15 shows the results of the compressive stress for the gap between the mean planes in the contact analysis. In the specimens $R_z = 6.3 \sim 3.2$, the influence of the initial alignment is small. In the specimens $R_z = 25 \sim 12.5$ and the specimens $R_z = 100 \sim 50$, the gap when the the compressive stress initiates was small in the additional contact analysis because the gap needs to be small enough in order for the convexities and the concavities to contact each other. On the other hand, in all of the specimen, the interfacial stiffness, i.e. the gradient of the compressive stress against the gap, was almost same at the sufficient compressive stress (over 20 MPa in the Al-Al specimen $R_z = 100 \sim 50$, for example) regardless of the initial alignment. Therefore, it can be presumed that even if the initial alignment of the asperities is different, such as the convexity facing to the concavity, the behavior of the asperities is different during the early stage of the contact, but similar after the contact progresses. Furthermore, seeing that the area of the interfaces is much larger than the size of the surface profile used in the contact analysis, the influence of the individual difference including the initial alignment of the asperities can be assumed to be small.

Fig. 14 Another FE models for additional contact analysis. Only the initial alignment is different from the FE models shown in Fig. 5. The convexities were set to face the concavities in the specimens at the initial alignment.
Fig. 15 Results of contact analysis. The data of “another alignment” show the results by the contact analysis using the FE models shown in Fig. 14. The other data are the same as the results shown in Fig. 9. The initial alignment affected the gap that initiates the compressive stress. On the other hand, the gradient of the compressive stress against the gap was almost same regardless of the initial alignment.

References


