ELASTO-PLASTIC BEHAVIOR OF 3-DIMENSIONAL REINFORCED CONCRETE ABUTMENTS CONSIDERING THE EFFECT OF THE WING WALL

Desy Setyowulan¹, Tomohisa Hamamoto², Toshitaka Yamao³

¹Graduate School of Science and Technology, Kumamoto University, 2-39-1 Kurokami, Kumamoto, 860-8555, Japan
²Department of Civil Engineering, Gunma National College of Technology, 580, Tribamachi, Maebashi, Gunma 371-8530, Japan
³Graduate School of Science and Technology, Kumamoto University, 2-39-1 Kurokami, Kumamoto, 860-8555, Japan

ABSTRACT

This study presents the elasto-plastic behavior of 3-dimensional reinforced concrete abutments with wing wall as a parametric. To analyze the structure, we used ABAQUS, the finite element software, with geometrical modeling of three dimensional solid elements (C3D8R) and two dimensional trusses (T3D2) for concrete and reinforcing bar, respectively. Four static horizontal loads were applied at half-height of parapet wall with the fixed boundary condition at the footing of abutment. Firstly, the analytical and experimental result of reinforced concrete beam was compared in this paper in order to check the validity of the numerical analysis. Then, numerical studies were carried out in four different abutment modeling approaches including the proposed model of Type 4 in order to generate an appropriate abutment model. According to the numerical results, it was demonstrated that effect of the wing wall in reducing the displacement of the proposed model was significant in the wing wall part. Furthermore, cracking and stress distribution in the proposed model were also affected by the wing wall.

Keywords: Reinforced concrete abutment, concrete damaged plasticity, elasto-plastic behavior, parapet wall, wing wall
I. INTRODUCTION

Reinforced concrete (RC) materials have been widely used as the main constituent material in the structural engineering structure, such as in the buildings, bridges, dams, etc. This material has the complex behavior which is mainly due to non-linear stress-strain relation of the concrete under multi-axial stress conditions, strain softening and anisotropic stiffness reduction, progressive cracking caused by tensile stresses, both between concrete and reinforcement, aggregation interlock and dowel action of reinforcement, time dependent behavior as creep and shrinkage [1]. Consequently, a proper material model should be capable to represent the behavior of materials, including elastic and elasto-plastic behavior of concrete in tension and compression, within finite element packages.

Recently, many studies have investigated the behavior of concrete using finite element analysis. Model verifications of RC beam structures with the damage identification by Concrete Damaged Plasticity model in ABAQUS [2] have performed by some researchers. This code has shown the adequate reliability and accuracy to perform nonlinear behavior of RC structure in comparison with the experimental results [3-5].  

In structural engineering, abutment refers to the end support of the bridge superstructure. It plays an important role on the performance of bridge, such as transferring load from superstructure to its foundation, resisting and/or transferring self-weight and lateral loads from earth pressure and wind loads, also supporting one end of an approach slab. Its behavior has been found to significantly influence the response of an entire bridge system under strong intensity dynamic excitation [6]. On many bridges, abutment damage was the only damage reported indicating that it attracted a large portion of seismic force [7].

The parapets and wing walls of abutment are designed based on the Japan Specification for Highway Bridges, Part IV Substructures [8]. In earthquake prone area, the unseating prevention structures (UPS) are needed to be installed in the bridge structure in order to prevent unseating of the superstructure against unforeseen conditions such as unexpected seismic force, destruction of the surrounding ground, or unexpectedly complicated vibrations in the structural members. However, falling of the superstructure may lead by the breakage of the parapet when UPS are installed to the parapet.

Previous study [9] has carried out an investigation on the elastic behavior of parapet considering to the wing wall. In comparison with the plate structure fixed three sides, it is found that the bending moment and shear force at the base of the parapet wall decrease and elastic deformation behavior is similar. In addition, an investigation on elastic and elasto-plastic behavior of the parapet-unified wing walls of abutment subjected to horizontal loads through unseating prevention structure of bridge has been conducted by another researcher [10]. From this study, it has been recognized that the effect of the wing walls on the behavior and bending moment at base of the parapet is very large.

In past large earthquake, many abutments collapsed due to strong intensity of dynamic excitation. The collapsed mainly due to high stress as the most common problems observed during the inspection of the abutment failure. Therefore, it is needed to analyze the new type of abutment as the proposed model in order to generate an appropriate abutment model. This study is focused on determination of the elasto-plastic behavior of 3-D reinforced concrete abutments considering to the effect of the wing wall. The finite element software of ABAQUS with the Concrete Damaged Plasticity method was used as the finite element analysis method. In verification method, modeling technique of reinforced concrete beam was verified by comparing the model prediction of RC beam to the experimental work in the previous research [10]. Then, numerical studies were carried out in four different abutment modeling approaches including a proposed model of abutment, subjected to horizontal loads through the unseating prevention structures. The results were used to predict the effect of the wing wall in the behavior of abutment.
II. NUMERICAL PROCEDURES

2.1 Finite element modeling

Numerical modeling of reinforced concrete beam and abutments were performed using ABAQUS [2] software. The concrete beam was idealized by homogenous material and modeled with eight-node solid (brick) elements, identified as C3D8R elements. Furthermore, the reinforcing bar was idealized by three dimensional truss elements called T3D2.

2.2 Constitutive model of steel reinforcing bar (rebar)

The constitutive model for steel rebar has approximately linear elastic behavior when the stiffness is governed by the Young’s modulus at low strain magnitudes. At higher strain magnitudes, it begins to behave nonlinear, inelastic behavior, which is referred to as plasticity [3]. The plastic behavior is described by its yield point and post-yield hardening, with the shift from elastic to plastic behaviors occurs at a yield point on a material stress-strain curve. Once the stress in rebar exceeds the yield stress, permanent (plastic) deformation begins with decreasing of the stiffness. The plastic deformation of this material increases its yield stress for subsequent loadings.

2.3 Constitutive model of concrete

In ABAQUS [2], there are three methods to simulate the damage criterion in reinforced concrete elements, including brittle crack concrete model, smeared crack concrete model and concrete damaged plasticity model (CDP). The CDP method allows in defining the complete inelastic behavior of concrete under tension and compression including the damage parameters. It provides a general capability for modeling concrete and other quasi-brittle materials in all types of structures (beams, trusses, shells, and solids), so it can be used with rebar to model concrete reinforcement. In the mechanical behavior of CDP, tensile cracking and compressive crushing are assumed to be the main failure mechanisms of concrete. Failure of the yield surface is controlled by tensile and compressive equivalent plastic strains, $\tilde{\varepsilon}_{pl}^t$ and $\tilde{\varepsilon}_{pl}^c$, respectively.

The concrete behavior is considered independently with rebar. Effect of the bond slip and dowel action are modeled by introducing some “tension stiffening”, as shown in Fig. 1(a), in order to simulate load transfer across cracks through the rebar and to define cracking and post-cracking properties for the concrete. The stress-strain response under uniaxial tension follows a linear elastic relationship up to the failure stress, $\sigma_{t0}$, corresponds to the onset of micro-cracking in the concrete material. The compressive equivalent cracking strain and plastic strain values, $\tilde{\varepsilon}_{tck}$ and $\tilde{\varepsilon}_{tpl}$, are defined by (1) and (2), respectively.

$$\tilde{\varepsilon}_{tck} = \varepsilon_t - \varepsilon_{ot}^{el}$$  \hspace{1cm} (1)

$$\tilde{\varepsilon}_{tpl} = \tilde{\varepsilon}_{tck} - \frac{d_t}{(1-d_t)} \frac{\sigma_t}{E_o}$$  \hspace{1cm} (2)

where: $\varepsilon_t$: total tensile strain

$\varepsilon_{ot}^{el}$: The elastic tensile strain corresponding to the undamaged material, $\varepsilon_{ot}^{el} = \sigma_t/E_o$

$E_o$: Young’s modulus

$\sigma_t$: tensile stress

$d_t$: tension damage parameter

The compressive behavior of plain concrete under uniaxial compression outside the elastic range can be defined, as shown in Fig. 1(b). The stress-strain response under uniaxial compression is
linear up to the initial yield, $\sigma_{c0}$. The response in plastic regime is characterized by stress hardening followed by strain softening beyond the ultimate stress, $\sigma_{cu}$. The inelastic (or crushing) strain and plastic strain values, $\varepsilon_{c}^{in}$ and $\varepsilon_{c}^{pl}$, are defined by (3) and (4), respectively. In addition, the reduction of the elastic modulus in compressive and tensile behavior, $E_c$ and $E_t$, are given in (5) and (6), respectively.

\[
\varepsilon_{c}^{in} = \varepsilon_{c} - \varepsilon_{oc}^{el}
\]

(3)

\[
\varepsilon_{c}^{pl} = \varepsilon_{c}^{in} - \frac{d_c \sigma_c}{(1-d_c) E_o}
\]

(4)

\[
E_c = (1-d_c) E_o
\]

(5)

\[
E_t = (1-d_t) E_o
\]

(6)

where:

$\varepsilon_{c}$: total compressive strain

$\varepsilon_{oc}^{el}$: The elastic tensile strain corresponding to the undamaged material, $\varepsilon_{oc}^{el} = \sigma_c / E_o$

$\sigma_c$: Compressive stress

$d_c$: compression damage parameter

---

(a) Tension behavior associated with tension stiffening
(b) Compression behavior associated with compression hardening

Figure 1. Concrete damaged plasticity model [2]

(b) Modified tension stiffening model [5]

Figure 2. Numerical properties of RC structure
In this analysis, modeling of the compressive stress-strain curve was developed based on the previous research [11], which can be used for normal strength concrete with $\sigma_{cu}$ less than 62 MPa, as shown in Fig. 2(a). The compressive stress value, $\sigma_c$, between the yield point at 0.5$\sigma_{cu}$ and the $0.3\sigma_{cu}$ in the descending portion is calculated by (7). Furthermore, modified tension stiffening model for the concrete is shown in Fig. 2(b). It is introduced in order to simulate load transfer across cracks through the rebar and to define cracking and post cracking properties for the concrete. The equations, $\sigma_c$, $\sigma_{cu}$ and $E_o$ are in kip/in² with the conversion factor of 1MPa = 0.145 kip/in².

\[
\sigma_c = \left( \frac{\beta (\varepsilon_c / \varepsilon_o)}{\beta - 1 + (\varepsilon_c / \varepsilon_o)^\beta} \right) \sigma_{cu} 
\]

(7)

\[
\beta = \frac{1}{1 - (\sigma_{cu} / (\varepsilon_o E_o) \ln)}
\]

(8)

\[
\varepsilon_o = 8.9 \times 10^{-5} \sigma_{cu} + 2.114 \times 10^{-3}
\]

(9)

\[
E_o = 1.243 \times 10^2 \sigma_{cu} + 3.283 \times 10^3
\]

(10)

### 2.4 Numerical modeling of reinforced concrete (RC) beam structure

In order to ensure the validity of the numerical analysis, a model comparison to actual behavior was performed. A simple model of RC beam structure shown in Fig. 3 was chosen, including its material properties [10]. As shown in this set-up, a rectangular RC beam with the width $b$ of 100 mm, the height $h$ of 400 mm, 900 mm long $L$ was positioned horizontally and supported at 230 mm from the beam ends. The shear span ratio ($a/d$) and the depth $d$ were 0.5 and 320 mm, respectively. Other structural properties were shown in Table 1. It was subjected to two identical point loads and applied symmetrically on the rigid plates. Moreover, the stress-strain relationship of the concrete and rebar are shown in Fig. 4(a) and Fig. 4(b), respectively. The deflection at mid-span was measured by means of LVDTs.

![Figure 3. Position of the tested RC beam [10]](image)

According to the experimental test setup data, RC beam was then simulated in X-Y-Z coordinate system with a half-length by symmetry condition, as shown in Fig. 5. X, Y, and Z axes were parallel to the longitudinal, depth and width of beam. The compressive strength of the concrete were 27.214 MPa (taken as yielding strength $f_{co}$) and 54.427 MPa (taken as ultimate strength $f_{cu}$). The density, Young modulus $E_{cr}$, tensile stress $\sigma_{ct}$, and Poisson’s ratio $\nu$ of concrete were 2400 kg/m$^3$, 33342.61 MPa, 3.315 MPa, and 0.167. In addition, rebar was assumed as steel material with the density of 7850 kg/m$^3$, yield stress $f_y$ of 375.3 MPa, Young modulus $E_s$ of 205939.65 MPa, and

<table>
<thead>
<tr>
<th>Descriptions</th>
<th>Dimensions (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$l$</td>
<td>440</td>
</tr>
<tr>
<td>$t$</td>
<td>30</td>
</tr>
<tr>
<td>$r$</td>
<td>100</td>
</tr>
</tbody>
</table>

Table 1. Structural properties of RC beam
Poisson’s ratio $\nu$, of 0.3. The damaged variables, $d_t$ and $d_c$, are defined based on plot presented in Fig.6(a) and Fig.6(b).

(a) Stress-strain curve for concrete  
(b) Stress-strain curve for steel rebar [10]

**Figure 4. Model input of RC beam**

(a) Applied load and boundary condition  
(b) Solid element  
(c) Rebar element

**Figure 5. Model of RC beam in ABAQUS**

(a) Tension damage  
(b) Compression damage

**Figure 6. Damage variables of the concrete**
2.5 Numerical modeling of reinforced concrete abutments

According to Japan Specifications for Highway Bridges, Part IV (Substructures) [8], there are three different types of the wing wall shapes. The shapes and dimensions vary according to the installation site of the abutment, backfill height, and gradient of the slope. The numerical model of RC abutment Type 2 as the typical model of abutment in Japan is representative for actual model, as shown in Fig. 7. In order to increase the strength of abutment, abutment Type 4 was analyzed as a proposed model to be developed from Type 1 (without wing wall) and Type 3 (full wing wall). Moreover, based on this specification, the wall of a T-shaped abutment can be designed as a cantilever with the fixed end at the node connected to the top of the footing. Therefore, in order to reduce the number of nodes and elements in the modeling, it needs to simplify the model by omitting the footing of abutment and modeling support with fixed boundary condition. Summary of the comparison results in Type 1, with and without footing are described in Table 2 and Fig. 8. It was recognized that the analytical method of RC abutment without footing is able to represent the behavior of abutment with footing.

![Numerical model and Real bridge](image)

**Figure 7. Model of abutment Type 2 as the typical model in Japan**

**Table 2. Summary of the maximum result in each model**

<table>
<thead>
<tr>
<th>Descriptions</th>
<th>Type 1 (with footing)</th>
<th>Type 1 (without footing)</th>
<th>Differences</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_{11}$</td>
<td>1.591 e+8</td>
<td>1.592 e+8</td>
<td>0.06%</td>
</tr>
<tr>
<td>$S_{33}$</td>
<td>2.493 e+6</td>
<td>2.507 e+6</td>
<td>0.56%</td>
</tr>
<tr>
<td>$PE_{11}$</td>
<td>2.955 e-4</td>
<td>2.956e-4</td>
<td>1.30%</td>
</tr>
<tr>
<td>$d_t$</td>
<td></td>
<td>Similar value and position</td>
<td></td>
</tr>
<tr>
<td>AC Yield</td>
<td></td>
<td>Similar value and position</td>
<td></td>
</tr>
</tbody>
</table>

© IAEME International Journal of Civil Engineering and Technology (IJCIET), ISSN 0976 – 6308 (Print), ISSN 0976 – 6316(Online), Volume 5, Issue 11, November (2014), pp. 97-113 © IAEME
Four different abutment modeling approach were carried out with a half-length model by symmetry condition, as shown in Fig. 9. The dimensional configurations are depicted in Fig. 10 and Table 3, respectively, with parameter of $t$ represents the thickness of parapet wall and wing wall. Furthermore, the rebar arrangement is given in Fig. 11.

![Figure 8. Load vs displacement](image)

**Figure 8. Load vs displacement**

![Figure 9. Numerical models of abutments](image)

(a) Type 1  (b) Type 2  (c) Type 3  (d) Type 4

**Figure 9. Numerical models of abutments**

![Figure 10. Dimensional configurations of abutments (unit: m)](image)

**Figure 10. Dimensional configurations of abutments (unit: m)**
Material property of the concrete was assumed from previous research [12], as shown in Fig. 12, while for the rebar was similar with RC beam input. The compressive strengths were 13.75 MPa (taken as yielding strength $f_{cu}$) and 27.5 MPa (taken as ultimate strength $f_{cu}$). The density, Young modulus $E_o$, tensile stress $\sigma_{ot}$, and Poisson’s ratio $\nu_c$ of concrete were 2400 kg/m$^3$, 25000 MPa, 2.75 MPa, and 0.2.

### 2.6 Loading conditions

There are two types of loading condition for RC beam and RC abutments, including the self-weight as a gravity load $g$ of 9.8 m/s$^2$ and the external load. The external load in RC beam was amplified under force control. Design seismic force of the unseating prevention structure, $H_F$, may be taken as 1.5 times dead load reaction, $R_D$ [13]. Previous research [14] developed a model of abutment with full wing wall and applied the unseating prevention structure (UPS), as shown in Fig. 13. In this research, four static horizontal loads through UPS were applied in the half-height of parapet wall, with the dead load reaction of the superstructure was 2900 kN, obtained from the deck weight. Therefore, each UPS will transfer the external load to the parapet wall of 543.75 kN.

$$H_F = 1.5 R_D$$  \hfill (11)
2.7 Interaction Properties

General contact surface algorithm was considered to determine the modeling of contact behavior in the interacting surfaces between concrete and steel plate in RC beam. The friction coefficients of contact point were set to be 0.1 and hard contact for pressure-over closure. In this analysis, the steel plate was used in order to transfer the external load to RC beam and to set down the support. The embedment was defined using an embedded constraint and was achieved by constraining rebar elements into solid element, in order to create a proper bond action.

2.8 Boundary conditions

The RC beam and abutments were modeled as a half-length model with the boundary conditions in support were hinged and fixed, respectively. The surface of YZ in the middle length of RC beam was constrained with boundary condition in X-direction ($U_x=0$), whether the surface of XY in the mid-span of RC abutments was constrained in Z direction ($U_z=0$).

III. RESULTS AND DISCUSSIONS

3.1 Analysis of RC beam structure

The load-displacement curves for experimental result and numerical analysis by ABAQUS are shown in Fig. 14. The displacement and load shown correspond to the vertical displacement at mid-span of RC beam structure and the total vertical force imposed on the structure, respectively. From this figure, it can be seen that the numerical result correlates well with those from experimental data up to 40 tf. Moreover, the maximum load to be transmitted in RC beam by numerical analysis is 48.54 tf, which is slightly larger than experimental result of 45 tf.

In order to visualize the cracking distribution within the FEM, the positive plastic strain can be mapped with color-coded contours shown in Fig. 15. The beam exhibits flexure-shear failure where flexural cracks initiate first in the mid-span of structure, followed by inclined flexure-shear cracks. The maximum plastic strain occurs in the “red” regions where the plastic strain equals 0.00255 corresponds to the maximum tension damage parameter of approximately 0.9. Dark blue regions correspond to areas of no tension damage. According to the analysis, it could be concluded that the cracking distribution of RC beam is more localized within the support and mid-span of the beam.

---

**Figure 14. Vertical load vs displacement curves**

**Figure 15. FEM magnitude of plastic strain**
3.2 Analysis of RC abutments

3.2.1 Longitudinal displacement

The longitudinal displacements of abutments were investigated in the wing wall part and half-length part of abutment, three levels in each side, as shown in Fig. 17. $UT_1$, $UB_1$ and $U_1$ represent the longitudinal displacement of the wing wall part in different level positions, top of parapet, bottom of parapet and bottom of abutment, respectively. Otherwise, in the middle part are depicted as $UT_2$, $UB_2$, and $U_2$.

![Figure 17. Position of the nodal displacement](image)

(a) Middle part  (b) Wing wall part

Figure 18. Longitudinal displacement of abutment (mm)
Total static horizontal load which can be expected to be transmitted through the unseating prevention structures for half-length model was 2175 kN. Applying external load affect the longitudinal displacement of each abutment, are depicted in Fig. 18(a) and Fig. 18(b) as displacement in the middle and wing wall part, respectively. Interesting results occur in the wing wall part U1 of 10.64 mm, 3.16 mm, 0.59 mm, and 0.36 mm for abutment Type 1, Type 2, Type 4, and Type 3, respectively. It signifies in reducing of 59.20%, 94.33% and 92.06% for abutment Type 2, Type 3 and Type 4, respectively. These results prove that installation of the wing wall in abutment have a capability in decreasing of the longitudinal displacement.

In addition, a static pushover analysis was carried out with the external load amplification in order to characterize the maximum load to be transmitted in abutment. The result of load versus displacement curve is shown in Fig. 19. The study reported that the maximum load for abutment Type 1 is the smallest one of 2792.25 kN. Type 3 and Type 4 have similar value of 3009.27 kN, which are smaller than Type 2 of 3244.43 kN. This condition is possibly due to design of the wing walls in Type 4 as slabs fixed on two sides to a wall and footing, which can reduce the longitudinal displacemnt and flexibility of the structure. Consequently, it leads crack at the parapet wall.

3.2.2 Cracking Distribution of RC Abutments

(a) Initial cracking at 1093.11 kN  (b) Final cracking at 2792.25 kN

Figure 20. Contour plot of tensile damage in RC abutment Type 1 at different external load
Figure 21. Contour plot of tensile damage in RC abutment Type 2 at different external load

(a) Initial cracking at 1093.11 kN  (b) Final cracking at 3244.43 kN

Figure 22. Contour plot of tensile damage in RC abutment Type 3 at different external load

(a) Initial cracking at 1093.11 kN  (b) Final cracking at 3009.27 kN

Figure 23. Contour plot of tensile damage in RC abutment Type 4 at different external load

(a) Initial cracking at 1093.11 kN  (b) Final cracking at 3009.27 kN
Figs. 20-23 show contour plot of output variable DAMAGET, which is a scalar degradation measure used to express the reduced tensile elastic modulus of concrete after it has sustained cracking damage. Sequence of the concrete cracking are described as initial and final cracking, as shown in section (a) and (b), respectively. “Dark blue” regions correspond to areas of no tension damage or no cracking. The maximum tension damage occurs in “red” regions, where the cracking strain equals 0.00264 corresponds to the maximum tension damage parameter of approximately 0.9. From these figures, it can be discovered that the initial cracking of all abutments occur in two different positions, in peripheral of the external load and intersection between parapet wall and abutment wall due the external load of 1093.11 kN. In addition, cracking is also found at bottom of abutment wall near the footing for Type 1 and Type 2. Type 1 is found as a structure with the most critical damage due to the cracking propagation in region through its width. Moreover, it propagates from the middle width to the edge position, lead the connection part between wing wall and parapet wall to crack.

3.2.3 Load-tensile strain plot for reinforcing bar

Fig. 24(a) shows the maximum tensile stress versus tensile strain relationship of the rebar, located at the reverse side of parapet wall. According to this figure, it can be seen that the tensile steel rebar is not yield at failure in abutment with all types of the wing wall. Otherwise, yielding occurs at Type 1. Furthermore, the relationship between load and maximum tensile strain in the rebar is depicted in Fig. 24(b). From these results, it is recognized that installation of the wing wall increases the stiffness of abutment, resulting on strain reduction in the rebar.

![Stress-strain ratio](image1)

![Load versus tensile strain](image2)

**Figure 24. Tensile strain plot for the rebar**

3.2.4 Shear stress distribution of RC abutments

Stress distribution is one of the important aspects to be evaluated in abutment. The areas subjected to high stress is the most common problems observed during the inspection of the failure of abutment in the past large earthquake. Shear stress distributions around vertical wall of abutments are studied in numerical analysis, with the results for each abutment are shown in Fig. 25. From these figure, it can be determined that maximum shear stresses occur at the region of parapet wall nearly the intersection with the wing wall.
IV. CONCLUSIONS

The elasto-plastic behavior of 3-dimensional reinforced concrete abutments considering the effect of the wing wall were investigated in this study. Numerical studies were carried out in four different abutment modeling approaches subjected to static horizontal loads through the unseating prevention structure with consideration of the wing wall. The conclusions are summarized as following.

1) The analytical method of Concrete Damaged Plasticity by ABAQUS was capable to analyze the elasto-plastic behavior of RC structure. The material modeling was said to be valid as experimental results in showing the behavior of RC beam in case of displacement and cracking distribution.

2) Installation of the wing wall in abutments had a capability in decreasing of the longitudinal displacement of 59.20%, 94.33% and 92.06% for abutment Type 2, Type 3 and Type 4 as the proposed type, respectively. This condition was affected by the increasing number of stiffness in abutment. Otherwise, there was no significant effect in half-length part of abutment.
3) According to the static pushover analysis, the maximum load that could be expected to be transmitted for the proposed type of abutment of 3009.27 kN was smaller than Type 2 of 3244.43 kN. It was possibly due to design of the wing walls in Type 2 as slabs fixed on two sides to a wall and footing, which reduced the longitudinal displacement and flexibility of the structure.

4) The initial cracking for all abutments occurred in two different positions, at the peripheral of the external load and intersection between parapet wall and abutment wall. Installation of the wing wall in the proposed model reduced the cracking at the bottom of wall and compressive strain of the concrete.

5) The stress distribution was affected by the wing wall. From numerical analysis, it was depicted that the maximum shear stress occurred at the region of parapet wall nearly the intersection of the wing wall. Furthermore, the tensile steel of the rebar was not yielded at failure in abutments with the wing wall which was affected by the reduction of the axial strain.

6) Although the present study found better evidence on the proposed model of abutment Type 4 in its behavior, no earth pressure was applied. Therefore, further studies are necessary in order to determine the behavior of RC abutment under earth pressure and strong ground excitation.

ACKNOWLEDGEMENT

The first author acknowledges DIKTI (Directorate General of Higher Education) as the financial supporter of the scholarship and University of Brawijaya as the home university. The support in completing the doctoral study in Kumamoto University is gratefully appreciated.

REFERENCES


