ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 4, Issue 6, December 2014

Simplification of Numerical Model to Analyze the Uniformly Heated one Way Reinforced Concrete Slabs Exposed by Fire

Wahyuniarsih Sutriso, Endah Wahyuni
Department of Civil Engineering Institut Teknologi Sepuluh Nopember
Department of Civil Engineering Institut Teknologi Sepuluh Nopember

Abstract*— This paper presented a simplification model for a reinforced concrete one way slab exposed by fire. A heat transfer modeling of reinforced concrete elements usually performed in 3D to get more accurate results. However, the complexities of 3D models sometimes produced the opposite result. That was due to a lot of things had to be considered when doing a 3D model thus the 3D analysis performed using finite element-based programs becomes more complex and time-consuming. Therefore, a 2D model was proposed to simplify the complicated 3D model. This study used a reinforced concrete slab model with 120 mm thickness, 1200 mm width and 3000 mm long which exposed to fire at the bottom part. The numerical modeling was performed by finite element based program and used the coupled temperature-displacement analysis. The result was compared with the experimental result and manual calculation using variation method. Based on the numerical result, the 2D model produced satisfying result. It was found that the reinforcement temperature and deflection difference between the 2D models and experimental reached 15% and 3% respectively when compared with experimental results.

Index Terms—**Fire, Simplification, Numerical Model, Slab, Reinforced Concrete.**

I. INTRODUCTION

Fire is one of the disasters which can affect the strength and durability of reinforced concrete structures. Fire can cause strength degradation of concrete structure which leads to collapse of the structure and make the users feel unsafe [1][2]. Design of the structural members actually includes several appropriate loads in the design. Some of the loads are static such as dead load, live load, and rain load. Others, such as wind load, seismic load and impact load are dynamics [3]. All of those loads are act to entire building and easy to applied. However, fire load is different compared with the other loads. Fire is not act to entire building but, it works at certain space in the building. The local effect of this load not only affect by structural arrangement and material properties of the building but also influenced by geometry of the space, ventilation, opening space of the building and also usage of that specific building space. The reason above makes fire load is difficult to applied in the structural design. However, the fire load is still very dangerous to the building's users because it can cause failure to the structure. Therefore, it is very important to consider this load in the design. The problems which mentioned before make the structural designer analyze the behavior of structure subjected to fire load and designing the whole structure separately. The behavior of fire load act as degradation of the constitutive materials in the structure. This condition will actually affect the behavior of the structure which subjected to specific fire loads. The technique use to analyze the ratio of the degradation of material properties is by find the average temperature of the element. Several code such as ACI and British standard gives graph regarding the relation between temperature and material properties which can used by the structural designer. Slabs as one of structural element have a great role to prevent the collapse of the structure due to fire load. Researchers found that slabs can resist the failure of the structure because the tensile membrane action develops at their large deformation [4]. Up until now there was several study was develop to understand the behavior of reinforced concrete slabs exposed to fire [1], [2], [4]–[8]. There are several ways used by several researchers to analyze the temperature and behavior of reinforced concrete slabs under fire such as numerical models and experimental tests. For few decades, experimental is one favorite methods to find the fire resistances of reinforced concrete structure because it produces a realistic result. However, this method is expensive and time consuming [9]. Thus, numerical analysis is chosen to analyze the structural member exposed to fire. In the numerical analysis, there were a lot of popular programs used to model the reinforced concrete elements subjected fire such as Safir, FEAST, Ansys and TAS [10] [11]. The software used the coupled thermal and structural analyses to gain satisfied result. However, the coupled thermal and structural analyses are complicated because they need to combine two different analyses. As the result, the analyst should choose between a two dimensional (2D) or three dimensional (3D) model to get the best result [12]. Many researchers use this model in their research to analyze the behavior of structural members when it exposed to fire [1], [7], [9], [13], [12]. The 3D modeling is often chosen to analyze reinforced concrete behavior exposed to fire because in some cases it produced more accurate result. However, the complexities of 3D models sometimes produced the opposite result. This kind of analysis have a lot of modeling consideration had to be considered such as the element type, interaction properties and meshing technique. Moreover, the 3D analysis performed using finite element-based programs is

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 4, Issue 6, December 2014

complex and time-consuming. Therefore, 2D model with one dimensional heat transfer analyses is proposed to simplify the complicated 3D model. The 2D model actually can easily applied by the designer because it is very simple and not time consuming to know the behavior and average temperature of the structural members. This paper provided a simplify method to analyze the behavior of reinforced concrete one way slab exposed to fire. The models performed by a finite element based program and used coupled temperature-displacement analysis. The result was compared with the experimental result and manual calculation using the variational method.

II. CONSTITUTIVE MATERIALS AND STANDARD FIRE

Fire resistance of concrete is quite good but a fire in a structure can lead strength degradation of the material [1]. The strength degradation of concrete and steel are caused by alteration of their properties. The properties of concrete and steel, such as thermal conductivity, specific heat and thermal elongation, have tendency to change as the rise of the temperature. The reinforcing steel also has same behavior when exposed to fire. The strength degradation of concrete and tension reinforcement in elevated temperature used in this research based on the Euro code [14], [15] The two common types of fire load are usually used in modeling and experimental test of reinforced concrete exposed to fire are natural fire and standard fire [8]. On fire resistance test which conducted to determine the fire resistance of the structure, it used a time and temperature relationship curve known as the standard fire curve. The natural fire or better known as compartment fires is more realistic when compared to standard fire when used in the experiment [2]. The natural fire curve considers two important factors which are the fire load and openings factor (holes). However, natural fire was very difficult when applied to a model or experiment because of its complexity. Based on the reason this paper used standard fire curves provided by ISO 834[16]. The standard fire exposure specified in ISO 834 is defined by the time temperature relationship as shown in this equation.

$$T = 345\log_{10}(8t+1) + T_o \qquad (1)$$

Where t is time in minute, *To* is the environmental temperature and T is temperature.

III. HEAT TRANSFER ANALYSIS WITH VARIATIONAL MODEL

The strength degradation in a reinforced concrete element is affected by additional thermal stress in the element. Therefore, to analyze the thermal stress it is really important to know temperature distribution in the particular element. The variational method is a simple method to analyze heat transfer process and temperature distribution of the element. This method assumed that the heat is spread only in the thickness of concrete. The basic one-dimensional temperature element, as shown in fig 1, used to modeling variation of the temperature within the thickness of the slab.

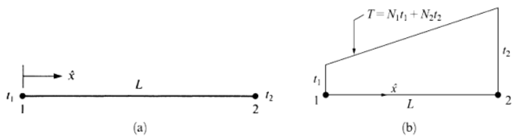

Fig 1. Basic One dimensional temperature element (b) Temperature Variation along the Length of the Element

The temperature function and temperature gradient relationship of the element are shown in eq. 2 and 3.

$$T(x) = N_1t_1+N_2t_2 \qquad (2)$$

$$qx = -[D]\{g\} \qquad (3)$$

For steady state condition, one dimensional heat conduction matrix can be written as:

$$[k_c] = \iiint [B]^T[D][B]\,dV = \int_0^L \begin{Bmatrix} -\frac{1}{L} \\ \frac{1}{L} \end{Bmatrix} [K_{xx}] \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} A\,dx$$

$$= \frac{AK_{xx}}{L^2}\int_0^L \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dx \qquad (4)$$

It can be simplified as

$$[k_c] = \frac{AK_{xx}}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \qquad (5)$$

And the convection matrix becomes

$$[k_h] = \iint h[N]^T[N]\,dS = hP\int_0^L \begin{Bmatrix} 1-\frac{x}{L} \\ \frac{x}{L} \end{Bmatrix} \begin{bmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{bmatrix} dx \qquad (6)$$

Or can be written as

$$[k_h] = \frac{hPL}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \qquad (7)$$

By adding the equation 6 and 7 so the stiffness matrix becomes

$$[k] = \frac{AK_{xx}}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + \frac{hPL}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \qquad (8)$$

For simplicity's sake, the convection assumed only occur from the right end of the element as shown in fig 2.

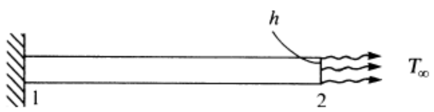

Fig 2. Convection Scheme

Therefore, the stiffness matrix at the end of the element given by

$$[k_h]_{end} = \iint h[N]^T[N]\,dS \qquad (9)$$

$$[k_h]_{end} = \iint h \begin{Bmatrix} 0 \\ 1 \end{Bmatrix} \begin{bmatrix} 0 & 1 \end{bmatrix} dS = h\,A\begin{bmatrix} 0 & 0 \\ 0 & 1 \end{bmatrix} \qquad (10)$$

The force matrix as follow

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 4, Issue 6, December 2014

$$\{f_h\} = \iint [N]^T h\, T_\infty dS \quad (11)$$

$$\{f_Q\} = \iiint_v [N]^T Q dV \quad (12)$$

$$\{f_q\} = \iint [N]^T h\, q^* dS \quad (13)$$

To simplify the equation so the value of Q, q^* and $h\, T_\infty$ assumed to be constant so the force matrix becomes

$$\{f_Q\} = \iiint_v [N]^T Q dV = QA \int_0^L \begin{Bmatrix} 1 - \frac{x}{L} \\ \frac{x}{L} \end{Bmatrix} dx = \frac{QAL}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \quad (14)$$

$$\{f_q\} = \iint [N]^T h\, q^* dS = q^* P \int_0^L \begin{Bmatrix} 1 - \frac{x}{L} \\ \frac{x}{L} \end{Bmatrix} dx = \frac{q^* PL}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \quad (15)$$

$$\{f_h\} = \iint [N]^T h\, T_\infty dS = \frac{h\, T_\infty PL}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \quad (16)$$

$$\{f_h\}_{end} = h\, T_\infty A \begin{Bmatrix} N_1\ (x = L) \\ N_2\ (x = L) \end{Bmatrix} = h\, T_\infty A \begin{Bmatrix} 0 \\ 1 \end{Bmatrix} \quad (17)$$

IV. MODELING CONFIGURATION AND PROPERTIES

Series of experimental test was conducted by Levuitsky, 2007 [11] by testing two type of one way slabs. This study only used the first type of slabs. This type used C15/20 concrete with longitudinal and transverse reinforcement. The longitudinals reinforcement used 10 mm diameters with 133.33 mm spacing and the transverse reinforcement used 6 mm diameters with 187.5 mm spacing. The slabs was exposed to 180 minutes fire at the bottom part. The slabs also subjected to uniformly distributed load at the upper part. The experimental mechanism shows in figure 3.

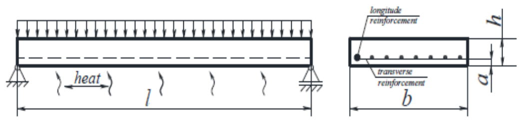

Fig 3. Slab Model and Validation [11]

The model in this study used same properties with the experimental test. The geometry of the model is a simply supported one way slab with 3000 mm long and 1200 mm. The thickness and cover of the slab used in this model is 120 mm and 25 mm respectively. The model was analyze in the 2D and 3D and the result compared with experimental test result conducted by Levuitsky, 2007 [11]. The slab was exposed in the bottom surface, thus the temperature will change as the change of exposure time. The temperature at upper part of the slab assumed to be constant at 20oC (indicate the environmental temperature). This model was analyze using couple temperature-displacement analysis to obtain both thermal and structural response.

V. RESULT AND DISCUSSION

Slabs were modeled in 3D (three dimensional) to study the thermal response of the element when it was exposed to fire. The material properties used in the model was temperature-dependent to get realistic and accurate result compared to the experimental result. Fig 4 showed the temperature distribution curves at the reinforcement compared to the experimental data.

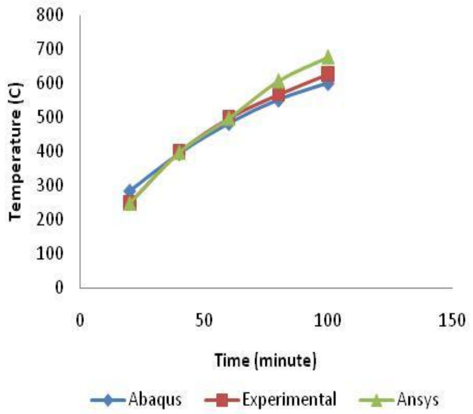

Fig 4. Comparison of Temperature at the Reinforcement Performed by Ansys, Abaqus 3D Model and Experimental Test.

Fig 4 shows the temperature distribution at the reinforcement performed by Ansys, Abaqus and experimental test had same trend. The temperature at the reinforcement was increased as increased of exposed time. The numerical analysis performed by Abaqus shows higher temperature in the beginning of exposure (t = 20 min). However, as increased of the exposure time, it tends to had lower temperature compared with experimental results and numerical analysis performed by Ansys. Fig 5 shows comparison of maximum deflection occurred at the middle of the slab. Modeling conducted by Abaqus inclined to has bigger deflection when compared with Ansys model and experimental test. The deflection at t = 20 minute until t = 60 minute had slightly different compared with Ansys and experimental test. But at the end of exposure

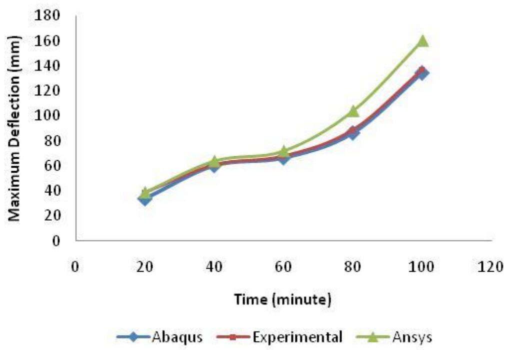

Fig 5 Comparison of Maximum Deflection at Mid Span of Slab Performed by Abaqus, Ansys and Experimental Test.

Based on the results of modeling, it can be seen that the stress was increased with increased of the exposure time (Fig 6 and 7). The stress on the concrete at the exposed surface, as shown in Figure 6, increased slowly at the beginning of combustion and significantly increased during mid-combustion until t = 100 minute. At the same time, the stress at the reinforcement, Fig 7, increased regularly from the beginning until the end of the combustion process (t = 100 min). Basically numerical analysis using Abaqus 3D modeling produced quite good output compared with

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 4, Issue 6, December 2014

experimental results. However, advanced modeling in 3D for coupled temperature-displacement analysis is very complicated, takes a lot of time and needs high computational demand. In addition, many considerations must be considered in 3D modeling such as the element type, the interaction between elements, meshing consideration etc. Based on that reason, this part focused to simplify the complicated three dimensional modeling to two dimensional modeling. The reason for this simplification is to make simpler model so the analysis can conducted with less time but, still generate accurate results. This model used two dimensional (2D) planar shells with same material properties as used in three dimensional model. The validation with analytical solution performed by varitional method as shown in Fig. 8. It conducted to convince the result of 2D modeling. Thing which compared in this validation is reinforcement temperature. The figure 8 shows the temperature at reinforcement from 2D modeling inclined to have same trend with the numerical calculation. It indicated that 2D modeling had been done correctly.

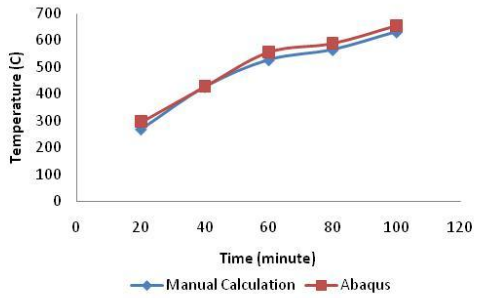

Fig 8. Comparison of Temperature at the Reinforcement Performed Abaqus 2D Model and Manual Calculation

Fig 9 shows comparison of temperature at the reinforcement performed by 2D Abaqus modeling, Ansys and experimental. It shows that the reinforcement temperature generated from 2D Abaqus model increased as increased of the exposure time. The curve also shows that the result of 2D Abaqus model has a slight different with the experimental test and Ansys modeling result. The difference only ranged from 2%-15%.

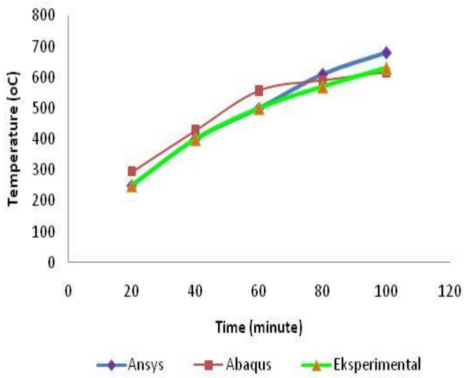

Fig 9. Comparison of Temperature at the Reinforcement Performed by Ansys, Abaqus 2D Model and Experimental Test.

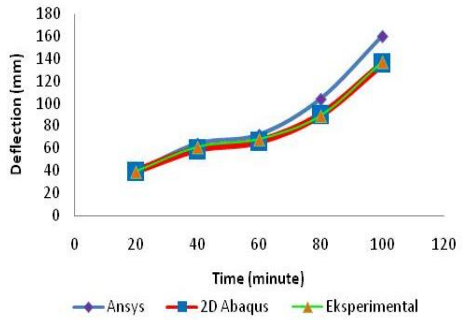

Fig 10. Shows the maximum deflection result at the mid span of the slab performed by Abaqus

The 2D modeling was extremely close compared with the experimental data with only 1% until 3% differences. The maximum deflection conducted by 2D modeling showed better result than 3D modeling when compared to experimental test (look at fig 5). Stress occurred at exposed surface and reinforcement showed in fig 11 and 12. The stress was tends to be increased as increased of the exposure time. It shows same trend with the 3D modeling. The stress at the exposed surface increased slowly at the beginning of combustion. However, after reached t = 40 minutes the stress was increased significantly as increased of the exposure time. Fig 6 also shows at the t = 80 min, the concrete stress has exceeded the allowable stress (fc') of concrete. This condition was faster when compared with experimental results and the results in 3D modeling where the time when stress exceeds the allowable stress occurs at t = 100 minute.

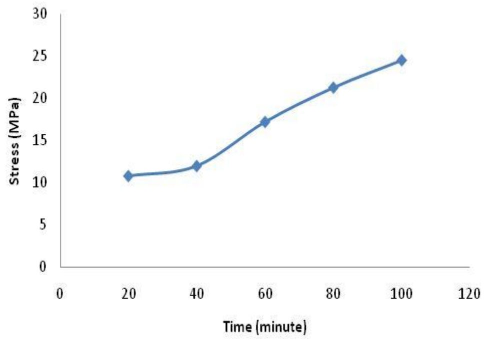

Fig 11. Stress at Exposed Surface for 2D Model

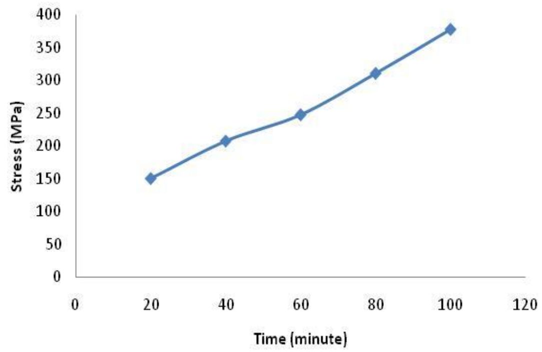

Fig 12. Stress at Reinforcement For 2D Model

Based on the 2D modeling result, it showed that this simplification model has good result. The result was close with the experimental data and analytical calculation. It also

ISSN: 2277-3754
ISO 9001:2008 Certified
International Journal of Engineering and Innovative Technology (IJEIT)
Volume 4, Issue 6, December 2014

less time consuming compared with complicated 3D model. However, this simplification has few limitations due to its ability to model non-uniformly heated slab and two way slabs. It is because this simplification only considers the cross section for the modeling so if it used for modeling non-uniformly heated slab (at length direction) and two way slabs the thermal response might be inaccurate. Therefore, this simplification only can be used for simple models which require rapid calculation of thermal and structural response.

VI. CONCLUSION

A simplification model was performed using finite element based program to study the behavior of reinforced concrete slab when subjected to fire. Based on the numerical modeling result, the 2D model showed a good result compared with experimental test and 3D modeling. It was found that the reinforcement temperature and deflection difference between the 2D modeling and experimental reached 15% and 3% respectively. The temperature and deflection of the slab changes according to time and it was found that their increase as increased of the time. The 2D modeling also less time consuming compared with 3D model. Due to the asumption of this simplification model that the temperature is assumed to be uniform along the slab surface so this technique is only recommended for simple model which require rapid thermal and structural response for uniformly heated slabs.

VII. ACKNOWLEDGMENT

Financial support from Indonesia Endowment Fund for Education (LPDP) with Grand No. 0018645/TK/T/8/LPDP 2013 is gratefully acknowledged. The authors also appreciate the fresh graduate scholarship from Sepuluh Nopember Institute of Technology.

REFERENCES

[1] C. Sangluaia, M. K. Haridharan, C. Natarajan, and A. Rajaraman, “Behaviour of Reinforced Concrete Slab Subjected To Fire,” Int. J. Comput. Eng. Res., vol. 3, pp. 195–206, 2013.

[2] E. Wahyuni and W. Sutrisno, “Evaluation Fire Resistance of Reinforced Concrete Slabs Using Analytical Solution and Numerical Modeling,” Second Int. Conf. Sustainable Infrastruct. Built Environ., vol. 2, no. November, pp. 1–11, 2013.

[3] R. W. Bukowski, “Fire as Building Loads,” Gaithersburg USA, 2001.

[4] Y. Wang, Y. Dong, and G. Zhou, “Nonlinear numerical modeling of two-way reinforced concrete slabs subjected to fire,” Comput. Struct., vol. 119, pp. 23–36, 2013.

[5] G. Wang, “Performance of Reinforced Concrete Flat Slabs Exposed to Fire,” University of Canterbury, 2006.

[6] L. Lim and C. Wade, “Experimental Fire Tests of Two-Way Concrete Slabs Experimental Fire Tests of Two-Way Concrete Slabs,” University of Canterbury, 2002.

[7] L. Lim, A. Buchanan, P. Moss, and J. Franssen, “Numerical modelling of two-way reinforced concrete slabs in fire,” Eng. Struct., vol. 26, pp. 1081–1091, 2004.

[8] A. Levesque and R. W. Fitzgerald, “Fire Performance of Reinforced Concrete Slabs,” WORCESTER POLYTECHNIC INSTITUTE, 2006.

[9] M. Saje, I. Planinc, S. Bratina, and C. Bojan, “Numerical modelling of behaviour of reinforced concrete columns in fire and comparison with Eurocode 2,” Int. J. Solids Struct., vol. 42, pp. 5715–5733, 2005.

[10] Y. . Wang, Steel and Composite Structures Behavior and Design for Fire Safety. New York: Spon Press, 2002.

[11] V. Kudryashov, N. T. Kien, and A. Lupandin, “Fire Resistance Evaluation of Reinforced Concrete Structures,” Saf. Technog. Environ., pp. 45–49, 2012.

[12] A. Jeffers, “Heat Transfer element for Modeling The Thermal Response of Non-uniformly Heated Plates,” Finite Elem. Anal. Des., vol. 63, pp. 62–68, 2013.

[13] E. Ellobody and C. G. Bailey, “Modelling of unbonded post-tensioned concrete slabs under fire conditions,” Fire Saf. J., vol. 44, pp. 159–167, 2009.

[14] BSI, “Eurocode 2 : Design of concrete structures,” no. July, 2003.

[15] BSI, “Eurocode 3 : Design of steel structures.” 2001.

AUTHOR’S PROFILE

Wahyuniarsih Sutrisno was born in Indonesia on June 12, 1990 She was accepted in ITS at 2008 trough governments scholarship. After completing her Bachelor Degree (Engineer) from ITS in 2012, she continued her study in ITS, and got her Master’s Degree (MT) in Civil Engineering in 2014. Since 2012, she active in some research related to durability of reinforced concrete structure. Now, she is a doctorate student of structural engineering in Civil Engineering Department of ITS and conducting research about corrosion in reinforced concrete structure.

Endah Wahyuni was born in Indonesia, on February 1, 1970. In 1998, she graduated from the Master Study Programme of UMIST. She got her Ph.D degree from The University of Manchester in 1991. Now, she is Lecturer at Civil Engineering Departement of ITS, Surabaya. She also has a duty as head of Civil Engineering Postgraduate Program. Her main research interest are structural analysis, design and modeling.