International Journal of Architectural, Civil and Construction Sciences
ISSN: 2415-1734
Vol:6, No:9, 2012

Simulation of non-Linear Behavior of Shear Wall under Seismic Loading

M. A. Ghorbani, M. Pasbani Khiavi

Abstract**—**The seismic response of steel shear wall system considering nonlinearity effects using finite element method is investigated in this paper. The non-linear finite element analysis has potential as usable and reliable means for analyzing of civil structures with the availability of computer technology. In this research the large displacements and materially nonlinear behavior of shear wall is presented with developing of finite element code. A numerical model based on the finite element method for the seismic analysis of shear wall is presented with developing of finite element code in this research. To develop the finite element code, the standard Galerkin weighted residual formulation is used. Two-dimensional plane stress model and total Lagrangian formulation was carried out to present the shear wall response and the Newton-Raphson method is applied for the solution of nonlinear transient equations. The presented model in this paper can be developed for analysis of civil engineering structures with different material behavior and complicated geometry.

Keywords**—**Finite element, steel shear wall, nonlinear, earthquake

I. Introduction

IN many practical applications the limitation of linear elasticity or more generally of linear behavior precludes obtaining an accurate assessment of the solution because of the presence of nonlinear effects or geometry having a thin dimension in one or more directions. Nonlinear behavior of solids takes two forms: material nonlinearity and geometric nonlinearity. The form of nonlinear material behavior is that of elasticity for which the stress is not linearly proportional to the strain. More general situations are those in which the loading and unloading response of the material is different. Typical here is the case of classical elasto-plastic behavior.

When the deformation of a solid reaches a state for which the undeformed and deformed shapes are substantially different a state of finite deformation occurs. In this case it is no longer possible to write linear strain-displacement or equilibrium equations on the undeformed geometry.

In this study, the analysis of shear shear wall is considered as an example to show the effect of nonlinearity. To formulate the problem and develop the numerical model, finite element method is selected because of its capability in analysis of structures with complicated geometry and materially nonlinear behavior.

Finite element procedures are now an important and frequently indispensable part of engineering analysis and design and the finite element programs are widely used in practically all branches of engineering for the analysis of structures. In the linear finite element formulation, it is usually assumed that the displacements of the finite element assemblage are infinitesimally small and that the material is linearly elastic.

M. A. Ghorbani, Department of Engineering, University of Mohaghegh Ardabili, Ardabil, Iran (Corresponding author, Phone: +98-451-5512804; fax: +98-451-5512904; e-mail: magorbani@ uma.ac.ir).

M. Pasbani Khiavi, Department of Engineering, University of Mohaghegh Ardabili, Ardabil, Iran (e-mail: pasbani@uma.ac.ir).

In addition, it is also assumed that the nature of the boundary conditions remains unchanged during the application of the loads on the finite element assemblage. With these assumptions, the finite equilibrium equations were derived for static analysis as following

$$KU = R \tag{1}$$

This equation corresponds to a linear analysis of a structural problem because the displacement response U is a function of the applied load vector R. If the loads are αR instead of R, where α is a constant, the corresponding displacements are αU. When this is not the case, the analysis will be nonlinear.

The linearity of a response prediction rests on the assumptions just stated and it is instructive to identify in detail where this assumptions have entered the equilibrium equation in equation (1). The fact that the displacements must be small has entered into the evaluation of the matrix K and load vector R because all integrations have been performed over the original volume of the finite elements and the strain-displacement matrix B of each element was assumed to be constant and independent of the element displacements. The assumption of a linear elastic material is implied in the use of a constant stress-strain matrix C and finally the assumption that the boundary conditions remain unchanged is reflected in the use of constant constrain relations for complete response. If during loading a displacement boundary condition should change, a degree of freedom which was free becomes restrained at a certain load level, the response is linear only prior to the change in boundary condition. This situation arises, for example, in the analysis of a contact problem [3].

The above discussion of the basic assumptions used in a linear analysis defines a nonlinear analysis and its categorizations. In a materially-nonlinear-only analysis, the nonlinear effect lies only in the nonlinear stress-strain relation. The displacements and strains are infinitesimally small; therefore the usual engineering stress and strain measures can be employed in the response description. Considering the large displacements but small strain conditions, in essence the material is subjected to infinitesimally small strains measured in a body-attached coordinate frame while this frame undergoes large rigid body displacements and rotations. The stress-strain relation of the material can be linear or nonlinear [4].

In actual analysis, it is necessary to decide whether a problem falls into one or the other category of analysis, and this dictates which formulation will be used to describe the actual physical situation. Conversely, it may be said that by use of a specific formulation, a model of actual physical situation is assumed, and the choice of formulation is part of the complete modeling process. Surely, the use of the most general large strain formulation will always be correct; however, the use of a more restrictive formulation may be computationally more effective and may also provide more insight into the response prediction.

II. FINITE ELEMENT FORMULATION

The basic problem in a general nonlinear analysis is to find the state of equilibrium of a body corresponding to the applied loads. Assuming that the externally applied loads are described as a function of time, the equilibrium conditions of a system of finite elements representing the body under consideration can be expressed as

$$^{t}R - {}^{t}F = 0 \tag{2}$$

Where ^{t}R lists the externally applied nodal point forces in the configuration at time *t* and the vector ^{t}F lists the nodal point forces that correspond to the element stresses in this configuration [3].

Considering the solution of the nonlinear response, it is recognized that the equilibrium relation in equation (2) must be satisfied throughout the complete history of load application and the time variable *t* may take on any value from zero to maximum time of interest. In a static analysis without time effects other than the definition of the load levels, time is only a convenient variable which denotes different intensities of load applications and correspondingly different configurations. However, in a dynamic analysis and in static analysis with material time effects, the time variable is an actual variable to be properly including in the modeling of the actual physical situation. Based on these considerations, it is realized that the use of time variable to describe the load application and history of solution responses is a very general approach and corresponds to assertion that a "dynamic analysis is basically a static analysis including inertia effects".

The basic approach in an incremental step-by-step solution is to assume that the solution for the discrete time *t* is known and that the solution for the discrete time $t + \Delta t$ is required, where Δt is a suitably chosen time increment. Hence, considering equation (2) at time $t + \Delta t$ we have

$$^{t+\Delta t}R - {}^{t+\Delta t}F = 0 \tag{3}$$

where the left superscript denotes "at time $t + \Delta t$". Assume that $^{t+\Delta t}R$ is independent of the deformations. Since the solution is known at time *t*, it can be written

$$^{t+\Delta t}F = {}^{t}F + F \tag{4}$$

In which F is the increment in nodal point forces corresponding to the increment in element displacements and stresses from time t to time $t + \Delta t$. This vector can be approximated using a tangent stiffness matrix ^{t}K which corresponds to the geometric and material conditions at time t,

$$F = {}^{t}KU \tag{5}$$

where U is a vector of incremental nodal point displacements and

$$^{t}K = \frac{\partial^{t}F}{\partial^{t}U} \tag{6}$$

Hence, the tangent stiffness matrix corresponds to the derivative of the internal element nodal point forces ^{t}F with respect to the nodal point displacements ^{t}U. Substituting equations (4) and (5) into equation (3), we obtain

$$^{t}KU = {}^{t+\Delta t}R - {}^{t}F \tag{7}$$

and solving for U, we can calculate an approximation to the displacements at time $t + \Delta t$,

$$^{t+\Delta t}U = {}^{t}U + U \tag{8}$$

The exact displacements at time $t + \Delta t$ are those that correspond to the applied loads $^{t+\Delta t}R$. We calculate in equation (8) only an approximation to these displacements because equation (5) was used.

Having evaluated an approximation to the displacements corresponding to time $t + \Delta t$, we could solve for an approximation to the stresses and corresponding nodal point forces at time $t + \Delta t$ and then proceed to the next time increment calculations. However, because of the assumption in equation (5), such a solution may be subject to very significant errors and depending on the time or load step size used, may indeed be unstable. In practice, it is therefore necessary to iterate until the solution of equation (3) is obtained to sufficient accuracy.

The widely used iteration methods in finite element analysis are based on the Newton-Raphson iteration and closely related technique. So in this research, the solution process is proceeding by using a Newton-Raphson scheme. A characteristic of this iteration is that a new tangent stiffness matrix is calculated in each iteration.

To obtain the matrix equation, we consider the motion of a general body in a stationary Cartesian coordinate system and assume that the body can experience large displacements, large strain and a nonlinear constitutive response. The aim is to evaluate the equilibrium positions of the complete body at the discrete time points $0, \Delta t, 2\Delta t, 3\Delta t, \ldots$ where Δt is an increment in time. To develop the solution strategy, assume that the solutions for the static and kinematic variables for all time steps from time 0 to time t, inclusive, have been obtained. Then the solution process for the next required equilibrium position corresponding to time $t + \Delta t$ is typical and is applied repetitively until the complete solution path has been solved for. Hence, in the analysis we follow all particles of the body in their motion, from the original to the final configuration of the body, which means that we adopt a total Lagrangian formulation of the problem and we used it in the finite element model. Considering the analysis of solids and structures, a Lagrangian formulation usually represents a more natural and effective analysis approach than other formulation.

III. MATRIX FORM OF EQUATIONS

The basic steps in the derivation of the governing finite element equations are the same as linear analysis. So, it should be selected the interpolation functions and the interpolation of the element coordinates and displacements with these functions in the governing continuum mechanics equations. By invoking the linearized principle of virtual displacements for each of the nodal point displacements in turn, the governing finite element equations are obtained. As in linear analysis, we need to consider only a single element of a specific type in this derivation because the governing equilibrium equations of an assemblage of elements can be constructed using the direct stiffness procedure.

Using of a standard Galerkin weighted residual and total Lagrangian formulation in developed finite element model, we derive the governing equations for this formulation and obtain the matrix form of equations used in present finite elements model. The stiffness matrix and load vector will be as following:

$$ {}_0^tK = {}_0^tK_L + {}_0^tK_{NL} \tag{9}$$

$$ {}_0^tK_L\hat{u} = \left(\int_{^0V} {}_0^tB_L^T \, {}_0C \, {}_0^tB_L \, d\,^0V\right)\hat{u} \tag{10}$$

$$ {}_0^tK_{NL}\hat{u} = \left(\int_{^0V} {}_0^tB_{NL}^T \, {}_0^tS \, {}_0^tB_{NL} \, d\,^0V\right)\hat{u} \tag{11}$$

$$ {}_0^tF = \int_{^0V} {}_0^tB_L^T \, {}_0^t\hat{S} d\,^0V \tag{12}$$

In which ${}_0^tK_L$ and ${}_0^tK_{NL}$ are the linear and nonlinear strain incremental stiffness matrices. ${}_0^tF$ is the vector of nodal point force equivalent to the element stresses at time *t*. ${}_0^tS$ and ${}_0^t\hat{S}$ are the matrix and vector of second Piola-Kirchhoff stresses and ${}_0C$ is the incremental stress-strain material property matrices.

IV. Two-dimensional plane stress elements

For the derivation of the required matrices and vectors for nonlinear analysis of shear wall, we consider a typical two dimensional plane strain element in its configuration at time 0 and at time t. The global coordinates of the nodal points of the element are at time 0, ${}^0x_1^k$, ${}^0x_2^k$ and at time *t*, ${}^tx_1^k$, ${}^tx_2^k$, where $K = 1,2,...,N$, and N denotes the total number of element nodes. Using the interpolation concepts, we have at time 0:

$$ {}^0x_1 = \sum_{k=1}^{N} h_k \, {}^0x_1^k \tag{13}$$

$$ {}^0x_2 = \sum_{k=1}^{N} h_k \, {}^0x_2^k \tag{14}$$

And at time *t*:

$$ {}^tx_1 = \sum_{k=1}^{N} h_k \, {}^tx_1^k \tag{15}$$

$$ {}^tx_2 = \sum_{k=1}^{N} h_k \, {}^tx_2^k \tag{16}$$

In which, the h_k are the interpolation functions.

Since we use the isoparametric finite element discretization, the element displacements are interpolated in the same way as the geometry. So we can write:

$$ {}^tu_1 = \sum_{k=1}^{N} h_k \, {}^tu_1^k \tag{17}$$

$$ {}^tu_2 = \sum_{k=1}^{N} h_k \, {}^tu_2^k \tag{18}$$

The evaluation of strains requires the following derivatives:

$$ \frac{\partial\,^tu_i}{\partial\,^0x_j} = \sum_{k=1}^{N}\left(\frac{\partial h_k}{\partial\,^0x_j}\right){}^tu_i^k \qquad i = 1,2 \tag{19}$$

The derivatives are calculated in the same way as in linear analysis using a Jacobian transformation. The chain rule relating ${}^tx_1$, ${}^tx_2$ to *r*, *s* derivatives is written as:

$$ \begin{bmatrix} \dfrac{\partial}{\partial r} \\ \dfrac{\partial}{\partial s} \end{bmatrix} = {}^tJ \begin{bmatrix} \dfrac{\partial}{\partial\,^tx_1} \\ \dfrac{\partial}{\partial\,^tx_2} \end{bmatrix} \tag{20}$$

In which

$$ {}^tJ = \begin{bmatrix} \dfrac{\partial\,^tx_1}{\partial r} & \dfrac{\partial\,^tx_2}{\partial r} \\ \dfrac{\partial\,^tx_1}{\partial s} & \dfrac{\partial\,^tx_1}{\partial s} \end{bmatrix} \tag{21}$$

Inverting the Jacobian operator J, we obtain:

$$ \begin{bmatrix} \dfrac{\partial}{\partial\,^tx_1} \\ \dfrac{\partial}{\partial\,^tx_2} \end{bmatrix} = \frac{1}{\det\,^tJ}\begin{bmatrix} \dfrac{\partial\,^tx_2}{\partial s} & -\dfrac{\partial\,^tx_2}{\partial r} \\ -\dfrac{\partial\,^tx_1}{\partial s} & \dfrac{\partial\,^tx_1}{\partial r} \end{bmatrix}\begin{bmatrix} \dfrac{\partial}{\partial r} \\ \dfrac{\partial}{\partial s} \end{bmatrix} \tag{22}$$

where the Jacobian determinants is

$$ \det\,^tJ = \frac{\partial\,^tx_1}{\partial r}\frac{\partial\,^tx_2}{\partial s} - \frac{\partial\,^tx_1}{\partial s}\frac{\partial\,^tx_2}{\partial r} \tag{23}$$

and the derivatives of the coordinates with respect to *r* and *s* are obtained using equation (15):

$$ \frac{\partial\,^tx_i}{\partial r} = \sum_{k=1}^{N}\frac{\partial h_k}{\partial r}\,{}^tx_i^k \tag{24}$$

$$ \frac{\partial\,^tx_i}{\partial s} = \sum_{k=1}^{N}\frac{\partial h_k}{\partial s}\,{}^tx_i^k \tag{25}$$

In which $i = 1,2$.

With all required derivatives defined, it is now possible to establish the strain-displacement transformation matrices for the elements in total Lagrangian formulation. The linear strain-displacement transformation matrix is obtained as following:

$$ {}_0^tB_L = {}_0^tB_{LO} + {}_0^tB_{LI} \tag{26}$$

In which

$$ {}_0^tB_{LO} = \begin{bmatrix} {}_0h_{1,1} & 0 & {}_0h_{2,1} & 0 & {}_0h_{3,1} & 0 & \cdots & {}_0h_{N,1} & 0 \\ 0 & {}_0h_{1,2} & 0 & {}_0h_{2,2} & 0 & {}_0h_{3,2} & \cdots & 0 & {}_0h_{N,2} \\ {}_0h_{1,2} & {}_0h_{1,1} & {}_0h_{2,2} & {}_0h_{2,1} & {}_0h_{3,2} & {}_0h_{3,1} & \cdots & {}_0h_{N,2} & {}_0h_{N,1} \end{bmatrix} \tag{27}$$

$$ {}_0^tB_{LI} = \begin{bmatrix} l_{11}\,{}_0h_{1,1} & l_{21}\,{}_0h_{1,1} & l_{11}\,{}_0h_{2,1} \\ l_{12}\,{}_0h_{1,2} & l_{22}\,{}_0h_{1,2} & l_{12}\,{}_0h_{2,2} \\ l_{11}\,{}_0h_{1,2} + l_{12}\,{}_0h_{1,1} & l_{21}\,{}_0h_{1,2} + l_{22}\,{}_0h_{1,1} & l_{11}\,{}_0h_{2,2} + l_{12}\,{}_0h_{2,1} \end{bmatrix}$$

$$\begin{matrix} l_{21\,0}h_{2,1} & \cdots & l_{11\,0}h_{N,1} & l_{21\,0}h_{N,1} \\ l_{22\,0}h_{2,2} & \cdots & l_{12\,0}h_{N,2} & l_{22\,0}h_{N,2} \\ l_{21\,0}h_{2,2}+l_{22\,0}h_{2,1} & \cdots & l_{11\,0}h_{N,2}+l_{12\,0}h_{N,1} & l_{21\,0}h_{N,2}+l_{22\,0}h_{N,1} \end{matrix}\Bigg] \tag{28}$$

where

$$l_{11}=\sum_{k=1}^{N} {}_0h_{k,1}\,{}^tu_1^k \tag{29}$$

$$l_{22}=\sum_{k=1}^{N} {}_0h_{k,2}\,{}^tu_2^k \tag{30}$$

$$l_{21}=\sum_{k=1}^{N} {}_0h_{k,1}\,{}^tu_2^k \tag{31}$$

$$l_{12}=\sum_{k=1}^{N} {}_0h_{k,2}\,{}^tu_1^k \tag{32}$$

The nonlinear strain-displacement transformation matrix will be as following:

$${}_0^tB_{NL}=\begin{bmatrix} {}_0h_{1,1} & 0 & {}_0h_{2,1} & 0 & {}_0h_{3,1} & 0 & \cdots & {}_0h_{N,1} & 0 \\ {}_0h_{1,2} & 0 & {}_0h_{2,2} & 0 & {}_0h_{3,2} & 0 & \cdots & {}_0h_{N,2} & 0 \\ 0 & {}_0h_{1,1} & 0 & {}_0h_{2,1} & 0 & {}_0h_{3,1} & \cdots & 0 & {}_0h_{N,1} \\ 0 & {}_0h_{1,2} & 0 & {}_0h_{2,2} & 0 & {}_0h_{3,2} & \cdots & 0 & {}_0h_{N,2} \end{bmatrix} \tag{33}$$

It is required to second Piola-Kirchhoff stress matrix and vector to obtain linear and nonlinear stiffness matrix and nodal load vector. These matrices can be written as

$${}_0^tS=\begin{bmatrix} {}_0^tS_{11} & {}_0^tS_{12} & 0 & 0 \\ {}_0^tS_{12} & {}_0^tS_{22} & 0 & 0 \\ 0 & 0 & {}_0^tS_{11} & {}_0^tS_{12} \\ 0 & 0 & {}_0^tS_{21} & {}_0^tS_{22} \end{bmatrix} \tag{34}$$

and

$${}_0^t\hat{S}=\begin{bmatrix} {}_0^tS_{11} \\ {}_0^tS_{22} \\ {}_0^tS_{12} \end{bmatrix} \tag{35}$$

A fundamental observation comparing elastic and inelastic analysis is that in elastic solutions the total stress can be evaluated from the total strain alone, whereas in an inelastic response calculation the total stress at time *t* also depends on the stress and strain history. The solution process can be interpreted to consist of, first, an elastic prediction of stress and then, if this stress prediction lies outside the yield surface, a stress correction. Using the Gauss numerical integration, obtained matrices are evaluated at the Gauss integration points.

V. Example

To d To demonstrate the effectiveness of the analysis procedure presented in this paper and the effect of the nonlinearity on seismic response steel shear wall, the behavior of steel shear wall, due to horizontal and vertical component of El Centro earthquake is presented.

To analyze the model, a steel shear wall with 48 m height and 6 m length subjected to earthquake loading, was considered as an example. The model was analyzed for nonlinear behavior of material and large displacement. 4-node isoparametric elements were used to represent the finite element modeling of the shear wall. Multilinear isotropic hardening for material model has been selected for steel non-linear behavior. Fig. 1 and 2 show the finite element meshing of model and strain-stress curve for selected materially-nonlinear behavior.

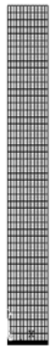

Fig. 1 Finite element discretization of the model

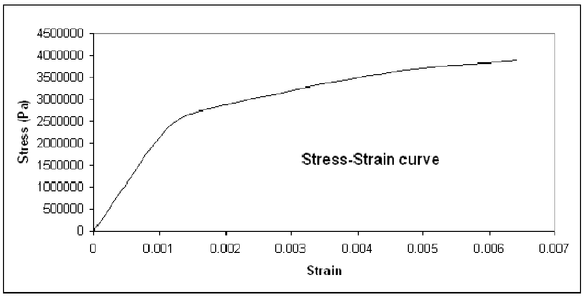

Fig. 2 Multilinear isotropic hardening for material model

Figures 3 and 4 show the horizontal and vertical component of the El Centro earthquake, which is selected for the purpose of seismic analysis. The values of integration parameters in Newmark method were taken as $\beta=0.25$ and $\gamma=0.5$ with time step equal to 0.02 second. The stiffness proportional damping (Rayleigh damping) is used for analysis.

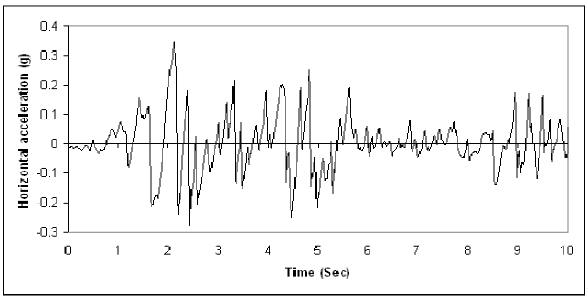

Fig. 3 Horizontal component of the El Centro earthquake

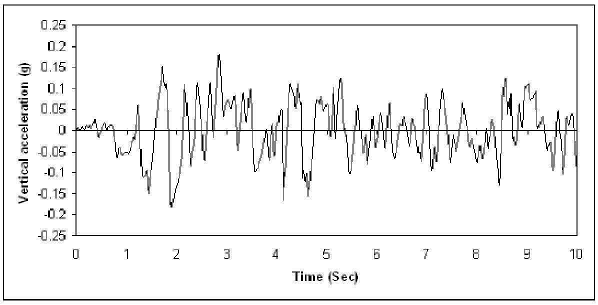

Fig. 4 Vertical component of the El Centro earthquake

After analyzing the model, the results were obtained considering horizontal and vertical components of El Centro earthquake.

Figures 5 and 6 show the time history of horizontal and vertical displacements at top of the steel shear wall.

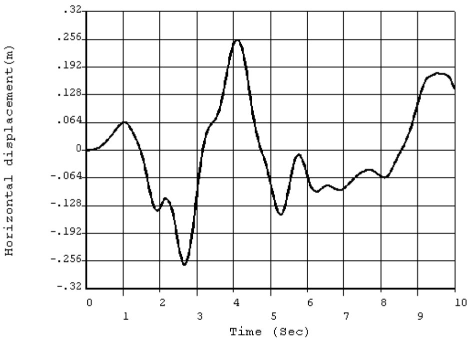

Fig. 5 Time history of horizontal displacement at top of the steel shear wall

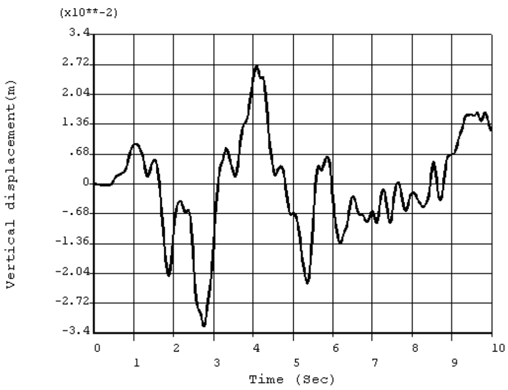

Fig. 6 Time history of vertical displacement at top of the steel shear wall

Furthermore, it is worth paying attention to the variation of principle stress during earthquake loading. The time histories of principle tensile and compressive stress on the bottom of wall are shown in figures 7 and 8.

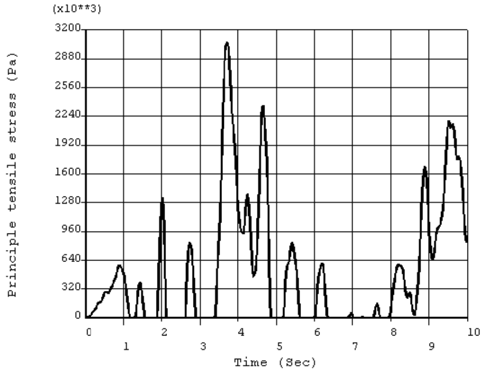

Fig. 7 Time history of principle tensile stress on the bottom of shear wall

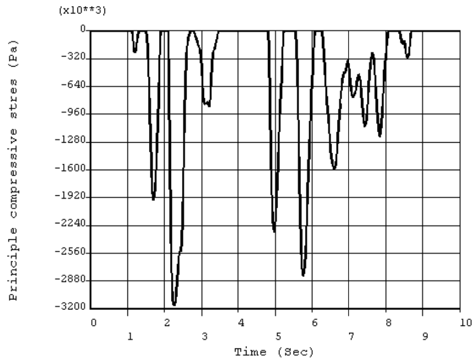

Fig. 8 Time history of principle compressive stress on the bottom of shear wall

VI. Conclusion

A numerical model for a transient nonlinear analysis of steel shear wall under seismic loading was presented in this paper. For numerical modeling, the finite element formulation for plane stress elements has been reviewed to include a standard Galerkin weighted residual formulation more general and concise than those existing in the literature. The technique is an enhanced represented for analysis with materially nonlinear behavior and large displacements. An example was considered to describe the numerical procedures and show the nonlinear transient behavior in such structures. This work can provide the further understanding of the characteristics of nonlinear behavior under different loading in structures and may be taken for a quantitative comparison to various analysis and numerical solutions.

References

[1] A. K. Marsono and L. S. Wee, “Nonlinear finite element analysis of reinforced concrete tube in tube of tall building,” Proceeding of the 6th Asia-Pacific structural engineering and construction conference, Kuala Lumpur, Malaysia, 2006.

[2] T. Kabeyasawa and J. I. Milev, “Modeling of reinforced concrete shear walls under varying axial load,” Transaction of the 14th international conference on structural mechanics in reactor technology”, Lyon, France, 1997, pp. 500-600.

[3] K. J. Bathe, “Finite element procedure,” Prentice-Hall, Inc, 1996.

[4] O. C. Zienkiewicz and R. L. Taylor, “The finite element method,: The basis,” vol. 1, 2000, pp. 200-246.