



## 동적 문제에 효율적인 적응적 유한요소망

윤 종 열<sup>1†</sup>

<sup>1</sup>홍익대학교 건설도시공학부

## Efficient Adaptive Finite Element Mesh Generation for Dynamics

Chongyul Yoon<sup>1†</sup>

<sup>1</sup>Department of Civil Engineering, Hongik University, Seoul, 121-791, Korea

### Abstract

The finite element method has become the most widely used method of structural analysis and recently, the method has often been applied to complex dynamic and nonlinear structural analyses problems. Even for these complex problems, where the responses are hard to predict, finite element analyses yield reliable results if appropriate element types and meshes are used. However, the dynamic and nonlinear behaviors of a structure often include large deformations in various portions of the structure and if the same mesh is used throughout the analysis, some elements may deform to shapes beyond the reliable limits; thus dynamically adapting finite element meshes are needed in order for the finite element analyses to be accurate. In addition, to satisfy the users requirement of quick real run time of finite element programs, the algorithms must be computationally efficient. This paper presents an adaptive finite element mesh generation scheme for dynamic analyses of structures that may adapt at each time step. Representative strain values are used for error estimates and combinations of the h-method(node movement) and the r-method(element division) are used for mesh refinements. A coefficient that depends on the shape of an element is used to limit overly distorted elements. A simple frame example shows the accuracy and computational efficiency of the scheme. The aim of the study is to outline the adaptive scheme and to demonstrate the potential use in general finite element analyses of dynamic and nonlinear structural problems commonly encountered.

**Keywords :** adaptive finite element mesh, finite element method, structural dynamics

### 1. Introduction

The finite element method(FEM) has become the most widely used method of structural analysis and recently the method has often been applied to complex dynamic and nonlinear structural analyses problems. Even for these complex problems, if appropriate element types and meshes are used, finite element analyses have produced reliable results (Bathe and Wilson, 1976; Belytschko *et al.*, 1996; Zienkiewicz *et al.*, 2005). However, the dynamic and nonlinear behaviors of a structure often include large

deformations in various portions of the structure and if the same mesh is used throughout the finite element analysis, some elements may deform to shapes beyond the reliable limits causing significant errors in the output results. To reduce this type of error, engineers in practice often use a fine uniform mesh for the entire finite element model. This is computationally inefficient and severely hinders the general users requirement of quick real run time of finite element programs. In addition, elements being small does not prevent them from becoming overly deformed during an unpredictable dynamic analysis.

\* Corresponding author:

Tel: +82-2-320-1478; E-mail: cyoon@hongik.ac.kr

Received August 6 2013; Revised September 30 2013;

Accepted October 1 2013

©2013 by Computational Structural Engineering Institute of Korea

This is an Open-Access article distributed under the terms of the Creative Commons Attribution Non-Commercial License(<http://creativecommons.org/licenses/by-nc/3.0>) which permits unrestricted non-commercial use, distribution, and reproduction in any medium, provided the original work is properly cited.

Thus a dynamically adapting finite element mesh is needed in order for a finite element analysis to be accurate and computationally efficient(Heesom and Mahdjoubi, 2001; Jang and Lee, 2011; Yoon and Park, 2010).

In this paper, a computationally efficient and dynamically adaptive finite element mesh generation scheme for dynamic analysis of structures is described. Through diverse studies on many practical problems, the scheme is an improved and refined algorithm of the basic procedure presented in reference Yoon and Park(2010) and Yoon(2012). A general adaptive mesh generation process involves two basic steps : estimation of error given a mesh and generation of a new improved mesh based on the error(Choi and Jung, 1998; de Las Casa, 1988; Zienkiewicz and Zhu, 1987). The dynamically adaptive scheme uses computationally simple error estimation based on representative strain(Yoon, 2005). Based on this error an optimal combination of the r-method(moving an existing node) and the h-method(dividing an element into smaller elements) is used to generate an improved mesh. The limits on distortions of elements are checked by an easily computed shape factors of elements(Yoon, 2005). The dynamic analysis considered is in time domain, based on direct integration. As an example, an undamped frame modeled with four node quadrilateral isoparametric elements for plane stress is considered. The results of the analyses of the example show that the proposed dynamically adaptive finite element mesh scheme using the error estimation based on the representative strain, the r-h method, and the shape factor is reasonably accurate and computationally efficient.

## 2. Adaptive mesh generation scheme for dynamic analysis

Error estimation given a mesh and a scheme for generating a refined mesh based on the errors are the two main steps in an adaptive mesh generation. In a time domain dynamic analysis or in a typical nonlinear analysis, at each time step, a new mesh

needs to be generated although not necessarily at each time step. For a new mesh, a check to prevent overly distorted element shape is also essential.

### 2.1 Error estimation with representative Strains

The results of a finite element analysis largely depend on the type of element formulation and the mesh used. An adaptive scheme attempts to generate an appropriate mesh for the finite element analysis. A scheme starts with error estimation given an initial mesh and iterate until an end condition is satisfied. Initial mesh and end conditions are important for the efficiency of the scheme(Jeong *et al.*, 2003; McFee and Giannacopoulos, 2001; Ohnimus *et al.*, 2001; Stampfle *et al.*, 2001; Zhu *et al.*, 1991). In engineering problems, accurate solutions are not known and thus accurate error estimation is inherently a difficult problem. Norm of a matrix is generally used to represent error of a mesh where the matrix includes values of stress, strain and displacements. Norm of error  $\|E\|$  in the domain  $\Omega$  may be represented as follows :

$$\|E\| = \left[ \int_{\Omega} (\epsilon - \hat{\epsilon})^T (\sigma - \hat{\sigma}) d\Omega \right] \quad (1)$$

Here  $\sigma$ ,  $\epsilon$  are the exact solutions for stress and strain and  $\hat{\sigma}$ ,  $\hat{\epsilon}$  are the finite element solutions. The error defined in Eqs. (2~5) are the representative strains of element based on the standard deviations of the strains at the Gauss points in the element computed during the previous finite element analysis. The  $z$  axis is not considered in  $x-y$  planar problems, and the components of representative strain values of element  $i$  are represented by the following equations:

$$\|\epsilon\|_{ix} = \sqrt{\frac{\sum_{j=1}^{n_{gx}} (\epsilon_{jx} - \epsilon_x^*)^2}{n_{gx} - 1}} \quad (2)$$

$$\|\epsilon\|_{iy} = \sqrt{\frac{\sum_{j=1}^{n_{gy}} (\epsilon_{jy} - \epsilon_y^*)^2}{n_{gy} - 1}} \quad (3)$$

$$\|e\|_{ixy} = \sqrt{\frac{\sum_{j=1}^{n_{gxy}} (\gamma_{jxy} - \gamma_{xy}^*)^2}{n_{gxy} - 1}} \quad (4)$$

$$\|e\|_i = \{\|e\|_{ix} + \|e\|_{iy} + \|e\|_{ixy}\} \times \frac{A_i}{A_{total}} \quad (5)$$

Here,  $\|e\|_{ix}$  is the  $x$  directional standard deviation of the strain,  $n_{gx}$  is the number of Gauss points in  $x$  direction,  $\epsilon_{jx}$  is the  $x$  directional strain of Gauss point  $j$ , and  $\epsilon_x^*$  is the  $x$  directional strain;  $\|e\|_{iy}$ ,  $n_{gy}$ ,  $\epsilon_{jy}$ ,  $\epsilon_y^*$  are the similar  $y$  directional values and  $\|e\|_{ixy}$ ,  $n_{gxy}$ ,  $\gamma_{jxy}$ ,  $\gamma_{xy}^*$  are the similar  $x-y$  shear strain values. In addition,  $A_i$  is the element area and  $A_{total}$  is the total area of the mesh.

## 2.2 Dynamic analysis algorithm

The direct numerical integration in the time domain is used for the dynamic analysis. Specifically, the Newmark- $\beta$  method, a reliable explicit method widely used in dynamic structural analysis, is used (Newmark, 1959). The equations in the Newmark- $\beta$  method may be summarized as follows :

$$\dot{u}_{i+1} = \dot{u}_i + (\Delta t)[(1-\gamma)\ddot{u}_i + \gamma\ddot{u}_{i+1}] \quad (6)$$

$$u_{i+1} = u_i + (\Delta t)\dot{u}_i + (\Delta t)^2[(\frac{1}{2}-\beta)\ddot{u}_i + \beta\ddot{u}_{i+1}] \quad (7)$$

Here,  $u_i$ ,  $\dot{u}_i$ ,  $\ddot{u}_i$  are the displacement, velocity, and acceleration vectors in the  $i$ th step and  $u_{i+1}$ ,  $\dot{u}_{i+1}$ ,  $\ddot{u}_{i+1}$  are the similar quantities in the  $i+1$ th step;  $\Delta t$  is the time step length;  $\gamma$  and  $\beta$  are parameters selected by the user and the recommended values of  $\gamma=1/2$  and  $\beta=1/4$  are commonly used.

The matrix equilibrium equations for the  $i$ th step may be represented as follows :

$$K' \vec{u}_{i+1} = F'_{i+1} \quad (8)$$

The matrices  $K'$  and  $F'_{i+1}$  represent the following:

$$K' = K + \frac{1}{\beta(\Delta t)^2} M \quad (9)$$

$$F'_{i+1} = F_{i+1} + \frac{M}{\beta(\Delta t)^2} [u_i + (\Delta t)^2 \dot{u}_i + (\frac{1}{2} - \beta)(\Delta t)^2 \ddot{u}_i] \quad (10)$$

Here,  $K$  is the stiffness matrix,  $M$  is the mass matrix, and  $F_{i+1}$  is the force vector for the  $i+1$ th time step.

## 2.3 The r-h mesh generation scheme

The adaptive mesh generation based on the error estimation with representative strains is formulated combining the r-method and the h-method. The r-method moves existing nodes based on the following equations for the new coordinates  $x_b$ ,  $y_b$  :

$$x_b = \frac{\sum_{i=1}^{n_m} x_{ci} (\|e\|_i)}{\sum_{i=1}^{n_m} (\|e\|_i)} \quad (11)$$

$$y_b = \frac{\sum_{i=1}^{n_m} y_{ci} (\|e\|_i)}{\sum_{i=1}^{n_m} (\|e\|_i)} \quad (12)$$

Here,  $x_{ci}$ ,  $y_{ci}$  are the coordinates of the centroid of the element,  $n_m$  is the number of elements and  $\|e\|_i$  is the representative strain value of element  $i$ . Eq. (11) and Eq. (12) for nodes on the boundary are modified according to the following equations :

$$x_b = \frac{\sum_{i=1}^2 x_{bci} (\|e\|_i)}{\sum_{i=1}^2 (\|e\|_i)} \quad (13)$$

$$y_b = \frac{\sum_{i=1}^2 y_{bci} (\|e\|_i)}{\sum_{i=1}^2 (\|e\|_i)} \quad (14)$$

Here,  $x_{bci}$ ,  $y_{bci}$  are the  $x$ ,  $y$  coordinates of the central point on the boundary.

The shapes of the elements are kept rectangular by limiting  $S_i$ , the shape factor of element  $i$ . Eq. (15) defines  $S_i$  and details may be found in reference Jeong *et al.*(2003).

$$S_i = \frac{\sqrt{A_i}}{0.25L_i} \quad (15)$$

Here,  $L_i$  is the total length of the boundary of element  $i$ . For quadrilaterals,  $S_i$  is 1 for square which is an ideal value. Limiting  $S_i$  to over 0.9 and keeping the lengths of four sides to be approximately equal are efficient means of preventing overly distorted element shapes.

The h-method divides an element into smaller elements of the same type. The elements to be subdivided are based on the discretization parameter  $d$ :

$$d = \alpha \times \text{mean}[\|e\|_{\text{initial}}] \div P_{\text{maximum}} \quad (16)$$

Here,  $\alpha$  is a constant,  $\text{mean}[\|e\|_{\text{initial}}]$  is the mean representative strain value of the initial mesh and  $P_{\text{maximum}}$  is the maximum value of the applied load. Experimental and parametric studies report that values between 12.0 and 15.0 are appropriate for  $\alpha$ .

Various means of combining the r-method and the h-method have been studied(de Las Casas, 1988; Yoon, 2005). To obtain an optimal combination of the r-method and the h-method, first the representative strain values are normalized using the following equations.

$$\text{minimum } [\|e\|_i] \times a + b = 0 \quad (17)$$

$$\text{maximum } [\|e\|_i] \times a + b = 100 \quad (18)$$

Here,  $\text{minimum } [\|e\|_i]$  and  $\text{maximum } [\|e\|_i]$  are the minimum and the maximum values of the representative strains. The constants  $a$  and  $b$  are determined from Eq. (17) and Eq. (18) and they are used to normalize the representative strains to range from 0 to 100.

A dispersion parameter  $D$  is defined as follows :

$$D = |A_e - M_e| \quad (19)$$

Here,  $A_e$  and  $M_e$  are the average and the mode of the distribution of the representative strain values. A value for  $D$  needs to be set to combine the r-method and the h-method and a reasonable value is between 18 and 20 where if the value is larger the r-method is used, and in other cases, the h-method is used.

In the h-method, the refinement of the mesh is terminated when the change in the representative strain values is less than 1% or when the element's discretization parameter is less than  $d$  in Eq. (16). In the r-method, the refinement of the mesh is terminated when the element's discretization parameter is less than  $d$ .

### 3. A simple frame example

The example considered is a two dimensional portal steel frame that is 600cm wide and 600cm high as shown in Fig. 1. The bottom of the two columns are fixed to the ground. The beam and columns are all 100cm deep. For material properties, Young's modulus is  $210 \times 10^5 \text{ N/cm}^2$ , Poisson's ratio is 0.3, and the unit mass is  $7.85 \times 10^{-3} \text{ kg/cm}^3$ . For modeling, 1cm thickness is assumed and 4 node isoparametric quadrilateral plane strain elements are used. The only applied

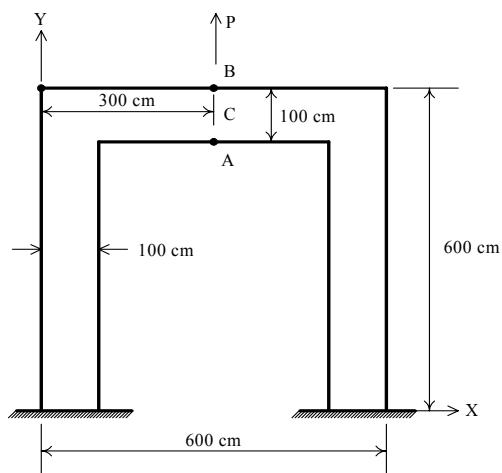


Fig. 1 Geometry of the frame example

dynamic load is a concentrated load  $P$  at the center of the frame(point B in Fig. 1) where between time  $t=0$  and  $t=1$  second, the function is given by the following equation and as shown in Fig. 2.

$$P = -500 \sin(2\pi t) \quad (20)$$

Free vibration response continues after the steady state response for 1 second, and the total response time considered is 5 seconds. The time step  $\Delta t$

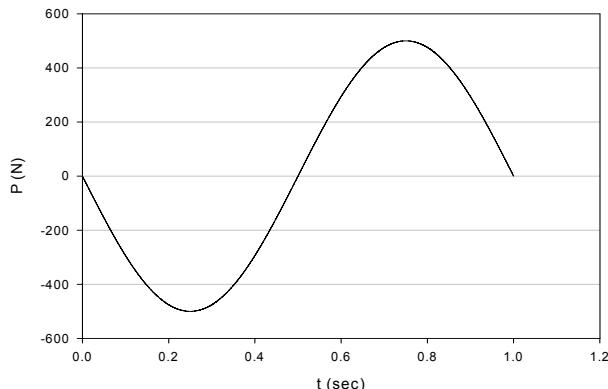
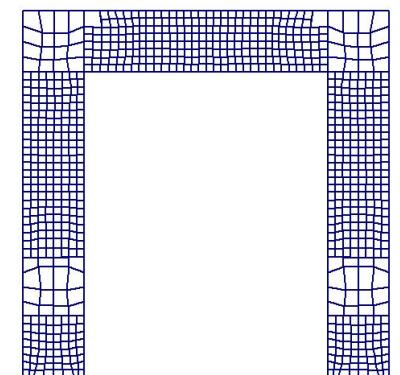
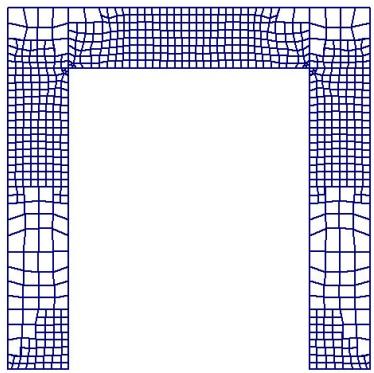


Fig. 2 Time variation of the applied vertical load  $P$

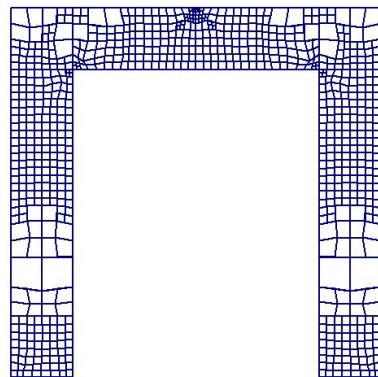


(a) 1.25sec(826 elements, 953 nodes)



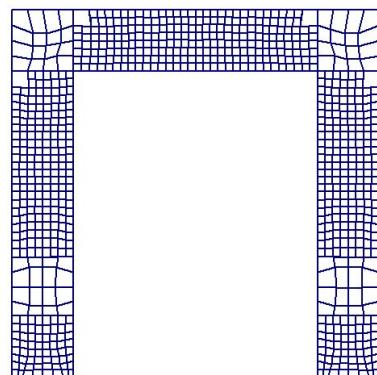
(c) 3.75sec(760 elements, 903 nodes)

selected for analysis is 0.01 seconds, yielding 500 steps for 5 seconds. The adaptive meshes are generated with dispersion parameter  $D=20$ , discretization parameter  $d=15$ , and limiting shape factor  $S_i$  to 0.9. The initial mesh is a regular mesh composed of 144 identical square elements. Fig. 3 shows the mesh at 0.27 seconds when the vertical deflection of point A is at the maximum which is during the steady state response. Fig. 4 shows some samples of the

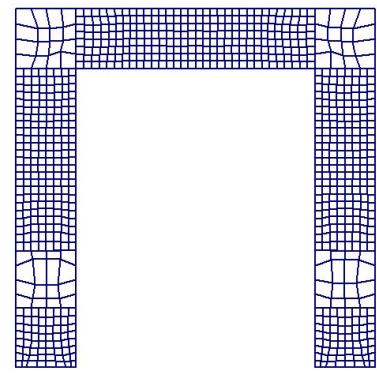


0.27sec, time when deflection at A is maximum  
(840 elements, 997 nodes)

Fig. 3 Mesh at the maximum deflection of point A during steady state



(b) 2.50sec(820 elements, 957 nodes)



(d) 5.00sec(832 elements, 969 nodes)

Fig. 4 Generated adaptive meshes during free vibration

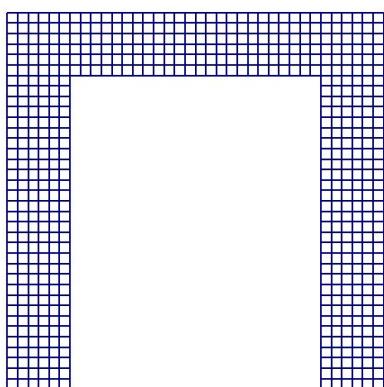


Fig. 5 Regular constant mesh used for the general solution

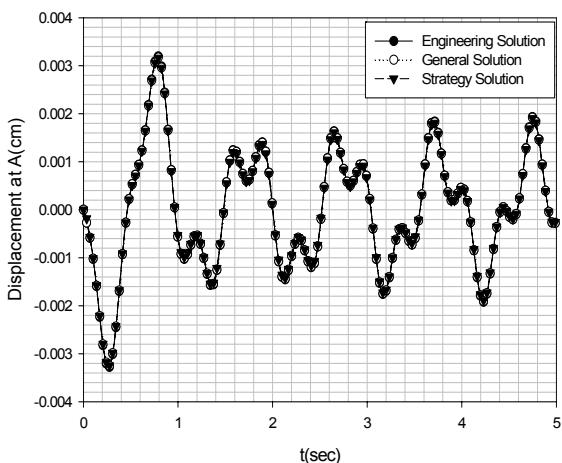


Fig. 6 Vertical displacement of bottom point A of the frame

generated meshes during the free vibration stages of analysis. As expected, note that the meshes are symmetric, fine meshes are under the load during the steady state response only(see Fig. 3), and during free vibration(see Fig. 4) coarse meshes are at the corners and away from the fixed end of columns. The most number of elements generated is 840 at around 0.27 seconds and the minimum number of elements generated is 760 at around 3.75 seconds. At the boundary between a fine and a coarse element, the node at the middle of the coarse mesh is slaved to the end nodes for displacement compatibility.

Solutions obtained by the dynamic adaptive meshes are termed the *strategic* solutions. To represent a typical mesh used for this type of analysis, a regularly discretized mesh of square shapes with 576 elements is shown in Fig. 5. Solutions obtained by using this mesh throughout the analysis are termed the *general*

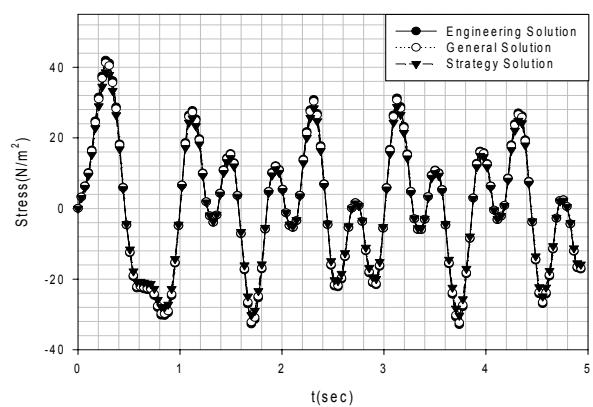


Fig. 7 Mid-horizontal normal stress at mid point C of the frame

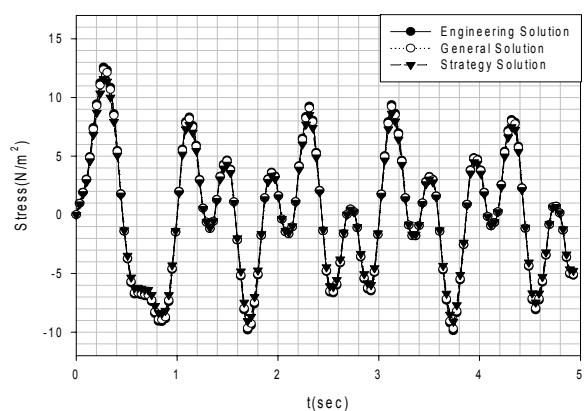


Fig. 8 Mid-vertical normal stress at mid point C of the frame

solutions. To estimate a more accurate engineering solutions without an adaptive scheme, a finer regular mesh with 2304 elements is generated by dividing each element used in the general solution into four identical square elements. Solutions obtained by using this fine mesh are termed the *engineering* solutions. Fig. 6, Fig. 7 and Fig. 8 show respectively, the comparisons of the vertical displacement of point A, the mid-horizontal( $x$  directional) normal stress of point C, and the mid-vertical( $y$  directional) normal stress of point C of the engineering, the general, and the strategy solutions. The graphs agree in general with the expected responses and graphically the solutions are in close agreement. However numeric values show that if the engineering solutions are assumed to be the most accurate, the errors from the general solutions are much larger than the errors from the strategic solutions.

Although the computing speed of computers is

**Table 1** Comparative computation times and error

| Solution Method                | Total Displacement Error<br>vs <i>Engineering</i> | Total Stress Error<br>vs <i>Engineering</i> | Computation Time<br>(Increase vs <i>General</i> ) |
|--------------------------------|---|---|---|
| Engineering<br>(2304 elements) | 0.00%<br>(assumed exact)                          | 0.00%<br>(assumed exact)                    | 32min. 23sec<br>(1063.47%)                        |
| General<br>(576 elements)      | 3.70%   | 2.82%                                       | 2min. 47sec<br>(0.00%)                            |
| Strategy<br>(760-840 elements) | 0.52%   | 0.34%                                       | 6min. 42sec<br>(140.72%)                          |

continuously increasing in general, the enormous amount of computing needed in the complex analysis of structures requires computing efficiency in every aspect of the algorithm in order for the method to be practical(Stampfle *et al.*, 2001; Zienkiewicz *et al.*, 2005). Table 1 shows the comparative computation times and error among the engineering, general, and strategic solutions. The total error is the square root of the sum of the errors at selected critical points where the engineering solutions are assumed to have no error. The analyses times are real run times where the program is run on Personal Computer with Pentium Dual Core CPU at 2.60 GHz, 2.0 GB RAM, and Windows XP. The data on the table shows that with the dynamic adaptive scheme, errors have been reduced significantly(3.70% to 0.52% and 2.82% to 0.34%) with a reasonable increase(140.72%) in real run time whereas the fine mesh for the entire structure for the engineering solutions required enormous increase in real run time of 1063.47%.

#### 4. Conclusions

A dynamically adaptive mesh generation scheme for dynamic analyses of structures in time domain is presented. The scheme uses representative strain values from each element computed from the previous time step for estimation of error and an efficient combination of the r-method and the h-method for mesh refinement. A refined coefficient that depends on the shape of an element is used to correct overly distorted elements. Analysis of the application of the scheme to a simple frame example shows that the adaptive scheme significantly reduces error with a reasonable increase in computational time.

The proposed adaptive mesh algorithm is computationally efficient and thus the scheme may be applied to analysis of large complex structures under erratic time dependent loads such as earthquakes and to non linear problems where there are many similarities with the time domain dynamic analysis. Some aspects of the scheme still need to be improved, one of which is the appropriate selection of the initial mesh that starts the algorithm; this may be achieved with an appropriate development of an expert system that accounts the geometry of the structure and the characteristics of the loading.

#### Acknowledgments

This work was supported by 2011 Hongik University Research Fund.

#### References

- Bathe, K.J., Wilson, E.L.** (1976) *Numerical Methods in Finite Element Analysis*, Prentice Hall, Englewood Cliffs.
- Belytschko, T., Hughes, J.R., Bathe, K.J.** (1996) *Finite Element Procedures*, Prentice-Hall, Englewood Cliffs.
- Choi, C., Jung, H.** (1998) Adaptive Mesh Generation for Dynamic Finite Element Analysis, *J. Korean Soc. of Civil Eng.* (in Korean), 18(I-2), pp.203~220.
- de Las Casas, E.B.** (1988) *R-H Mesh Improvement Algorithms for the Finite Element Method*, Ph.D. Dissertation, Purdue University, West Lafayette.
- Heesom, D., Mahdjoubi, L.** (2001) Effect of Grid Resolution and Terrain Characteristics on Data from DTM, *J. Comp. in Civil Eng.*, ASCE, 15(2), pp.137~143.

- Jang, H., Lee, J.** (2011) Seismic Analysis of Chemical Pump Using Automatic Mesh Generation System, *J. Comp. Struct. Eng. Inst. of Korea*, 24(6), pp.685~690.
- Jeong, Y.C., Yoon, C., Hong, S.** (2003) Adaptive Mesh Generation Scheme for Planar problems using Representative Strain Values for Error (in Korean), *Proc., Korean Soc. Comp. Structural Eng.*, 16(2-31), pp.403~409.
- McFee, S., Giannacopoulos, D.** (2001) Optimal Discretizations in Adaptive Finite Element Electromagnetics, *Int. J. Numer. Meth. Eng.*, 52(9), pp.939~978.
- Newmark, N.M.** (1959) A Method of Computation for Structural Dynamics, *J. Eng. Mech. Division, ASCE*, 85(EM3), pp.67~94.
- Ohnimus, S., Stein, E., Walhorn, E.** (2001) Local Error Estimates of FEM for Displacements and Stresses in Linear Elasticity by Solving Local Neumann Problems. *Int. J. Numer. Meth. Eng.*, 52(7), pp.727~746.
- Stampfle, M., Hunt, K.J., Kalkkuhl, J.** (2001) Efficient Simulation of Parameter-Dependent Vehicle Dynamics, *Int. J. Numer. Meth. Eng.*, 2(11), pp.1273~1299.
- Yoon, C.** (2005) Adaptive Mesh Generation for Dynamic Finite Element Analysis (in Korean), *J. Korean Soc. Civil Eng.*, 25(6A), pp.989~998.
- Yoon, C.** (2012) Adaptive Finite Element Mesh Generation for Dynamic Plana Problems, *J. Korean Soc. Hazard Mitigation*, 12(6), pp.151~156.
- Yoon, C., Park, J.** (2010) Dynamically Adaptive Finite Element Mesh Generation Schemes, *J. Comp. Struct. Eng. Inst. of Korea*, 23(6), pp.659~665.
- Zhu, J.Z., Zienkiewicz, O.C., Hinton, E., Wu, J.** (1991) A New Approach to the Development of Automatic Quadrilateral Mesh Generation, *Int. J. Numer. Meth. Eng.*, 32, pp.849~866.
- Zienkiewicz, O.C., Zhu, J.Z.** (1987) A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis, *Int. J. Numer. Meth. Eng.*, 24, pp.337~357.
- Zienkiewicz, O.C., Taylor, R.L., Zhu, J.Z.** (2005) *The Finite Element Method: Its Basis and Fundamentals*, 6th Ed., Elsevier Butterworth-Heinemann, Oxford.

## 요지

유한요소법은 구조해석법으로 가장 많이 사용되는 방법으로 자리잡고 있으며, 근래에는 다소 복잡한 동적 및 비선형 문제에도 사용이 일반화되고 있다. 이러한 거동 예측이 어려운 구조해석에도 구조물을 적절한 유한요소와 요소망으로 표현하면 신뢰있는 해석 결과를 얻을 수 있다. 구조물의 동적 또는 비선형 거동에는 예상하지 않은 부분에서 큰 변형이 일어날 수 있으며, 유한요소해석 과정에서 같은 요소망을 계속 사용하면 요소의 모양이 신뢰 범위 밖으로 변형될 수 있으므로 요소망 역시 동적으로 적응할 필요가 있다. 또한, 유한요소 프로그램의 사용자 요구 사항 중 하나가 실시간으로 빠르게 진행되는 것으로 연산면에서 효율적이어야 한다. 본 연구는 시간영역 동적해석에서 전 단계 해석 결과를 사용하여 계산된 대표 변형률 값을 오차 평가에 사용하여 절점 이동인  $r$ -법과 요소 분할인  $h$ -법의 조합으로 요소 세분화를 진행하여 동적으로 적응하는 요소망 형성 과정을 기술한다. 해석 중 과대하게 변형되는 요소는 모양계수 개념으로 방지한다. 간단한 프레임의 동적 유한요소해석을 예제로 정확성과 연산 효율성을 보여준다. 본 연구에서 제시하는 적응적 유한요소망 형성 전략은 복잡한 동적 및 비선형 해석에 일반적으로 적용될 수 있다.

**핵심용어** : 적응적 유한요소망, 유한요소법, 구조 동역학