Effect of Columns Discretization on P-δ Analysis

Dr. Salah R. Al Zaidee
Lecturer, Civil Eng. Dept.,
College of Engineering, University of Baghdad

Dr. Hayder A. Al-Baghdadi
Lecturer, Civil Eng. Dept.,
College of Engineering, University of Baghdad.

Aqeel T. Fadhil
Assistant lecturer, Civil Eng. Dept.,
College of Engineering, University of Baghdad

Abstract

Utilizing recently available high-strength materials and design codes along with the current complex design layouts have led to frequent use of slender compression members where P-δ analysis should be included in the analysis and design. This type of analysis is usually executed using commercial software. Therefore, designers need to be fully aware of the aspects that affect the results of P-δ analysis including the method of analysis used by the program, matrix method or the finite element method, and column discretization ratio. Ideal column mesh size is not covered by software manuals and has been a controversial topic for engineers' discussion as there is no available guidance for engineers to follow. Improper selection of column mesh size may lead to inaccurate or even misleading analysis results. The interest of this study is to investigate the influence of column discretization on P-δ analysis in order to provide a scientific base for engineers to rely on. The research also diagnoses the difference between software that utilizes matrix method and those that utilizes finite element method when adopted in P-δ analysis of columns. Loading case is also examined in this study as a factor that could affect the ideal mesh size for each method of analysis. Two loading scenarios are investigated; nodal loads and member loads. A Matlab code has been written to show the difference between the two approaches, matrix and finite element methods, when used for a column with nodal forces without the influence of different solving techniques used by software. Two programs are then implemented to study the effect of column discretization on P-δ analysis under different solving approaches and different level of axial loading. STAAD Pro software has been considered as an example on software that adopts matrix method for skeleton structures and that is used in routine design works. On the other hand, Abaqus software is taken as an example for software that completely implements finite element method and is usually used in research field. Results for different case studies indicate that the solving approach, matrix or finite element methods, significantly affect the choice of ideal column discretization ratio. The results also show that columns that are subjected to member forces seem more sensitive to mesh size comparing to the corresponding columns that are subjected to nodal forces.

Keywords: - P-δ analysis, matrix method, finite element analysis, mesh size, Matlab code, STAAD Pro software, Abaqus software.

I. Introduction

In traditional structural analysis, equilibrium equations are generally formulated in terms of undeformed shape. This formulation may be adequate to estimate the interaction between axial forces and bending moments in short columns.

For long columns (or slender columns) equilibrium equations should be formulated in terms of deformed shape to include the secondary moment that is resulted from the product of axial forces by the lateral deformations due to primary moments.

In general, design codes offer three different approaches to deal with the secondary moments of slender columns. In the first approach, usually called the Moment Magnification Method, both of the geometric and material nonlinearities are treated empirically by modification of member stiffness to simulate possible cracking and yielding and by magnify the primary moments to simulate the geometric nonlinearity **[4]**

In the second approach, called Elastic Second Order Analysis, the material nonlinearity is still simulated empirically, while the geometric nonlinear is modeled rationally by adopting the deformed shape in the preparation of the stiffness matrices for columns and connected beams. Finally, in the third approach, both of material and geometric nonlinearities are simulated rationally in a Nonlinear Second Order Analysis.

The second approach is relatively simple and can be adopted in the finite element models of commercial software. Unfortunately, the built-in stiffness matrix for most commercial software has been developed in terms of Hermite cubical shape function. When P-δ effect increases, column behavior asymptotically approaches the critical buckling case where the mode shapes are trigonometric in nature. The difference between assumed cubical shape and actual trigonometric shape leads to an inaccurate analysis **[9]**

During their practical works and discussions, the authors have noted that usually a single element is used to simulate the whole column length without giving adequate cautions to the aforementioned drawback in the stiffness formulation. What makes this issue deserves more concern is that most of the related literature do not discuss the discretization aspect when presenting details and provisions related to the Elastic Nonlinear Analysis.

This paper aims to show how column discretization can affect the result of Elastic Nonlinear Analysis. For this propose, a Matlab code has been written to simulate the difference between matrix method and finite element method when adopted in P-δ analysis. Different case studies have been analyzed using STAAD.Pro software and Abaqus software. The former has been employed to represent the software that is based on displacement method and is usually used in routine design works while the latter is adopted as an example on more sophisticated software that is completely based on finite element method and is usually used in research field.

II. Matrix versus Finite Element Formulations

For framed structures, stiffness matrix, that relates nodal forces to nodal displacements, can be formulated either based on matrix method or based on finite element method **[11].**

In the matrix method, the stiffness matrix is derived from an analytical solution of the governing differential equation **[16].** Except the approximation during factorization of the resulting simultaneous equations, solutions with matrix approach are closed forms in nature **[8].**

On the other hand, in the finite element formulation, the stiffness matrix is formulated based on an approximate displacement field with a variational principle **[14].**

A. Stiffness Matrix Based on Matrix Approach

Referring to the deflected shape of column indicated in **Fig. 1** , and by adopting the second derivative, v'', to approximate the curvature of the elastic curve, the governing differential equation would be as indicated in Eq. (1) **[15].**

$$\frac{d^4v}{dx^4}+\frac{P}{EI}\frac{d^2v}{dx^2}=\frac{q}{EI} \qquad \text{Eq. (1)}$$

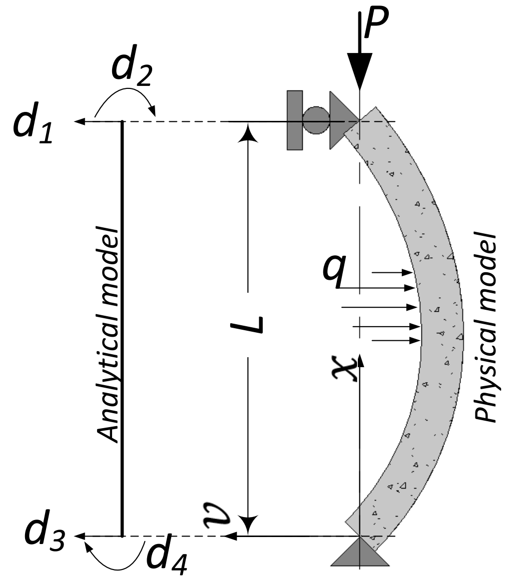

Fig.1 Deflected shape of a single column with physical and analytical models.

The analytical solution of Eq. (1) is presented in Eq. (2) **[9].**

$$v=A_1\sin\left(u\frac{x}{L}\right)+A_2\cos\left(u\frac{x}{L}\right)+A_3\,x+A_4 \qquad \text{Eq. (2)}$$

where A_i are the integration constants that can be determined from boundary conditions, while the coefficient u is a non-dimensional coefficient that is defined as follows:

$$u=L\sqrt{\frac{P}{EI}}$$

Accordingly **[9]**, when Eq. (2) is reformulated in a matrix form and when integration constants, A_i , are expressed in terms of degrees of freedom indicated in **Fig. 1** , the stiffness matrix would be as indicated in Eq. (3).

$$[K] = EI \times \begin{bmatrix} \frac{u^3\, s}{l^3(2-2c-us)} & & \textit{Symmetrical} & \\ \frac{u^2(1-c)}{l^2(2-2c-us)} & \frac{u(s-uc)}{l(2-2c-us)} & & \\ \frac{-u^3\, s}{l^3(2-2c-us)} & \frac{-u^2(1-c)}{l^2(2-2c-us)} & \frac{u^3\, s}{l^3(2-2c-us)} & \\ \frac{u^2(1-c)}{l^2(2-2c-us)} & \frac{u(u-s)}{l(2-2c-us)} & \frac{-u^2(1-c)}{l^2(2-2c-us)} & \frac{u(s-uc)}{l(2-2c-us)} \end{bmatrix}$$

Eq. (3)

B. Stiffness Matrix Based on Hermite Displacement Field

In finite element method, Hermite cubical displacement field indicated in Eq. (4) is usually adopted in the approximate formulation of stiffness matrix of a frame element **[6].**

$$v = \alpha_0 + \alpha_1 x + \alpha_2 x^2 + \alpha_3 x^3 \qquad \text{Eq. (4)}$$

According to **[7],** with Hermite displacement field of Eq. (4) , and by using degrees of freedom of **Fig. 1** , the stiffness matrix for the beam would be as indicated in Eq. (5).

$$[K] = \begin{bmatrix} 12\, EI/L^3 & & \textit{Symmetrical} & \\ 6\, EI/L^2 & 4\, EI/L & & \\ -12\, EI/L^3 & -6\, EI/L^2 & 12\, EI/L^3 & \\ 6\, EI/L^2 & 2\, EI/L & -6\, EI/L^2 & 4\, EI/L \end{bmatrix} \qquad \text{Eq.(5)}$$

When a compression axial force, P, acts, the beam would be more flexible and its stiffness matrix is reduced to the following formula **[7].**

$$[K_R] = [K] - [K_G] \qquad \text{Eq. (6)}$$

where

K_R is the reduced stiffness,

K_G is the geometric stiffness determined from the following relation **[7].**

$$[K_G] = \frac{P}{30L} \times \begin{bmatrix} 36 & & & \\ 3L & 4L^2 & \textit{Symmetrical} & \\ -36 & -3L & 36 & \\ 3L & -L^2 & -3L & 4L^2 \end{bmatrix} \qquad \text{Eq.(7)}$$

C. Load Vectors

The way of generating load vector from applied loads represents another point where finite element modeling significantly differs from that of the matrix method.

To obtain an exact solution, the applied load in the stiffness matrix are broken down into nodal forces and fixed end moment as indicated in **Fig. 2**. For each load type, the fixed end moments are determined based on classical methods, usually using column analogy method, while the unbalance forces is included in the load vector for the member element **[13].**

On the other hand, load vector in the finite element formulation is generated based on the principle of potential energy, where the applied loads are lumped into nodal forces that produce equivalent external work **[6].**

As same displacement field is adopted in the formulation of stiffness matrix and in lumping of the applied loads, the resulting load vector is called a consistent load vector **[6].**

Based on the aforementioned discussion, finite element simulation is considered as an approximate solution compared with stiffness matrix analysis in the aspects of generating the stiffness matrix and the load vector.

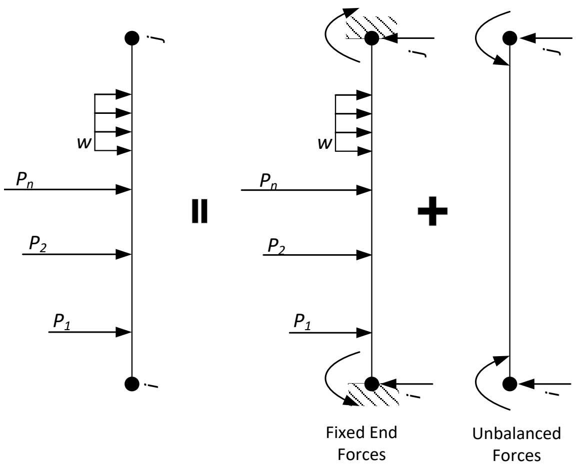

Fig.2 Load vector generating in matrix method.

III. MATLAB Code

Matlab code has been developed for P-δ analyses based on stiffness matrix approach, Eq. (3) , and based on finite element method, Eq. (6) .

To be positive definite, stiffness matrix has been modified for boundary conditions. Penalty approach has been adopted to impose the boundary conditions, where supports have been simulated as stiff springs and have been added to the corresponding DOF in the main diagonal of the stiffness matrix. Stiffness for stiff springs has been assumed as 1000 times the maximum stiffness on the main diagonal **[6].**

Finally, the resulting simultaneous equations have been solved by Gauss elimination approach **[10].**

IV. P-δ Analysis in Commercial Software

To show how column discretization process affect analysis and design aspects, three commercial software, namely the aforementioned Matlab code, STAAD.Pro V8i (SS 6), and Abaqus 6.12-1, have been considered.

STAAD.Pro V8i (SS6) software has been considered as an example on software usually used in routine design works while Abaqus software has been considered as an example on more sophisticated software that is used in research field.

As discussed earlier, adopting finite element method with different mesh size lead to approximate formulation of stiffness matrix and load vector. In order to show partial effects of approximation in stiffness matrix and in load vector formulation, two case studies indicated in **Fig. 3** and **Fig. 4** are examined.

In the case study with member forces, Hermite displacement field can be used to approximate the stiffness matrix and load vector. On the other hand, Hermite displacement field can only be used to approximate the stiffness matrix for the case study with nodal forces shown in **Fig. 3**. Therefore, it may be concluded that

Hermite cubical displacement field is more accurate to simulate nodal forces of **Fig. 3** than the member forces of **Fig. 4**.

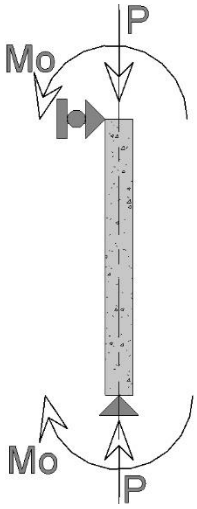

Fig.3 Case studies with nodal forces.

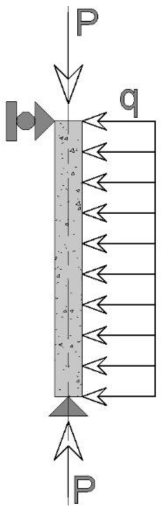

Fig.4 Case studies with member forces.

A. Matlab Case Studies

Adopting the Matlab code of Article III and referring to the column with nodal forces indicated in **Fig. 3**, P-δ response for different levels of axial forces have been determined, normalized in terms of the first order analysis, and presented in **Fig. 5**. The figure indicates that the closer the axial force to the buckling load, the more divergence between matrix method and finite element method. This trend can be interpreted in terms of the difference between the approximated Hermite displacement field of the finite element method and the more accurate trigonometric displacement field of the stiffness method.

From practical point of view, a value of 1.4 represents an upper bound for the permissible P-δ effect adopted by the **[4].** According to **Fig. 5** this upper bound level occurs with an axial load level in the range of one-quarter of the critical load and with an error of fifty percent in finite element analysis compared to that of matrix method. This emphasizes how meshing is important in second order effect even for columns with nodal forces only.

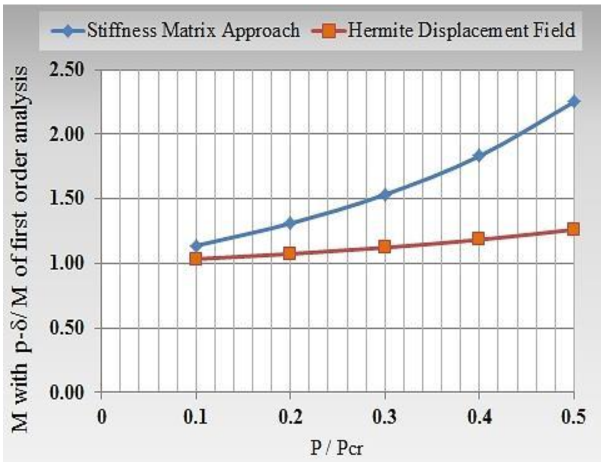

Fig.5 Normalized responses for matrix method and finite element method of a column simulated with single element and subjected to nodal forces.

B. Effect of Column Discretization in STAAD Environment

STAAD Pro adopts stiffness approach to simulate skeleton structures while uses finite element method for analysis of continuum structures including plates, shells, and solids **[5].**

As it starts with a stiffness matrix similar to that indicated in Eq. (3) , STAAD can capture $P-\delta$ effects even with a single element for the whole column height.

In STAAD environment, two-dimensional model with frame element has been adopted to simulate the column. Axial and lateral forces have been applied simultaneously to model the $P-\delta$ effects. Based on the applied primary load, the software determines the geometric stiffness, K_G. The primary load is then modified by the K_G times the displacement of previous iteration, $i-1$, and solve for the displacement of the current iteration i as indicated .

$$[K]\{v\}_i = \{P\} + [K_G]\{v\}_{i-1}$$

$$\{v\}_i = [k]^{-1}(\{P\} + [K_G]\{v\}_{i-1}) \qquad \text{Eq.(8)}$$

where

$\{v\}_i$ and $\{v\}_{i-1}$ are the displacement vectors at current and previous iterations respectively.

$\{P\}$ is the load vector due to primary vertical and lateral loads when applied simultaneously.

This procedure permits the formulating and factorization of the global stiffness matrix, $[K]$, only once at the beginning of the analysis process **[5].**

According to **[5].**, 5 to 25 iterations are usually adopted to ensure an accurate analysis within a reasonable run time. Ten iterations have been adopted in this study.

The $P-\delta$ effect for a single column with different mesh sizes, different levels of axial force have been determined and presented **Fig. 6** and **Fig. 7** for nodal and member forces respectively. These figures indicate that the STAAD can capture $P-\delta$ effects even when single element is used to simulate the whole column height. This seems logical since STAAD relies on the matrix method technique in the solution.

As the proposed displacement filed is used to approximately generate the nodal load vector in the case of member forces, the response for nodal forces would be less sensitive for mesh size as the forces are already applied at nodes with no further approximation.

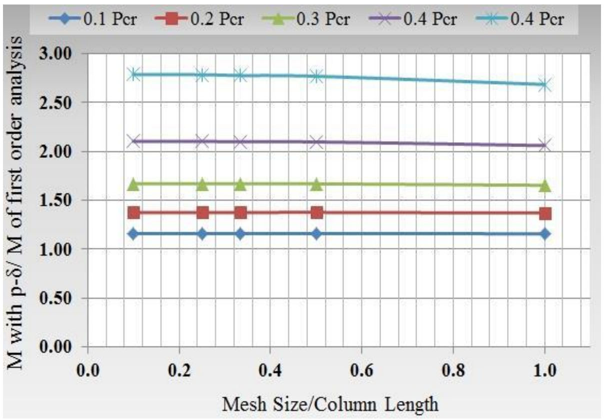

Fig.6 Effect of mesh size for different levels of axial force with nodal moments in STAAD environment.

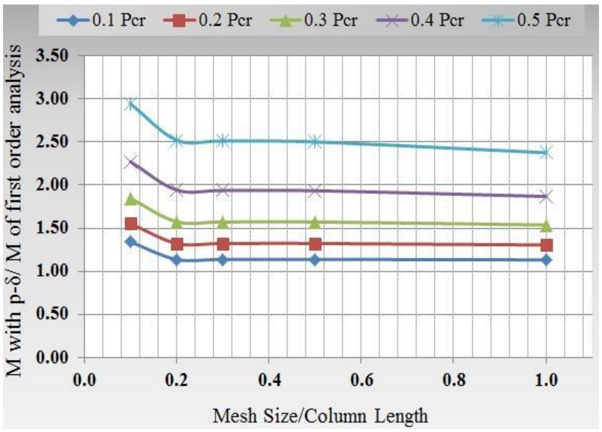

Fig.7 Effect of mesh size for different levels of axial force with uniform member load in STAAD environment.

C. Effect of Column Discretization in Abaqus Environment

Abaqus is one of the most powerful general finite element software. It can be utilized to simulate almost all problems in stress-strain analysis. As a general software, Abaqus adopts finite element method to simulate skeleton and continuum structures **[12].**

In Abaqus environment, a single deformable two-dimensional wire has been adopted to simulate the column with focusing on the bending about major axis **[3].**

Frame element is used to discretize the wire body. This element is a 2-node element with Hermite displacement field that can exactly simulate the behavior of nodal forces without $P-\delta$ effects. Mesh size of 6m, 3m, 1.8m, 1.2m, and 0.6m have been used to show how element size can affect $P-\delta$ response with different levels of axial force **[2].**

Two load steps are produced; the first step is a default one that is generated automatically by the software to accommodate the hinge and roller boundary conditions indicated in **Fig. 3** and **Fig. 4**. The second step is a general step that is used to apply the nodal forces of **Fig. 3** and the member forces of **Fig. 4** with including the $P-\delta$ effects **[1].**

Results for different mesh size and different load levels have been presented in **Fig. 8** and **Fig. 9** for nodal and member forces respectively. These figures indicate that no $P-\delta$ effect can be simulated when single element is used for the whole column height. For member forces, the single element cannot even simulate the primary moment as seen in **Fig. 9**. In general, fine mesh, in the range of 0.1 of column height, should be adopted to simulate the $P-\delta$ effects accurately.

Even with fine mesh, one can notice some differences between STAAD and Abaqus results. These differences are due to different displacement fields that are utilized by STAAD and Abaqus software and due to different analysis techniques that are employed. In contrast to STAAD that maintains the global stiffness matrix, $[K]$, and only updates the load vector according to Eq. (8), full nonlinear analysis with updating stiffness matrix and load vector is adopted by Abaqus software.

V. Conclusions

Based on the different case studies that are examined in this research, one can conclude the following:

1. In general, the user should be completely aware about the built-in displacement field that is employed by the software before executing P-δ analysis.
2. A fine mesh, in the range of 0.1 of column height, is generally essential in software that utilizes Hermit displacement field.
3. With software that is based on stiffness method, an accurate $P-\delta$ analysis can be obtained even with a single element for the whole column height.
4. When starting with Hermite displacement filed, columns that are subjected to member forces are more sensitive to mesh size than those that are subjected to nodal forces. This seems reasonable as member forces in Hermite displacement field are approximately lumped to the corresponding nodes.
5. Finally, results of members with fine mesh revealed that the software that employs full nonlinear analysis provides a more accurate $P-\delta$ response than the software that utilizes matrix method in spite of the fact that finite-element software starts with an approximate displacement field.

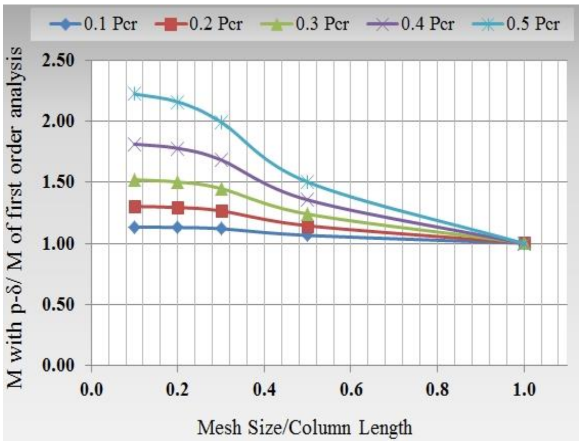

Fig.8 Effect of mesh size for different levels of axial force with nodal moments in Abaqus environment.

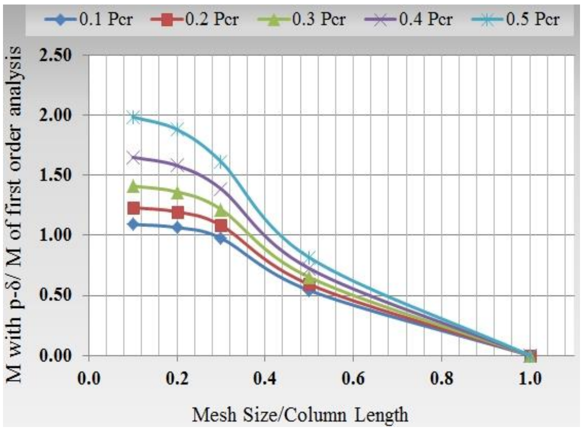

Fig.9 Effect of mesh size for different levels of axial force with uniform member load in Abaqus environment.

VI. Recommendations

1. For future works, it would useful to assess that effect of mesh size of P-δ analysis with material nonlinearity.
2. As this study indicates that the accuracy of P-δ analysis is sensitive to load nature, including the inertia forces in a dynamic P-δ analysis is worth investigating in order to enhance the outcome of this study. This aspect has a substantial practical importance for frames that are subjected to seismic forces.

VII. Acknowledgements

First and foremost, praise be to **Allah**. Without **Allah**'s guidance and help we could not have presented this work.

The authors would like to thank the staff of Civil Engineering Department at the College of Engineering/University of Baghdad for their encouragement and support during the preparation of this study.

VIII. References

[1] Abaqus 2009, *Abaqus Analysis User's Manual Volume II: Analysis*, Dassault Systèmes.

[2] Abaqus 2009, *Abaqus Analysis User's Manual: Volume IV: Elements*, Dassault Systèmes.

[3] Abaqus 2009, *Analysis User's Manual: Volume I: Introduction, Spatial Modeling, Execution and Output*, Dassault Systèmes.

[4] American Concrete Institute 2014, *ACI318M; Building Code Requirements for Structural Concrete (ACI 318M-14) and Commentary (ACI 318RM-14)*, Farmington Hills, MI, USA.

[5] Bentley 2012, *STAAD.Pro V8i (SELECTseries 4): Technical Reference Manual*, Bentley Systems, Inc.

[6] Chandrupatla, TR and Belegundu, AD 1996, *Introduction to Finite Elements in Engineering*, Prentice Hall of India.

[7] Cook, RD 1995, *Finite Element Modeling for Stress Analysis*, John Wiley and Sons, Inc.

[8] Gerald, CF and Wheatley, PO 1999, *Applied Numerical Analysis, 6th Edition*, Addison-Wesley.

[9] Ghali, A, Neville, AM and Brown, TG 2009, *Structural Analysis, A Unified Classical and Matrix Approach, 6th Edition*, Spon Press.

[10] Moore H., 2012, *MATLAB for Engineers, 3rd Edition*, Pearson.

[11] Kassimali, A 1999, *Matrix Analysis of Structures*, Cengage Learning.

[12] Khennane, A 2013, *Introduction to Finite Element Analysis Using MATLAB and Abaqus*, CRC Press.

[13] Norris, CH, Wilbur, JB and Utku, S 1976, *Elementary Structural Analysis, Third Edition*, McGraw-Hill Book Company.

[14] Stasa, FL 1985, *Applied Finite Element Analysis for Engineers*, Holt, Rinehart, and Winston.

[15] Timoshenko, SP and Gere, JM 1961,

Theory of Elastic Stability. 2nd Edition, Dover Publications, Inc.

[16] Wang, PC 1966, *Numerical and Matrix Methods in Structural Mechanics*, John Wiley and Sons, Inc.

تأثير التجزئة للأعمدة على تحليل القوى-الازاحة (P-δ)

د. صلاح رحيمة الزيدي
مدرس في قسم الهندسة المدنية
كلية الهندسة / جامعة بغداد

د. حيدر عامر البغدادي
مدرس في قسم الهندسة المدنية
كلية الهندسة / جامعة بغداد

عقيل طالب فاضل
مدرس مساعد في قسم الهندسة المدنية
كلية الهندسة / جامعة بغداد

الخلاصة:

أدى استخدام المواد ذات المقاومة العالية المتاحة مؤخرا واحدث اصدارات مدونات التصميم بالإضافة الى اشكال التصاميم المعقدة الحالية إلى الاستخدام المتكرر للأعمدة النحيفة التي تتطلب تضمين حسابات القوى-الازاحة (P-δ) للأعمدة في التحليل والتصميم. عادة ما يتم تنفيذ هذا النوع من التحليل باستخدام البرامج الهندسية التجارية. لذلك يحتاج المصممون إلى الالمام بكل العناصر التي تؤثر على نتائج تحليل القوى-الازاحة (P-δ) بما في ذلك طريقة التحليل المستخدمة من قبل البرنامج (طريقة مصفوفة الصلابة أو طريقة العناصر المحددة) ونسبة تجزئة العمود. نسبة تجزئة العمود المثالية لا يتم تغطيتها في الدليل المرفق مع البرامج وهو موضوع تقليدي للجدل بين المهندسين نتيجة عدم وجود دليل علمي متوفر للمهندسين لاختيار هذه النسبة. الاختيار غير المثالي لنسبة تجزئة العمود قد يؤدي إلى نتائج غير دقيقة أو حتى خاطئة في بعض الاحيان. يهتم هذا البحث بالتحقيق في تأثير نسبة تجزئة العمود على على نتائج تحليل القوى-الازاحة (P-δ) من أجل توفير قاعدة علمية للمهندسين للاعتماد عليها. كما يشخص البحث الفرق بين البرامج التي تستخدم طريقة مصفوفة الصلابة وتلك التي تستخدم طريقة العناصر المحددة عند اعتمادها في تحليل القوى-الازاحة (P-δ) للأعمدة. يتم دراسة نوع تحميل العمود كعامل يمكن أن تؤثر على نسبة التجزئة المثالية لكل طريقة من وسائل التحليل. يجري التحقيق في حالتين للتحميل: الأحمال العقدية والأحمال العنصرية للأعمدة. تم كتابة كود ماتلاب (MatLab) لإظهار الفرق بين النهجين (طريقة مصفوفة الصلابة أو طريقة العناصر المحددة) عند استخدامها لعمود مع قوى عقدية دون تأثير تقنيات الحل المختلفة التي تستخدمها البرنامج. تم استخدام برنامجين لدراسة تأثير نسبة تجزئة العمود باستخدام طرق تحليل مختلفة ومستوى مختلف من التحميل المحوري.

تم اخذ برنامج ستاد برو (STAAD.Pro) مثالا على البرامج التي تعتمد طريقة مصفوفة الصلابة للعناصر الاطارية ويتم استخدامه في أعمال التصميم الروتينية. من ناحية أخرى، تم الاستفادة من برنامج أباكوس كمثال للبرامج التي توظف بالكامل طريقة العناصر المحددة وعادة ما يستخدم في الجوانب البحثية. نتائج دراسات الحالة المختلفة تشير إلى أن طريقة التحليل المستخدمة (طريقة مصفوفة الصلابة أو طريقة العناصر المحددة) تؤثر بشكل كبير على اختيار النسبة المثالية لتجزئة العمود. تظهر النتائج أيضا أن الأعمدة التي تتعرض للأحمال العنصرية تبدو أكثر حساسية لنسبة التجزئة مقارنة بالأعمدة المقابلة التي تتعرض لاحمال عقدية.

الكلمات المفتاحية: - تحليل القوى-الازحة (P- δ)، طريقة مصفوفة الصلابة، طريقة العناصر المحددة، حجم التجزئة، كود ماتلاب (MatLab)، برنامج ستاد برو (STAAD.Pro)، برنامج اباكوس.