Scientific Review Engineering and Environmental Sciences (2022), 31 (3), 203–222

Sci. Rev. Eng. Env. Sci. (2022), 31 (3) https://srees.sggw.edu.pl

ISSN 1732-9353 (suspended) eISSN 2543-7496 DOI 10.22630/srees.3897

Received: 14.08.2022
Accepted: 30.09.2022

Mahmoud Mohamed BADAWY[1] https://orcid.org/0000-0002-1099-100X
Suzan Ali A. MUSTAFA[2]✉ https://orcid.org/0000-0001-9647-2456
Atef Eraky BAKRY[3] https://orcid.org/0000-0002-7514-3677
Zagazig University, Faculty of Engineering, Egypt

BEHAVIOR AND FAILURE TRACKING OF STRUCTURAL ELEMENTS USING APPLIED ELEMENT METHOD

Key words: applied element method (AEM), numerical analysis, stiffness matrix, large deformation failure tracking

Introduction

Structural analysis numerical techniques have two categories, continuum method and discrete element method. The first upholds the well-known finite element method (FEM) (Tagel-Din & Meguro, 1999; Meguro & Tagel-Din, 2000, 2001). Joints in the FEM can identify major cracks, however the position and direction of fracture propagation must be specified prior to analysis application (Meguro & Tagel-Din, 1997, 2001, 2002). Observing structural failure behavior in this method is difficult. If the behavior is considerably non-linear in the FEM, plenty of issues arise. For example, it is exceedingly difficult or impossible to use the FEM to evaluate the behavior of materials that move from a continuum to a totally discrete condition, like the behavior of structures before and during failure (Tagel-Din & Meguro, 1999).

The second category is discrete element methods, which upholds the distinct element method (DEM) and the rigid body and spring model (RBSM). The DEM assumes that the objective materials are composed of discrete pieces and may represent discrete material behavior. The extended DEM (EDEM) was primarily used for constraining structural analysis, and is often used to model and recontact structural components with extremely large deformations (Meguro & Tagel-Din, 2001). The RBSM analysis could not be done until the system is collapsed (Meguro

✉samustafa@eng.zu.edu.eg

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

& Tagel-Din, 2000). The main drawback of these rigid element methods is that the simulation results are heavily influenced by the element shape, dimension, and arrangement (Meguro & Tagel-Din, 1997, 2001). Furthermore, in the small deformation condition, both methods are less accurate than the FEM (Meguro & Tagel-Din, 1997, 2001, 2002; Tagel-Din & Meguro, 2000a, 2000b). Based on the foregoing, it is possible to infer that the present methodologies are unsuitable for tracking whole structural behavior from zero to collapse within an appropriate time frame (Meguro & Tagel-Din, 2000).

Applied element method (AEM)

The applied element method (AEM) is a recently developed displacement method (Meguro & Tagel-Din, 1997, 2000; Tagel-Din & Meguro, 1999). The ability of AEM to follow the behavior of the structural failure at various phases is impressive. This comprises the application of load, elastic stage, fracture initiation and propagation, the yielding of reinforcement, non-linear behavior, large deformation condition, the separation and collision of elements, and energy dissipation during collision (Lupoae & Bucur, 2009; Tokal-Ahmed, 2009; Wibowo, Reshotkina & Lau, 2009; Gohel, Patel & Joshi, 2013; Eraky, Mustafa, & Badawy, 2021). This paper sheds some light on AEM as a vital technique of structural analysis. A MATLAB program was created to apply the AEM method on structural elements. Moreover, some other points will be highlighted, the verification of the proposed program, the parameters affecting the analysis results and tracking the elements failure. In AEM, the structure is split into small rigid elements interlinked by springs (Meguro & Tagel-Din, 1997; Gohel et al., 2013; Shakeri & Bargi, 2015). Superior to FEM is AEM, because it requires less degrees of freedom (DOFs). This reduces the amount of time and memory required for processing (Meguro & Tagel-Din, 2001). By correctly arranging the springs, any form may be modeled without increasing the computing work (Christy, Pillai & Nagarajan, 2020). To transmit normal and shear stress, AEM elements are linked by a series of normal and shear springs. Springs, by their characteristics, stresses, and strains, characterize a specific volume of the elements (Meguro & Tagel-Din, 2000; Gohel et al., 2013; Shakeri & Bargi, 2015; Christy, Pillai, & Nagarajan, 2018). A spring is disconnected when the stress exceeds the permitted limit. Following the location of such springs might reveal the fracture pattern. Therefore, the structure behavior and crack propagation may be assessed at all stages of loading. The AEM also enables easy modeling, rapid processing, and high accuracy of results (Meguro & Tagel-Din, 2000; Christy et al., 2018).

To analyze the structures in AEM, they are split into minor rigid elements. Normal and shear springs are used to link the two parts as in Figure 1 at a single point of contact. Three DOFs are used to analyze the elements in 2D while six DOFs are used in 3D. These degrees of freedom reflect the element's translations and rotations. For every pair of springs surrounding the element, the stiffness matrix is derived using a unit displacement at the center of the element, as well as the forces at the other DOFs are determined if they are restricted (Meguro & Tagel-Din, 1997; Shakeri & Bargi, 2015; Christy et al., 2018). The stiffness matrix size for each spring is 6×6. The stiffness matrices of every spring surrounding each element are assembled to produce the total stiffness matrix. Equation (1) shows the upper-left fourth of the stiffness matrix. The element stiffness matrix in this formulation is given by the contact point location (distance L and angles ϴ and α), and the stiffness of normal and shear springs (Meguro & Tagel-Din, 1997). The AEM is quicker than FEM since it does not need the definition of shape functions or the integration methods to construct the stiffness matrix (Shakeri & Bargi, 2015). The FEM is mostly used to simulate structural components like frames and shells. Because neighboring elements are connected by their shared nodes, partial connections are not permitted, and hence element failure cannot be simulated (Tokal-Ahmed, 2009). Although AEM modeling is similar to FEM, the primary difference is the element combination. The separation of elements in AEM is simpler to simulate than in FEM due to the use of springs between the elements. When the size of the element's changes, a transition zone is necessary in FEM for transferring large elements from smaller elements, but not in AEM. This leads to many fewer elements as shown in Figure 2. Moreover, AEM employs springs to link two components on a section of the surface (rather than the entire surface), whereas FEM cannot model connection without utilizing a different meshing technique (Tagel-Din & Rahman, 2006; Tokal-Ahmed, 2009; Shakeri & Bargi, 2015), as shown in Figure 3. The AEM assumes that every normal and shear spring stiffness represents a distinct area of the connected elements. The location of springs across the element's edges shown in Figure 4. Spring stiffness is calculated using Equation (2) (Meguro & Tagel-Din, 1997; Lupoae & Bucur, 2009).

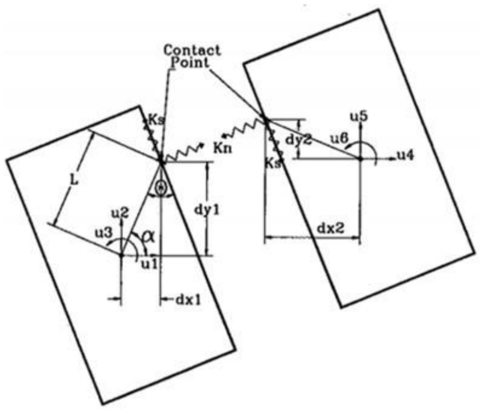


FIGURE 1. Element shape, contact point and degrees of freedom for two elements (Meguro & Tagel-Din, 1997)

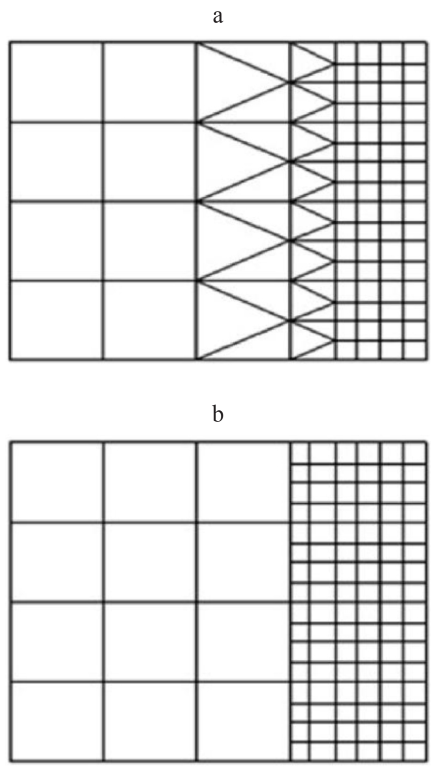


FIGURE 2. Transition from large elements to small elements in (a) FEM and (b) AEM (Shakeri & Bargi, 2015)

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

a b

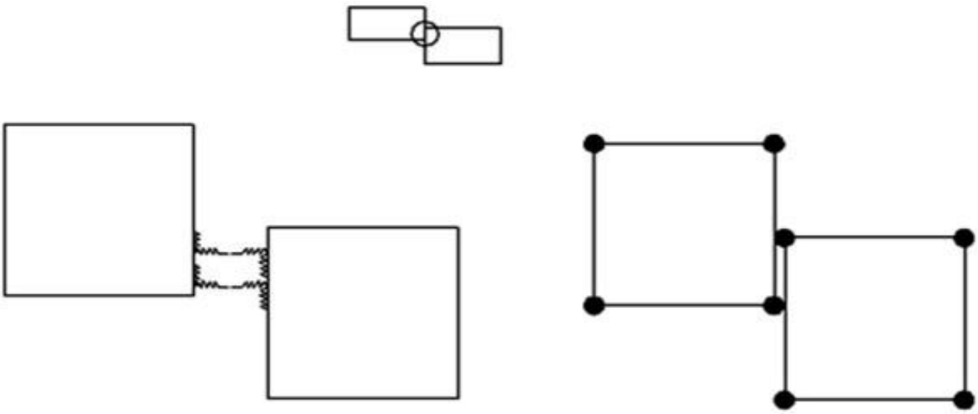


FIGURE 3. Element connectivity: a – AEM (connectivity via springs); b – FEM (no connectivity) (Shakeri & Bargi, 2015)

a b

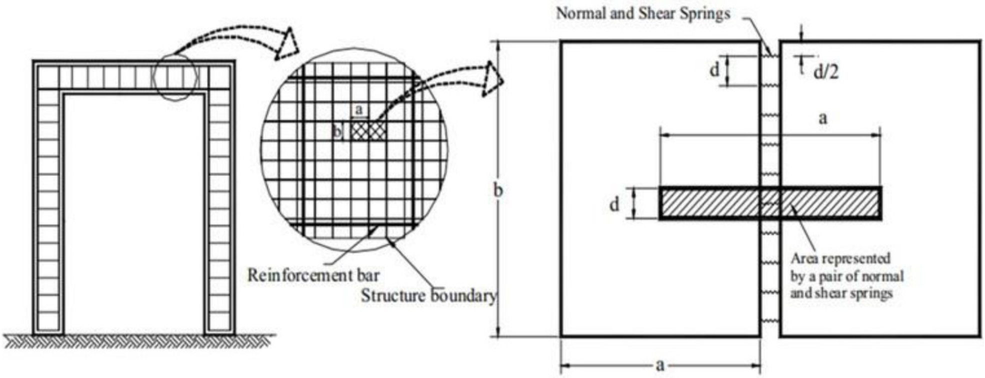


FIGURE 4. Modelling of structure in AEM: a – element generation; b – spring distributions and area of influence of each spring (Wibowo et al., 2009)

$$\begin{vmatrix} \begin{matrix} k_n \sin^2(\theta+\alpha) \\ +k_s \cos^2(\theta+\alpha) \end{matrix} & \begin{matrix} -k_n \sin(\theta+\alpha)\cos(\theta+\alpha) \\ +k_s \sin(\theta+\alpha)\cos(\theta+\alpha) \end{matrix} & \begin{matrix} k_s L \sin(\alpha)\cos(\theta+\alpha) \\ -k_n L \cos(\alpha)\sin(\theta+\alpha) \end{matrix} \\ \begin{matrix} -k_n \sin(\theta+\alpha)\cos(\theta+\alpha) \\ +k_s \sin(\theta+\alpha)\cos(\theta+\alpha) \end{matrix} & \begin{matrix} k_s \sin^2(\theta+\alpha) \\ +k_n \cos^2(\theta+\alpha) \end{matrix} & \begin{matrix} k_n L \cos(\alpha)\cos(\theta+\alpha) \\ +k_s L \sin(\alpha)\sin(\theta+\alpha) \end{matrix} \\ \begin{matrix} k_s L \sin(\alpha)\cos(\theta+\alpha) \\ -k_n L \cos(\alpha)\sin(\theta+\alpha) \end{matrix} & \begin{matrix} k_n L \cos(\alpha)\cos(\theta+\alpha) \\ +k_s L \sin(\alpha)\sin(\theta+\alpha) \end{matrix} & \begin{matrix} k_n L^2 \cos^2(\alpha) \\ +k_s L^2 \sin^2(\alpha) \end{matrix} \end{vmatrix} . \quad (1)$$

$$k_n = \frac{E \cdot d \cdot t}{a}, k_s = \frac{G \cdot d \cdot t}{a}, \quad (2)$$

where:
k_n – normal spring stiffness,
E – Young modulus of the concrete,
d – distance between the linked springs,
T – element's thickness,
a – representative area,
k_s – shear spring stiffness,
G – shear modulus of the concrete.

Proposed MATLAB open source program for AEM

A 2D MATLAB open source program was created to analyze various structures with varying boundary conditions using the AEM method and to permit researchers for enhancing the method. Figure 5 depicts the flow chart for the proposed program. The program's operation consisted of three steps: preparation, processing, and post-processing (Karad & Patel, 2020).

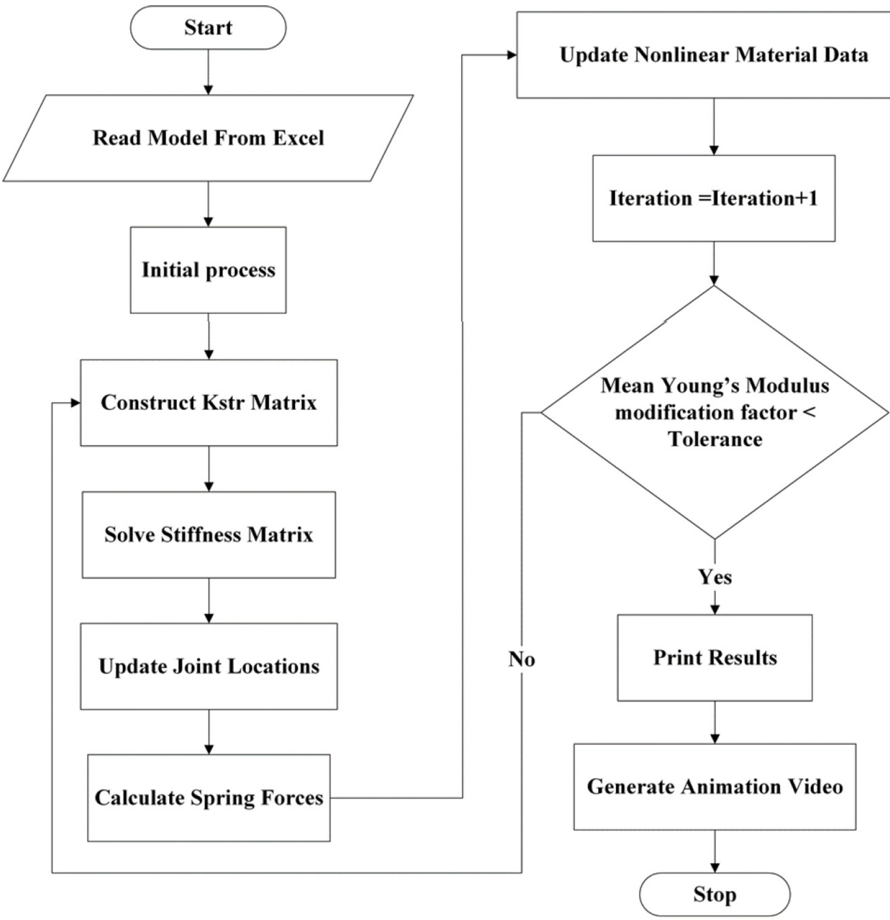


FIGURE 5. Flowchart of proposed MATLAB program for AEM

The MATLAB was chosen for its capacity to handle large matrix multiplication, its robust programming language, and its converting functions and scripts into machine code that run efficiently.

Verification of the proposed program

The proposed program described in the preceding section was used to simulate various structural elements in verifying its precision and applicability. Various boundary conditions and loading were used on these structural elements. The verification was mostly based on stress and deformation values. Two scenarios were used to verify the program: linear static and non-linear geometric analysis in the case of large deformation. Using the MATLAB program in linear static analysis, three problems were used to verify the AEM method.

Cantilever beam

The first verified model was a cantilever beam, presented by (Meguro & Tagel-Din, 2000). This beam was also verified by Moss (2020). It was a six-meter long cantilever beam loaded by a concentrated force at its tip, as shown in Figure 6. The beam cross-section was 0.25 × 1.0 m. The modulus of elasticity was taken 840 MPa. The considered value of Poisson ratio was 0.15. Both studies concerned how mesh discretization affects solution precision. The same scenario was utilized using the proposed program. Many square mesh sizes were applied revealing meshes ranged from 1.0 × 1.0 to 0.083 × 0.083 m. In all cases, 10 linked springs were installed between the edges of the elements. Figure 6 shows the beam model with the lowest mesh size (0.083 m).

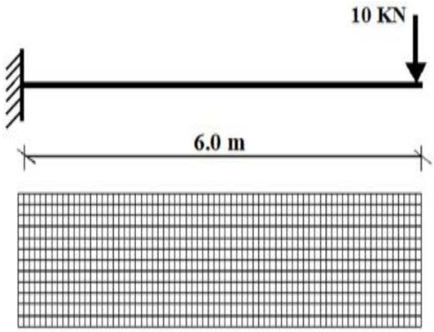


FIGURE 6. Cantilever beam with a point load on the free end and meshing

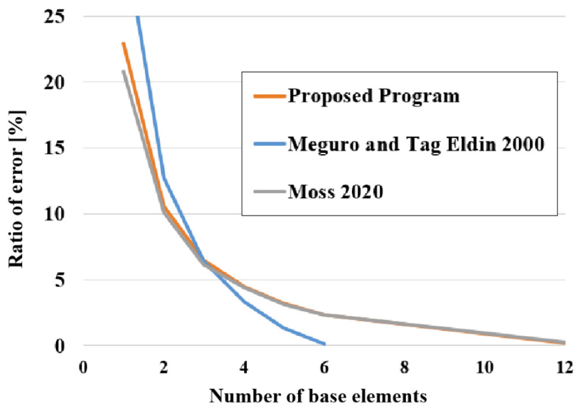


FIGURE 7. Accuracy of deflection of a cantilever beam with different mesh sizes

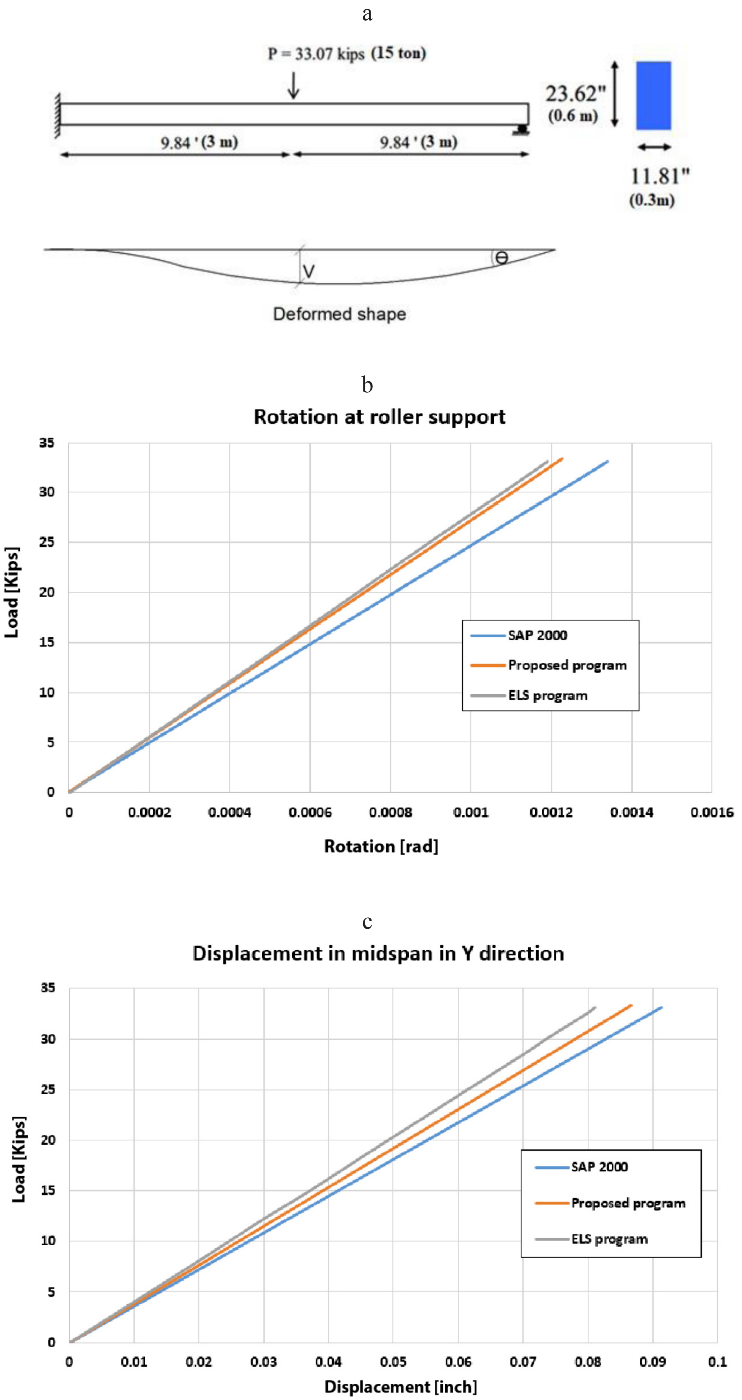


FIGURE 8. Results of proposed program, ELS and SAP2000 model: a – loading and geometry of the beam; b – load–deflection relation; c – load–rotation relation at the hinged support

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

The obtained tip deflection of the beam was compared with the deflection computed using Euler–Bernoulli beam theory. The deviation of the proposed program's outcomes from theory was calculated as (error percentage), as were the deviations of both of these two previous programs (Meguro & Tagel-Din, 2000; Moss, 2020). Figure 7 displays the solution's convergence by charting the expected percent difference for every mesh size.

The deflection caused by the course meshes clearly resulted in substantial inaccuracy. Using a finer mesh, helped to eliminate the error. It is also clear that the proposed program's outcomes closely match those of Moss (2020).

Fixed-roller beam

The second problem used for verifying the linear static analysis was modeled using the Extreme Load Structures (ELS) software help (Extreme Load Analysis (ELS) Program, 2021). A six-meter long fixed roller beam was subjected to a static linear analysis. As illustrated in Figure 8, a concentrated force was applied at mid-span. The dimensions and material properties were maintained the same for the sake of comparison. The beam cross-section was 11.81 × 23.62 inch (0.3 × 0.6 m). The modulus of elasticity considered was 3,499.35 ksi. The value of Poisson ratio was 0.25.

The mid-span deflection and the rotation at the roller support were computed with the proposed AEM program, and elementary beam theory. Moreover, a finite element (FE) model analysis was performed using SAP2000 software. The proposed program's findings were also compared to the available ELS results. Figure 8 depicts the load–deflection and load–rotation relationships at the mid-span and roller support. When compared to other programs and the beam theory, the proposed program revealed in satisfactory results.

Fixed-fixed beam

The proposed program was verified using a third problem that is a fixed-fixed supported beam. The model was verified using the problem presented by Christy et al. (2018). As illustrated in Figure 9, a 5 kN concentrated force was applied at mid-span. The dimensions and material properties were maintained the same for the sake of comparison. The beam cross-section was 0.2 × 0.4 m. The modulus of elasticity considered was 25,000 kN·m^{-2}. The value of Poisson ratio was 0.2. The beam was split into 120 square elements, with five springs to investigate the deflection at 400 mm distant from the point load. The MATLAB code, FEM using SAP2000, and the beam theory were used to determine the deflection.

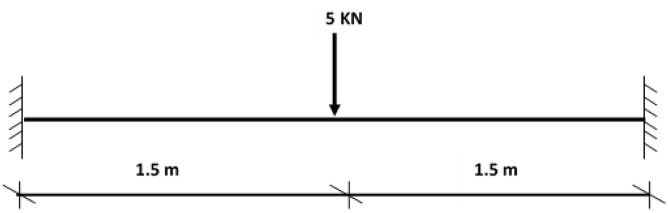


FIGURE 9. Fixed-fixed beam with center point load

The theoretical value of the deflection was 0.02624 and the calculated values from the proposed program and SAP2000 were 0.02506 and 0.0264 m respectively. The bending moment at mid-span was also calculated. The theoretical value of the moment was 1.875 kN·m^{-1}, while the calculated moment from SAP2000 was 1.877 kN·m^{-1} and from the proposed MATLAB program was 1.81 kN·m^{-1}. In comparison to the elementary beam theory and the SAP2000 software, the proposed program produced reasonable results.

Verification of large deformation static analysis

The proposed program was utilized to model a large deformation static analysis program (Meguro & Tagel-Din, 2002). A 12-meter simply supported beam was subjected to a steadily increasing point load at mid-span of the beam. The beam cross-section was 1 × 1 m. The modulus of elasticity considered was 210 MPa, and the Poisson ratio was 0.2. The verified beam's mesh discretization was 0.2 × 0.2 m. Figure 10 shows the beam's undeformed deformed shapes. The large deformation of the beam can be seen in the arched shape and the slipping of the roller towards to the loading point. The arching curvature of the beam indicates its increasing stiffness under load. The proposed program's load–displacement response was verified using the FE software ABAQUS.

In Figure 11, the maximum vertical deformation and the horizontal displacement at the roller support were plotted against the steadily rising load.

Both models' vertical displacement curves showed good agreement. The highest difference was 7.57%. While the horizontal displacement values were similar until roughly one-third of the maximum applied force. As the load level increased, the disparity between the two models' values increased, as indicated in Figure 11. This might be due to the ABAQUS program's distortion control, as well as the fact that local element deformation happened in the FE model but not in the proposed AEM model. This might results in discrepancies in the behavior of the two models, especially at high degrees of large deformation.

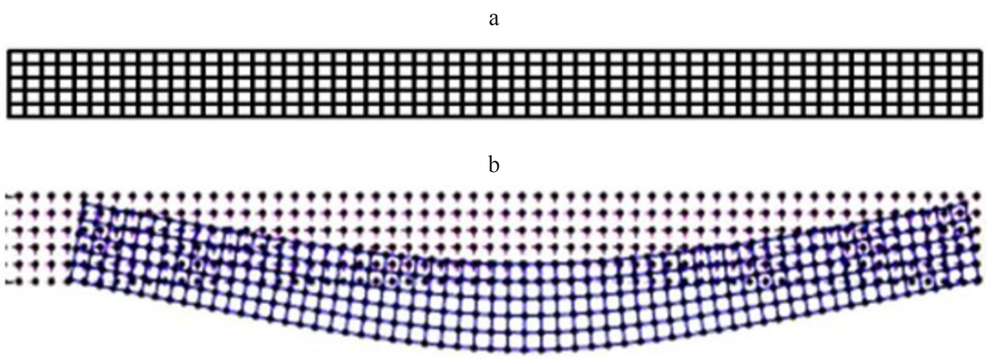


FIGURE 10. Large deformation static analysis model: a – 120 elements of 0.1 × 0.1 mesh; b – deformed shape

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

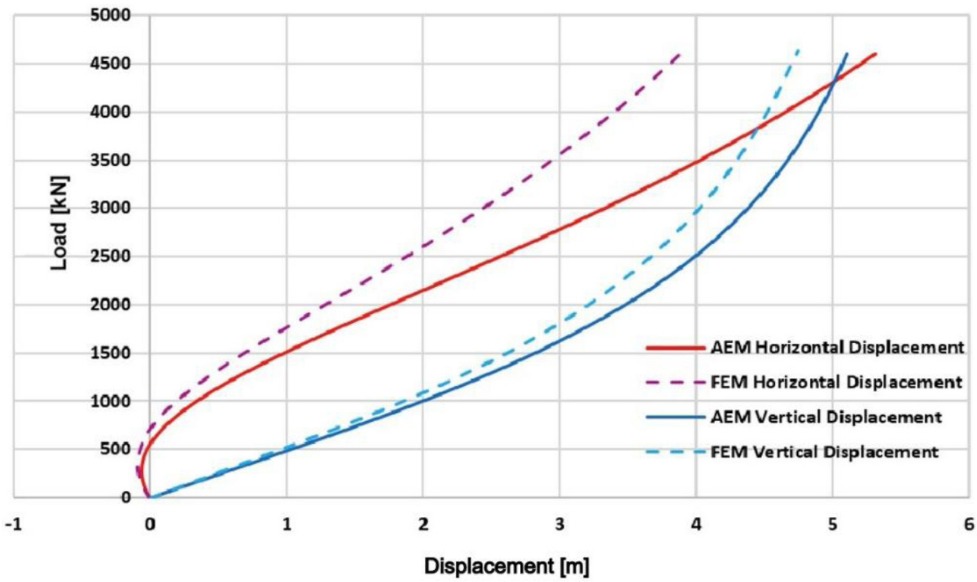


FIGURE 11. Load–displacement curves for AEM and FE models in large deformation static analysis problem

Parametric analysis

Some parameters were investigated to improve the proposed program's effectiveness in modelling various sorts of structures under various loading schemes. The size of elements and the number of connecting springs between the elements were the most effective ones. These two parameters were studied using a simply supported beam and a portal frame.

Simply supported beam

A three-meter long simple beam with cross section 0.2 × 0.45 m was subjected to two types of loading to study the effect of the size of elements and the number of sprigs connecting each pair on the modeling results. As illustrated in Figure 12, the first type of loading was a mid-span concentrated load of 60 t, while the second was two loads of 30 t at 0.9 m from the supports. The modulus of elasticity considered was 2.2 × 106 $t \cdot m^{-2}$, and Poisson ratio was 0.2. The beams in the two cases were divided into 20, 60 and 240 square elements, to study the effect of increasing the number of elements (finer mesh size) on the proposed program outcomes. Meanwhile, each two elements in each trial were connected together by using three, five, seven and nine springs, to check the impact of using different number of springs on the proposed program results. This resulted in using 12 models to study these two parameters. The result concerning the simple beam with one concentrated load at mid-span were displayed in Figure 12. These results were compared to the calculated theoretical maximum displacement at mid-span. It can be noticed that, increasing the number of elements which were used in meshing the beam increased the accuracy of the program. The difference between the proposed model and the theoretical calculation decreased to 0.65% which is considered a very good result. It is worthy note that the number of the connecting springs has a slight effect on the results. However, in case of the finest mesh, there was no effect of the different number of theses springs on the results, as can be seen in Figure 12.

The other loading scheme: four-point--bending was analyzed under the effect of the same two parameters as well. The second beam was divided into the same numbers of elements; 20, 60 and 240. More even, the different number of springs was utilized between the elements as well. The conclusion was noticed under this loading scheme. Figure 13 shows the maximum deflection of the beam for the different element numbers and springs. The finer the mesh, the more accurate results were obtained compared to the theoretical value of the maximum vertical deflection. The accuracy of the proposed program reached 97.4%.

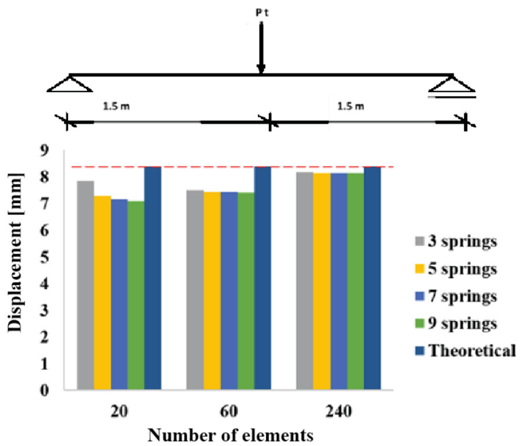


FIGURE 12. Effect of element size and number of connecting springs on the results of the simple beam under three-point bending

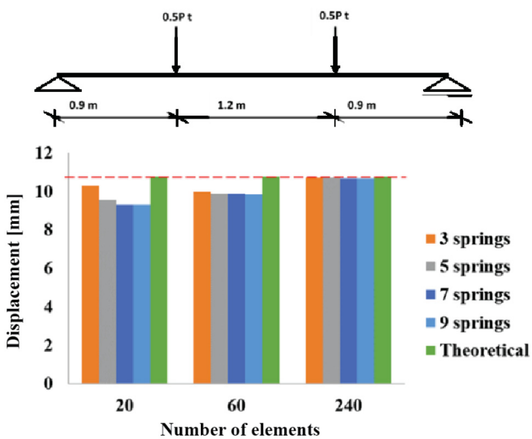


FIGURE 13. Effect of element size and number of connecting springs on the results of the simple beam under four-point bending

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

Figure 13 illustrates the effect of the number of springs on the model accuracy. The difference between the results using different number of springs almost vanished in case of using a fine mesh.

Portal frame

A portal frame with the shown dimension in Figure 14 loaded by a horizontal lateral load was analyzed. The dimensions of the beam and columns cross-section were 0.3 × 0.6 m. The considered modulus of elasticity was 25 GPa. The Poisson ratio was 0.15. The frame was divided into different number of elements to represent the fineness of the mesh. The used element size were 30, 150, 100 and 50 mm revealing to number of elements as 64, 256, 576 and 2,304 elements as shown in Figure 15. Each two adjacent elements were connected by using three, five, seven and nine springs, to examine its effect on the frame results.

Figure 16 shows the results of these parameters; represented as maximum lateral displacements, compared to the calculated theoretical value. The figure shows that the difference between the proposed model decreases with the increase of the number of the elements. The difference decreased to 4.1%. It was noticed that using different numbers of springs affected the accuracy in case of coarse meshes. However, using small element size in the fine meshes, as in the last two cases, revealed in decreasing the effect of the number of springs used between each two adjacent elements.

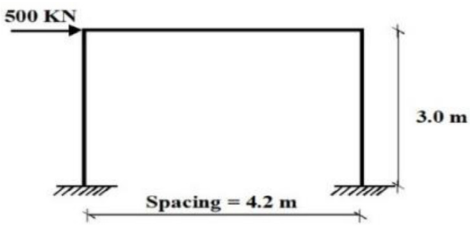


FIGURE 14. Portal frame exposed to point lateral load at top

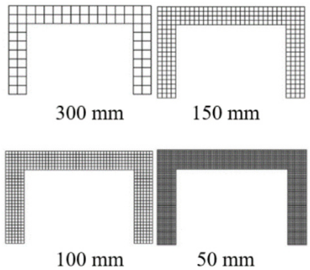


FIGURE 15. Structure discretization with different element size

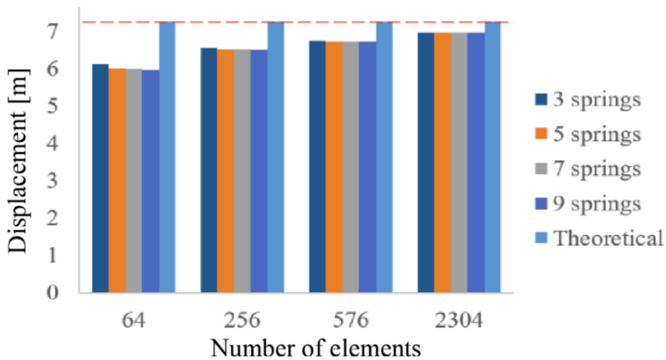


FIGURE 16. Effect of element size and number of connecting springs on the results of the portal frame with lateral load

It was noticed as well that using lower number of springs had a high precision on the processing time of the program. So, it is recommended to use fine mesh as possible with small number of springs to obtain accurate results with reasonable run time.

Failure tracking

An important feature was added to the proposed applied element program, which is the ability to track the failure in a loaded structure. This feature depended mainly on calculating the principal stress in each element during the incremented loading procedure (step by step loading). Failure was considered when the calculated principal stress reaches the specified ultimate stress according to the material type. The program was set so that each represented element whose stress reaches the ultimate tensile stress is colored in blue. While the element whose stress reaches the ultimate compressive stress is colored in red. After the on-set of failure, the failed elements' effect on the global stiffness matrix of the structure is omitted. In addition, the deformed shape at each load step could be extracted giving a motion of the structure. These features were applied on three different structures to show how the failure was tracked.

Cantilever beam

A three-meter long cantilever beam subjected to an incremental load on the free end was studied using the proposed program to track the failed elements and its spread. The cross-section of the beam was 0.2 × 0.6 m. The elasticity modulus used was 21.5 GPa. The Poisson ratio was 0.2. In each step, the increment was raised by 0.5 t. Figure 17a depicts the failure tracking. When the load reached 7 t, the first failed element was detected. Since the stress in this element exceeded the tensile strength. This can be seen by the user since the failed element is colored by blue in this case. By increasing the load, the failed elements increased.

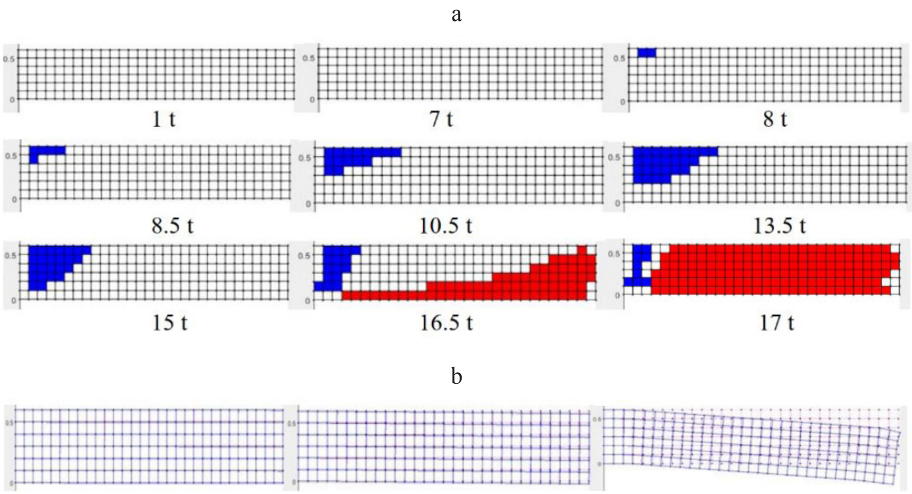


FIGURE 17. Results of cantilever beam: a – tracking of failed elements in a cantilever; b – deformed shape at different load steps of a cantilever beam

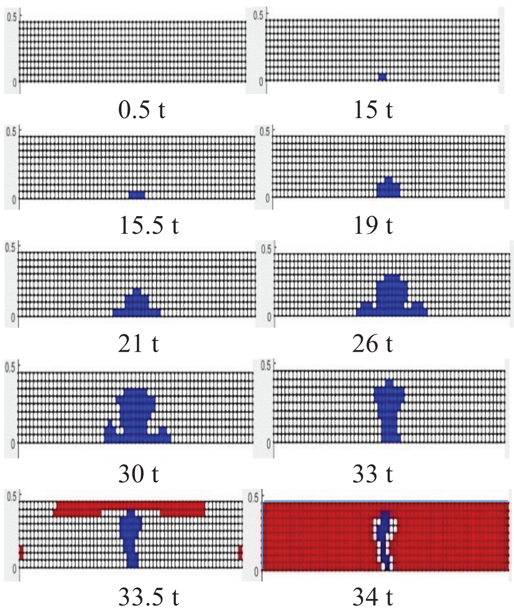
Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

The colored elements increased then it was noticed that after a load of 16 t crushed elements started to appear and were colored by red, as shown in the figure. The deformed shape can be obtained as a result from the program as well. The motion of the deformed shape was obtained, and Figure 17b shows the deformed shape at different load steps.

Simply supported beam

A three-meter long simply supported beam subjected to an incremental load at mid-span of the beam was studied using the proposed program to follow the failure, start and spread. The cross-section of the beam was 0.2 × 0.45 m.

a



b

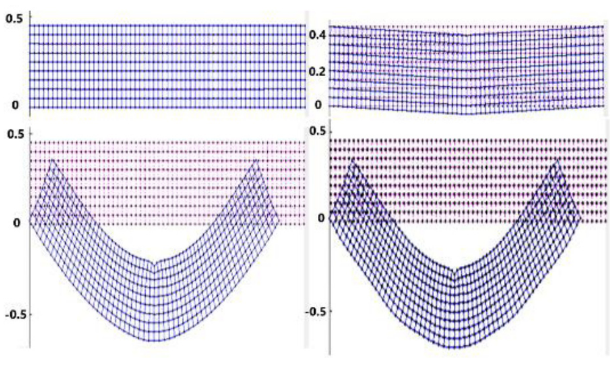


FIGURE 18. Results of simply supported beam: a – tracking of the failed elements of simply supported beam under mid-span concentrated load; b – deformed shape at different load steps of a simply supported beam

The elasticity modulus used was 21.5 GPa. The Poisson ratio was found to be 0.2. In each step, the increment was raised by 0.5 t. Figure 18a depicts the failure tracking. When the load reached 15 t, the first failed element was depicted. The first element failed because it exceeded the tensile strength. The number of the failed elements in tension increased gradually with increasing the load, then the failed elements under compression started to show, as can be seen in the figure. As described above, the deformed shape at different load steps is shown in Figure 18b.

Portal frame

The proposed program was used to follow the failure initiation and propagation of a portal frame subjected to increased lateral load. The frame's dimensions are shown in Figure 19. Both the beam and the columns have cross-sectional dimensions of 300 × 600 mm. Young modulus was set to 25 GPa. The Poisson ratio had a value of 0.2. The element sizes used were 100 mm. Three springs were attached between each pair of adjacent elements. In each step, the value of the load (P) raised by 1 N. Figure 20 depicts the failure tracking. The first failed element was detected when it was surpassed the specified tensile strength.

With increasing the applied load, more elements in different positions of the frame reached the ultimate tensile strength. Therefore, these elements were colored to show the position of failed elements, as instructed in the proposed program. On the other hand, when the elements reached the specified ultimate compressive stress, the crushed elements were colored by red in the regions of compression. It was inferred from these results that the proposed program designed to use the AEM method could accurately anticipate the failure pattern.

Infilled frame

The proposed program was used to follow the failure initiation and propagation of an infilled frame subjected to increased lateral load. This was to examine applicability of the proposed program to model a structure composed of two different materials. The structure was a plain concrete frame infilled with a brick wall, as shown in Figure 21. Young modulus for concrete and bricks were 21.5 and 7 GPa, respectively (Nichols & Totoev, 1997). The Poisson ratio for concrete and bricks were 0.2 and

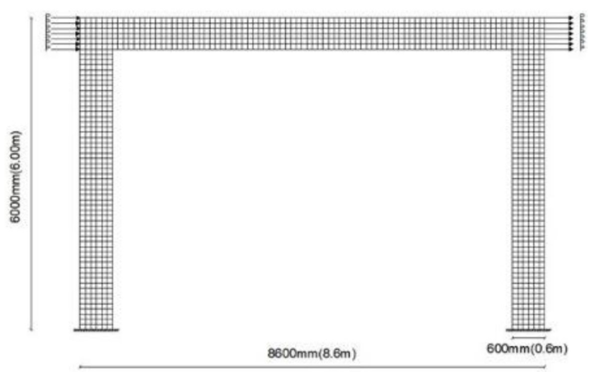


FIGURE 19. Portal frame under increased lateral loads

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

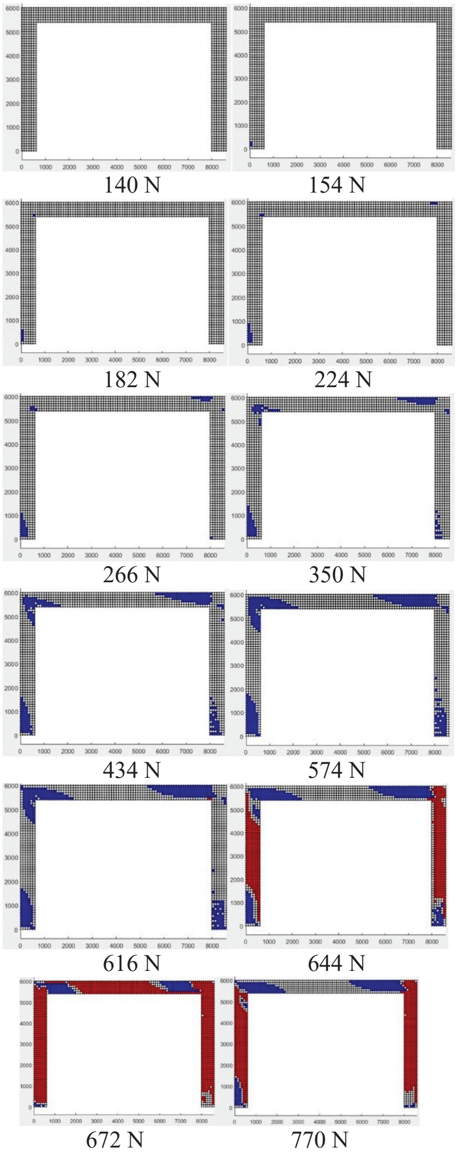


FIGURE 20. Results of portal frame with incremental lateral loads: tracking failed elements

0.21, respectively. The structure consisted of concrete frame supported by concrete ground beam and filled with bricks, with the dimensions shown in the figure. Both the beam and the columns have cross-sectional

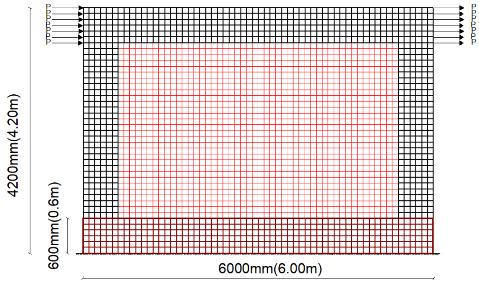


FIGURE 21. Infilled frame under increased lateral loads

dimensions of 250 × 600 mm, while the cross-sectional dimensions of ground beam are 400 × 600 mm. The width of bricks was 120 mm. The element size used were 100 mm, as seen in Figures 21, 22 and 23. Three springs were attached between each pair of adjacent elements. The applied load step size was 0.5 t to ease tracking the failed elements accurately. The first failed element was detected when it surpassed the permissible tensile strength of the bricks at the boundary between concrete and bricks, as shown in Figure 22.

On the other hand, when other elements reached the specified permissible compressive stress, the crushed elements were colored by red. Figure 23 shows the deformed shape of the infilled frame. It was inferred from these results that the proposed program designed to use the AEM method could accurately anticipate the failure pattern in the case of using two different materials.

Conclusions

Through the presented study in this research, the following conclusions were derived:

– Using AEM, the proposed program showed good accuracy in simulating different structural elements with different

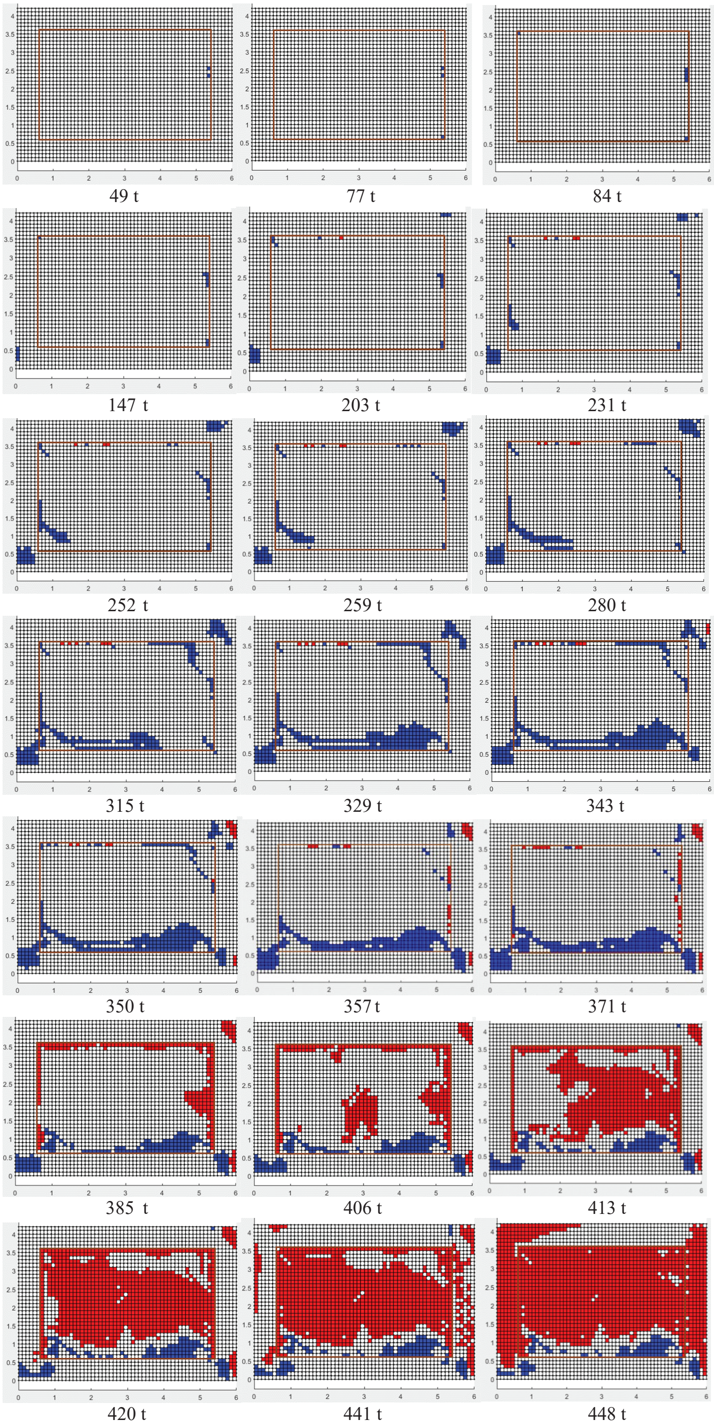


FIGURE 22. Results of infilled frames: tracking failed elements of an infilled frame

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

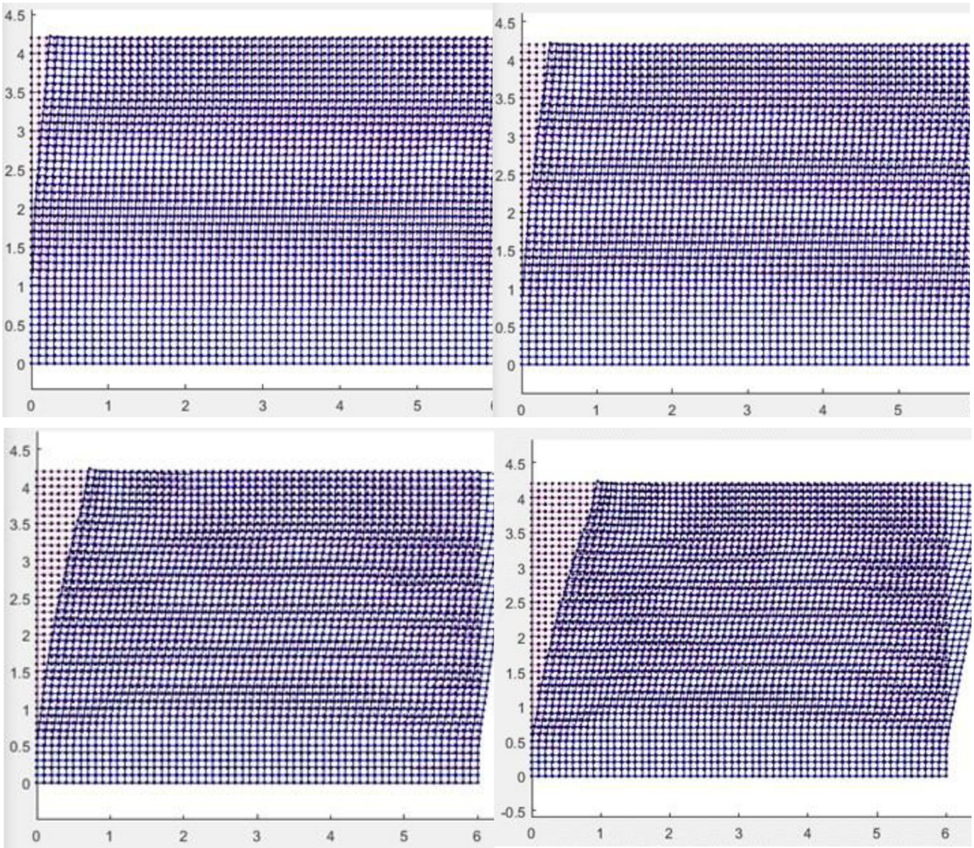


FIGURE 23. Results of infilled frames: deformed shape at different load steps

boundary conditions. The results were verified using available software packages, results computed using beam theory and previous examples from literature.

– More accurate results were obtained from increasing the number of elements in meshing a structure. However, increasing the number of springs between each two adjacent elements had a slight effect on the results.
– To achieve good results with a reasonable run time, fine mesh with a modest number of springs is advised.
– The proposed program managed to predict failure initiation and propagation during the different loading steps of the model unlike FE programs.
– When elements exceed the permissible tensile or compressive stresses, they were directed to be colored by two different colors, to differentiate the failure kind.

The proposed program could accurately represent failure propagation and deformation of a structure composed of more than one material.

References

Eraky, A., Mustafa, S. A. & Badawy, M. (2021). Structural analysis using Applied Element Method: a review. *Egyptian Journal for Engineering Sciences and Technology*, *34* (1), 16–27. https://doi.org/10.21608/eijest.2021.56786.1043

Extreme Load Analysis (ELS) *Program (2021). Linear analysis of a statically indeterminate beam*. Retrieved from: https://www.extremeloading.com/download/Static-Loading-Verification-Samples/Elastic-Cases/General/1-001.pdf [acccessed: 13.08.2022].

Gohel, V., Patel, P. V. & Joshi, D. (2013). Analysis of frame using applied element method (AEM). *Procedia Engineering*, *51*, 176–183. https://doi.org/10.1016/j.proeng.2013.01.026

Christy, D. L., Pillai, T. M. & Nagarajan, P. (2018). Analysis of concrete beams using applied element method. *IOP Conference Series: Materials Science and Engineering*, *330* (1), 012117. https://doi.org/10.1088/1757-899X/330/1/012117

Christy, D. L., Pillai, T. M. & Nagarajan, P. (2020). Annular Sector Element for Applied Element Method. *Journal of the Institution of Engineers (India): Series A*, *101* (4), 571–578. https://doi.org/10.1007/s40030-020-00455-3

Karad, P. & Patel, P. (2020). Analysis of reinforced concrete element using Applied Element Method (AEM). *Proceedings of the International Conference on Recent Advances in Computational Techniques*, *2020*, 1–5. https://doi.org/10.2139/ssrn.3697555

Lupoae, M. & Bucur, C. (2009). Building demolition – positive aspect of progressive collapse. *Military Technical Academy Review*, *19* (4), 399–408.

Meguro, K. & Tagel-Din, H. (1997). A new efficient technique for fracture analysis of structures. *Bulletin of Earthquake Resistant Structure Research Center*, *30*, 103–116.

Meguro, K. & Tagel-Din, H. (2000). Applied element method for structural analysis: theory and application for linear materials. *Structural Engineering/Earthquake Engineering*, *17* (1), 31–45.

Meguro, K. & Tagel-Din, H. (2001). Applied Element Simulation of RC Structures under cyclic loading. *Journal of Structural Engineering*, *127* (Nov.), 1295–1305.

Meguro, K. & Tagel-Din, H. (2002). Applied Element Method used for large displacement structural analysis. *Journal of Natural Disaster Science*, *24* (1), 25–34.

Moss, J. M. (2020). *Development, verification, and validation of an applied Element Method simulation framework for glass lite fracture, fragmentation, and debris field prediction* (doctoral dissertation). The University of North Carolina at Charlotte, Charlotte.

Nichols, J. & Totoev, Y. (1997). Experimental determination of the dynamic Modulus of Elasticity of masonry units. *15th Australian Conference on the Mechanics of Structures and Materials*, *2015*, 1–7.

Shakeri, A. & Bargi, K. (2015). Use of applied element method for structural analysis. *KSCE Journal of Civil Engineering*, *19* (5), 1375–1384. https://doi.org/10.1007/s12205-015-0625-4

Tagel-Din, H. & Meguro, K. (1999). Applied Element Method: a new efficient tool for design of structure considering its failure behavior. *5th US National Conference on Lifeline Earthquake Engineering*, *13*, 1–30.

Tagel-Din, H. & Meguro, K. (2000a). Analysis of small scale RC building subjected to shaking table tests using applied element method. *Proceedings of the 12th World Conference on Earthquake Engineering*, 1–8. Retrieved from: http://www.iitk.ac.in/nicee/wcee/article/0464.pdf [acccessed: 13.08.2022].

Tagel-Din, H. & Meguro, K. (2000b). Applied element method for dynamic large deformation analysis of structures. *Doboku Gakkai Ronbunshu*, *2000* (661), 1–10.

Tagel-Din, H. & Rahman, N. A. (2006). The Applied Element Method: the ultimate analysis of progressive collapse. *STRUCTURE Magazine*, *4*, 30–33.

Tokal-Ahmed, Y. M. (2009). *Response of bridge structures subjected to blast loads and protection techniques to mitigate the effect of blast hazards on bridges*. Rutgers-New Brunswick: The State University of New Jersey.

Badawy, M. M., Mustafa, S. A. A., Bakry, A. E. (2022). Behavior and failure tracking of structural elements using applied element method. *Sci. Rev. Eng. Env. Sci.*, *31* (3), 203–222. DOI 10.22630/srees.3897

Wibowo, H., Reshotkina, S. S. & Lau, D. T. (2009). Modelling progressive collapse of RC bridges during earthquakes. *Proceedings, Annual Conference – Canadian Society for Civil Engineering*, *2*, 899–909.

Summary

Behavior and failure tracking of structural elements using applied element method. Applied element method (AEM) is a recently displacement-based structural analysis method. It provides the benefits of both the finite element method (FEM) and the discrete element method (DEM). This method relies on those structures are segmented into rigid elements linked by normal and shear springs. In this paper a brief note of the AEM is given. Then, using the AEM, a 2D MATLAB open source program was created to analyze different structures with varied boundary conditions and to permit researchers for enhancing the method. The proposed program was verified using linear elastic analysis and large deformation static analysis. The influence of element size and the number of connecting springs between elements was studied. Finally, the proposed program was capable of tracking failed elements and their spread. In addition, the program could predict deflection values and structure deformed shape.