2011 International Conference on Modeling, Simulation and Control
IPCSIT vol.10 (2011) © (2011) IACSIT Press, Singapore

Finite Elements for Engineering Analysis: A Brief Review

Logah Perumal [1] and Daw Thet Thet Mon [2]

[1, 2] Faculty of Engineering and Technology, Multimedia University, Malacca Campus,
Jalan Ayer Keroh Lama, Bukit Beruang, 75450 Melaka, Malaysia

Abstract. Finite element analysis (FEA) employs piecewise approximation in which the continuum (domain) of interest is divided into several sub regions called finite elements. Each finite element is solved independently and later, overall solution for the continuum is obtained by combining these individual finite element results. Various finite elements have been proposed to facilitate analysis of new phenomena and to improve existing methods. This paper is written to provide brief introduction to FEA and evolution of finite elements used in engineering analysis. From the review reported in this paper, it can be seen that finite element has undergone vast development since its formulation. There are still rooms for further development to suite for analysis of new phenomena and more specific elements are being developed to facilitate new engineering application.

Keywords: Element, Finite element, finite element analysis, element formulation.

1. Introduction

Finite elements play an important role in FEA since it enables a continuum to be analysed with ease, by discretizing the continuum into several sub domains. Apart from that, finite elements enable analysis of a continuum with complex geometries, since finite elements can be formulated with arbitrary shape. This enables a continuum to be represented without need for any actual geometry simplification. Finite elements are formulated to carry out specific analysis and they are used in various engineering applications, especially in engineering design. Use of FEA enables verification of a proposed design to be safe and meets required specifications even before the design is manufactured. Finite elements play an important role in providing accurate and reliable results, since it would impact important decisions which are taken based on the FEA results.

Early studies showed that accuracy of a FEA depends on the element's geometry, geometry distortion such as isoparametric elements, shape function used in development of finite element, principles and laws used in developing the governing equation and material to be analysed. Since there are many choices of elements to be used for an analysis, studies have been conducted to ease selection of suitable element for an analysis. For 1-dimensional analysis, specific elements can be directly used to solve certain analysis. Subsequent research results indicated that for 2-dimensional analysis, bilinear quadrilateral elements are superior compared to simple linear triangular elements in terms of meshing and accuracy. Accuracy of simple linear triangular element can be improved by using higher order elements, but it leads to a problem known as mesh locking, which is main drawback of triangular element when the analysis is done onto incompressible materials [1]. Later, h-p adaptive techniques were introduced to increase accuracy of finite element method. It is found out that quadrilateral elements give better accuracy compared to triangular elements when tested using the newly formulated h-p adaptive technique [2]. As for 3-dimensional analysis, hexahedron elements are preferred compared to tetrahedron elements. For same degree of order, hexahedron elements give more accurate results compared to tetrahedron elements. Nevertheless, both quadratic hexahedron and tetrahedron elements give similar performance and accuracy.

This paper is hoped to give a brief idea to the reader on FEA and finite elements that have been developed so far. Organization of the paper is as follows. Following section 2 is used to provide brief introduction to FEA. Formulation of a finite element is explained in section 3. Section 4 is used to highlight evolution of finite elements since its formation and the paper is finally concluded in section 5.

2. Finite element analysis

Behaviour of a phenomenon can be represented using mathematical models (approximate models), which are derived based on principles and laws. There are several principles which are used to formulate finite elements. Principle of static equilibrium (also known as direct method) is used for phenomena which can be represented by simple governing equation, and theorem of Castigliano and principle of minimum potential energy are applied for complicated elastic structural systems. Higher mathematical principles, known as variational methods are used to formulate finite element analysis for phenomena governed by complex mathematical model, involving derivative terms. There are several variational methods such as Ritz, Galerkin, collocation, and least-squares methods [3].

Once approximate model has been developed using principles and laws described above, shape functions are then applied according to element geometry to complete the finite element formulation. General equation for a single finite element is represented in form shown below:

$$[k]\{q\} = \{Q\} \tag{1}$$

where $[k]$ is the matrix representing characteristics of the continuum, $\{q\}$is the column matrix representing nodal values (output variable of interest), and $\{Q\}$ represents input to the continuum. In case of stress analysis, $[k]$ represents stiffness matrix, $\{q\}$ represents vector of nodal displacements, and $\{Q\}$ represents vector of nodal forces. Once individual finite elements are formulated, these finite elements would then be assembled to form global/assemblage equations which are represented generally in form below:

$$[K]\{r\} = \{R\} \tag{2}$$

where $[K]$ represents assemblage property matrix, $\{r\}$ represents assemblage vector of nodal unknowns, and $\{R\}$ represents assemblage vector of nodal forcing parameters. Figure 1 summarises steps involved in FEA.

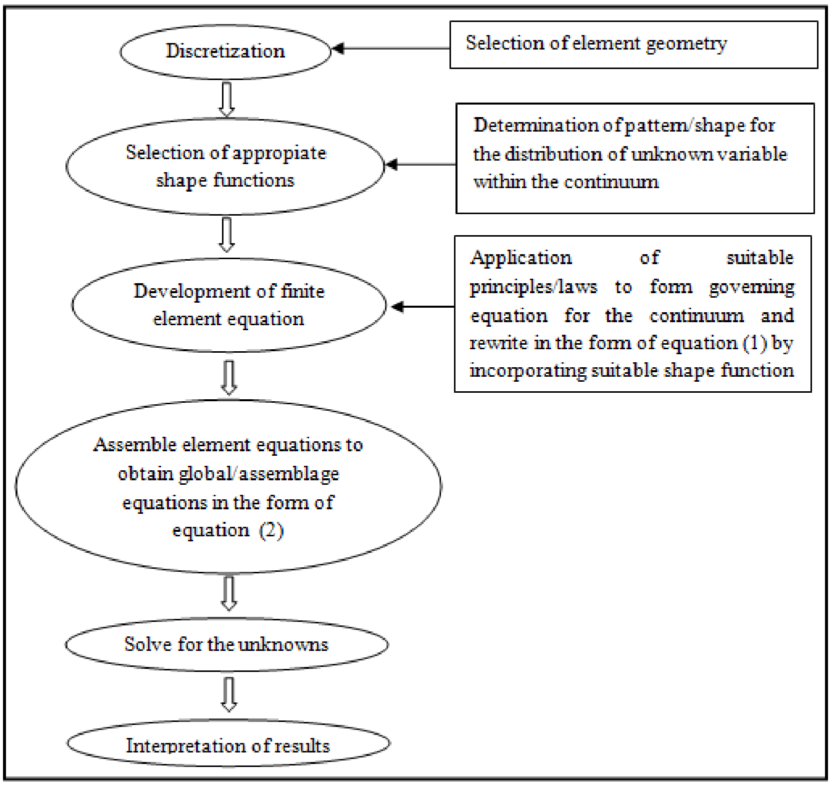

Fig. 1: Steps involved in finite element analysis.

3. Formulation of a finite element

Finite element equation, which is represented by equation (1), is obtained by incorporating governing equation into element equation. Element equation is derived from shape function and geometry equation. Number of equations for an element depicts number of nodes for the element. For example, considering two degree of freedom for each node, linear shape function for triangular and rectangular elements will consists of 2 unknowns (represents 2 degree of freedom). Number of element equations for a triangular element would be 3 whereas rectangular element would have 4 equations. In other words, number of equations for an element represents total number of nodes for the particular element and number of unknowns in each element equation represents degree of freedom of the particular node. Figure 2 below shows example of triangular and rectangular elements with 2 degree of freedom for each node. Assuming one field variable to be analysed, linear shape function for elements in figure 2 are given by equations 3 and 4.

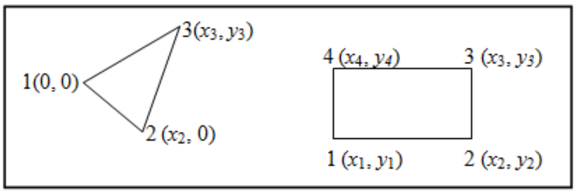

Fig. 2: Triangular and rectangular elements.

Shape function for linear triangular element (3 nodes): $\Phi(x, y) = a_0 + a_1 x + a_2 y$ (3)

Shape function for linear rectangular element (4 nodes): $\Phi(x, y) = a_0 + a_1 x + a_2 y + a_3 xy$ (4)

where coefficients a_0, a_1, a_2, and a_3 are the unknowns. Element equations for triangular element are later obtained by incorporating nodal conditions into the shape function and rewriting:

$$
\begin{aligned}
N_1(x, y) &= \frac{1}{x_2 y_3}\left[x_2 y_3 - y_3 x + (x_3 - x_2) y\right] \\
N_2(x, y) &= \frac{1}{x_2 y_3}\left[y_3 x - x_3 y\right] \\
N_3(x, y) &= \frac{y}{y_3}
\end{aligned}
\tag{5}
$$

Where x and y represents the coordinate systems and N_1, N_2 and N_3 represents element equations. Similarly for rectangular element:

$$
\begin{aligned}
N_1(r, s) &= \frac{1}{4}(1-r)(1-s) \\
N_2(r, s) &= \frac{1}{4}(1+r)(1-s) \\
N_3(r, s) &= \frac{1}{4}(1+r)(1+s) \\
N_4(r, s) &= \frac{1}{4}(1-r)(1+s)
\end{aligned}
\tag{6}
$$

where r and s represent the normalized coordinate systems. Governing equation is then incorporated into the element equation to form the specific finite element. For example, governing equations for 2-dimensional plane stress are given by:

$$
\begin{aligned}
\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial x} &= 0 \\
\frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{xy}}{\partial y} &= 0
\end{aligned}
\tag{7}
$$

Substituting governing equations above into the element equation for a triangular element produces the finite element formula:

$$\{\varepsilon\} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial x} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{\partial N_1}{\partial y} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial x} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ v_1 \\ v_2 \\ v_3 \end{Bmatrix} \tag{8}$$

$$\{\varepsilon\} = [B]\{\delta\}$$
$$similarly \;\; for \;\; \{\sigma\} = [D][B]\{\delta\}$$

Where matrix ε represents element strain matrix, [B] represents strain displacement matrix, δ represents element displacement matrix and [D] represents material property matrix. Application of minimum energy principle later yields the following equation:

$$[k]\{\delta\} = \{f\} \tag{9}$$
$$[k] = V^e [B]^T [D][B] \tag{10}$$

Where V^e represents volume of the element and {f} represents nodal forces applied onto the element. Figure 3 below summarises steps involved in development of a finite element equation.

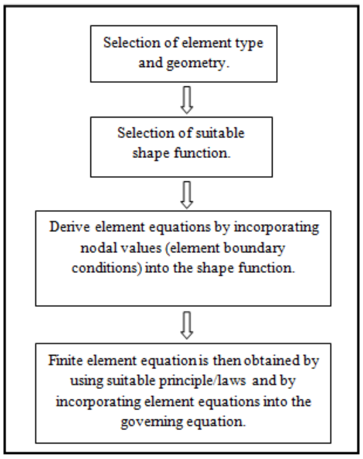

Fig. 3: Summary of development of an element equation.

4. Evolution of finite elements

Basically, there are three groups of elements and those are; line elements (for one dimensional analysis), planar elements (also known as membrane elements; for two dimensional analyses) and solid elements (for three dimensional analyses). There are various types of elements formulated under each group. For instance, spring element, bar element, and flexure elements are specific elements categorised under line elements. Specific elements are used in particular cases in which the element has been formulated for. On the other hand, general elements can be used for any cases, by simply changing governing equation according to problem type. Triangular elements and rectangular elements are examples of general planar elements. Pentahedron (also known as triangular prism or wedge), hexahedron (also known as brick) and tetrahedron (also known as tet) elements are examples of general solid elements. Since then, more types of elements have been formulated as described below.

4.1. One dimensional elements

Simplest line element is the spring element (a specific element developed using principle of static equilibrium) and can be used to analyse linear member within its elastic range. Since its formulation, spring elements have been successfully applied in various applications. One of the applications which benefited plenty from line-spring element is simulation and analysis of surface crack growth. Line-spring model was developed and used to carry out stress analysis for cracked surface, but limited to linear elastic analysis [4]. Nonlinear elastic-line-spring model was then developed [5]. In order to incorporate plastic deformation, the authors in [6] introduced elastic-plastic model of the line-spring finite element using the incremental theory of plasticity. Eventually, fully plastic crack growth has been formulated using line spring element in [7].

Analysis of surface crack requires large computational time, especially for 3-dimensional analysis. It is found out that this problem can be solved by coupling shell and line-spring finite elements together [8] (Shell elements are used to model the surface while line-spring elements are used to model the crack growth), which reduces 3-dimensional problem to equivalent simpler 2-dimensional problem and thus reduces computational effort. These observations are reported in [9]. Another application of coupled shell and line-spring finite elements is found in fracture mechanic analysis on welded wide plates [10]. Spring element has also been used in seismic response analysis, by representing the contact bodies by finite number of small rigid bodies which are connected with springs distributed over the contact area of two neighbouring bodies [11]. This method is called rigid body spring element method (RBSM) and proved to be more economical and reduces computational time compared to conventional finite elements [12]. RBSM is also used in structural analysis, fracture analysis and damage analyses. Spring element has also been used in analysis of fiber-reinforced composites by representing the fiber with longitudinal spring element and the matrix with transverse shear spring element. The mathematical model representing fiber-reinforced composites using spring elements as described above is known as spring element model (SEM) [13].

New types of spring elements have also been formulated. Such new development is shear spring elements which were formulated to carry out stress and energy analysis of a centre-crack panel reinforced by a rectangular patch [14]. In [15], the authors have formulated generalized beam/spring track element by coupling spring elements at the periodic rail/tie intersections to analyze natural vibration of rail track within its elastic range.

Concept of spring element has then been extended to bar element, (also known as spar, link, beam or truss element, which are also specific elements) by introduction of “interpolation function, which is also known as “shape function or blending function” to evaluate values in between boundaries of element. From this point onwards, the term “shape function” will be used. Bar elements are used to analyse skeletal-type systems such as planar trusses, space trusses, beams, continuous beams, planar frames, grid systems, and space frames. One of applications which benefitted a lot from this element is analysis of truss and beam structures. The authors in [16] have developed a beam finite element to analyse dynamic problems especially vibration, which is arising in beam-like truss structures. Later, elastic beam finite element was formulated based on elasto–plastic fracture mechanics to simulate formation and effects of cracks in beams [17]. Analysis of hyperbolic cooling tower which was formerly focussing solely on the tower was made complete by using beam elements to include the supporting columns in the analysis [18]. A new element known as rigid-ended beam element was formulated to represent brackets in beam modelling of ship structures (Formerly, the brackets were represented by rigid elements). The newly formulated rigid-ended beam element has been found to increase computational efficiency as compared to previously used rigid elements [19].

Beam element has been also found to be applied for linear static analysis of laminated composites and in wavelets analysis. Another type of beam element is cubic beam elements which are used in practical analysis and design of steel frames [20]. Bar elements discussed above takes only axial loads and can be used to analyse simple trusses, since bending effects are not included in the analysis. Later, elementary beam theory is used to include transverse bending effect and thus new element known as flexure element was developed. Flexure element developed earlier is then enhanced using shear-deformable theory in order to avoid “locking” problem which arises in analysis of thin beam/plate [21]. Flexure elements are also used as a design tool to determine optimal geometry.

4.2. Two dimensional elements

Spring element, bar element and flexure element discussed above are examples of line elements, which are used for analysis of simple one dimensional problem. For problems involving differential equations, variational methods are used to formulate the finite element, as mentioned earlier in section 3. Prior to the year 1952, line elements were the only source used to carry out structural analysis. Need for a planar element arises when line elements failed to produce satisfactory results when used in two dimensional analyses. This shortcoming was fulfilled in 1954 when head of the structural dynamics unit of Boeing Airplane Company, Turner successfully formulated triangular element which is known as the constant strain triangular (CST) element [22] to solve for bending and torsional flexibility influence coefficients on low aspect wings. The CST was developed when results obtained by using one dimensional element did not agree with the experimental results. In addition to CST, rectangular element was also formulated based on equilibrium stress patterns and presented at the same time.

Initially, CST and rectangular elements developed by Turner is specifically used to analyse discrete structure. Later, the elements were further enhanced and many different methods to solve for continuous structures were proposed in [23]. Later in 1958 and 1960, computer programs were developed to solve finite element equations automatically for triangular elements and rectangular elements in structural analysis [24]. Analysis on plate bending using computer program was first initiated using rectangular element [25] and results showed that the problem can be modelled accurately using the rectangular element. It has been found out that rectangular elements require fewer computational efforts compared to triangular element. Nonetheless, triangular elements can be used to represent irregular boundaries accurately compared to rectangular elements.

Triangular element has been used in many other areas. One of the early applications of triangular element is in the thin plate bending analysis hence known as plate bending element. The triangular element used for the development of plate bending element is given the name HCT, named after the founders; Hsieh, Clough and Tocher [26]. In [27], the author has combined HCT with a triangular element to solve shell structures of arbitrary geometry and in [28], the author has combined HCT and CST to solve buckling of plates. HCT element is later replaced by Discrete Kirchhoff Triangular (DKT) element. Triangular and rectangular elements were later used in solving time dependant problem. In [29], the author has successfully used these elements to determine factors causing creep stresses and construction stresses in dams prior to hydrostatic loading. After the first introduction of finite elements in solution of time dependant problems, similar method were then used to rapidly investigate the stress concentration within underground concrete and rock structures [30]. Shortly, CST and a frequency domain time solution method were used in [31] to analyze dam interaction during earthquake (the analysis consists both earth and concrete dam). Later, these elements were used in stress analysis of a cross section of a solid rocket propellant subjected to internal pressure (quadrilateral element) and a rocket nozzle (triangular element). It is proved that these elements can be used for analysis of objects with arbitrary geometry.

These two dimensional elements were then vastly implemented in many engineering areas such as in heat transfer, analysis of nearly incompressible materials, large displacements, analysis involving nonlinear materials and fluid-flow in porous media. Since then, triangular and rectangular elements have been reviewed and enhanced further. Accuracy of analysis is improved by increasing number of nodes of the element. Thus, higher order elements were introduced and consequently two families of elements emerged; serendipity family and lagrangian family. Elements with external nodes (nodes are located on the boundaries of the element) are classified under serendipity family and elements with both internal and external nodes are classified under lagrangian family. Serendipity elements are more efficient and stable compared to Lagrangian elements, even though serendipity elements are less accurate.

Six node triangular elements were first introduced by De Veubeke and John Argyris. In [32], the author used the higher order six node triangular elements to solve problems in which nonlinear behaviour and buckling are important and proved that six node triangular element produces more accurate result compared to three node triangular element. Other new types of planar elements were formulated. Such new developments are the isoparametric elements. Isoparametric elements enables elements to have curved boundaries and flexible hence more compatible as shown in figure 4. Quadrilateral element as shown by figure 4(a) is an example of isoparametric element, which was derived from the basic rectangular element while figure 4(b) shows a triangular isoparametric element formulated from basic triangular element. Quadrilateral element was first applied in thermal stress analysis of the Apollo Spacecraft during re-entry to the Earth’s atmosphere. The structure was categorised as axisymmetric structure subjected to non axisymmetric loading.

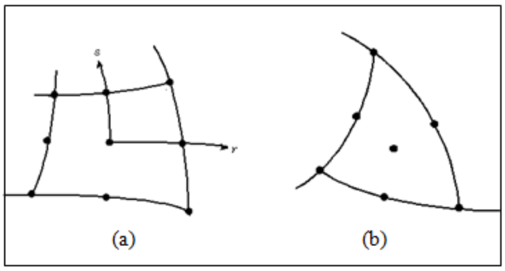

Fig. 4: Examples of isoparametric elements (a) quadrilateral element formulated from rectangular element (b) triangular isoparametric element.

Polygon shaped elements were later developed such as pentagonal element, hexagonal element and *n* – sided elements. These polygonal finite elements are used extensively in solid mechanics as well as in heat transfer applications. Polygonal elements are advantages in such way that polygonal elements provide more accurate results, easier to mesh arbitrary geometries especially those commonly found in biomechanics, used as mid-edge elements (transition element), suitable in material design and in the modelling of polycrystalline materials [33], and less sensitive to lock. In addition, circular elements were developed to facilitate analysis of circular or annular plate. The circular element is then improved and applied in analysis of stress distribution along free boundary of a circular hole in a plate [34].

4.3. Three dimensional elements

Advancement in computer era enabled development of new and advanced elements such as 3-dimensional elements which are used in analysis of solids, plates and shells. Most of 3-dimensional elements were first formulated and presented in a conference held at Ohio [35]. The papers also presented sample applications of 3-dimensional solid elements in solid mechanics. One of the new 3-dimensional elements formulated and presented in the conference is the ten node solid tetrahedral element. Since then, 3-dimensional elements were used in analysis of fluid flow (in 1969 using eight node isoparametric element) and structural analysis of three dimensional solids.

There are 3 basic solid elements; hexahedron (also known as brick), tetrahedron (also known as tet), and pentahedron (also known as triangular prism or wedge) elements. Hexahedron elements are generally used in modelling 3-dimensional solids and it also has been the motivating factor for development of "Ahmad-Pawsey" shell elements. Specific hexahedron element known as thin piezoelectric hexahedron finite element is developed to be used in design analysis of "smart" continua, which is an elastic plate laminated with segmented distributed piezoelectric sensors and actuators [36]. Recently, hexahedron elements have been used to simulate deformation of elastic objects in real time on desktop PCs [37].

Tetrahedron elements are mostly used in mesh generation and adaptive mesh refinement schemes. Pentahedron elements are relatively used in thick shell analysis. Pentahedron elements are also used in investigation of formation formation, propagation and interaction of cracks and interlaminar delaminations in composites due to impact loadings [38]. These solid elements are then refined with additional nodes to increase accuracy. For example, a new 6 noded pentahedron element was developed in [39] to prevent shear, trapezoidal and thickness lockings and eight to 26-node hexahedron is proposed in [40] for thermal-elasto-plastic finite element analysis.

Other types of solid elements were developed to facilitate specific tasks. Such elements are the pyramid and wrick elements and they are used in vibration analysis. Pyramid elements are also used as mid-edge element (transition element) between hexahedron and tetrahedron elements [41]. Later, new technique has been studied whereby tetrahedron elements connect directly with hexahedron elements without using pyramid elements. Another type of 3-dimensional element formulated is the shell elements. Shell elements are 4 to 8 node isoparametric quadrilaterals or 3- to 6-node triangular elements in any 3-D orientation. These elements are used to model both thick and thin shell problems as well as curved shells. Equivalent 3-dimensional elements are also formulated for polygon type elements discussed in previous section.

5. Conclusion

Finite element formulation is one of vital processes in FEA and there have been vast researches in the area of development of element geometries (to facilitate meshing) and finite elements (to facilitate analysis of new physical phenomena) as reported in this paper. Active research is still being carried out in FEA including invention of new elements to facilitate analysis of new phenomena and more specific elements are being developed to facilitate new engineering application. Although variety of element types have been developed and being used, there are still rooms to improve and develop new elements, particularly specific elements for specific analysis.

6. References

[1] Hughes, T.R.J., The Finite Element Method, Prentice Hall, Englewood Cliffs, N.J.,1987.

[2] Lo, S.H. and Lee, C.K. "On using Meshes of Mixed Element Types in Adaptive Finite Element Analyses," Finite Elements in Analysis and Design, 11 (1992) 307-336

[3] J.N. Reddy, An introduction to the finite element method, 3rd ed. New York: McGraw – Hill, 2004.

[4] Wang, Y.-Y. and Parks, D. M. (1992). Evaluation of the elastic T-stress in surface-cracked plates using the line spring method. Int. J. Fract. 56,25-4O.

[5] Kumar, V. and German, M. D. (1985). Studies of the line-spring model for nonlinear crack problems. J. Press. Vessel Tech. 107,412-420.

[6] Hyungyil lee and david m. parks "Enhanced Elastic-Plastic Line-Spring Finite Element", int. J. Solids Structures Vol. 32, No. 16, pi. 2393-4418, 1995

[7] Hyungyil Lee and David M. Parks, "Line-spring finite element for fully plastic crack growth—II. Surface-cracked plates and pipes**"**, International Journal of Solids and Structures, Volume 35, Issue 36, December 1998, Pages 5139-5158

[8] d. i. nwosu and d. o. olowokere," evaluation of stress intensity factors for steel tubular t-joints using line spring and shell elements", engineering failure analysis, vol 2, no. 1 pp. 31-44, 1995

[9] E. Berg, B. Skallerud and C. Thaulow, "Two-parameter fracture mechanics and circumferential crack growth in surface cracked pipelines using line-spring elements**"**, Engineering Fracture Mechanics, Volume 75, Issue 1, January 2008, Pages 17-30

[10] Matteo Chiesa, Bjorn Skallerud, and Christian Thaulow, "Fracture analysis of strenght-mismatched welded wide plates by line spring elements", Engineering Fracture Mechanics Volume 68, Issue 8, May 2001, Pages 987-1001

[11] Jian-Hai Zhang, Jiang-Da He, and Jing-Wei Fan," Static and dynamic stability assessment of slopes or dam foundations using a rigid body–spring element method", International Journal of Rock Mechanics & Mining Sciences 38 (2001) 1081–1090

[12] tadahiko kawai**, "**new discrete models and their application to seismic response analysis of structures", nuclear engineering and design volume 48, issue 1, june 1978, pages 207-229

[13] T. Okabe, H. Sekine, K. Ishii, M. Nishikawa, N. Takeda, "Numerical method for failure simulation of unidirectional fiber-reinforced composites with spring element model", Composites Science and Technology 65 (2005) 921–933

[14] R.C. Chu and T.C. Ko,"Isoparametric shear spring element applied to crack patching and instability", Theoretical and Applied Fracture Mechanics Volume 11, Issue 2, May 1989, Pages 93-102

[15] Z. Cai and G.P. Raymond, "Use of a generalized beam/spring element to analyze natural vibration of rail track and its application", International Journal of Mechanical Sciences Volume 36, Issue 9, September 1994, Pages 863-876

[16] B. Necib and C.T. Sun, "Analysis of truss beams using a high order Timoshenko beam finite element," Journal of Sound and Vibration Volume 130, Issue 1, 8 April 1989, Pages 149-159

[17] M. Krawczuk, A. Zak and W. Ostachowicz "Elastic beam finite element with a transverse elasto-plastic crack," Finite Elements in Analysis and Design Volume 34, Issue 1, 1 January 2000, Pages 61-73.

[18] Kara" S. Surana' and Steve. H. Nguyen," Higher-order shear-deformable two dimensional hierarchical," Math/ Compur. Mode&g, Vol. 14, pp. 893-898, 1990

[19] Seung I1 Seo, Chang Doo Jang, "Development of a Rigid-ended Beam Element For Analysis of Bracketed Frame Structures," *Marine Structures Vol.* 9, 1996, pp. 971-990

[20] Lip H. Teh “Cubic beam elements in practical analysis and design of steel Frames”, Engineering Structures Vol. 23, 2001, pp. 1243–1255

[21] Tarun Kant and Pradeep B. Kulkarni “A *C*° continuous linear beam/bilinear plate flexure element”, Computers & Structures Volume 22, Issue 3, 1986, Pages 413-425

[22] Turner, M., R. W. Clough, H. C.Martin and L. J. Topp, “Stiffness and Deflection Analysis of Complex Structures”, J. Aeronautical Science 23 (9), pp. 805-823, Sept. 1956

[23] Argyris,J., “Energy Theorems and Structural Analysis”, Aircraft Engineering, 1954 and 1955. In 1960 these papers were consolidated in a book by Butterworths Scientific Publications titled *Energy Theorems and Structural Analysis*.

[24] Clough, R. W., “The Finite Element Method in Plane Stress Analysis”, Proc. 2nd ASCE Conf. On Electronic Computation, Pittsburg, Pa. Sept. 1960.

[25] Adini, A. and Clough, R. W., “Analysis of Plate Bending by the Finite Element Method”, NSF Report, Grant G7337, 1960.

[26] Clough, R. W. and J. L. Tocher, “Finite Element Stiffness Matrices For the Analysis of Plate Bending’, *Proc. Matrix Methods in Structural Analysis*, Wright-Patterson Air Force Base, Ohio, October 26-28, 1965.

[27] Johnson, C. P. “The Analysis of Thin Shells by the Finite Element Method”, UCB/SESM Report No. 67/22, University of California, Berkeley, 1967.

[28] Murray, D. W., “Large Deflection Analysis of Plates”, UCB/SESM Report No. 67/44, University of California, Berkeley, .Ph.D. Dissertation, 1967.

[29] King, I. P.,”Finite Element Analysis of Two-Dimensional Time-Dependent Stress Problems”, UCB/SESM Report No. 65/1, University of California, Berkeley, January 1965.

[30] Raphael, J. M. and R. W. Clough, “Construction Stresses in Dworshak Dam”, UCB/SESM Report No. 65/3, University of California, Berkeley, April 1965

[31] Chopra, A. K., “Hydrodynamic Pressures on Dams During Earthquakes”, UCB/SESM Report No. 66/02a, University of California, Berkeley, 1966.

[32] Felippa, C. A., “Refined Finite Element of Linear and Nonlinear Two - Dimensional Solids”, UCB/SESM Report No. 66/22, University of California, Berkeley, October 1966.

[33] Ghosh S, Moorthy S. Elastic-plastic analysis of arbitrary heterogeneous materials with the Voronoi cell finite-element method. Computer Methods in Applied Mechanics and Engineering 1995; 121(1–4):373 –409.

[34] A.K. Soh, Z.F. Long, “Development of two-dimensional elements with a central circular hole” Computer Methods in Applied Mechanics and Engineering, Volume 188, Issues 1-3, 21 July 2000, Pages 431-440

[35] *Proc. Matrix Methods in Structural Analysis*, Wright-Patterson Air Force Base, Ohio, October 26-28, 1965.

[36] H. S. Tzou, C. I. Tseng and H. Bahrami,”A thin piezoelectric hexahedron finite element applied to design of smart continua”, Finite Elements in Analysis and Design, Volume 16, Issue 1, April 1994, Pages 27-42

[37] Christian Dick, Joachim Georgii and Rüdiger Westermann **“**A real-time multigrid finite hexahedra method for elasticity simulation using CUDA” Simulation Modelling Practice and Theory, Volume 19, Issue 2, February 2011, Pages 801-816

[38] Mohammadi, S., Owen, D. R. J. & Peric, D. (1998). A combined finite/discrete element algorithm for delamination analysis of composites. *Finite Elements in Analysis and Design*, *28*, 321-336.

[39] Sze, KY and Chan, WK “A six-node pentagonal assumed natural strain solid–shell element” Finite Elements in Analysis and Design, 2001, v. 37 n. 8, p. 639-655

[40] J. M. J. McDill and A. S. Oddy “A nonconforming eight to 26-node hexahedron for three dimensional thermal-elasto-plastic finite element analysis”, Computers & Structures Volume 54, Issue 2, 17 January 1995, Pages 183-189

[41] Steven J. Owen and Sunil Saigal “Formation of pyramid elements for hexahedra to tetrahedra transitions”, Computer Methods in Applied Mechanics and Engineering Volume 190, Issue 34, 25 May 2001, Pages 4505-4518