

Stress Distribution In a Shear Wall – Frame Structure Using Unstructured – Refined Finite Element Mesh

Bahadır ALYAVUZ*

Gazi University, Faculty of Engineering & Architecture, Department of Civil Engineering, 06570, Ankara

Received: 01.09.2005 Accepted: 26.01.2007

ABSTRACT

A semi-automatic algorithm for finite element analysis is presented to obtain the stress and strain distribution in shear wall-frame structures. In the study, a constant strain triangle with six degrees of freedom and mesh refinement - coarsening algorithms were used in Matlab® environment. Initially the proposed algorithm generates a coarse mesh automatically for the whole domain and the user refines this finite element mesh at required regions. These regions are mostly the regions of geometric discontinuities. Deformation, normal and shear stresses are presented for an illustrative example. Consistent displacement and stress results have been obtained from comparisons with widely used engineering software.

Key Words: Shear wall, FEM, Unstructured mesh, Refinement.

1. INTRODUCTION

In the last two decades, shear walls became an important part of our mid and high rise residential buildings in Turkey. As part of an earthquake resistant building design, these walls are placed in building plans reducing lateral displacements under earthquake loads so shear-wall frame structures are obtained. Since the 1960's several approaches have been adopted to solve displacements and stress distribution of shear wall structures. Continuous medium approaches, and frame analogy models are the examples of these approaches [1-4]. In the past and today, numerical solution methods are the main effort area because of the accuracy of solution and the ease of usage in 2D and 3D analysis of shear walls [5-7].

Shear walls with openings, coupled shear walls and combined shear wall frame structures can be modeled as thin plates where the loading is uniformly distributed over the thickness, in the plane of the plate. This 2D domain can be subdivided into a finite number of geometrical shapes. In the finite element method (FEM), these simple shaped elements such as triangles

or quadrilaterals (in 2D) are called elements. The connection of these individual elements at nodes and along interelement boundaries covering the whole problem domain is called finite element mesh or grid. In the literature meshes can be grouped into two main categories such as structured and unstructured meshes. Structured meshes are constructed with geometrically similar triangular or quadrilateral elements. They are suitable especially for problems with simple geometry and boundary shapes (Figure 1-a). Although structured meshes can be constructed as simple-time saving routines, regarding complicated domains with complex boundaries, it is a problem to fit the boundary shape. To circumvent this difficulty, unstructured meshes are used to discretize the complicated domains with internal boundaries (Figure 1-b). While it is a time consuming procedure, unstructured meshes are also suitable for local mesh refinement and coarsening. The aim of this work is to get a good quality unstructured mesh which will have smaller elements at the geometric discontinuities and bigger elements at other regions for a shear wall frame geometry.

* Corresponding author, e-mail: balyavuz@gazi.edu.tr

2. TRIANGULAR FINITE ELEMENTS

2.1. Element Formulation

The first advantage of using triangular elements is that almost any plane geometry may be discretized using triangles. These elements have six degrees of freedom, two translations at each node (Figure 2). Because of the three nodes, the element has linear shape functions that are an additional benefit because of simplified mathematics. However, these functions generate constant strain and stress throughout the element. To surmount this disadvantage, smaller elements must be employed where strain and stress vary rapidly.

CST element has displacement functions and shape functions as follows,

$$u(x, y) = N_1 u_1 + N_2 u_2 + N_3 u_3, \quad (1)$$

$$v(x, y) = N_1 v_1 + N_2 v_2 + N_3 v_3 \quad (1)$$

$$N_1(x, y) = \frac{1}{2A_e} [(x_2 y_3 - x_3 y_2) - (y_3 - y_2)x + (x_3 - x_2)y] \quad (2)$$

$$N_2(x, y) = \frac{1}{2A_e} [(x_3 y_1 - x_1 y_3) - (y_1 - y_3)x + (x_1 - x_3)y] \quad (3)$$

$$N_3(x, y) = \frac{1}{2A_e} [(x_1 y_2 - x_2 y_1) - (y_2 - y_1)x + (x_2 - x_1)y] \quad (4)$$

where u_1 , u_2 and u_3 nodal displacements in x direction corresponding to nodes 1, 2 and 3 respectively. v_1 , v_2 and v_3 nodal displacements in y direction and N_1 , N_2 and N_3 are linear shape functions. x and y are the coordinates of corresponding nodes and A_e is area of the element. In the finite element method, nodal displacements are obtained from the solution of the linear system of equations, that is

$$\mathbf{K}\mathbf{u} = \mathbf{f} \quad (5)$$

where, \mathbf{K} is stiffness matrix, \mathbf{u} is nodal displacement vector, and \mathbf{f} is nodal load vector. Stiffness matrix may be calculated as

$$\mathbf{K} = A_e t (\mathbf{S}\mathbf{N})^T \mathbf{C}(\mathbf{S}\mathbf{N}) \quad (6)$$

where t is thickness of the element,

$$\mathbf{N} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \quad (7)$$

and differential operator is,

$$\mathbf{S} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial y} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}^T \quad (8)$$

and the elasticity matrix is defined by

$$\mathbf{C} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \quad (9)$$

where E is modulus of elasticity and ν is the Poisson's ratio.

2.2. Mesh Generation and Refinement

Unstructured mesh generation procedure consists of some basic steps. These are the generations of boundary and interior nodes and connection of these nodes which has a specific name as triangulation for triangular finite elements. In this work, a random point generation scheme is used to form interior nodes that were explained by Fukuda and Suhura in detail [8]. This procedure uses subsquares in which only one node is generated randomly. Although, here, node generation is a random process, we can define some restriction on this generation. For example, we can say that the distance between any two nodes must not be less than a minimum value which can be taken as the width of a subsquare. In fact this random procedure is a time consuming procedure, but this is not too long for today's desktop computers for initial node generation. Other algorithms can be used in large scale finite element problems. For example, instead of generating interior nodes by random trials, a rectangular grid could be used creating one node in each rectangle. The width and height of the rectangles are in the ratio of $2/\sqrt{3}$ and they are placed in a zigzag manner and nodes are placed at the center of the rectangles. The ratio helps ensure equilateral triangles [9]. Another node generation scheme in which imaginary horizontal lines cut the domain in an even number of points could also be used. Interior nodes are generated on the horizontal line between the cuts according to a prescribed spacing parameter [10].

For the triangulation step, a number of algorithms have been suggested by various authors. These techniques involve simple automatic triangulation methods [8,9], advancing front methods [10,11], domain decomposition methods [12,13], and coordinate transformation methods [14]. In the present work, triangular elements in the problem domain are obtained using a condition known as Delaunay or empty circle criterion [15-18]. According to this rule, in a circumcircle of a triangle no node must exist in the problem domain (Figure 3). The nodes in the problem domain are scanned from the first node to the last node selecting three candidate nodes which will obey the empty circle criterion. This criterion is very useful to eliminate intersection check of the interelement boundaries.

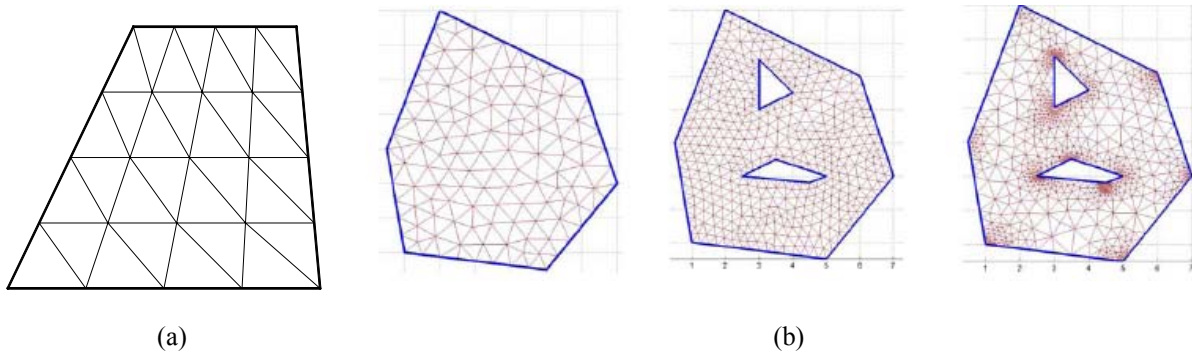


Figure 1. Structured (a) and unstructured mesh (b)

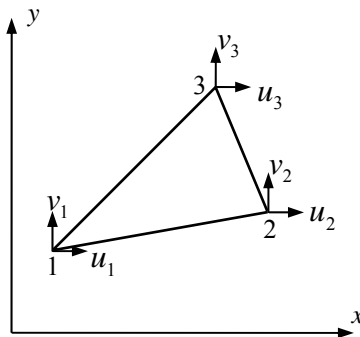


Figure 2. Constant strain triangular finite element

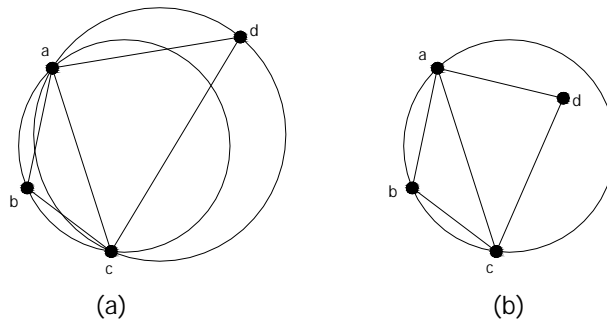


Figure 3. A triangulation which obeys the empty circle criterion (a), triangulation which does not obey the criterion (b).

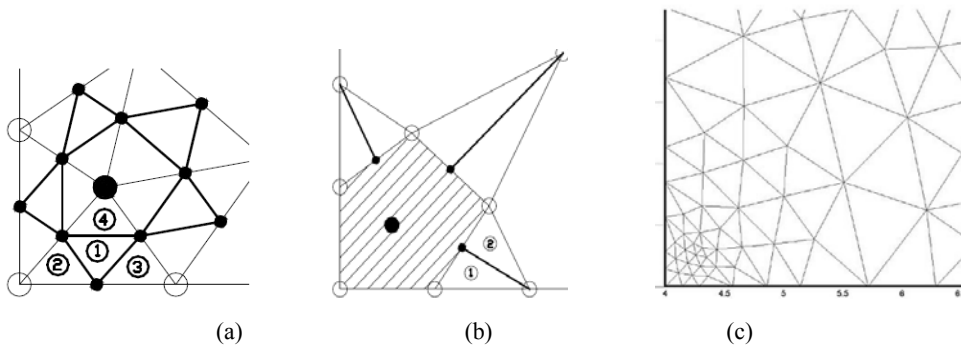


Figure 4. First kind of triangles (a), second kind of triangles (b), and after refinement (c)
 ● selected point for refinement.

Initially we could not know the necessary degree of smallness of finite elements to represent the stress distribution accurately. For that reason, a few numbers of initial nodes are used for triangulation giving an initial triangulation. Around geometrical discontinuities such as interior holes at shear walls and shear wall frame connection points, a local mesh refinement must be done by inserting new nodes and repeating the triangulation procedure. At this stage, a number of alternative algorithms could be employed to get a fine mesh where it is necessary. Those are,

- One point mesh refinement
- Line or polygon mesh refinement
- Central point mesh refinement
- and, Delaunay point mesh refinement

In FEM, all generated nodes and elements have some special id denoted by numbers. When this number is known for a node, it is easy to search surrounding elements of this node. For the first algorithm mentioned above, once these elements are determined, they are grouped forming *first kind of triangles* which are affected by refining. Some *second kinds of triangles exist*, which are the neighboring triangles to the first kind. According to the one point local mesh refinement algorithm, four new triangles are generated in the first kind of triangles and two new triangles are generated in the second kind of triangles and old ones are deleted (Figure 4-a and b). Line and polygon local mesh refinement algorithms also use the same logic which differs from the first one in such a way that, selected nodes make a line or a closed polygon. In the third refinement algorithm, a point is added into a triangle, using the geometrical center of apex nodes. The last algorithm, named the Delaunay point mesh refinement, adds a node into the triangulation at the center of the circumcircle of a selected triangle.

A mesh smoothing process was made to improve the quality of triangles after triangulation. In any branch of mechanics, the shape of triangles is an important factor for finite element meshes. Especially in solid mechanics triangles are regarded as good if they are nearly equiangular [19]. In computational fluid dynamics, problems concerning boundary layers and shocks, skinny and long triangles provide a better solution [20]. In order to obtain well-distributed good quality triangles, here, we used a Laplacian smoothing method which uses neighboring triangles for node repositioning (Figure 5). This is an iterative method in which a node is moved to the centroid of the nodes to which it is connected [21]. One or two iterations for moving the nodes are sufficient but iterations could be continued until each movement satisfies a convergence distance. The new coordinates defined as,

$$(x, y) = (x, y)_0 + \Delta u \quad (10)$$

where

$$\Delta u = \frac{1}{n} \sum_{i=1}^n U_i \quad (11)$$

$$U_i = (x_i - x_0, y_i - y_0) \quad (12)$$

n represents the number of surrounding nodes and zero indices represent initial values.

3. THE SHEAR WALL - FRAME PROBLEM

A shear wall model with small window openings connected to a beam column system was considered to perform a FEM analysis using unstructured mesh generation with refinement. This shear wall frame structure was loaded as tip load but the loading was distributed to the nodes at the top of the frame. The necessary dimensions of geometry are given in Figure 6-a. Boundary nodes were generated in a direct manner using the width of a column as a spacing parameter. A few initial nodes were obtained using the random procedure given in section 2 (Figure 6-b). Line mesh refinement algorithm was used to refine the mesh at the beams and columns. One point mesh refinement algorithm was used around sharp corners of window openings and beam shear-wall connections, in order to get the final mesh (Figure 7). The deformed shape and stress distributions obtained using the final mesh is given in Figure 8.

4. CONCLUSIONS

In order to get an accurate solution for stress distribution at beam to column and beam to shear wall connections, we developed a finite element program in MATLAB environment. Although the geometry of the problem is quite complex because of interior holes, the problem domain was discretized with an unstructured finite element with refinement.

- 1) The element mesh was refined employing small triangles at the regions of geometric discontinuities. Smooth transitions from small to large triangles were obtained. Good quality equiangular elements were obtained using empty circle criterion and Laplace smoothing.
- 2) A good agreement exists between ANSYS (PLANE2 elements) results and the work presented here in lateral displacements and shear stresses. Comparisons are given in Table 1 and Table 2 for lateral displacements and shear stresses respectively for a load value of $P=100N$.
- 3) The algorithm reviewed here falls into the semi automatic approach category. There is an automatic mesh generation for arbitrary domain with inner holes. A manual decision is required for the refinement regions.
- 4) In fact, the work presented here provided us with a springboard for further developments to get an adaptive finite element procedure for shear wall frame structures.

Table 1. Lateral displacement of shear wall structure with openings.

Location	u_x, Lateral disp. (m)	u_x, Lateral disp. (m) ANSYS (PLANE2)	Error (%)
N_{8m}	2.73×10^{-4}	2.83×10^{-4}	4.8
N_{4m}	1.03×10^{-4}	1.06×10^{-4}	2.8

N_{8m} : Nodes at 8 m height.

N_{4m} : Nodes at 4 m height.

Table 2. Shear stress, σ_{xy} , values at some points given in Figure 5.

Location	Stress (Pa)	Stress (Pa) ANSYS (PLANE2)	Error (%)
P1	205.6	222.15	7.4
P2	189.25	175.06	8,1
P3	36.75	40.13	8.4
P4	180.35	164.38	9.8

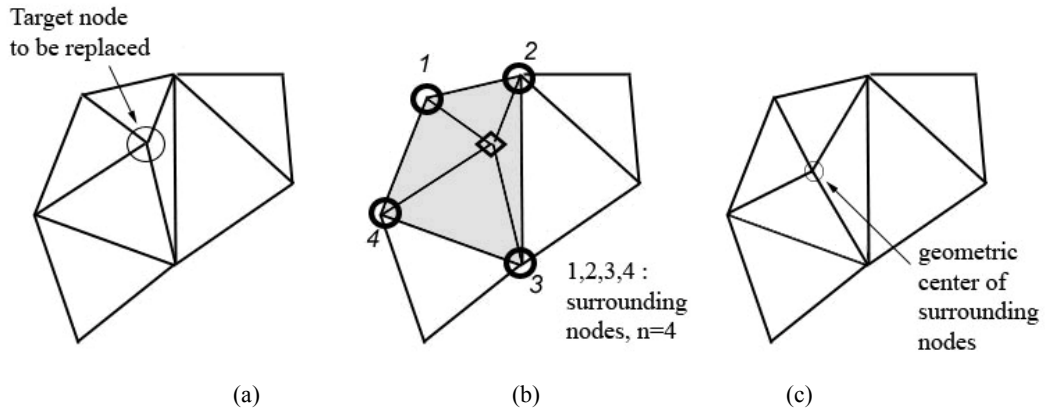


Figure 5. Before smoothing (a), neighboring nodes (b), after Laplacian smoothing (c).

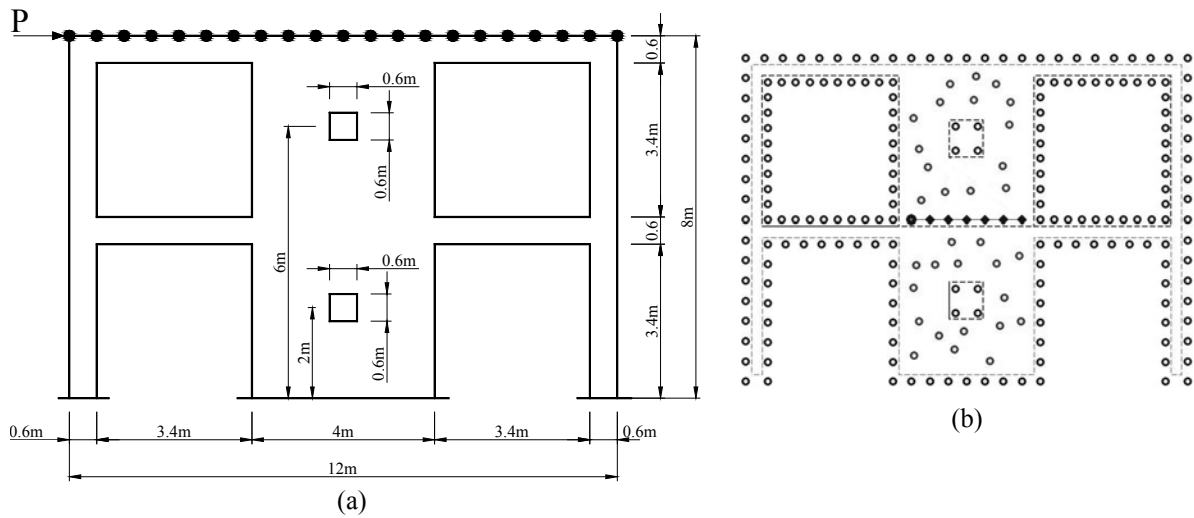


Figure 6. Example problem a) dimensions, loading and b) initial node generation.

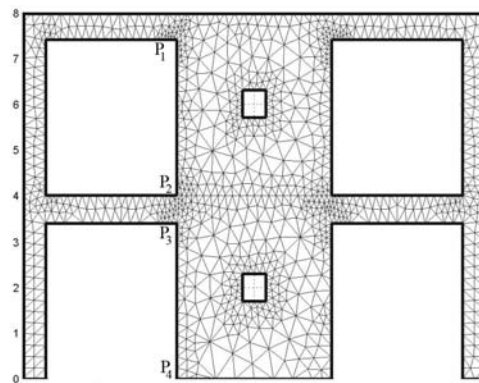


Figure 7. After mesh refinement.

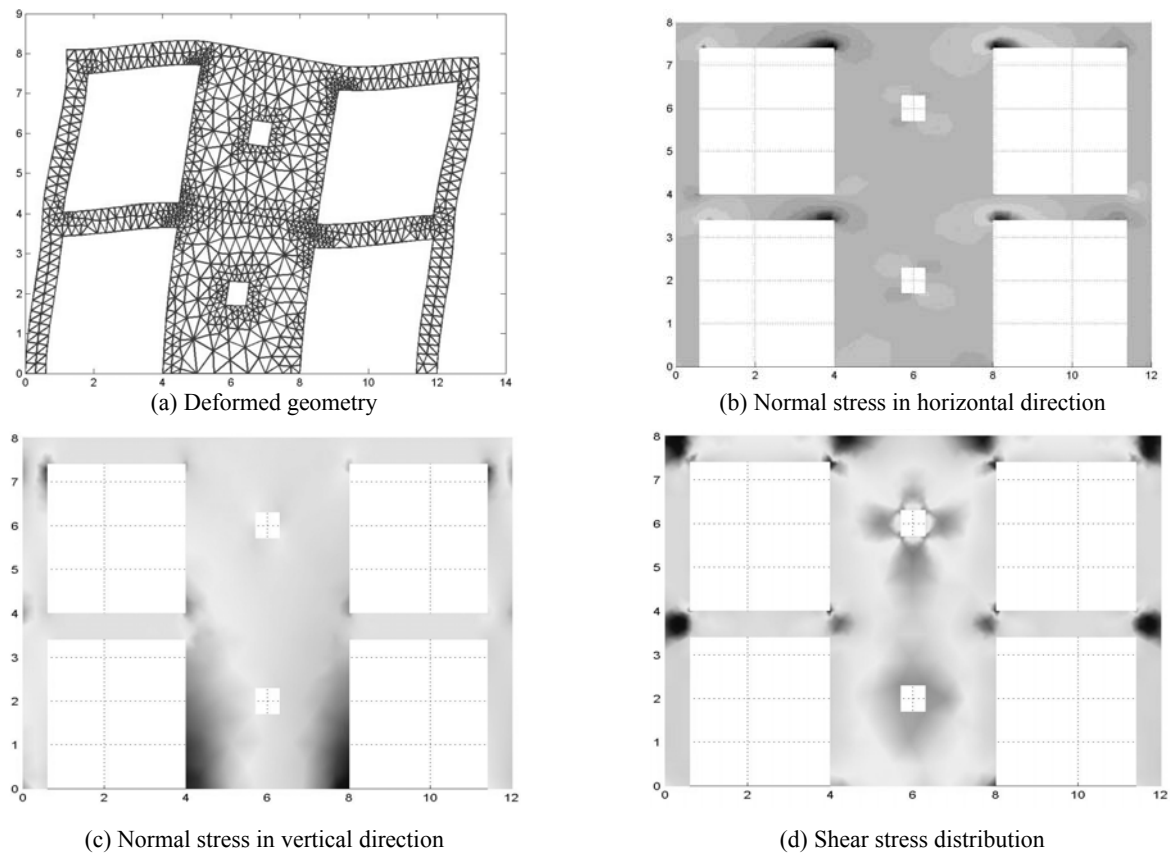


Figure 8. Graphical results after analysis.

ACKNOWLEDGMENT

I am grateful to Professor Tekin Gultop for his guidance and for many fruitful discussions.

REFERENCES

- [1] Rosman, R., "Approximate analysis of shear walls subject to lateral loads", *Journal of American Concrete Institute*, 61, No 6, 711-733, 1964.
- [2] Coull, A., Choudhury, J. R., "Analysis of coupled shear walls", *Journal of American Concrete Institute*, 64, 587-593, 1967.
- [3] Stafford Smith, B., Abergel, D. P., "Approximate analysis of high-rise structures comprising coupled walls and shear walls", *Building and Environment*, 18, No 1/2, 91-96, 1983.
- [4] Stafford Smith, B., Girgis, A., "Simple analogous frames for shear wall analysis", *ASCE Journal of Structural Engineering*, 110, 2655-2666, 1984.
- [5] Chan, H.C., Cheung, Y.K., "Analysis of shear walls using higher order finite elements", *Building and Environment*, 14, 217-224, 1979.
- [6] Ozturun, N.K., Citipitioglu, E., Akkas, N., "Three-dimensional finite element analysis of shear wall buildings", *Computes and Structures*, 68, 41-55, 1998.
- [7] Kim, H.S., Lee, D.G., "Analysis of shear wall with openings using super elements", *Engineering Structures*, 25, 981-991, 2003.
- [8] Suhura, J., Fukuda, J., "Automatic mesh generation for finite element analysis", advances in computational methods in structural mechanics and design, *2nd U.S.-Japan seminar on matrix methods of structural analysis and design*, UAH Press, Huntsville, Alabama, 607-624, 1972.
- [9] Shaw, R.D., Pitchen, R.G., "Modification to the Suhura-Fukuda method of network generation", *International Journal For Numerical Methods In Engineering*, 12, 93-99, 1978.

- [10] Lo, S.H., “A new mesh generation scheme for arbitrary planar domains”, *International Journal For Numerical Methods In Engineering*, 21, 1403-1426, 1985.
- [11] Zhu, Z.Q., Wang, P., Tuo, S.F., Liu, Z., “A structured/unstructured grid generation method and its application”, *Acta Mechanica*, 167, 197–211, 2004.
- [12] Löhner, R., “Extensions and improvements of the advancing front grid generation technique”, *Communications In Numerical Methods In Engineering*, 12,683-702, 1996.
- [13] Joe, B., Simpson, R.B., “Triangular meshes for regions of complicated shape”, *International Journal For Numerical Methods In Engineering*, 23, 751-778, 1986.
- [14] Zienkiewicz, O.C., Phillips, D.V., “An automatic mesh generation scheme for plane and curved surfaces by isoparametric co-ordinate”, *International Journal For Numerical Methods In Engineering*, 3, 519-528, 1971.
- [15] Watson, D.F., “Computing the n-dimensional Delaunay tessellation with application to Voronoi polytopes”, *The Computer Journal*, 24, 167-172, 1981.
- [16] Bowyer, A., “Computing Dirichlet tessellations”, *The Computer Journal*, 24 (2), 162 – 166, 1981.
- [17] Bern, M., Plassmann, P., “Handbook of Computational Geometry Edited by J.R. Sack and J. Urritia”, *Elsevier Science*, 303-308, 2000.
- [18] Du, C., “A note on finding nearest neighbors and constructing Delaunay triangulation in the plane”, *Communications in Numerical Methods in Engineering*, 14, 871-877, 1998.
- [19] Sibson, R., “Locally equiangular triangulations”, *The Computer Journal*, 21(3), 243-245, 1978.
- [20] Tam, A., Ait-Ali-Yahi, D., Robichaud, M.P., Moore, M., Kozel, V., Habashi, W.G., “Anisotropic mesh adaptation for 3D flows on structured and unstructured grids”, *Computer Methods in Applied Mechanics and Engineering*, 189, 1205-1230, 2000.
- [21] Hyun, S., Lindgren, L. E., “Smoothing and adaptive remeshing schemes for graded element”, *Communications in Numerical Methods in Engineering*, 17 (1), 1-17, 2001.