Athens Institute for Education and Research
ATINER

ATINER's Conference Paper Series
CIV2015-1752

Assessment of Mesh Size Refinement Influence on FEM Solution of Shear Wall Structural Systems

Enkeleda Kokona
PhD Student
Polytechnic University of Tirana
Albania

Helidon Kokona
PhD Student
Institute of Earthquake Engineering and Engineering Seismology
IZIIS
FYROM

Altin Bidaj
Lecturer
Polytechnic University of Tirana
Albania

An Introduction to ATINER's Conference Paper Series

ATINER started to publish this conference papers series in 2012. It includes only the papers submitted for publication after they were presented at one of the conferences organized by our Institute every year. This paper has been peer reviewed by at least two academic members of ATINER.

Dr. Gregory T. Papanikos
President
Athens Institute for Education and Research

This paper should be cited as follows:

Kokona, E., Kokona, H. and Bidaj, A. (2015). **"Assessment of Mesh Size Refinement Influence on FEM Solution of Shear Wall Structural Systems",** Athens: ATINER'S Conference Paper Series, No: **CIV2015-1752.**

Athens Institute for Education and Research
8 Valaoritou Street, Kolonaki, 10671 Athens, Greece
Tel: + 30 210 3634210 Fax: + 30 210 3634209 Email: info@atiner.gr URL: www.atiner.gr
URL Conference Papers Series: www.atiner.gr/papers.htm
Printed in Athens, Greece by the Athens Institute for Education and Research.
ISSN: 2241-2891
14/12/2015

Assessment of Mesh Size Refinement Influence on FEM Solution of Shear Wall Structural Systems

Enkeleda Kokona

Helidon Kokona

Altin Bidaj

Abstract

This paper presents FEM modelling through a structural analyses code of a Reinforced Concrete Structure "4ever green" tower. The structure is located in the center of Tirana, the capital of Albania. The tower has 6 levels underground, (pit depth 26 m) and 24 levels above ground (height 95 m). The structural system applied is reinforced concrete, composed of coupled walls located in perimeter line and staircase shafts. So, the vertical and lateral forces are fully resisted by shear walls. From a structural point of view it's necessary to develop a structural model in different types of mesh refinement to achieve better results, avoiding the solution errors on the stress-strain and deformation state over the structural elements. FEM detailed description is not the goal of this paper. The principal goal of this paper is to present case studies with respective results that help to achieve realistic structural behaviour directly connected to themesh refinement applied. Quadrilateral displacement elements meshing as practical and accurate procedures are used. After the computational solution, forces and displacements analyses results are presented for three levels of mesh densities. As it is known the basic idea that the FEM solution of a real problem is replacing it by a simpler one, we are able to find only an approximate and not exact solution. Thus in the absence of the exact solution that defines theoretically an asymptote line we can only improve or refine the FEM solution by spending a more computational effort. Small differences in analyses results in between solutions for two last consecutive mesh densities, yields practically asymptote line, accepted as a satisfactory solution.

Keywords: Asymptote, mesh, refinement, reinforced concrete, shear wall

Acknowledgments: Our thanks to colleagues of “aei progetti” structural design studio, Ing. Niccolò De Robertis, Ing. Stefano Valentini for their helpful collaboration and support to achive the optimal solution of “4 Ever Green" Tower, Tirana.

Introduction

The behaviour of a real structure depends upon the geometry, the property of the material used, the boundary and the initial and loading conditions. In general, it is very difficult to solve the differential equations of the structural model [1].

In practice, the problems are solved using numerical methods. The methods of model discretization like the FEM are popular, due to its practicality. The main principle of FEM is the modeling of real physical structure as an assemblage of individual elements. In structural modeling there are three basic element types: displacement elements, equilibrium elements and hybrid elements [2]. The software package SAP2000 used for the structure presented in this paper employs displacement elements.

The computational modeling using the FEM consists of four steps:

- Modeling of the geometry.
- Meshing (discretization).
- Specification of material property.
- Specification of boundary, initial and loading conditions.

Modeling of the Geometry

Real complex structures have to be reduced by simpler geometry. The geometry is eventually presented by a collection of elements at different shapes, approximated by straight lines or flat surfaces.

The accuracy of representation is controlled by the number of elements interconnected. It is obvious that with more elements, the representation would be more accurate. Because of the constraints on computational efforts, it is always recommended to limit the number of elements.

Compromises are usually made in order to decide a suitable number of elements. Hence, fine geometry needs to be modeled only if very accurate results are required in specific regions. The engineers have to interpret the results having in mind these geometric approximations.

There are many ways to create a suitable geometry in the computer for the FE mesh. Points called nodes are specified by their coordinates. Lines are specified connecting the points or nodes. Surfaces are specified by connecting, rotating or translating lines; and solids can be specified by connecting, rotating or translating surfaces.

Graphic interfaces are used to provide the structural geometry. There are software packages which can significantly save time creating the model of structural geometry.

Engineering experience and judgment are very important in modeling the geometry of a system, using necessary simplifications required. For example, a physical plate geometrically has three dimensions but a mathematical model is presented in two dimensions.

It is similar in shells presented by a two-dimensional flat surface. Shell elements are used in meshing surfaces.

A physical beam geometrically has also three dimensions. The beam is presented mathematically in one dimension, so the mathematical model is a one-dimensional line. Beam elements are used to mesh the lines of models.

Meshing

Meshing is used to discretize the geometry into small pieces called elements. The solution for a structural problem is complex, using a variety of functions over the whole geometry of the problem.

The geometry can be divided (meshed) into subdivisions or elements using grids or nodes.

The solution within an element can be assumed using suitable functions such as polynomials. The solutions assumed for all elements, simple from a computational point of view satisfying convergence requirements, form the solution over the whole geometry of the structure.

The element connectivity information given along with meshing will be used later in the FEM equations.

Generally the solution is taken in polynomial form.

Triangulation is the most flexible way to create meshes. It can be made for two-dimensional (2D) planes, and even three-dimensional (3D) spaces. It is commonly used as a meshing type in most of the pre-processors. The advantage of triangulation is the flexible modeling of complex geometry and boundaries as well.

Quadrilateral element meshing is more difficult and more accurate way to create meshes.

Specification of Material Properties

The property of materials used in the structure can be defined for a group of elements or individually. Different sets of material properties like Young's modulus, shear modulus etc, are required for the FEM analysis of structures.

Boundaries, Initial and Loading Conditions

Boundaries, initial and loading conditions are decisive in FEM solutions. Those conditions are easily defined using commercial pre-processors, through a graphic interface. It can be specified to the geometrical identities (points, lines, surfaces, solids), or to the elements as well.

Engineering experience and judgment is required to correctly define the boundaries, initial and loading conditions for structural systems.

Mesh Refinement Techniques

The mesh defines the shape of an element and the shape is significant for an element to produce accurate stress levels within the structure.

Finite element analysis would proceed starting from the selection of a mesh. Experience is practically the only way in determining whether or not the mesh is optimal for the analysis [2].

The mesh needs to be graded in areas of importance, such as zones of stress concentration, rapid change in stress in a specific direction, corners and holes, change in material properties etc.

Element Distortion

Usually it is not possible to have always regular shaped elements for variational geometries. Irregular or distorted elements are acceptable in the FEM, but there are limitations. So, it needs to control the degree of element distortion in the process of mesh generation [4], [6].

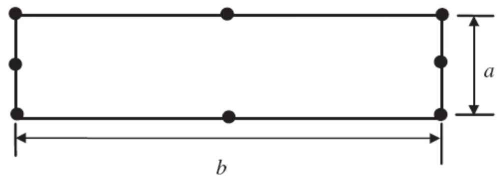
The distortions are measured regarding to the basic shape of the element, which are

• Quadrilateral elements
• Triangle elements
• Hexahedron elements
• Tetrahedron elements

Five possible forms of element distortions and their rough limits are as follows:

1. Aspect ratio distortion (elongation of element), (Figure 1).

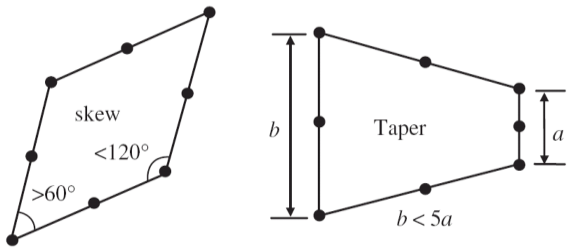
Figure 1. *Aspect Distortion*



(b/a)≤3 for stress analysis, (b/a)≤10 for displacement analysis

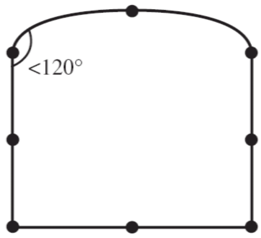
2. Angular distortion of the element (Figure 2), where any included angle between edges (skew and taper).

Figure 2. *Angular Distortion*



3. Curvature distortion of element (Figure 3), where the straight edges from the element are distorted into curves when matching the nodes to the geometric points.

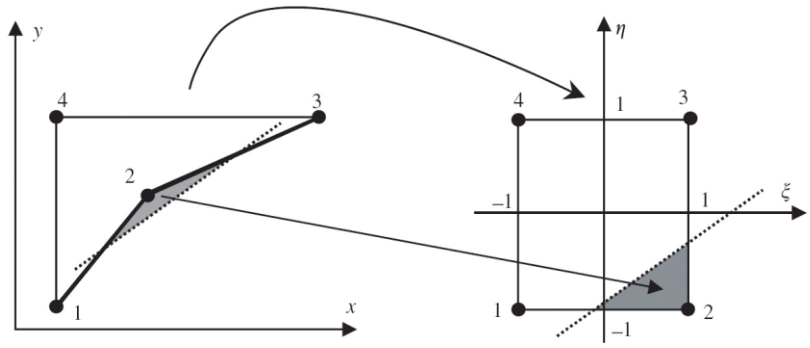
Figure 3. *Curvature Distortion*



4. Volumetric distortion occurs in concave elements. A mapping is performed in order to transfer the irregular shape of the element in the physical coordinate system into a regular one in the non-dimensional natural coordinate system.

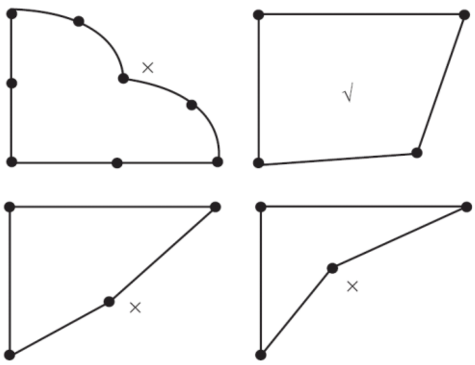
For concave elements, there are areas outside the elements (shadowed area in Figure 4) that will be transformed into an internal area in the natural coordinate system. The element volume integration for the shadowed area based on the natural coordinate system will result in a negative value.

Figure 4. Mapping of an outside area of the physical element into an interior area in the natural coordinates.



A few unacceptable shapes of quadrilateral elements are shown in Figure 5.

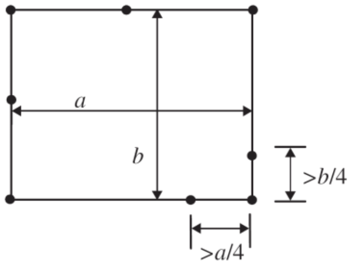
Figure 5. *Unacceptable Shapes of Quadrilateral Elements*



5. Mid-node position distortion occurs with higher order elements where there are mid nodes. The mid node should be placed close to the middle

of the element edge. The limit for mid-node displacement away from the middle edge of the element is a quarter of the element edge, as shown in Figure 6. The reason is that this shifting of mid nodes can result in a singular stress field in the elements.

Figure 6. *The Limit for Mid-node Displacing Away from the Middle Edge of the Element*



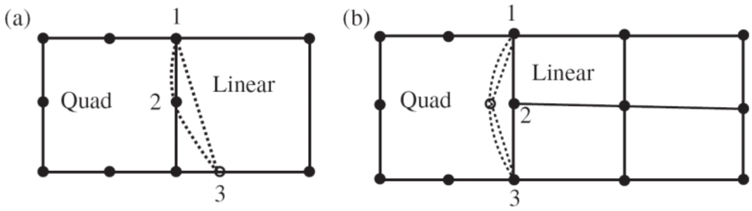
Mesh Compatibility

A mesh is said to be compatible if the displacements are continuous along all edges between all the elements in the mesh. The use of different types of elements in the same mesh or improper connection of elements can result in an incompatible mesh [10].

Different Order of Elements

Mesh incompatibility issues can arise when we have a transition between different mesh densities, or when we have meshes comprised of different element types. When a quadratic element is joined with one or more linear elements, as shown in Figure 7, incompatibility arises from the difference in the orders of shape functions used [12]. The eight-node quadratic element in Figure 7 has a quadratic shape function, which implies that the deformation along the edge follows a quadratic function. Also, the linear shape function used in the four-node linear element in Figure 7 will result in a linear deformation along each element edge.

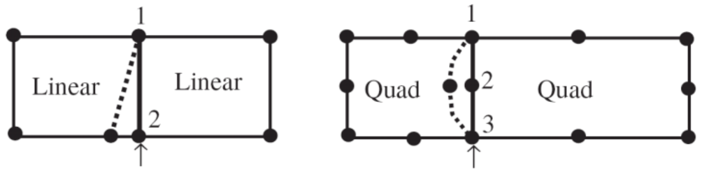
Figure 7. *Incompatible Mesh Caused by the Different Shape Functions. (a) A Quadratic Element Connected to One Linear Element; (b) A Quadratic Element Connected to Two Linear Elements*



Solutions for an incompatible mesh are:

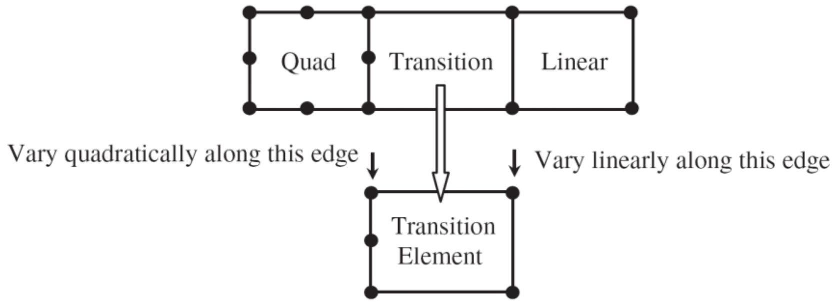
1. Use the same type of elements throughout the entire model. This is the simplest solution and it is a usual practice, as complete compatibility is automatically satisfied if the same elements are used as in Figure 8.

Figure 8. *Elements of the Same Type with Complete Edge-to-Edge Connection Ensures Mesh Compatibility*



2. When elements of different orders of shape functions have to be used for some reason, such as in p-adaptive analysis, use transition elements whose shape functions have different orders on different edges. An example of a transition element is shown in Figure 9. The five-node element shown can behave in a quadratic fashion on the left edge and linearly on the other edges. In this way, the compatibility of the mesh can be guaranteed.

Figure 9. *Five Nodes Transition Element Used to Connect Linear and Quadratic Wlements to Ensure Mesh Compatibility*

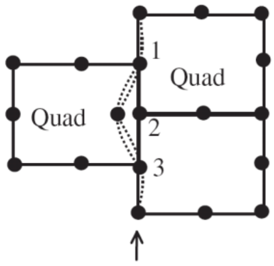


3. Another method used to enforce mesh compatibility is to use multipoint constraints (MPC) equations. MPCs can be used to enforce compatibility for the cases shown in Figure 7(a).

Straddling Elements

Straddling elements can also result in mesh incompatibility, as illustrated in Figure 10. Although the shape functions order of these connected elements is the same, the straddling can result in an incompatible deformation of edges 1-2, and 2-3, shown by dotted lines in Figure 10. This is because in the assembly of elements, the FEM requires only the continuity of the displacements (not the derivatives) at nodes between elements. The method for fixing the problem of the mesh incompatibility is to avoid straddling elements in the mesh [14].

Figure 10. *Incompatible Mesh Caused by Straddling along the Common Edge of the Same Order of Elements*



Meshing Accuracy Ladder

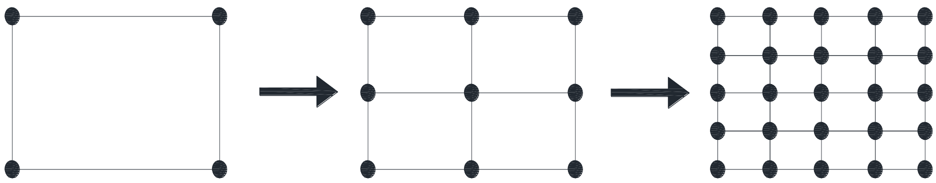
The solution accuracy requires the generation mesh refining, coarsening, relocating, or adjusting locally polynomial degree.

The computation starts with a coarse mesh solution with a low order of polynomial degree [1],[8]. If accuracy isn't achieved, the following adaptive procedures can be used:

• Local refinement and/or coarsening of a mesh (h-refinement)

The easy manner to move up the accuracy ladder is to employ finite element codes that automatically increase the number of elements used in an analysis. Increasing the number of elements within a model without changing the order of the polynomial used to approximate the displacements within the element automatically is known as h-adaption. This adaption process is illustrated in Figure 11, where a 2-D 4-noded membrane element is used.

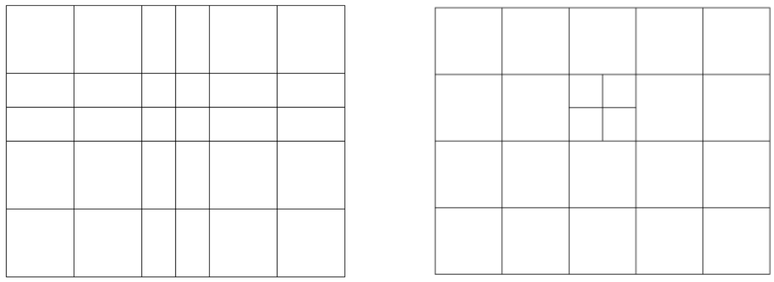
Figure 11. *h-refinement*



The usual way of avoiding the excess h-refinement is to introduce irregular nodes where the edges of a refined element meet at the midsides of a coarser one, Figure 12.

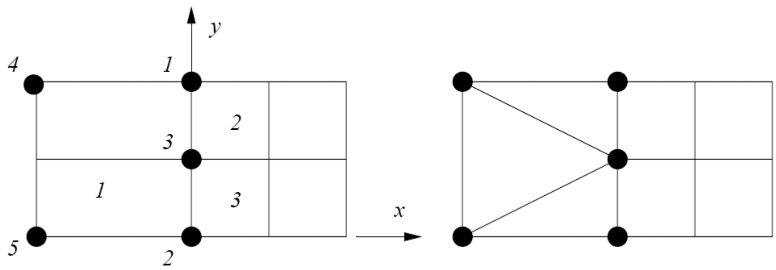
The way to retain continuity at irregular nodes is to restrict displacement for irregular node i, $u_i(x_i,y_i)$ constructing shape functions on each element [11]. The difficulties arise when there are too many irregular nodes on an edge. To overcome this, typically we use "one irregular" and "three neighbor"rules. The "one irregular" rule limits the number of irregular nodes on an element edge to one. The "three neighbor" rule states that any element having irregular nodes on three of its four edges must be refined [13].

Figure 12. *Bisection of an Element Mesh (Left). Mesh Lines Removed by Creating Irregular Nodes (Right)*



Irregular nodes can be avoided by using transition elements as shown in Figure 13. On the right are used triangular elements as a transition between the coarse and fine elements. If triangular elements are not desirable the transition element on the left uses rectangles but only adds a mid-edge shape functions at Node 3.

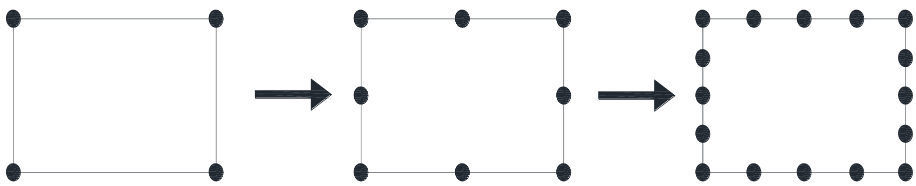
Figure 13. *Transition Elements between Coarse and Fine Elements Using Rectangles (Left) and Triangles (Right)*



• Locally varying the polynomial degree (p-refinement)

An alternative to employing more elements is to move up the accuracy ladder by increasing the order of the polynomial used within the element to model the displacement field. This process is known as p-refinement. The number of nodes per element increases, with the same number of elements. This is demonstrated using a simple 2-D model in Figure 14.

Figure 14. *p-refinement*



• relocating or moving a mesh (r-refinement)

The r-refinement is not capable of finding an accurate solution. If the mesh is too coarse it might be impossible to achieve a high degree of precision.

The above procedures can be used separately or in combination [4]. So far, h-refinement is the most popular. It can increase the convergence rate particularly when singularities are present. The p-refinement is most natural with a hierarchical basis, since parts of the stiffness, mass matrices and load vector will remain unchanged while increasing the polynomial degree [5].

Mesh Refinement Influence in Practical Solution

"4 Ever Green" Tower General Description

The tower structure is 95 m high above the ground (24 floors) with a development in depth down -26.0 m from ground level (6 levels).

The foundation slab of 2.40m thickness is nearly a rectangular shape with longitudinal sides equal to 39m and transversal side variable from 27m to 28.6m. The underground levels of 2.8m inter-storey height are reinforced concrete slabs of 30cm thickness, reduced to 24cm at ducts and pipelines holes passing through slabs, vertical elements and RC walls and columns of 40cm thick.

The underground levels from the sixth to the second floor are intended mainly for parking, on the sixth floor there are also technical spaces intended for sanitary and firefighting water tanks. The entrance to the parking area is realized through the RC ramp that goes down from the ground floor to the second basement level.

At the first underground level it is a rigid basement of 1.4m thickness for the tower vertical structure.

The upper levels of the structure have inscribed a variation edges shape, starting from 26mx22m at the tower base, to 30mx26m on the top. In the center of the first four levels above the ground circular openings, to allow the installation of escalators, are provided.

The fifth level is intended for conference room venue. The inter-storey height of these levels is 5m.

Starting from the sixth level +25.07m, up to the top level +95.00m, the inter-storey height 3.5m remains constant. The destinations for levels, from six to ten are offices, from eleven to twenty are hotels, the twenty-first and twenty-second, residential.

In the center of the floor slabs, free span dimensions are 13mx13m. There are no vertical elements supporting the loads of stairs that are applied on the perimeter walls of the tower through the slab, the thickness of which is equal to 40cm. The slabs are made of reinforced concrete lightened, using polyethylene hollow spheres in high density.

The perimeter walls of the tower have tapered thickness with the height and vary from 40cm at the base to 25cm at the top. The walls of the elevator cores and stair shafts have a constant thickness of 40cm throughout the height of the tower.

"4 Ever Green" Tower Structural Modelling, Meshing and Analyses Results

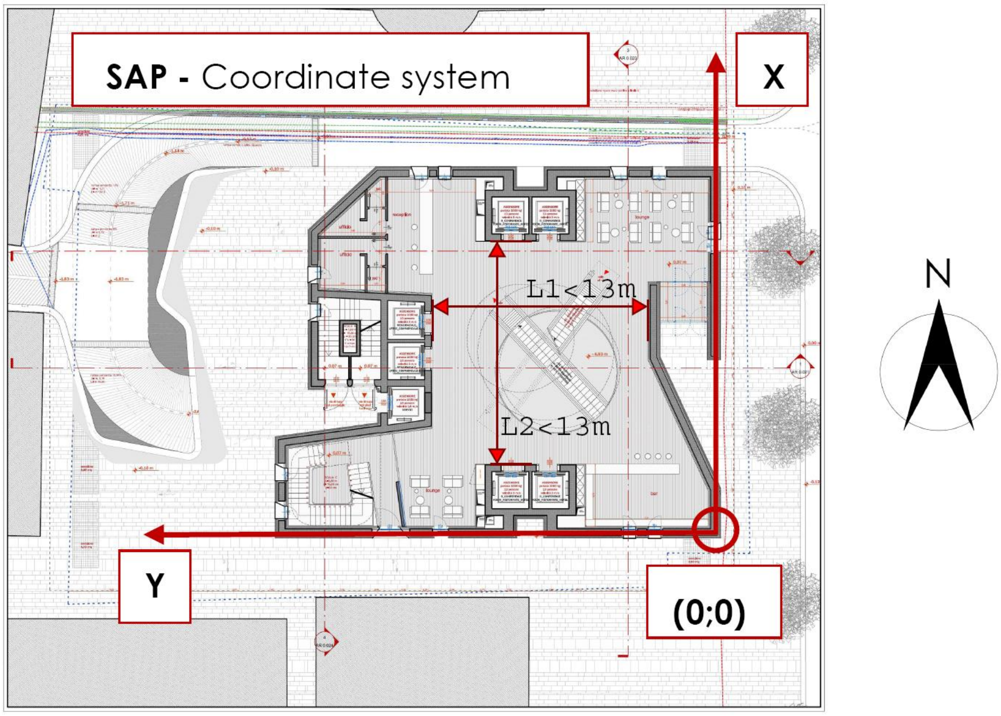
A structural analysis is performed using finite element modeling, using the computer program SAP 2000. Some of the major assumptions used in FEM modeling are presented below:

FEM model refers to three dimensional right-handed rectangular coordinate systems, (Figure 15)

Coordinate directions are:

-X axis in the transversal (minor) direction of the structure;
-Y axis in the longitudinal (major) direction of the structure;
- Z axis in the vertical direction, with the positive direction upward.

Figure 15. *Coordinate System of FEM*



The structure is modeled using two types of finite elements:
- "frame" a two-node linear element, for the modeling of beams and columns;
- "shell" a three- or four-node planar element mix in membrane and plate-bending behavior for the modeling of walls and slabs [3].

The local coordinate systems for the elements "shell" are arranged as follows:

- floor slabs, elements are in the global plan (X-Y),
the local 1 axis is parallel to global X, the local 2 axis is parallel to global Y and local 3 axis is arranged in the vertical global Z.
- walls, elements in the global plane (X-Z)
the local 1 axis is parallel to global X, the local 2 axis is parallel to global Z.
- walls, elements in the global plane (Y-Z)

the local 1 axis is parallel to global Y, the local 2 axis is parallel to global Z. The weight of the elements modeled is automatically computed by program (SelfWtMult = 1); Vertical loads due to imposed and permanent actions are modeled as distributed loads on the elements.

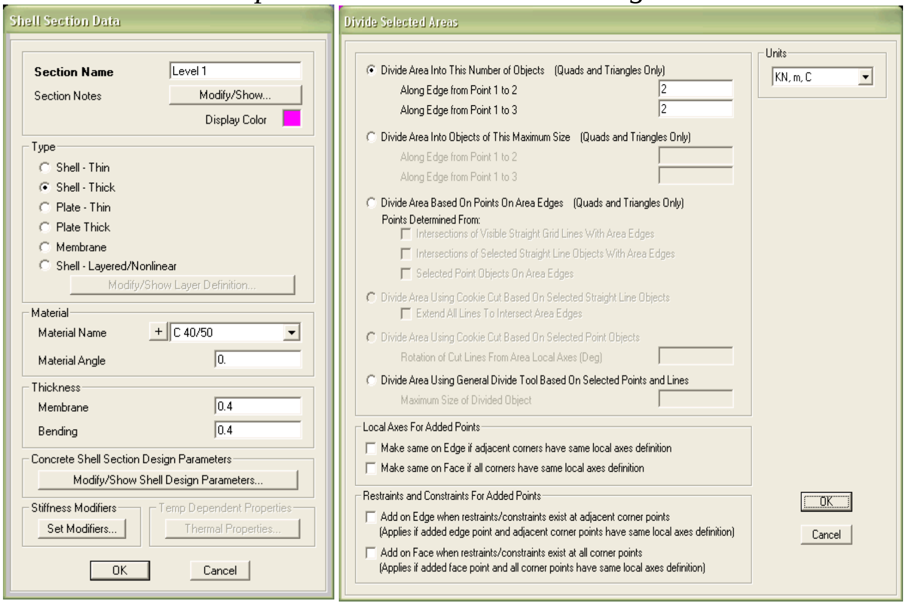
All characteristic values of imposed loads are applied according to EN 1991-1-1. Load combination actions are applied according to EN 1990, prEN

1998-1. Material properties and partial factors for materials are applied according to prEN 1992-1-1.

The units used in the model are kN and meter.

Material properties definition and shell element meshing for perimetral RC walls at Level 1 are shown according to SAP 2000 window, (Figure 16):

Figure 16. *Material Properties Shell Element Meshing*



A four-point numerical integration is used for the shell elements.

Internal forces, moments and stresses, in the element local coordinate system, are evaluated at the 2-by-2 Gauss integration points and extrapolated to the joints of the element [9].

An error in the element stresses or internal forces can be estimated from the difference calculated from different elements attached to a common joint. This indicates the accuracy of a finite-element approximation selected that can be used as the basis for the new selection of more refined finite element mesh.

Mindlin/Reissner (thick-plate) formulation is used to include the effects of transverse shear deformation on shell elements [7].

Using h-refinement for 3D tower structure, (Figure 17) at different mesh densities 1x1, 2x2 and 4x4, (Figure 18), the internal forces F22 for Level L1, resulting from earthquake action Ex_Dynamic Load, are presented in (Figures 19, 20, 21), respectively.

Figure 17. *FEM Modelling 3D View*

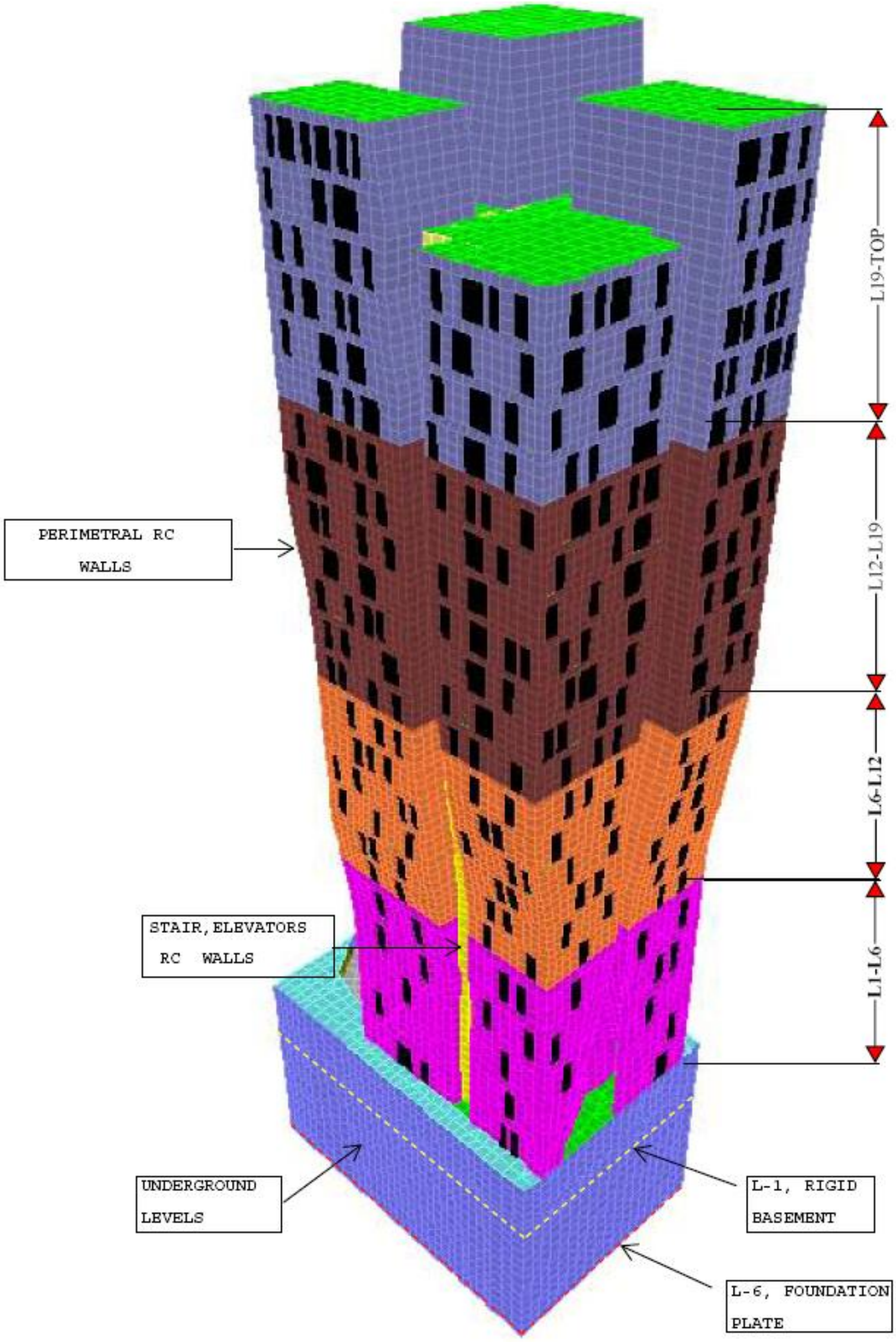
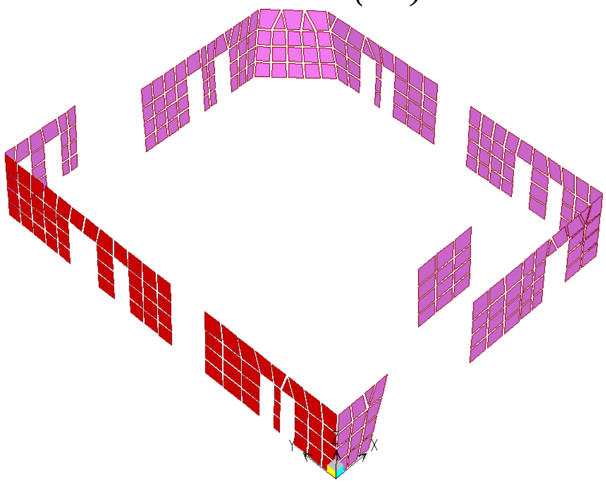
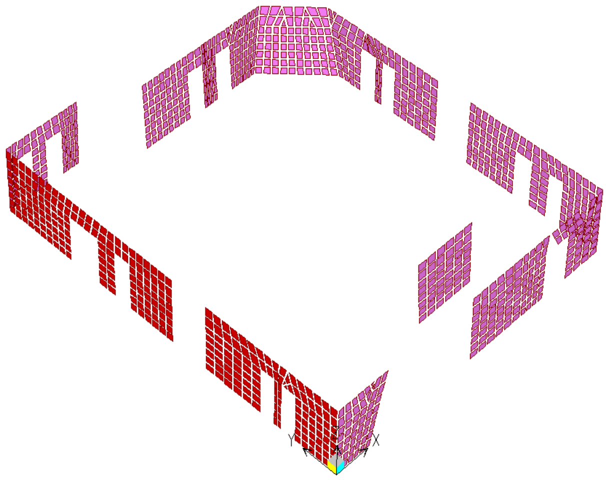


Figure 18. *Mesh Densities*

Coarse Mesh (1x1)



Medium Mesh (2x2)



Fine Mesh (4x4)

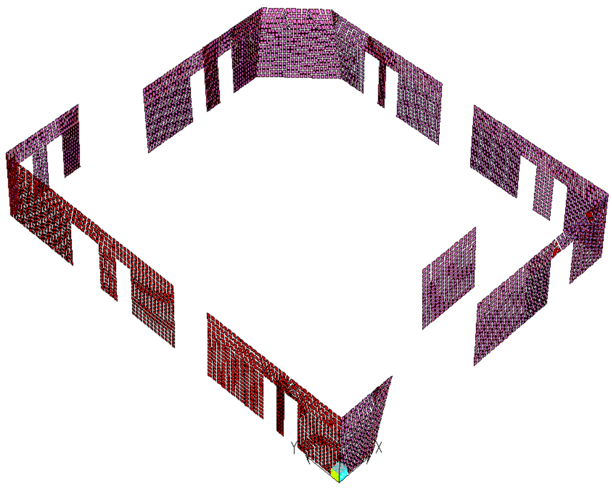


Figure 19. *Element Forces F22, Ex_Dynamic Load, Mesh Density (1x1)*

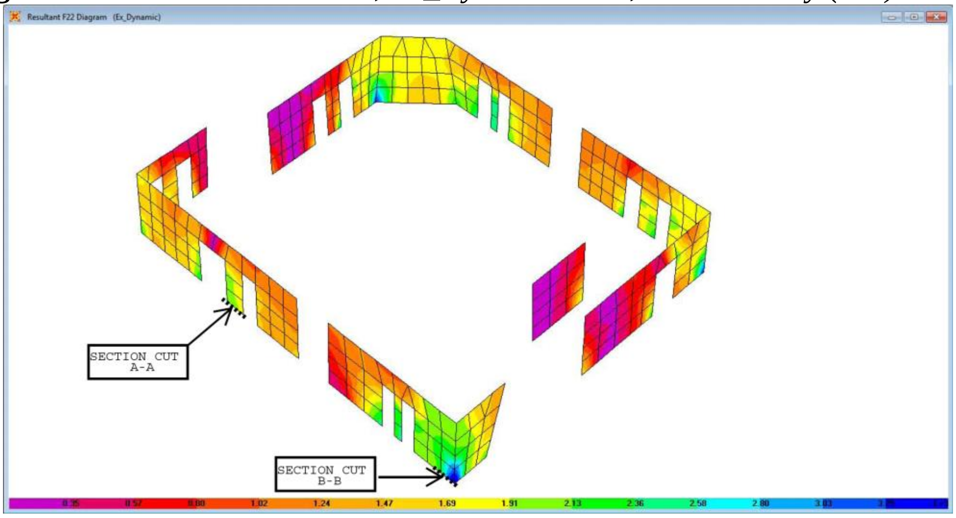


Figure 20. *Element Forces F22, Ex_Dynamic Load, Mesh Density (2x2)*

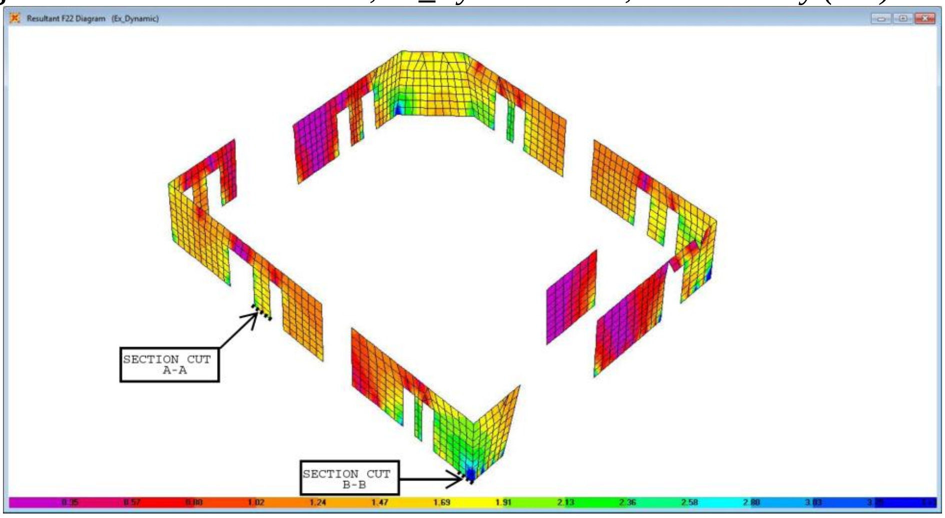
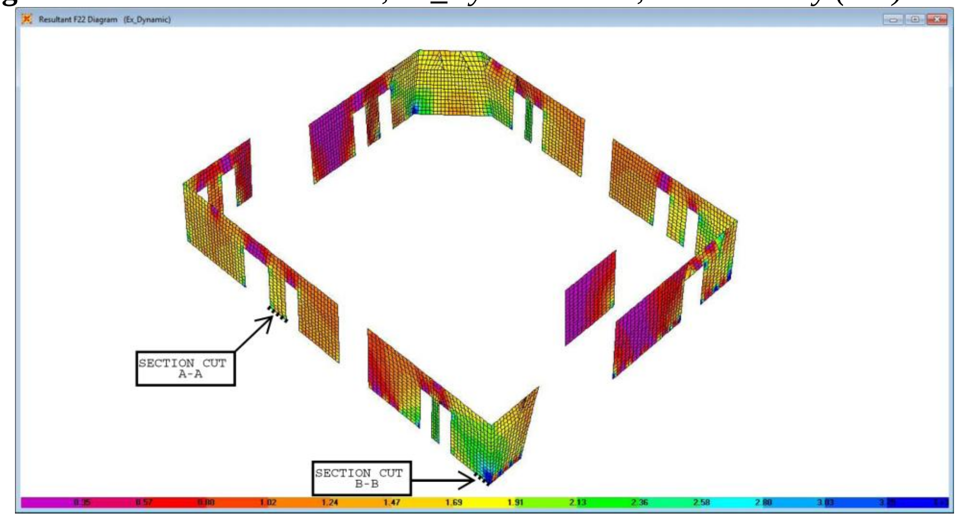


Figure 21. *Element Forces F22, Ex_Dynamic Load, Mesh Density (4x4)*



Convergency diagrams corresponding to different mesh densities, of internal forces F2, F3 for two sections cuts A-A and B-B, resulting from earthquake action Ex_Dynamic Load, Ey_Dynamic Load are presented in (Figures 22, 23).

Figure 22. *Convergency Diagram Sec. A-A*

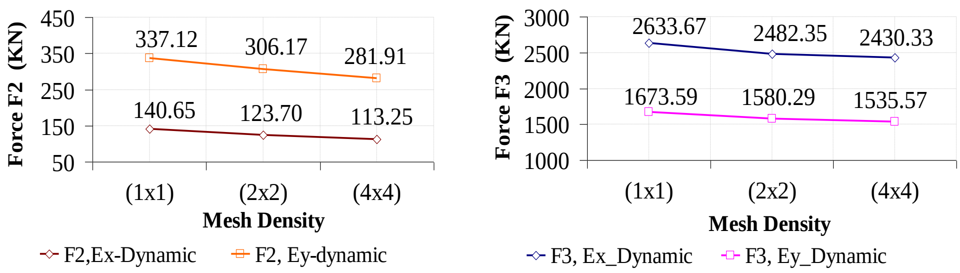
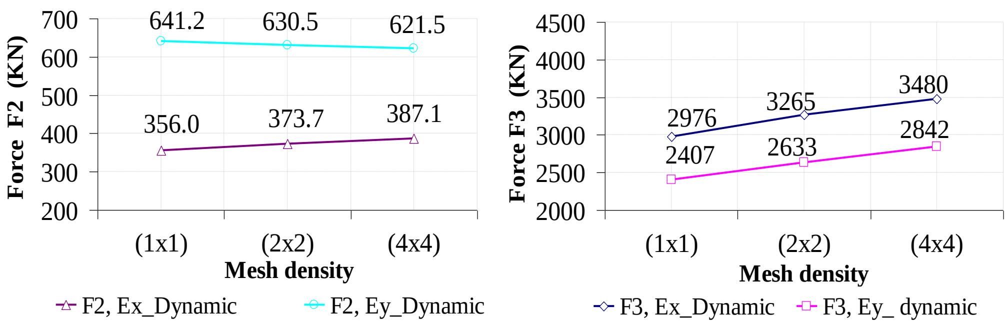
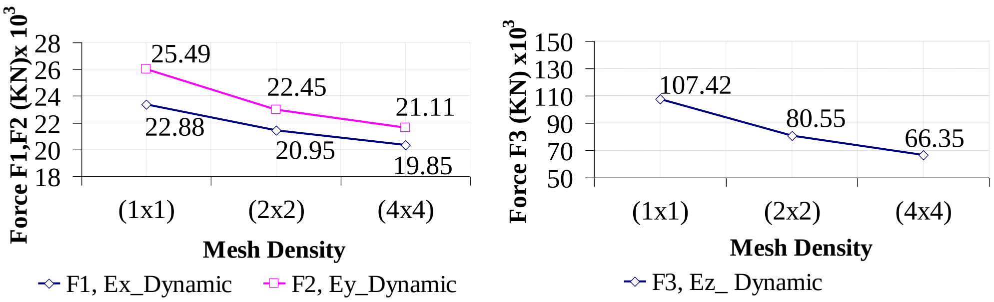


Figure 23. *Convergency Diagram Sec B-B*



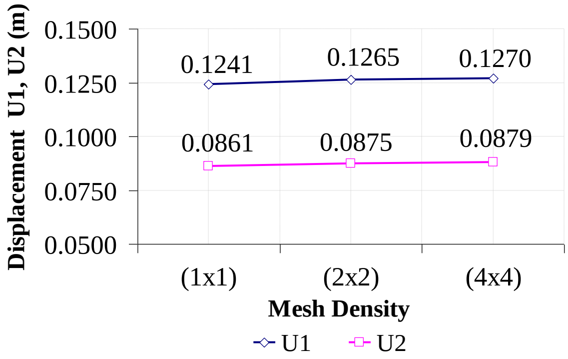
The convergency diagram corresponding to different mesh densities, of base shear (F1,F2), resulting from earthquake action Ex_Dynamic Load, Ey_Dynamic Load and base F3 resulting from earthquake action Ez_Dynamic Load is presented in (Figure 24).

Figure 24. *Convergency Diagram, Base Shear (F1,F2), Base F3*



The convergency diagram corresponding to different mesh densities, of top displacements U1,U2, resulting from earthquake action Ex_Dynamic Load, Ey_Dynamic Load is presented in (Figure 25).

Figure 25. *Convergency Diagram, Top Displacements*



From the convergency diagrams, it is found that numerical results in between two last mesh densities give small variations despite significant mesh density changes. That means, spending more computational efforts of further mesh refinement it is not effective.

Conclusions

The different meshes are created using the SAP2000 structural analysis program. Well shaped quadrilateral and triangular shell elements are graded from original mesh density coarse 1x1, to medium 2x2 and fine 4x4 according to h-refinement procedure.

- The h-refinement procedure used in FEM solution produce accurate results at a monotonic convergency. Mesh refinement 4x4 obtained by a subdivision of existing elements exploits full limits of computer capacity.
- Internal element forces converge smoothly to accurate results, increasing mesh density from coarse 1x1 to fine 4x4.
- The shape of an element has a significant impact on its ability to produce accurate element forces within the structure. The quadrilateral shape elements obtain more accurate results than triangular elements.
- In order to obtain accurate results sudden changes in the shell element size must be avoided.
- The refinement mesh should be applied to structural parts where rapid change of internal forces and material properties is expected.
- Mesh refinement it is a very important task of the pre-processing. It can be a very time consuming task but an experienced engineer will produce a more credible mesh for a complex problem. The structural model has to be meshed properly into elements of specific shapes such as quadrilaterals, triangles etc., using as many advantages of automated mesh generators as possible.
- Because there is not a priori method of an efficient finite element model that insures a specified degree of accuracy, numerical tests of different

mesh refinement analyses can be used to assess the solution convergence.

References

[1] Alan Morris, *"A practical Guide to Reliable Finite Element Modelling"*, British Library Cataloguing in Publication Data, p. 143-200, England 2008.

[2] G. R. Liu and S. S. Quek, *"The Finite Element Method. A practical course",* Library of Congress Cataloguing in Publication Data, p 246-256, England 2003.

[3] Robert D. Cook, *"Finite Element Modeling for Stress Analysis"*, Library of Congress Cataloguing Data, p 138- 155, England 1995.

[4] Zhu, J. Z.; Hinton, E.; Zienkiewicz, C., *"Mesh enrichment against mesh regeneration using quadrilateral elements"***,** Communications in Numerical Methods in Engineering, v 9, n 7, Jul 1993, p. 547-554.

[5] Zhu, J. Z., Zienkiewicz, O. C., Hinton, E., and Wu, J., *"A New Approach to the Development of Automatic Quadrilateral Mesh Generation"*, IJNME, Vol. 32, p. 849-866, 1991.

[6] Lee, C. K. and Hobbs, R. E., "*Automatic Adaptive Finite Element Mesh Generation Over Arbitrary Two-dimensional Domain Using Advancing Front Technique"*, Computers and Structures, Vol.71, p. 9-34, 1999.

[7] Lee, C. K. and Hobbs, R. E., *"Automatic Adaptive Refinement for Plate Bending Problems Using Reissner-Mindlin Plate Bending Elements,"* International Journal for in Numerical Methods in Engineering, Vol. 41, p. 1-63, 1998.

[8] Diez, P. and Huerta, A., *"A Unified Approach to Remeshing Strategies for Finite Element h- Adaptivity,"* Comput. Methods Appl. Mech. Engrg., vol. 176, (1999), p. 215-229.

[9] Blacker, T. D. and Stevenson, M. B., *"Paving: A new approach to automatic quadrilateral meshing technique*," Advances in Engineering Software, 13(5-6):811-847, 1991.

[10] Owen, S. J., *"A Survey of Unstructured Mesh Generation Technology"*, The 7th International Meshing Roundtable, Dearborn, Michigan, October 26 - 28, 1998, p. 239-267.

[11] Zienkiewicz, O. C and Zhu, J. Z., *"A simple Error Estimator and Adaptive Procedure* for Practical Engineering Analysis," Int. J. Numer. Methods Eng., Vol. 24, p. 337-357, 1987.

[12] S. J. Owen, M. L. Staten, S. A. Canann and S. Saigal, *"Q -Morph: An Indirect Approach to Advancing Front Quad Meshing"*, Int. J. Numer. Meth. Engng., 44, 1317-1340 (1999).

[13] C. K. Lee and S. H. Lo, *"A New Scheme for the Generation of a Graded Quadrilateral Mesh"*, Comput. Struct., 52, 847-857 (1994).

[14] P. Kinney, *"Clean Up: Improving Quadrilateral Finite Element Meshes"*, Proc. 6th Int. Meshing Roundtable, 1995, 449-461.