ICSBE2016-252

APPLICATIONS OF FINITE ELEMENT METHOD IN STRUCTURAL ENGINEERING

S.M.C.M. Randiligama*, R.G.S.N. Jayathilake and K.K. Wijesundara

University of Peradeniya, Sri Lanka
**E-Mail: chathurangimadusha7 @ gmail.com, TP: +94775158479*

Abstract: It is a challenge for structural engineers to analyze complicated structural forms effectively using conventional analytical methods. Therefore, structural engineers prefer to use a commercial finite element software rather than using analytical methods. Many finite element software are based on the displacement based finite element method. As it is an approximate method, many drawbacks have been identified in applications in structural engineering due to misconceptions of users. Therefore, the objective of this study is to identify the limitations of using different finite elements to model structural components in buildings such as trusses, beams, slabs, foundations and their connections. To achieve the above objective, several case studies are selected and analysed using SAP2000 software. Based on the results, proper guidelines have been proposed for structural modelling. It is important to note that this study was limited to materials in linearly elastic behaviour.

Keywords Displacement based Finite Element Method; Guideline; Structural Engineering

1. Introduction

Many finite element software are based on the approximate method of analysis called the displacement based finite element procedure (DBFE). The misconception that the DBF provides exact answers could result in malpractices by structural engineers [3].

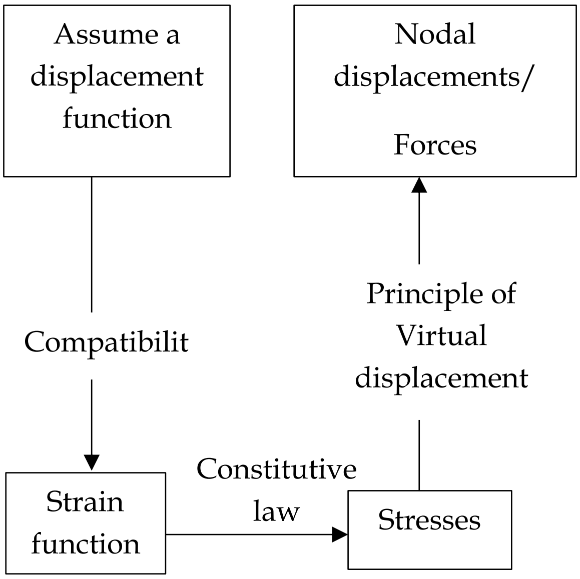

Figure 1: Main steps in DBFE procedure

Displacement based finite element method can present as a three step procedure as given in Figure 1. In the first step, a displacement polynomial function is assumed based on the independent degrees of freedom in an element and hence, strain function can be obtained by using compatibility relationships. In the second step, stresses can be obtained by using constitive law. In the final step of the procedure, the element stiffness matrix can be found by applying principle of virtual displacement.

Problems and limitations of finite element formulation of different elements had been identified when assuming a displacement function and when obtaining the stiffness matrix. Problems can be occurred due to the assumption of lower order displacement polynomial function and use of a numerical integration to obtain stiffness matrix [3].

Furthermore, analysing a structural system the different elements may have to be connected together. Elements are connected at the nodes. As number of degree of freedom having at a node varying from element to element, there may be problems occurred in connecting different element into one structure [2].

h-refinement and *p*–refinement are the two methods to minimize the errors in the displacement based finite element method [1]. The main objective of this study was to identify those drawbacks and propose guidelines to structural engineers by using *h*–refinement through case studies. This study was limited to the linearly elastic behaviour of material.

2. Selected case studies

As discussed before there are two methods to minimize the errors in displacement based finite element method. They are called *h*-refinement and *p*–refinement. *h*-refinement increases the accuracy of results by increasing the number of elements while *p*-refinement increases the accuracy of by changing the order of the displacement polynomial function. However, it is important to note that following case studies are selected only considering the *h*–refinement to improve the results.

2.1Case study 1: Modelling of truss elements

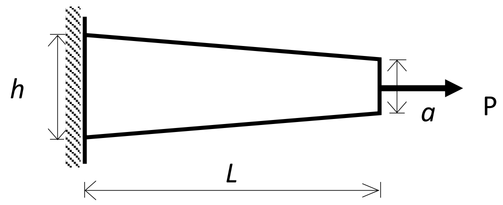

Figure 2: Elevation of doubly tapered truss

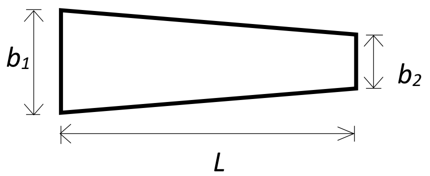

Figure 3: Plan view of doubly tapered truss

A doubly tapered truss with square cross section was considered in this case study. Height (*h*) and width (*b_1*) are remained in constant for all case studies. Height (a) and width (*b_2*) are varied from 0.05m to 0.15m in 0.025m intervals while length of the truss (*L*) is varied from 1m to 5m in 1m intervals resulting 25 case studies. They were analysed by restraining all the degrees of freedoms at one end and applying an axial force at the free end as shown in Figure 2.

2.2Case study 2: Modelling of frame elements

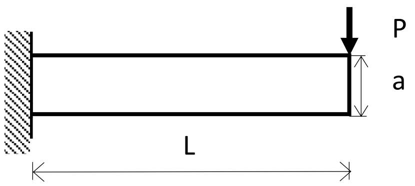

Figure 4: Elevation of cantilever prismatic beam

A prismatic beam with rectangular cross section was considered in this case study. Depth (*a*) is varied from 0.1m to 0.5m in 0.1m intervals while length of the truss (*L*) is varied from 1m to 5m in 1m intervals resulting 25 case studies. The width (*b*) of the cross section for all case studies is remained in constant. They were analysed by restraining all the degrees of freedoms at one end and applying a vertical force at the free end as shown in Figure 4.

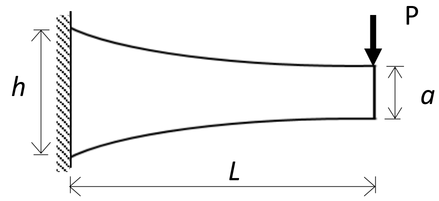

Figure 5: Elevation of cantiliver curved tapered beam

A curved tapered beam with rectangular cross section was considered in this case study. Height (*h*) and width (*b*) are remained in constant for all case studies. Height (*a*) is varied from 0.1m to 0.5m in 0.1m intervals while length of the truss (*L*) is varied from 1m to 5m in 1m intervals resulting 25 case studies. They were analysed by restraining all the degrees of freedoms at one end and applying an axial force at the free end as shown in Figure 5.

2.3Case study 3: Modelling of plate bending problem by using plate, shell and solid elements

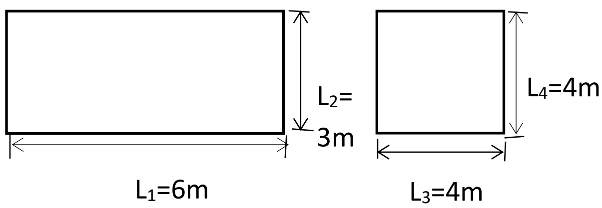

Figure 6: Pin supported one way and two way slabs

2.3.1 Effect of mesh size of the slab

A one-way and a two-way slabs shown in Figure 6 were modelled by using square mesh with the mesh size varying from 0.1 m to 2 m of shell and solid elements. All the lengths (L_i) and thicknesses of the slabs were constant. They were analysed for a uniformly distributed area load acting perpendicular to the plane of the slabs by restraining all the translational degrees of freedoms at boundaries. When the slabs were modelled by using solid elements, the restrained conditions were given at nodes in the centre of the slab thickness.

2.3.2 Effect of span/depth ratio of the slab

The one-way and the two-way slabs were modelled by using the converged square mesh size from the previous case study by varying the span-to-depth ratio from 2 to 40. The span-to-depth ratio is varied by only changing the thickness of the slab from 0.1m to 1.5m resulting 14 case studies. They were also analysed by restraining all the translational degrees of freedoms at boundary and applying a vertical uniformly distributed area load.

2.3.3 Effect of shear locking of the slab

For the same models of the slabs used in the section 2.3.1, the span-to-depth ratio was further increased up to 350 by varying the thickness of the slab. They were modelled separately by using thin shell, thick shell and solid elements and analysed for a uniformly distributed load acting perpendicular to the plane of the slabs by restraining all the translational degrees of freedoms at boundary.

2.3.4 Effect of distorted angle of the slab

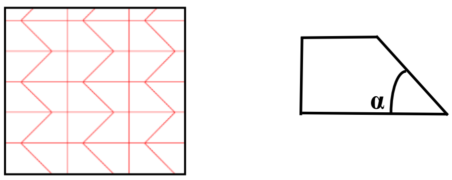

Figure 7: Mesh refinement by using distorted elements

For the same case study, effect of distorted angle was analysed by using distorted element with varying the angle of the element (α) from 10^0 to 90^0. All the translational degrees of freedoms at boundary were restrained and applied a vertical uniformly distributed area load.

2.3.5. Analysis of slab panels

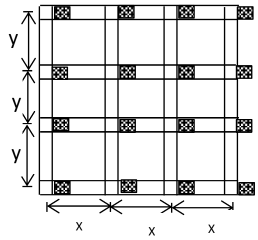

Figure 8: Plan view of slab panel

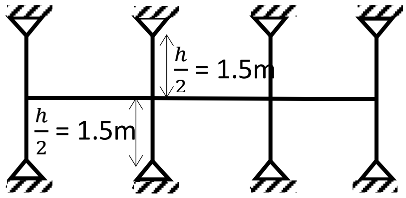

Figure 9: Sectional elevation of slab pan

One floor slab including interior, exterior and corner slabs was extracted from a three dimensional building model as shown in Figure 8. In the extracted model far end of the half height columns were assumed to be pin restrained as shown in Figure 9. A vertical uniformly distributed area load was applied. Frame and shell elements with and without insertion option were used for two models and solid elements were used for one model. Compare the results of two models with solid model.

2.4 Case study 4: Element connectivity problems

As discussed before, analysing a structural system the different elements may have to be connected together. As number of degree of freedom having at a node varying from element to element. Therefore some degrees of freedom are not restrained. To improve that connections following methods were used in that case studies.

2.4.1 Frame to solid connectivity

Column to foundation connection was considered in this case study. Foundation was modelled by using solid elements. And the column was modelled by using frame element. Then the frame element insert in to the solid mesh by layer and layer. Then, the horizontal force and vertical force were applied at the top of the column as shown in Figure 10.

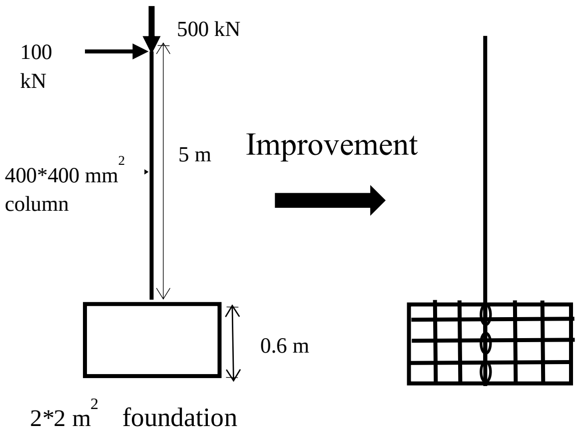

Figure 10: Frame to solid connection

2.4.2 Shell to solid connectivity

Retaining wall was considered in this case study. Wall was modelled by using shell elements and the foundation part was modelled by using solid elements. Then, the shell element insert in to the solid mesh by layer and layer. Then, the uniformly distributed area load was applied to the wall as shown in Figure 11.

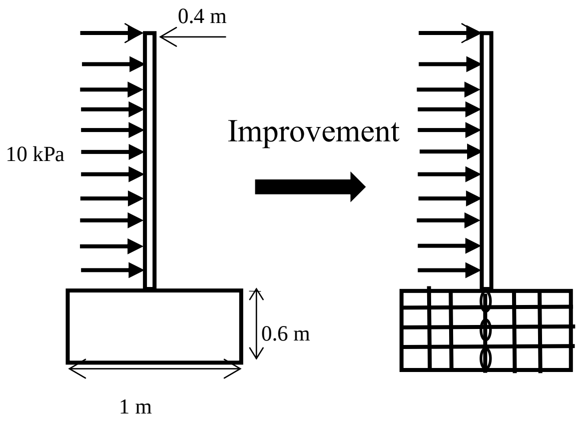

Figure 11: Shell to solid connectivity

3. Results and Discussion

3.1Case study 1: Modelling of truss elements

Cantilever doubly tapered truss element (width and height varying along the length) was analysed by increasing the number of elements. Results were compared with theoretical solutions. Figure 12 shows that variation of error percentage versus the number of divisions. The error percentage (EP) is calculated using the following equation.

$$EP = \frac{\text{Actual Result} - \text{Sap Model Result}}{\text{Actual Result}} * 100\%$$

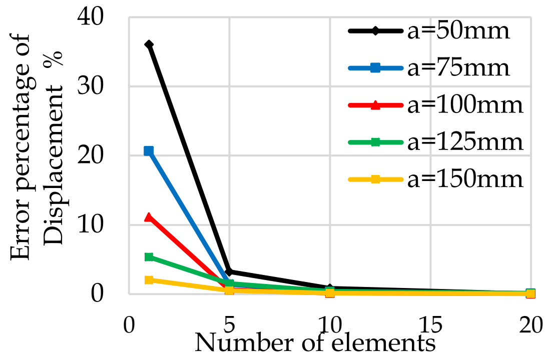

Figure 12: Selection of number of element for the double tapered truss.

It is observed that when number of elements getting increase (greater than 10), finite element solution is given the exact solution. However significant deviation of the results is observed when number of element less than 10.

3.2Case study 2: Modelling of frame elements

Cantilever prismatic beam was analysed by varying the span/depth ratio and results were compared with theoretical solutions. Figure 13 shows the displacement deviation with respect to the span / depth ratio of the beam.

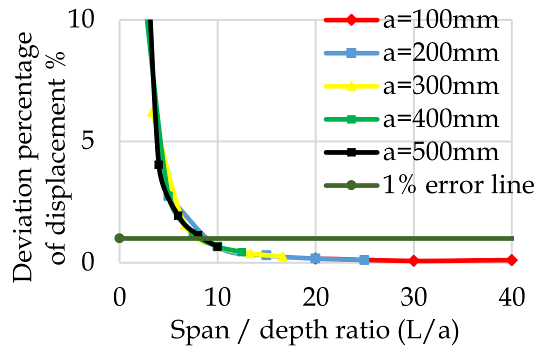

Figure 13: Deviation percentage vs. span/depth ratio for prismatic beam

Closed form solution can be obtained for prismatic beam using the stiffness matrix derived by beam theories. It is observed that significant deviation of the results is observed when span/depth ratio is less than 10. If span/depth ratio is lower than 10, displacement due to shear effect is dominant.

In sap2000 tapered option is only limited to the linear, parabolic and cubic functions of "EI" variation. EI function of curved tapered beam element has 6th order variation. Therefore it is necessary to do the h-refinement for the curved tapered beam element analysis. If the displacement based finite element software has EI function of 6th order variation in the tapered option, curved tapered beam can be analysed by using a single element.

Therefore, the curved tapered beams were analysed by increasing the number of elements. Results were compared by using theoretical solutions only considering flexural deformations. Figure 14 shows the displacement deviation.

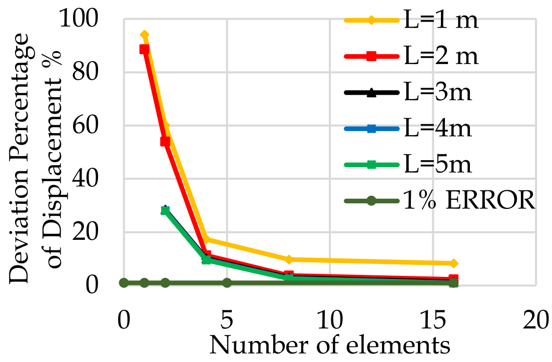

Figure 14: For b/a = 6 ,deviation percentage vs number of elements for curved tapered beam.

It is observed that, the number of elements are getting increased, finite element solution is given the exact solution. Hence Single element should be discretised at least to 16 elements for span/depth ratio 2-50 to obtain results with less than 1% error.

3.3Case study 3: Modelling of plate bending problem by using plate, shell and solid elements

3.3.1 Effect of mesh size of the slab

Pin supported one way and two way slab panels subjected to a uniformly distributed area load were analysed. Figure 15 illustrates the effects of mesh size on the mid displacement and the mid moment of the two-way slab.

It is observed that accurate results can be obtained by using the mesh size less than 0.5 m for moments and it can be obtained less than 1 m for displacement.

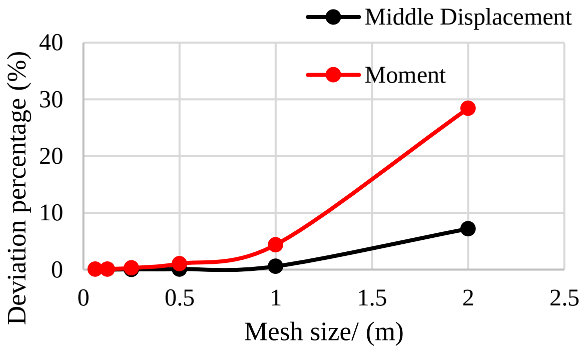

Figure 15: Effect of mesh size for slabs

3.3.2Effect of span/depth ratio of the slab

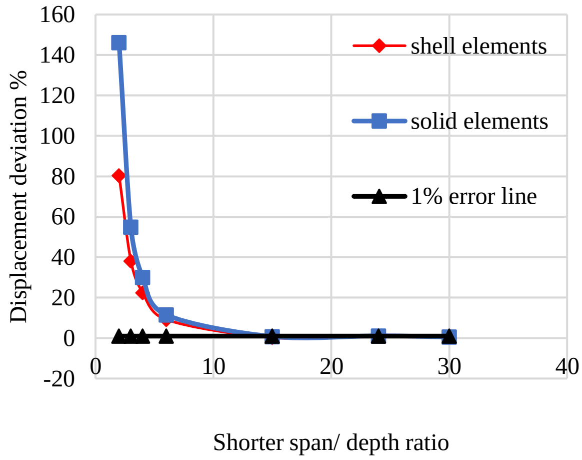

Figure 16: Effects of span/depth ratio for slabs

Figure 16 shows the variation of the results due to aspect ratio in the one way slab. Theoretical values were obtained considering effect of flexure only to identify the limit for shear effect. It is observed that shear effect is dominant when shorter span/ depth ratio of the slab is less than 15.

3.3.3 Effect of shear locking of the slab

Artificial moments are induced due to the shear effect. Hence moments and stresses are increased infinitely when span to depth ratio became larger. This phenomenon is called shear locking. Effect of the shear locking is also included when the Span / Depth ratio getting large in the slab as shown in Figure 17.

When span/ depth ratio is greater than 100, shear locking problem is occurred.

Therefore, a slab which having span/depth ratio greater than 100 cannot be analysed by using thin shell elements accurately.

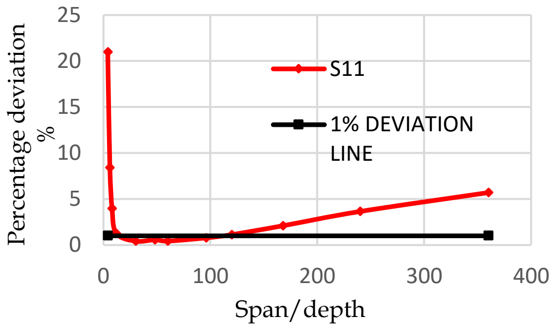

Figure 17: Effects of shear locking

3.3.4 Effect of distorted angle of the slab

Converged number of elements were used in the analysis by varying the angle of the element (α). Theoretical values were obtained by using basic plate theory and deviation percentages of stresses, moments and middle displacement were shown in Figure 18.

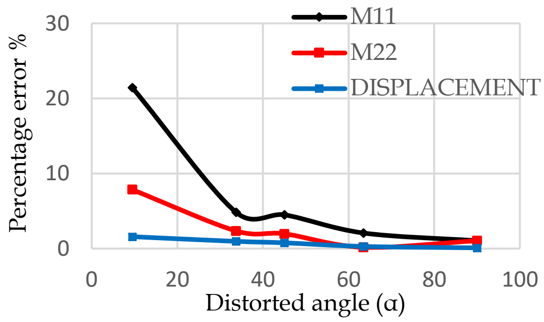

Figure 18: Effects of distorted angle-two way slab

Due to the distorted element, there was a large variation in Jacorbian matrix. It imply highly distorted mappings and affects the accuracy of results. Least distorted angle $\propto > 40^0$ gives the results with less than 5% error.

3.3.5 Analysis of slab panels

Slab panels were analysed by using Shell, Plate (using insertion option) and solid elements. Results were obtained by considering interior, corner and edge slabs. Figure 19 shows the effect of the centerline modeling and the neutral axis shift of the shell to beam connectivity. It shows the effect of the neutral axis shift is higher in middle displacement. But it is not that much of affected on the moments.

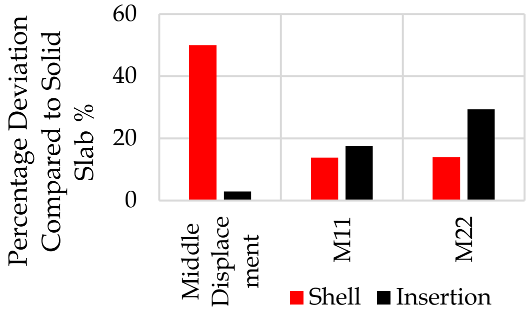

Figure 19: Effect of restrain conditions for Corner Slab

3.4 Case study 4: Element connectivity problems

Frame element and shell element were inserted in to the solid foundation by layer by layer to resist rotation. Table 1 shows that deviation percentage of horizontal displacement with theoretical solutions.

Table 1: The deviation percentage of horizontal displacement with theoretical solutions

Embedded length/ thickness	Frame to solid connectivity	Shell to solid connectivity
0	4.73E+17	4.48 E+12
0.25	79.46	36
0.5	30.51	15
0.75	22.17	13
1	20.63	12

4. Conclusion

4.1Guidelines for modelling linear elements

Closed form solution can be obtained for prismatic and linear tapered elements using the stiffness matrix derived by beam theories using the finite element method with direct stiffness approach.

Modelling of truss elements

a). For doubly tapered truss element (width and thickness varying along the length): Single element should at least be discretised into 10 for span/depth ratio 6-100 to obtain results with less than 1% error.

Modelling of frame elements

a). Prismatic beam - If the span / depth ratio is lower than 10, displacement due to shear effect is dominant and shear locking problem is not included in prismatic beam element.

b). If the displacement based finite element software has EI function of 6th order variation in the tapered option, curved tapered beam can be analysed by using a single element. Otherwise, discretization should be used for modelling of curved tapered frame elements. Single element should be discretised at least to 16 elements for span/depth ratio 2-50 to obtain results with less than 1% error.

4.2. Guidelines for modelling plate bending problem

I). Optimum mesh size can be obtained from figure 20 by using square elements.

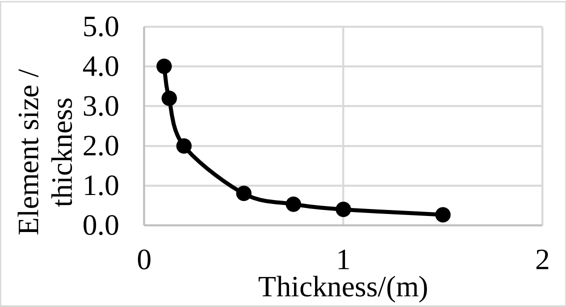

Figure 20: Guide for effect of mesh size

II). Selecting element type based on span / depth ratio for plate bending problem as given in the table 2.

Table 2: Limits for span/depth ratio

	Thick Shell	Thin Shell	Shear Locking
Span/Depth ratio of slab	Less than 15	15 - 100	More than 100

III). Minimizing errors due to distorted angle.

When using the quadrilateral elements, least distorted angle $\propto > 40^0$ gives the results with less than 5% error

IV). Minimizing errors by neutral axis shift

It is necessary to use INSERTION option when obtaining the results for Displacements.

When obtaining results for moments and stresses, insertion is not necessary.

4.3. Guideline for element connectivity problems

I). Insertion of frame element into solid mesh is proposed to improve the frame to solid connectivity. (As an example: when designing Foundations)

II). Insertion of shell elements into solid mesh is proposed to improve the frame to solid connectivity. (As an example: when designing Retaining Wall).

Acknowledgement

We take this opportunity to thank all the people who help us to finish this research project successfully.

First we wish to thank Department of Civil Engineering, University of Peradeniya to giving such an opportunity to conduct a research project.

References

[1]. Knupp P. M., 2000, Achieving finite element mesh quality via optimization of the Jacobian matrix norm and associated quantities- Part 1 –a framework for surface mesh optimization, International journal for numerical methods in engineering, Published by John Wiley and Sons, Ltd.

[2]. Shashikant T. More, Bindu R.S., May 2015, Effect of mesh size on finite element analysis of plate structure, International Journal of Engineering science and Innovative Technology, Volume 4, Issue 3,

[3]. Zienkiewicz O.C., Taylor R.L., Zhu J.Z., April 6, 2010, The Finite Element Method: Its Basis and Fundamentals, Sixth Edition.