International Journal of Bridge Engineering (IJBE), Vol. 6, No. 3, (2018), pp. 01-17

OPTIMAL MESH DENSITY FOR FEM ANALYSIS OF A BRIDGE DECK UNDER SELF-WEIGHT AND DYNAMIC VEHICULAR LOADS

Rohit Kumar Dubey

Lucknow Institute of Technology, Dept. Of Civil Engineering, Lucknow, India
e-mail: rohit16may@gmail.com

ABSTRACT: Analytical Modelling is the initial step in a software based Finite Element Analysis of any sophisticated structure associated with irregular shape, complex loading pattern and combination of different loads along with complex boundary condition. In the modelling of structure, Mesh Density plays a major role and it becomes a critical issue of finite element analysis, which closely relates to the accuracy of finite element model while directly determine their complexity level. This project report presents a systematic study on finding the effects of Mesh density on accuracy of numerical analysis results, based on which brief guidelines of choosing the best mesh strategy in finite element modelling is provided. In the project work, for studying the effect of Mesh Density, a bridge deck has been considered and analysis has been performed separately for Self-weight of the structure as well as Vehicular loading which is dynamic in nature. The modelling of the Bridge deck is performed several times with different mesh sizes for studying the analysis results to accomplish the best Mesh strategy that maintain the complexity level of the modelling without compromising in the accuracy of the analysis results. In addition, a case study on similar kinds of attempts to establish perfect Mesh size has been discussed briefly and a comparison has been made between the conclusions of this project work with the case study results, which are in line up to a great extent.

KEYWORDS: Bridge Deck, Finite Element Analysis, Mesh Density

1. INTRODUCTION

1.1 Finite element analysis and its importance

Finite Element Method is a numerical method for solving problems of engineering and mathematical physics. The application of Finite Element method is spread over a large area ranging from Structural mechanics, Fluid mechanics, and thermodynamics to electromagnetic potential. FEA has been gaining its popularity due to its unique nature of analyzing multiple complex problems easily and it has given a new direction in the field of engineering analysis. One of its major applications is in Structural Mechanics problems. Structural Analysis becomes a major issue in front of Professional engineering or even in academic research when it is a complex situations like analyzing a multidimensional extremely irregular shape, structure associated with different elements with different material properties, sophisticated boundary condition and complex loading pattern with combined effect of different kinds of loads. To overcome with these kinds of problems, we have the tool Finite Element Method. At present time, there is lots of analysis software for structural mechanics problems based on finite element method. These all helps in dealing with complex structural engineering project.

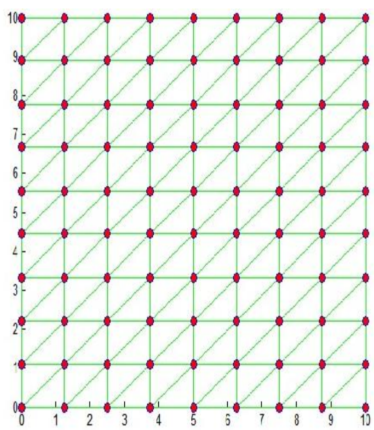
1.2 Basic principle of finite element analysis

The basic principle of Finite element method in structural mechanics is the discretization of the member, which needs to be analyzed; breaking down the member into infinitely small element, and prepare stiffness matrix of each element and applying boundary condition, loading equation and all gives the global stiffness matrix that does the stress strain analysis of the structure. The same concept has been coded in the analysis software to do complex calculation involving higher level of mathematics. The analysis depends on the discretization, higher the level of discretization closer the analysis results towards the exact solution. In case of dealing with an analysis of structural problem with an FEM based software, we have to define all the parameters (loading, material property, boundary condition etc.) first followed by a modeling of the structure. In this analytical modeling, the finer the members or the more insertion of nodes gives the higher accuracy of the analysis. Insertion of more nodes to create finer element can be referred as discretization of structure, in another terms the same phenomena is defined as mesh density.

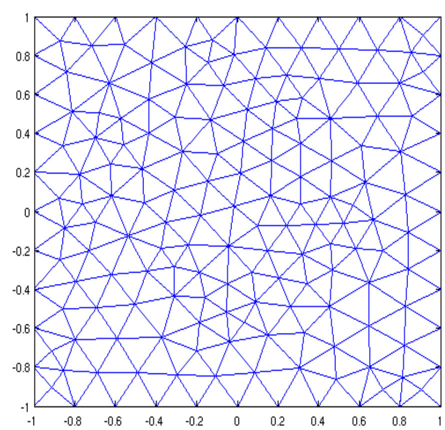
1.3 Mesh density in finite element analysis

In a simplest way, Mesh density in a finite element analysis can be defined as number of elementary mesh present in an analytical model of a member to be analysed. It can be referred as number of mesh present on plan area of the model or in case of a linear analysis by number of divisions in the linear modelling of the member.

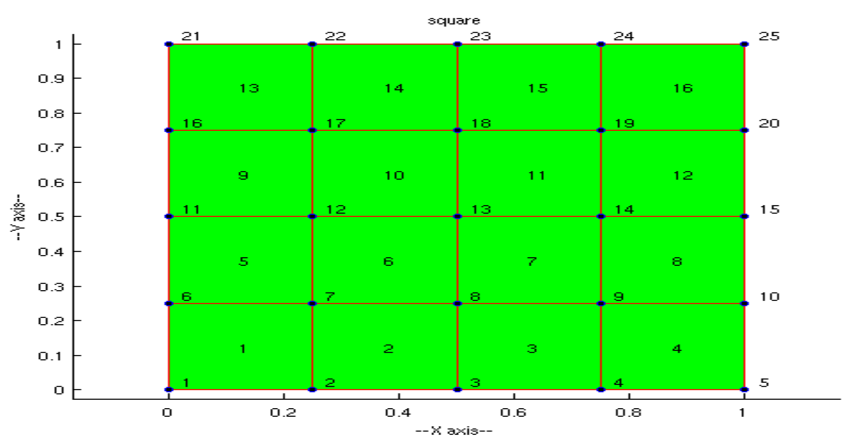
One of the basic principles of finite element analysis is it always deals with node to node and analysis results (Bending Moment, Shear Force etc.) are presented in nodes only. So defining nodes is the very initial step if Finite Element Modelling. Then beams are defined to connect these nodes and connection of these beams in two perpendicular directions creates a plane mesh. These kinds of Mesh are either square or rectangular and they are considered to be the best for giving good analysis results. Apart from this conventional mesh, triangular mesh, diamond mesh, octagonal or irregular shape mesh arepossible and they are used as per special requirement of the analysis. Different kinds of meshing based on shape have been shown below.



Triangular Meshing



Diamond or Irregular Meshing

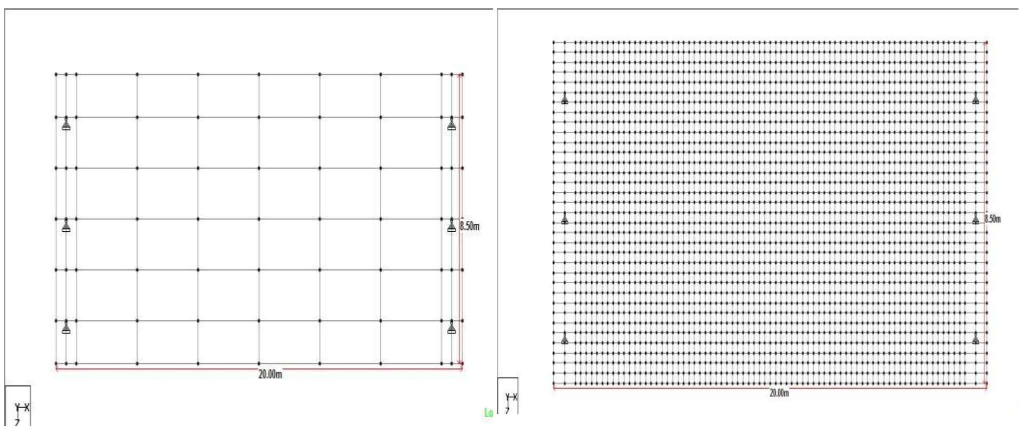


Desired Square Meshing for better Analysis

Figure 1. Different types of meshing based on shape

1.4 Classification of mesh based on size

Apart from different shapes, Mesh in Finite element analysis can be further categorized into Coarser Mesh and Finer Mesh based on its size. The classification will be clearer with the following figure.



Coarser Meshing of plane area 20x8.5 m2 Finer Meshing of plane area 20x8.5 m2

Figure.2. Coarser and finer meshing

As we increase the number of node in modeling, the beam element becomes finer and hence we get an increased mesh density, which can be treated as higher level of discretization and finally it gives a better analysis results responding to the actual structure condition.

2 PROJECT OBJECTIVE

It is clear that Analytical Modelling in Finite analysis with finer Meshing gives results, which converges towards the actual results correlating exact site condition of the structure. However, as we increase the level of finer meshing in our modelling, the complexity level of the modelling keeps on increasing. This kind of complex model always increase the chances of committing mistakes during modelling and as results, the analysis will show error. For example, the finer meshing shown on above figure representing plan of a bridge deck of size 20m x 8.5m consist of more than 2500 nodes and over 6000 beams associated with it and an intense level of concentration is required during performing this kind of finer mesh to avoid any kinds of mistake. Even source of error is very difficult to track down because of huge numbers of members associated with this kind of modelling. Apart from the error and possible chances of mistake, Finer-meshing analysis always takes significantly long amount of time for post processing and showing results, which is not desirable. To avoid these problems we can carry our finite element analysis with coarser meshing which gives quick results with almost zero percentage chances of error. But, that is not an acceptable solution as Analysis results with coarser mesh, sometimes shows variation far away from the actual results. This variation is often towards un-safer side and it can lead to a collapse of the structure if coarser meshing analysis results, has been adopted for designing of the structure. To overcome both the problems, a mesh size need to be fixed which is neither Finer nor Coarser and shows both advantages of finer and coarser meshing in terms of least computing time with no error maintaining a significantly higher level of proximity to the actual analysis results. The mesh density corresponding to this particular mesh size is termed as Optimal Mesh density and it is very important for a realistic analysis without error and delay in computing.

Several studies, researches has been carried out in different institutes around the globe to establish optimal density in Finite element analysis because of its importance in dealing with complex structural analysis required in industrial purpose as well as academic area. This optimal density may vary depending upon the type of structure to be analysed and its parameters. Studying the variation of analysis results corresponding to different Mesh density modelling is prime purpose of the projects. This study also helps in developing an optimal density of a bridge super structure, which is under the effect of self-weight as well as vehicular load which is another aim of this project work.

3 CASE STUDY

3.1 Source of study

A number of investigators have studied the effect of element size on the accuracy of numerical results of different types of analysis and important conclusions are drawn from previous researches. Several studies have been made based on various FEM based software to determine optimal mesh density or element size/division of elements for analytical modeling to give a realistic analysis. Out of these studies, one particular experiments conducted at Mississippi State University has been referred as case study to compare results and conclusion.

Based on previous works and achievements, a systematic investigation was conducted at Mississippi State University to fully discuss the size effect on simulation accuracy of static, modal and impact analysis for fundamental structural components such as plates and thin-walled beams. Out of these, static analysis of plate element corresponding to different mesh density and their results has been discussed thoroughly in this project report.

3.2 Experimental background and results

Static analysis were performed on a rectangular steel plate with dimension 300mm x 200mm and a thickness of 3mm. Material properties of the plate are – Young's Modulus 207 GPa, Density – 7830 Kg/m3, Yield Stress – 200 MPa, Ultimate Stress – 448 MPa and Poisson's ratio – 0.3

During the analysis, one end of the plate was fully constrained and 1 N-m moment was applied at the other end for a duration of 1 second. 10 times step were used to record the data so that 10 data points were collected during the analysis. A series of FE models were generated for that plate whose long side was meshed 2 (coarsest mesh) to 160 (finest mesh). Von mises stress and bending deformation yielded from each model were calculated and compared to study the influence of element size on the static analysis. Static analysis results and comparison are listed in Table 1 below. In that table, it is assumed that the FE models with finest mesh generate the most accurate results and percentage approximate errors were calculated by comparing other results to the most accurate ones.

Table 1.

Mesh density (No of Divisions)	Stresses (Mpa)	% of Error	Computing Time
2	6.290	5.08	3 sec
5	6.370	3.88	3 sec
10	6.580	0.76	3 sec
20	6.570	0.85	3 sec
30	6.607	0.3	3 sec
40	6.613	0.21	4 sec
50	6.607	0.3	6 sec
60	6.620	0.11	7 sec
70	6.621	0.09	9 sec
80	6.616	0.16	13 sec
90	6.624	0.05	18 sec
100	6.624	0.04	26 sec
120	6.626	0.01	40 sec
140	6.623	0.06	66 sec
160	6.627		124 sec

Observations made by comparing the results of table 1

1. The difference of Von Mises stresses generated from the model with 10 elements along the long side of the plate and from the finest mesh is less than 1% which is very good in engineering simulation. However the computing time for the coarse mesh model is only 3 seconds, which is less than 1/40 of the time cost by the finest model.
2. In can also be observed from the figure 3 that when the number of elements on the long side is higher than 60, the increase of mesh density doesn't significantly improve the accuracy of Von Mises stress any more. Such phenomenon was also observed in comparing other static analysis results.
3. Finally, it can be concluded that for static analysis, the FE model whose longest side is meshed by 10 elements can give us optimal combination of accuracy and efficiency.

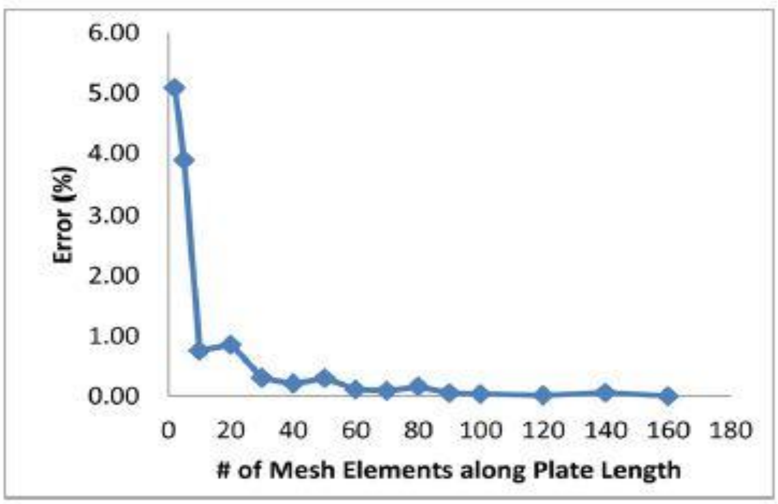


Figure 3. Variation of % of error with respect to Mesh density (case study plot)

4 PROJECT WORK: ANALYSIS OF A BRIDGE DECK WITH DIFFERENT MESH DENSITY

4.1 Data assumption and model development

Following the results of case study, on a similar track finite element analysis results has been studied for a bridge deck corresponding to different analytical modelling of same bridge associated with different mesh density. The basic assumptions made for developing the bridge super structure model are as follows-

1. The span of the bridge is 20.0m with a distance of 19.0m between centre to centre of supports.
2. The bridge is a 2- Lane bridge with outer width 8.5m.
3. The super structure of the Bridge consist of three RCC I girders with constant sectional properties though out the length of the bridge, connected with bridge deck and analysis results of the central girder has been used in this experiment.
4. Each ends of the bridge has three supports connected the girder to the bearing pedestal and it gives total six supports in the bridge system and all the supports are assumed to be pinned.
5. End diaphragm & intermediate diaphragm in girders and crash barrier and side walk on the bridge deck has been neglected for simplification of analysis and calculation.

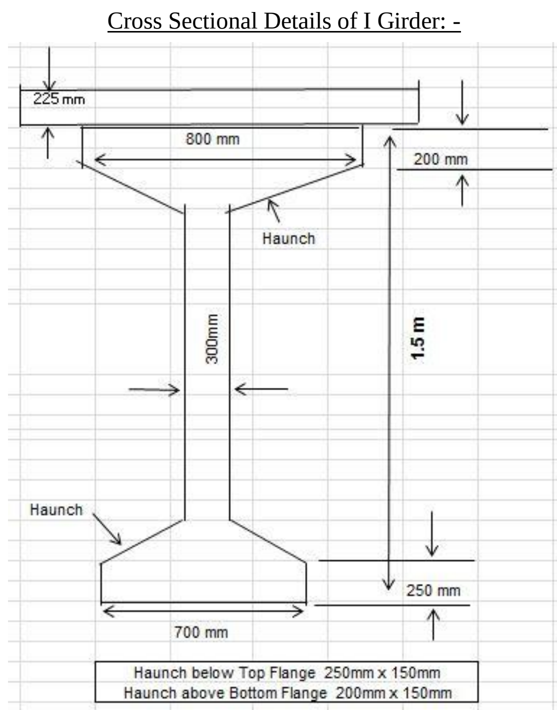


Figure 4. Cross sectional details of I girder

Depth of girder- 1.5m	Web thickness – 300 mm
Top flange width – 800mm	Top flange depth – 200mm
Bottom Flange width – 700mm	Bottom Flange depth – 250mm
Top Haunch – 250mm x 150mm	Bottom Haunch – 200mm x 150mm

Thickness of Deck slab – 225 mm

3 D Rendered view of the Bridge Deck

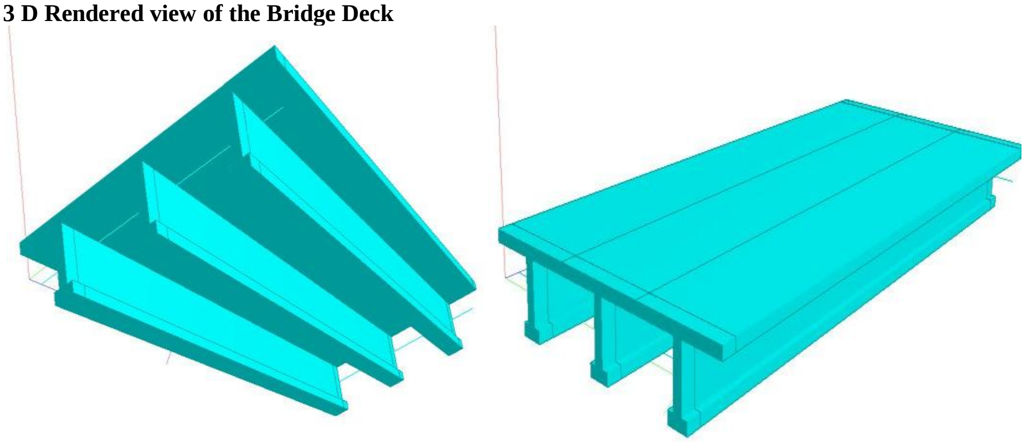
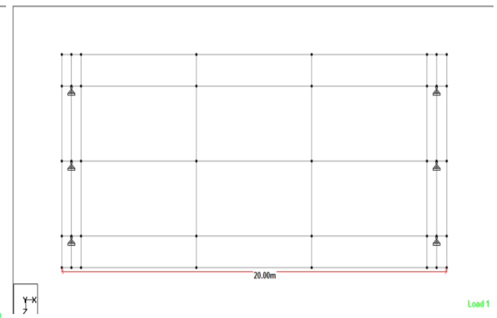


Figure 5. 3D view of the bridge deck with girders

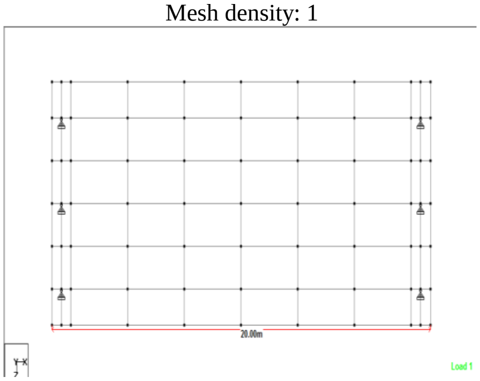
4.2 Analytical model development with different mesh density

The plan area of the bridge deck has been kept constant (20x8.5m2) and modelling has been performed six different times with a mesh density ranging from 1 to 72. As mentioned earlier, all girders are assumed to be pinned supported and therefore different mesh density has been arrived for the analytical model by diving beam girder in to different numbers of divisions. For example, if all girders in between the supports are represented by a single beam element is corresponding to a mesh density of 1. Hence in other modelling, the girder are divided into 3,6,12,36 and 72 numbers of elements in between supports to generate mesh density of 3,6,12,36 and 72 respectively. Out of these mesh density, 1 to 12 can be treated as coarser, 36 finer and 72 divisions as finest meshing.

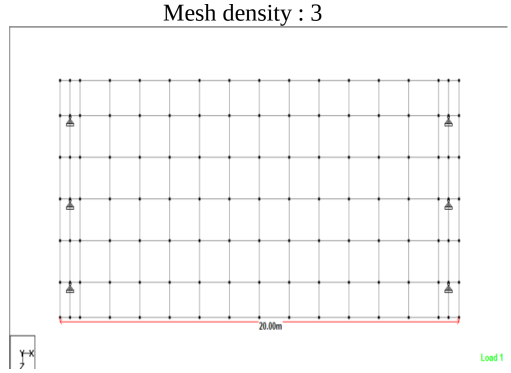
Mesh density: 1



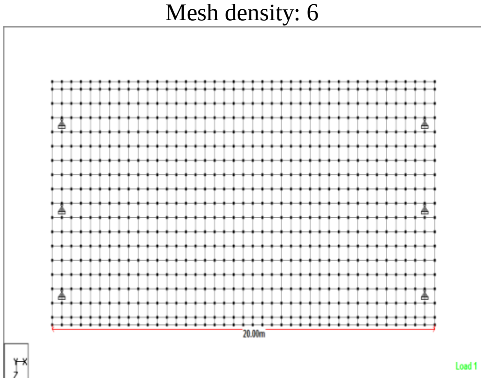
Mesh density : 3



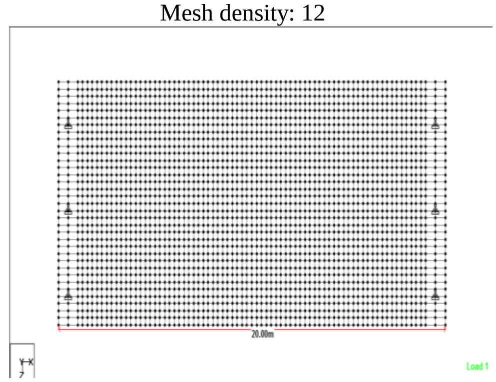
Mesh density: 6



Mesh density: 12



Mesh density: 36



Mesh density: 72

Figure 6. Modelling with different Mesh Density

4.3 Studies of analysis result and interpretation

After completion of the modelling with different mesh density, structure has been analyses for self-weight load effect and Max bending moment value obtained, corresponding to each different mesh density, have been recorded and presented on a tabular form below. Since the analysis has been carried out for self-weight only, critical moment is at mid-point and therefore, central girder mid-point readings are taken that represents the max bending moment.

Table 2. Max BM due to self-weight for different Mesh density

Sr No	*No of divisions along the Span*	*Max BM at Mid span of Central Girder (KN.m)*	*Increase in Max BM (KN.m) per number of increased Nodes*	*% of Increase in Max BM per number of Increased Node*
1	1	1440	-	-
2	3	1550	55.000	3.819
3	6	1680	43.333	2.796
4	12	1850	28.333	1.687
5	36	2210	15.000	0.811
6	72	2480	7.500	0.339

The graphical representation of the above data of table 2 is shown below.

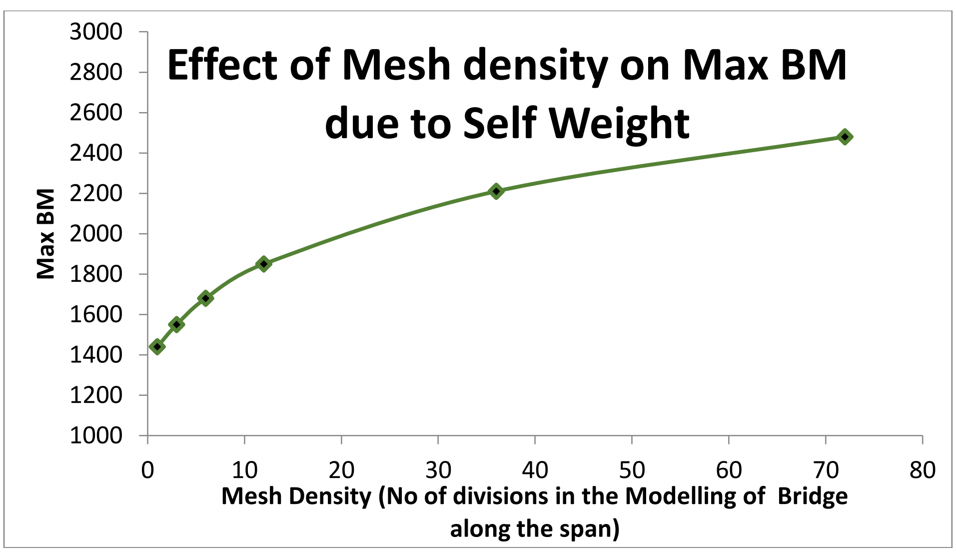


Figure 7. Variation of Max BM due to self-weight as per Mesh density

From the plot, it has been clear that Max BM in a bridge deck due to self-weight increases as we keep on increasing mesh density or adopting finer mesh

In last page of the report, Table no 2 shows the results of Max Bending Moment corresponding to different Mesh density. In our next study, a comparative plot between percentages of Increased in Max Bending Moment per number of increased nodesand increase of Mesh Density has been evaluated. As we know, the third column of the table 2, gives the Max BM due to self-weight corresponding to the different Mesh density shown on 2nd column of the table. For example, increase in Max BM is 110 KN.m (1550 – 1440) when mesh density is increased to 3 from 1. So the increase in Max BM per numbers of increased mesh density is, 110/(3-1) = 55KN.m, which is shown on the 4th column of the table and on the 5th column same figure same has been converted to percentages with respect to max BM of the earlier Mesh density. Example, against mesh density 3, we got a percentage of increase 3.819 which is the percentage fraction of increased max BM per increased nodes (55 KN.m) to the actual Max BM corresponding to mesh density 1 (1440.0 KN.m). The Plot has been shown below, for further analysis of the Results.

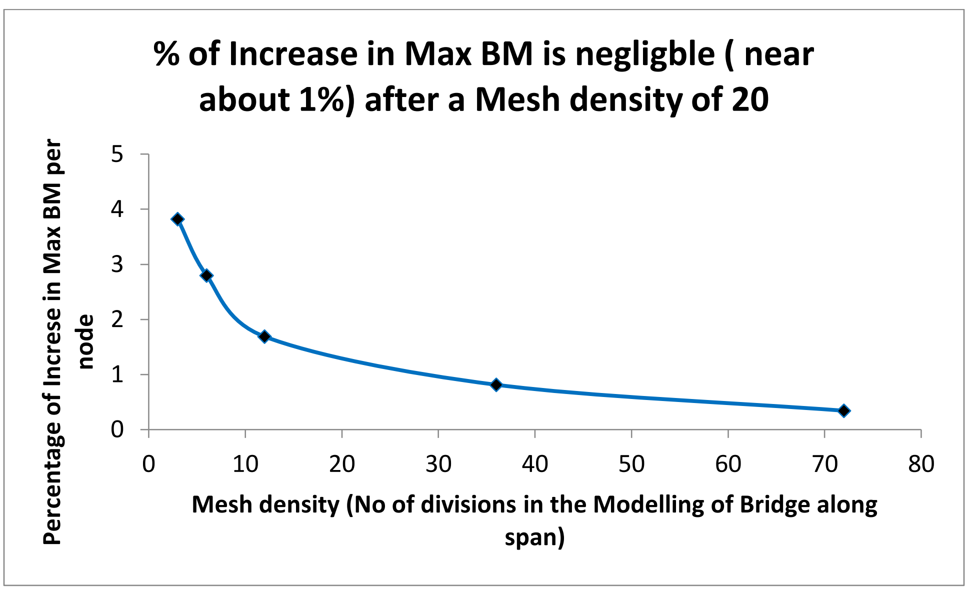


Figure 8. Variation in percentage Increased of Max BM with respect to Mesh density

From the graph, it has observed that the percentages of variation in Max BM keep on decreasing as we increase the Mesh density. In case of a finest Meshing of

density 72, the percentage value even goes down 0.5%. Besides, the percentage of decrease in Max BM goes down to 1% with a mesh density near about 20. So, practically it implies that a meshing of 20 is good enough to take care of accurate analysis as further increase in mesh density doesn’t give a significant change in percentage of increase in Max BM. Whereas, it is feasible to perform an error less analytical modelling with a mesh density in the range of 20 that’s gives a quick post processing and a less computing time. Therefore, in this analysis conducted for self-weight of a bridge deck shows an optimal mesh density of 20 to maintain an accurate finite element analysis with least computing time and error less analysis.
The optimal mesh density arrived from the results of case study is 10, as the whole analysis was carried out for a simple plate element of size 300mm x 200mm whereas a significantly long bridge of span of 20m necessarily demands a higher level of optimal Mesh density.

5 DYNAMIC LOAD ANALYSIS

5.1 Vehicular loading details

Vehicular load has been considered to study the analysis results for varying mesh density under the effect of dynamic load. The vehicular loading has been taken from CANADIAN HIGHWAY BRIDGE DESIGN CODE (CHBDC- CAN/CSA-S6-06) with an assumption that this hypothetical bridge is located somewhere in Canada. The section -3 of this code CAN/CSA-S6-06 deals with the details of various patterns of load can be acted on a typical highway bridge. Out of this section 3 of the manual, clause no 3.8.3.2 gives the details of vehicular loading need to be considered for a highway bridges. There are basically, two different vehicular loads as per the code and they are 1. CL-625 Truck loading and 2.CL-625 Lane loading. In this expression CL stands for Canadian loading and 625 represents the total weight of the vehicular load in KN. Out of these two loads, CL-625 truck loading is more critical and governs the Max BM, shear force and other parameters based on what design of a typical bridge is carried out. Therefore, in this part of the project vehicular load dynamic analysis has been carried out for CL-625 truckload only. Again, the Max BM result is recorded for the central girder only as done in self-weight analysis.

The CL-625 Truck load is comprised of five axles load and two tyres are associated with each axle distributing the axle load equally. The distribution of the 625 KN load is spreading over a length of 18.0m and following figure has been appended from the code CAN/CSA-S6-06 for a better understanding of Wheel spacing and weight distribution.

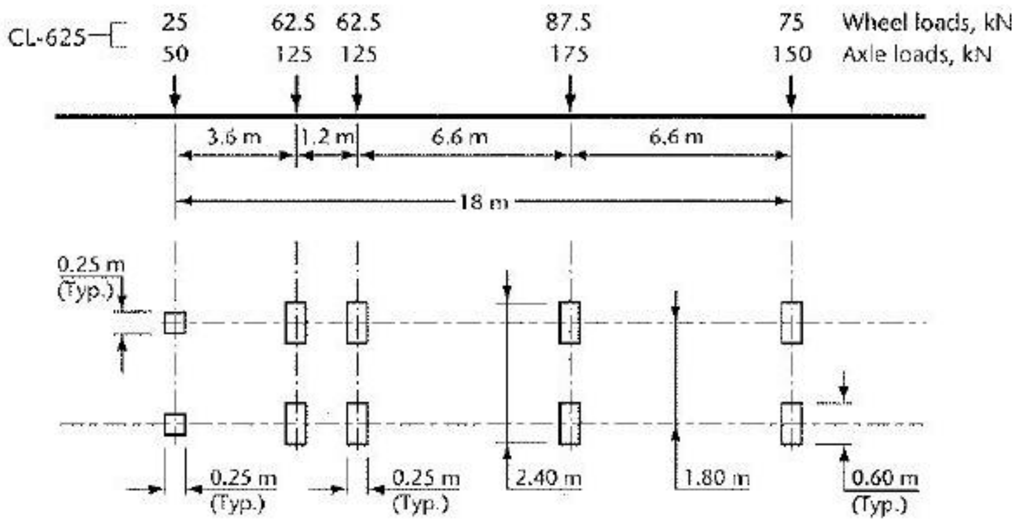


Figure 9. Axle spacing and load distribution of CL-625 Truck load

5.2 Results and interpretation from dynamic analysis

Dynamic load analysis has been carried out considering CL-625 Truck load for several times for the same bridge deck corresponding to six different modelling associated with different mesh density. The results of Max Bending Moment in the central girder corresponding to different mesh density has been recorded in a tabular form and shown below- Variation of Maximum Bending Moment due Vehicle load as per variation of Mesh size

Table 3. Max BM due to Vehicular load for different Mesh density

Sr No	*No of divisions along the Span*	*Max BM at Central Girder (KN.m)*	*Decrease in Max BM (KN.m) per increased Nodes*	*% of Decrease in Max BM per Node*
1	1	1010	-	-
2	3	965.66	22.170	2.195
3	6	948.22	5.813	0.602
4	12	931.798	2.737	0.289
5	36	872.296	2.479	0.266
6	72	796.569	2.104	0.241

The graphical representation of the Data shown on table 3 is as follows

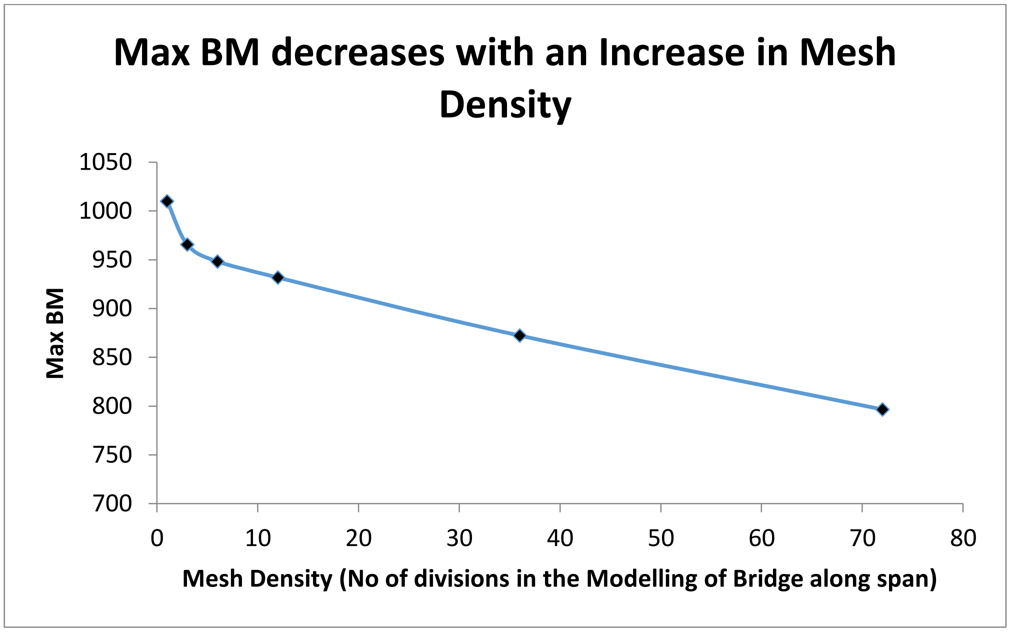


Figure 10. Variation of Max BM due to live load as per increase in Mesh density

From the previous plot shown in figure 10, it is clear that the Max BM due to vehicular live load decreases as we increase the Mesh density of the same analytical model. This is quite opposite in nature to the response of finite element analysis carried out for a static load like Self-weight which shows an increase in max BM with increased mesh density.

The reason behind this opposite nature of variation of Max BM due to live load is still not found out clearly. However, the main possible cause of the decrease in Max BM is because of the dispersive nature of the vehicular live load and higher density of Meshing gives freedom to disperse it widely and as a result decrease the concentrated action of load and Max BM accordingly.

The percentage of decrease in max BM due to live load corresponding to different mesh density has been calculated in the same way as described in case of Self-weight analysis and results are shown on the table 3 itself. The graphical representation of this Max BM due to live load variation is shown below.

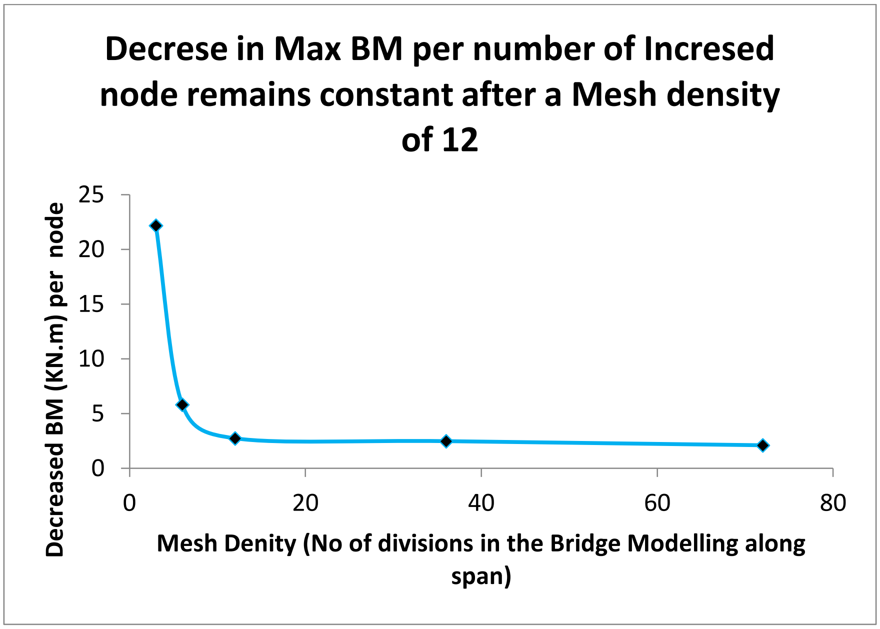


Figure 11. Variation of Decreased BM per increase node with respect to increase mesh density

Max BM due to live load keeps on decreasing as we increase the mesh density. However, from the above plot, it is clearly visible that this decreased in Max BM due to live load per increased number of mesh density is very negligible after a mesh density of 12 and it remains constant (near about 0.25%) even after further increase in mesh density.

6 CONCLUSIONS

In this project work several finite element analysis has been performed for a bridge deck under different loading (Static- Self weight and Dynamic- Vehicular CL-625 Truck load). Analyses are carried out for different modelling results from different mesh density varying from coarsest one to finest meshing of 72. Several interesting facts has been observed related to the effect of meshing size and Mesh density on the analytical modelling of Finite Element Analysis. Out of these studies some important conclusions are drawn and they are as follows –

1. Increase in Mesh Density in Finite Element Analysis results in increasing the Maximum Bending Moment due to static load (Self Weight) whereas the effect is reverse in case of Live Load Analysis.
2. An Optimal Mesh Density of 20 can be considered for a bridge deck Analysis

with a span in the range of 15m to 30m. In a practical application, it means diving the span of the bridge into 20 elements for finite element analysis gives us high level of accuracy with least possibility of error and least computing time involved in it.

3. Comparative Case study shows a mesh density of 10, which is for a small plate element of size 300mm x200mm and with consideration of this case study result, mesh density finalized from the project work for a bridge deck as 20 is seems to be valid and practical.

REFERENCES

1. H Alicia Kim, Osvaldo Querin, Grant P. Steven, “Improving Efficiency of Evolutionary Structural Optimization by implementing fixed grid mesh” International Society for Structural and Mulitdisciplinary Optimization, Springer Verlag
2. Brocca M. and Bazant Z. P., “ Size Effect in concrete Column: Finite Element Analysis with microplane Model”, Journal of Structural Engineering, 127(12),2001.1382-1390
3. R. Michael Biggs, Furman W. Barton, Finite Element Modeling and Analysis of Reinforced Concrete Bridge Decks” Virginia Transportation Research Council, September 2004.
4. Finite Element Analysis of Bridge Decks by “Mohammed R. Abdelraouf, Hudson Matlock”, Center for Highway Research, The University of Texas at Austin.
5. Ashford and Sitar, “Effect of Element size on the static finite element analysis of steep slopes”, International Journal for Numerical Analytical Methods in Geomechanics.25(14), 2001,1361-1376.
6. Saouma V.E., Natekar D. and Sbaizero O, “Non Linear finite Element analysis and size effect study in a metal reinforced ceramic composite”, Material Science and Engineering.
7. Liu Y –C. and Day M.L., “Simplified Modelling of thin walled box section beam”, International Journal of Crashwothiness, 11(3), 2006,263-272.
8. CAN/CSA-S6-06/ A national Standard of Canada, CANADIAN HIGHWAY BRIDGE DESIGN CODE, Section 3 – Loads
9. Staad Pro V8i User Manual