A Comparative Study on Deflection of Beams Using ANSYS

Akhilesh Methre[1], Vinodkrishna M Savadi[2], Uma Shankar Yaligar[3], Puneeth Kumar[4]
[1]Assistant Professor, [3]Assistant Professor and HOD Department of Civil Engineering, Guru Nanak Dev Engineering College Bidar
[2]Assistant Professor, Dept. of Civil Engineering, Acharya Institue of Technology, Bangalore
[4]Visvesvaraya Technological University Belagavi, Karnataka

***Abstract*—Analytical Determination of Deflections in Reinforced Concrete Beams with different Boundary Conditions was a difficult task and Engineers had to rely on the empirical formulas for calculations of deflections. The study consisted of modelling various specimens in Ansys and deflections were obtained. The study revealed that the deflections obtained through Theoretical Calculations and Ansys showed similar results.**

***Index Terms*—Ansys, Beams, Boundary Conditions, Deflection.**

I. INTRODUCTION

A. General: -

The basic concept behind FEM is that a body or structure is divided into smaller elements of finite dimensions called finite elements. The original structure is then considered as an assemblage of these elements at a finite number of joints called nodes. The properties of the elements are formulated and combined to obtain the solution for the entire structure. The shape functions are chosen to approximate the variation of displacement within an element in terms of displacement at the nodes of the element. The strains and stresses within an element will also be expressed in terms of the nodal displacement. The principle of virtual displacement is used to derive the equations of equilibrium for the element and the nodal displacement will be the unknowns in the equations. The boundary conditions are imposed and the equations of equilibrium are solved for the nodal displacement. From the values of the nodal displacement for each element, the stresses and strains are evaluated using the element properties.

Thus, instead of solving the problem for the entire structure in one operation, in this Finite Element Method attention is mainly developed to the formulation of properties of the constituent elements.

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature. For numerical approach, the solutions will be approximate when any of these relations are only approximately satisfied. The numerical method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. Few numerical methods which are commonly used to solve solid and fluid mechanics problems are given below.

- Finite Difference Method
- Finite Volume Method
- Finite Element Method
- Boundary Element Method
- Meshless Method

Degrees of Freedom: - A structure can have infinite number of displacements. Approximation with a reasonable level of accuracy can be achieved by assuming a limited number of displacements. This finite number of displacements is the number of degrees of freedom of the structure. For example, the truss member will undergo only axial deformation. Therefore, the degrees of freedom of a truss member with respect to its own coordinate system will be one at each node. If a two-dimension structure is modeled by truss elements, then the deformation with respect to structural coordinate system will be two and therefore degrees of freedom will also become two. The degrees of freedom for various types of element are shown in Fig 1 for easy understanding. Here (u, v, w) and (θ_x, θ_y ,θ_z) represent displacement and rotation respectively.

B. Need for Present study: -

The need for present study is as follows:

- Analytic study focuses on the prediction of behavior of beams which in turn reduces the time, manpower and material incurred during the experimental investigation.
- The study aids in the prediction of deflection of beam subjected to various loadings.
- 1.3 Objectives of the Present study: -
- From the earlier literature review the following objectives have been set.
- To model and analyze the models in Ansys software.
- To study the stress behaviour of beam using 3D finite element analysis (using Ansys 16.0)
- To compare the beam results with empirical methods.

II. METHODOLOGY

ANSYS is a general-purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers. So ANSYS, which enables to simulate tests or working conditions, enables to test in virtual environment before manufacturing prototypes of products. Furthermore, determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulations in virtual environment.

A. Analytical Data

ANSYS (version 16.0) has been chosen for the purpose of analyzing

Table 1.1: The properties of the materials are as shown in below table

Property	Concrete (M20)	HYST steel (Fe 415)
Tensile Yield strength (MPa)	0.2	2.5E+08
Ultimate Yield strength (MPa)	20	2.5E+08
Youngs modulus (MPa)	2.1E+5	2 E+11
Poissons Ratio	0.3	0.3
Shear modolus (MPa)	80769	1.6667E+11
Bulk modulus (MPa)	1.75E+05	7.6923E+10

B. MODELING OF STRUCTURE IN ANSYS: -

- Modeling of concrete slab
- Element modeling of geometry of concrete structure is either created in AutoCAD or in any 3D modeling software. A typical model of a Beam can also be created in ANSYS using the sketching command.
- Once the 3D model of the specimen to be tested is drawn in any 3D software the file needs to be saved in. iges or. igs or .stp or.stnp format.
- These are the formats which can be imported into ANSYS.

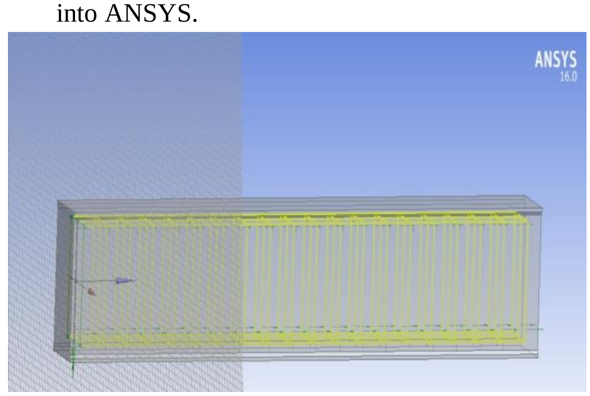

Figure: 2.1 RC beam with steel bars (yellow)

- Assigning of materials and connections between the materials
- Now once the geometry is ready open ANSYS.
- Now open Static Structral from the left table of contents present in the software.
- A dialogue box on the right of the screen appears as shown in fig 3.2. Open the engineering data and select the materials steel and concrete from

the materials table as shown in fig 3.3. We can also change the grade of concrete and steel by defining the properties.

- In the engineering data, you can observe material field variables, it's the drop-down box where we can change the properties of materials as per the requirements.
- Now open the geometry, design modeler will open and, on the top, you will get an option click import and select the .iges file containing the 3D model of specimen and then click generate and close the design modeler.

C. Meshing of slab

- Meshing is process in which the specimen is divided into different number of elements higher the elements higher is the accuracy of results. For example, grids in the figure 3.4 indicates the elements.

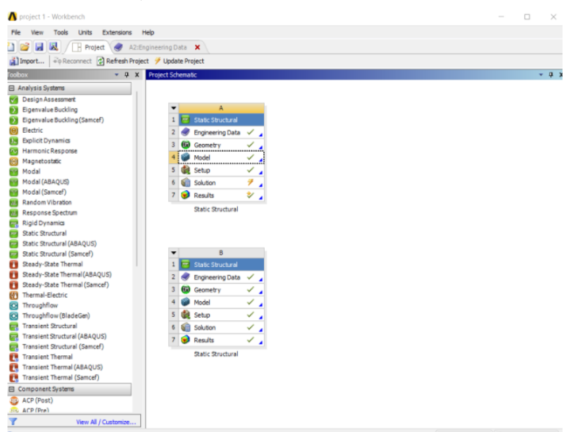

Figure2.2 Selection of Analysis system

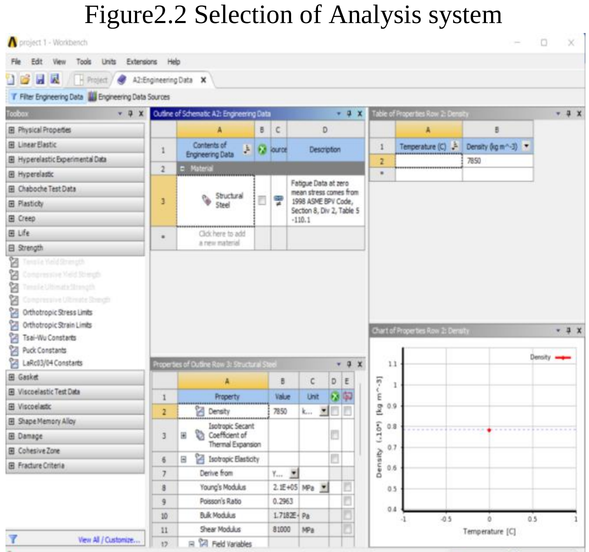

Figure 2.3 Engineering Data Tab

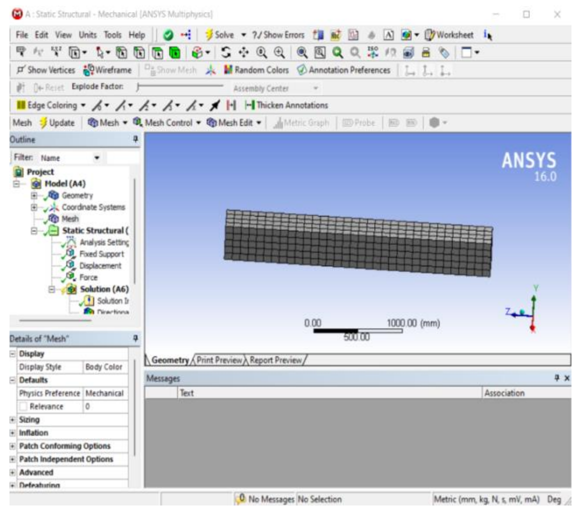

Figure 2.4 Meshing of Beam

D. Selection of required parameters

- Right click on the initial conditions in the explicit dynamics and select the type of support you want to provide. Now assign the support conditions to the corners of slab.
- Right click on solution select the required data to be extracted for example stress, deformations etc (figure 3.5).

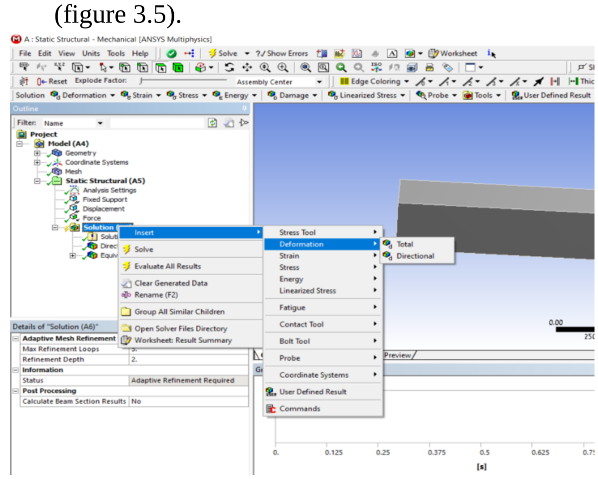

Figure 2.5 Selection of Parameters

E. Obtaining of solution

Now right click on the solution button and click solve or you can also press F5 button to solve and wait for solutions to be generated.

F. Deflection of beams

The deflection Deflection is the degree to which a particular structural element can be displaced with the help of a considerable amount of load. It can also be referred to as the angle or distance. The distance of deflection of a member under a load is directly related to the slope of the deflected shape of the body under that load. It can be computed by integrating the function which is used to describe the slope of the member under that load.

The Beam is a long piece of a body that is capable to hold the load by resisting the bending. The deflection of the beam towards in a particular direction when force is applied to it is known as Beam deflection.

The beam can be bent or moved away from its original position. This distance at each point along the member is the representation of the deflection. There are mainly four variables, which can determine the magnitude of the beam deflections. These include:

- How much loading is on the structure?
- The length of the unsupported member
- The material, specifically the Young's Modulus
- The Cross-Section Size, specifically the Moment of Inertia (I)
- There is a variety of range of beam deflection equations that can be used to calculate a basic value for deflection in different types of beams. Generally, we calculate deflection by taking the double integral of the Bending Moment Equation means M(x) divided by the product of E and I (i.e. Young's Modulus and Moment of Inertia).
- The unit of deflection, or displacement, will be a length unit and normally we measure it in a millimetre. This number defines the distance in which the beam can be deflected from its original position.

III. RESULTS AND DISCUSSIONS

A. General

Two Specimens of beams were analyzed using ANSYS 16.0 for deflection parameter and with the Empirical formula.

3.1.1 SPECIMEN A

The dimensions and the properties of specimen A is as shown below: -

- Length of the beam = 1000 mm
- Breadth of the beam = 40 mm
- Depth of the beam = 40mm
- Young's Modulus = 2.1 x 10^5 N/mm^2
- Distance from Neutral Axis = 20mm
- Moment of Inertia = 213333.3 mm^4
- Sectional Modulus = 10667.67 mm^3
- Type of Loading = Uniformly deforming load (UDL)
- Load Applied = 5 kN/m, 6 kN/m, 7 kN/m

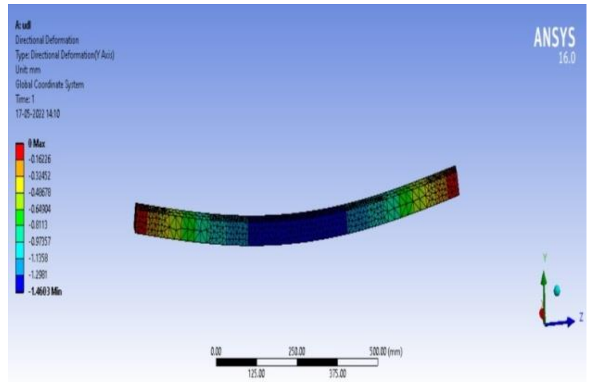

Figure 3.1 Defection of beam corresponding to udl 5 kN/m

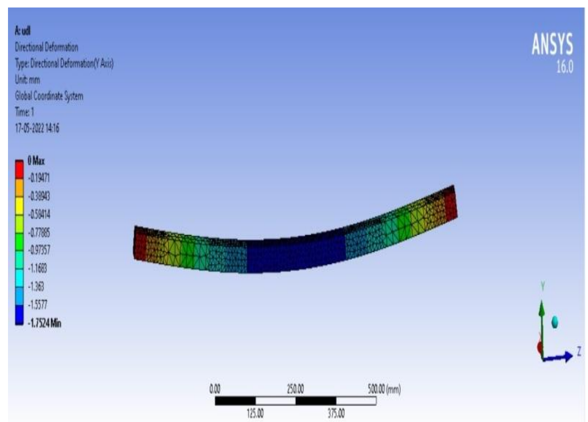

Figure 3.2 Defection of beam corresponding to udl 6 kN/m

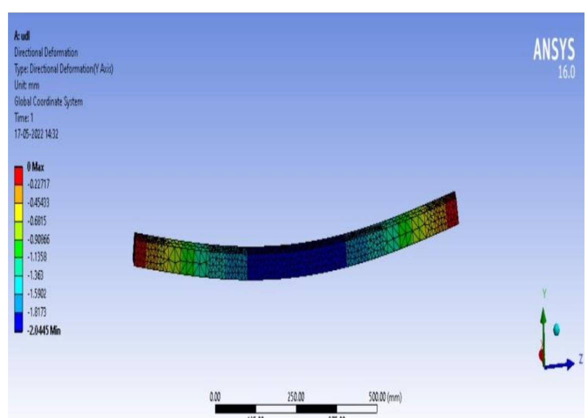

Figure 3.3 Defection of beam corresponding to udl 7 kN/m

Table 3.1 Comparison of theoretical results with simulation results of deflection

Load applied	Theoretical Results (in mm)	Simulation Results in ANSYS (in mm)
5 kN/m	1.453	1.46
6 kN/m	1.744	1.75
7 kN/m	2.035	2.04

B. SPECIMEN B

The dimensions and the properties of specimen B is as shown below: -

- Length of the beam = 1000 mm
- Breadth of the beam = 40 mm
- Depth of the beam = 40mm
- Young's Modulus = 2.1 x 10^5 N/mm^2
- Distance from Neutral Axis = 20mm
- Moment of Inertia = 213333.3 mm^4
- Sectional Modulus = 10667.67 mm^3
- Type of Loading = Central Point Load
- Load Applied = 5 kN/m, 6 kN/m, 7 kN/m

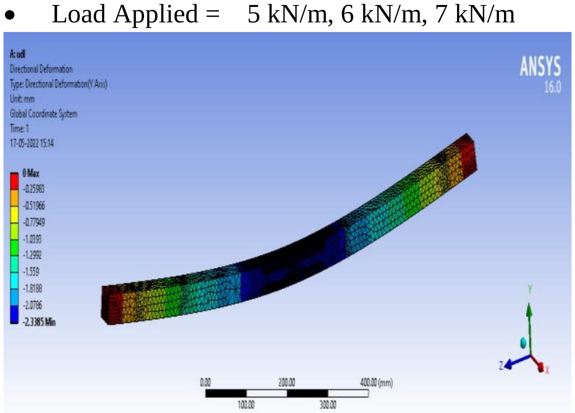

Figure 3.4 Defection of beam corresponding to Central Point Load 5 kN

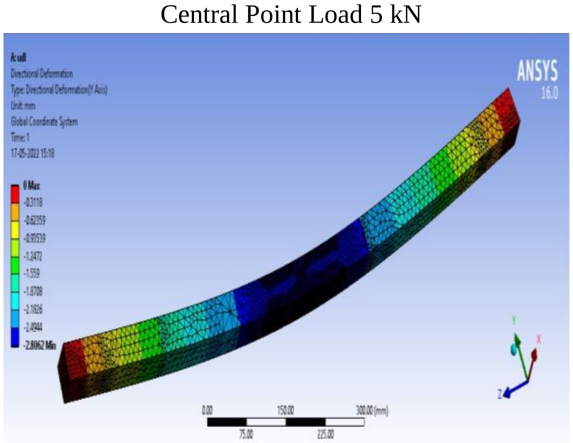

Figure 3.5 Defection of beam corresponding to Central Point Load 6 kN

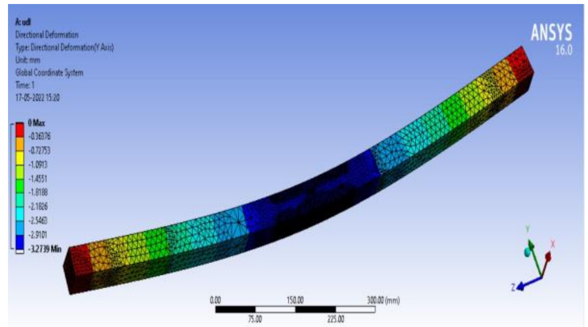

Figure 3.6 Defection of beam corresponding to Central Point Load 7 kN

Table 3.2 Comparison of theoretical results with simulation results of deflection

Central Point Load (kN)	Theoretical Results (in mm)	Simulation Results in ANSYS (in mm)
5	2.33	2.33
6	2.79	2.80
7	3.26	3.27

C. SPECIMEN C

The dimensions and the properties of specimen C is as shown below: -

- Length of the beam = 1000 mm
- Breadth of the beam = 40 mm
- Depth of the beam = 40mm
- Young's Modulus = 2.1 x 10^5 N/mm^2
- Distance from Neutral Axis = 20mm
- Moment of Inertia = 213333.3 mm^4
- Sectional Modulus = 10667.67 mm^3
- Type of Loading = Uniformly deforming load (UDL) and Central Point Load
- Load Applied = 5 kN/m and 5kN, 6 kN/m and 6 kN, 7 kN/m and 7 kN

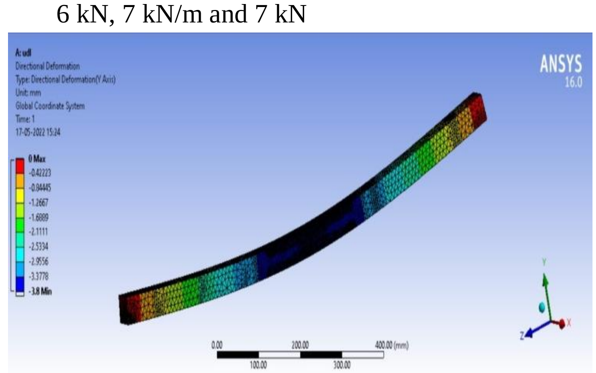

Figure 3.7 Defection of beam corresponding to UDL 5kN/m and Point load 5kN

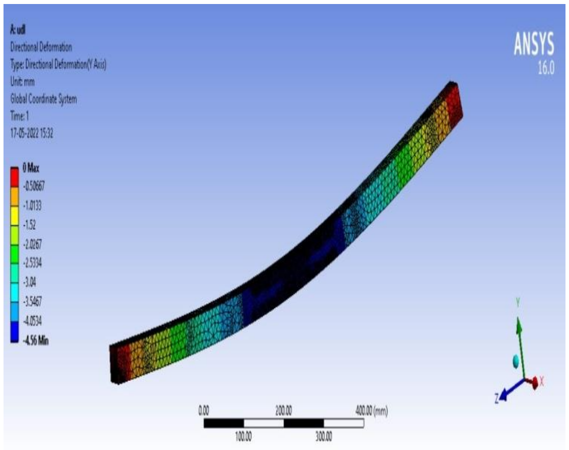

Figure 3.8 Defection of beam corresponding to UDL 6kN/m and Point load 6kN

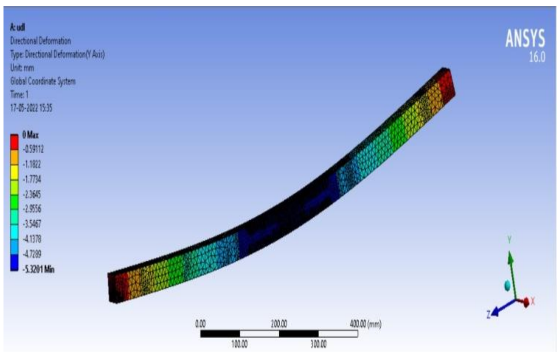

Figure 3.9 Defection of beam corresponding to UDL 5kN/m and Point load 5kN

Table 3.3 Comparison of theoretical results with simulation results of deflection

Load applied (UDL + Point Load)	Theoretical Results (in mm)	Simulation Results in ANSYS (in mm)
5 kN/m + 5kN	3.783	3.80
6 kN/m + 6kN	4.534	4.56
7 kN/m + 7kN	5.295	5.32

IV. CONCLUSIONS

From the results obtained in this analytical study following conclusions were made.

1. By comparing the theoretical values and the analysis with ANSYS, it was found that the deflection results are accurate and quite perfect.
2. The results are accurate as ANSYS module is based upon FEM.
3. As modelling can be done in ANSYS, it can be used to analyses different geometries of irregular shape since other software's don't give us the liberty to do that.
4. Ansys has better chances of giving the same results for validation of experimental results.

V. REFERENCES

[1] Samal, M. Rashied "Punching Shear Resistance of flat slabs with openings", International journal of civil engineering & technology (IJCIET) April2015.

[2] Ashraf Mohamed Mahmoud "Finite element implementation of punching shear behaviors in shear-reinforced flat slabs" Ain Shams Engineeri ng Journal February 2015.

[3] Chee Khoon Ng, et al. "Theoretical Evaluation on Effects of Opening on Ultimate Load - carrying Capacity of Square Slabs" Electronic Journal of Structural Engineering 2008.

[4] Jonas Gustafsson Keivan Noghabai, "Steel fibers as shear reinforcement in high strength concrete beams" IJET.

[5] Yoon-Keun, et al. "Shear strength of steel fiber reinforced concrete beams Without stirrups" ACI.

[6] Fanning, P., "Nonlinear Models of Reinforced and Post-tensioned Concrete Beams", Electronic Journal of Structural Engineering. vol.2, 2001, pp.111-119.

[7] Ngo D. and Scordelis, A.C., "Finite element analysis of reinforced concrete beams", Journal of ACI., Vol.64, 1967, pp. 152-163.

[8] Sheetal Gawas and Dr. S.V.Itti (2014), Study On Two Way RC Slab Using ANSYS With And Without Central Opening, International Journal Of Scientific Engineering And Technology (ISSN : 2277-1581).

[9] Koh Heng Boon, et al. (2009), Flexural Behaviour Of Reinforced Concrete Slab with Opening, Malaysian Technical Universities Conference on Engineering and Technology.

[10] Mr.Yadav Jaideep Purushottam, "Analytical Study of Solid Flat Slab and Voided Slab Using ANSYS Workbench",(ISSN :2348-4845).

[11] Liana L. J. Borges, et al. (2013), Punching Shear of Reinforced Concrete Flat Plates with Openings, ACI Structural Journal.

[12] M. Suneel Kumar, et al. (2007): Ultimate Strength of Square Plate with Rectangular

Opening Under Axial Compression, Journal of Naval Architecture and Marine Engineering.
[13] Prof. Dr.Nazar K, et al. (2014), Punching Shear Strength of Reinforced Concrete Flat Plates with Openings, Journal of Engineering.
[14] Hosam A. Daham (2010), Analytical Study of Reinforced Concrete Two Way Slabs with And Without Opening Having Different Boundary Conditions, journal of Al-Rafidain Engineering.
[15] Sunil Kumar M. S. and B.S.Suresh Chandra (2014), “Experimental Study On Self Compacting RC Slab With And With Out Opening”, International Journal Of Engineering Research & Technology (IJERT).
[16] Vera Nikolic, et al. (2011), Application of Finite Element Analysis of Thin Steel Plate with Holes, Technical journal.
[17] IS 456:2000, “Plain and Reinforced concrete code of practice”. 4th revision.