JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

PERFORMANCE EVALUATION OF PREFABRICATED SLAB PANALS IN BUILDING

Dr. M.S. Kuttimarks[1], Abhishek S. Ubale[2], Kalpesh V. Rathod[3], Harsha P. Pusal[4], Vaishali P. Harad[5] and Dr.Menbere Aklilu[6]
Associate Professor[1], Shivajirao S. Jondhle College of Engineering & Technology Asangaon, Thane, MH – 421601.
Student[2345], Shivajirao S. Jondhle College of Engineering & Technology Asangaon, Thane, MH – 421601
Assistant professor[6], Arbaminch University, Ethiopia.
drkuttimarks.ssjcet@gmail.com[1]

ABSTRACT

The prefabricated composite slab (PCS) is an essential horizontal component in a building, which is made of a precast part and a cast-in-place concrete layer. In practice, the floor should be split into many small PCSs for the convenience of manufacturing and installation. Conventional concrete-cast-implacable slab construction is widely used in worldwide and requires a long time and a lot of materials, strength and density. When constructing buildings, the implementation of a relatively cheaper and modern automated system will not only have economic benefits, but will also reduce the reliance on conventional systems. The use of the steel-concrete composite prefabricated slabs is significantly increased, as it leads to further reduction of the overall floor weight, primary energy and resources consumption in addition to the overall building cost comparing with traditional composite slabs. Prefabrication was made feasible with the advancement of production techniques and equipment for transportation and erection. Prefabricated slabs are usually designed and constructed as simply supported and discontinuous elements. However, on-site composite slabs are continuous and connected to steel beams using shear connectors. Hence, the capacity of the composite slab constructed on-site is higher than that of conventional prefabricated slabs. This study proposes interlocking connection systems for prefabricated steel– concrete composite slabs. The proposed prefabricated composite slabs with an interlocking system are analysed using finite element (FE) modelling and compared the results with the conventional slabs. Three-dimensional (3D) finite element (FE) models using ANSYS software were developed to investigate the load-deformation behaviour of prefabricated composite slabs with or without the interlocking connection systems.

Key Words:- Performance evaluation, Prefabricated slab, Finite Element Method, ANSYS workbench.

1. INTRODUCTION

The construction industry is increasingly turning to prefabricated techniques to address persistent challenges such as lengthy construction schedules, cost overruns, and variability in quality. Prefabrication involves the manufacturing (Figure 1.1 and Figure 1.2) of building components off-site in controlled factory conditions before transporting them to the construction site for assembly. This approach offers several advantages over traditional construction methods, including reduced construction time, minimized waste generation, and enhanced safety on-site. Prefabricated slab panels, in particular, have gained popularity in building construction due to their ability to significantly reduce construction time and costs. These panels, typically made of reinforced concrete or steel, are fabricated to precise specifications off-site and then transported to the construction site for installation.

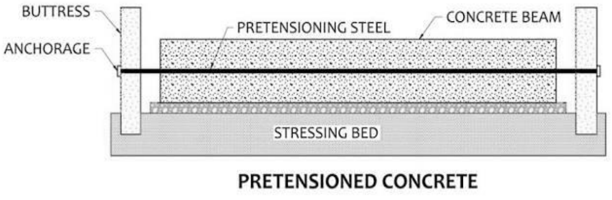


Figure 1.1: *Pre-tensioned concrete*

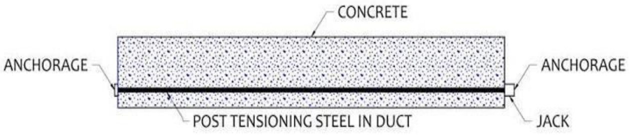


Figure 1.2: *Post-tensioned concrete*

The various types, such as beam and block systems and hollow core slabs (Figure 1.3), each offering unique benefits in terms of structural support and efficiency. Finite Element Analysis (FEA) is a crucial tool used to study the structural behaviour of prefabricated slabs (Figure 1.3). By utilizing FEA, engineers can gain insights into stress distribution, deformation characteristics, and potential failure mechanisms, allowing for optimized design and performance assessment.

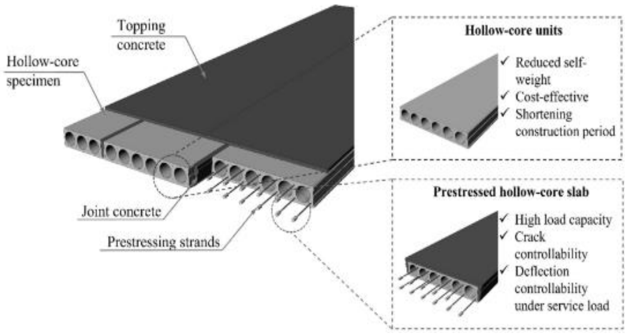


Figure 1.3: *Pre-stressed Hollow Core Slab*

ANSYS software, renowned for its robust computational capabilities, is commonly employed for such analyses, providing accurate predictions of slab performance under different loading conditions.

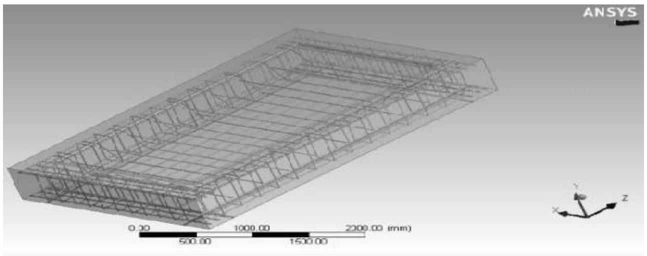


Figure 1.4: *Conventional type of rectangular RCC slab*

The objective of studies evaluating prefabricated slab performance is to provide comprehensive insights into their behaviour under various loads and conditions. By understanding how these slabs perform structurally, engineers can make informed decisions to enhance their design, construction, and overall efficiency. Ultimately, the goal is to promote the widespread adoption of prefabricated construction methods and contribute to the development of more sustainable and resilient built environments. The Overall objective to evaluate the performance of prefabricated slabs under loading conditions using ANSYS finite element analysis software. To investigate the stress-strain behaviour of prefabricated slabs under static loading conditions. To determine the deflection characteristics of prefabricated slabs under service loads. To provide recommendations for improving the design and performance of prefabricated slabs. The Scope of the Study This project deals about Software analysis of (Finite Element Method/Analysis) ANSYS Workbench in Pretensioned slab.

2. LITERATURE REVIEW

Nonlinear coupled thermal-structural analysis of precast concrete Noor Azim Mohd. Radzi, (2023) Muhaimin Kamal, Shareen Azuha, Nur Izzatie Aqilah Mansor, Roszilah Hamide carried out research on presents a nonlinear coupled thermal structural analysis using ANSYS Workbench to determine precast beam-to-column connections' thermal and structural behavior. Three precast connection models, a concrete corbel, a concrete nib, and an inverted E steel nib, are exposed to ambient and cellulose fire curves. Firstly, the precast connection models are verified based on the previous experimental result at ambient temperature. Then, the verified precast connection models are exposed to the cellulose fire curve for two hours before being loaded to failure. The results are compared with the recent experimental fire test conducted by the authors. Based on the result, finite element models at ambient temperature were validated with a percentage difference of less than 10%. However, finite element models at high temperatures were not verified due to the percentage difference exceeding 10 %. The significant difference was due to the nonuniformity of sample dimensions and different test setups in the previous experiment. Finite element models for concrete corbel and inverted E steel nib have a higher stiffness than the experimental sample.

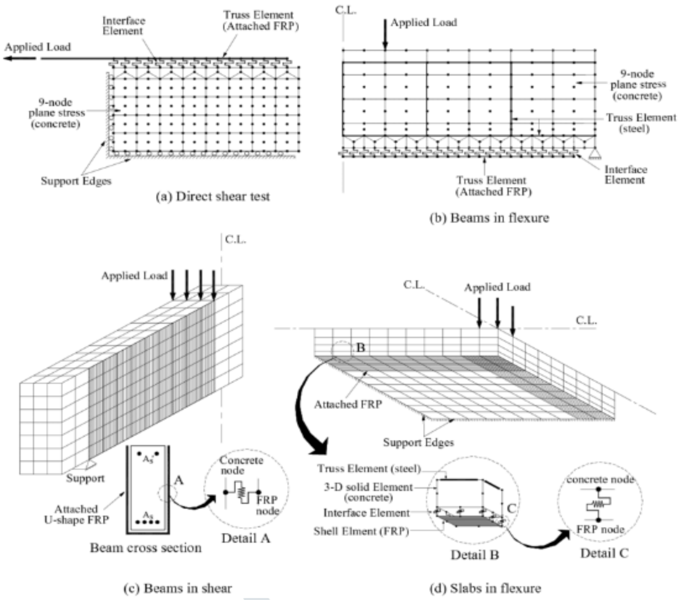
Nonlinear Thermo-Structural Analysis of Composite Slabs. This study investigates the fire resistance of lightweight concrete (LWC) composite slabs with trapezoidal steel decking, an area with limited existing research. Twelve experimental fire tests were conducted on slabs of varying compositions, with a focus on LWC slabs. Key observations included vaporization of water within LWC, debonding between steel decking and concrete, and changes in material properties affecting structural performance. Findings indicated that while lighter slabs didn't meet minimum structural requirements, they demonstrated improved thermal performance due to lower thermal transmittance. Advanced numerical models were developed to predict thermal and structural behavior, incorporating factors like vaporization and debonding. Comparison between experimental and numerical results showed good agreement. The study also compared LWC composite slabs with normal concrete (NC) ones, yielding practical insights. **Nonlinear Analysis of Reinforced Concrete Column with ANSYS** V. S. Pawar1, P. M. Pawar2 (2016) Different methods have been utilized in order to assess the behavior and phenomenon of typical failures like flexural, shear, torsion, buckling etc. of the Reinforced concrete structures. Typically, the behavior of reinforced concrete is studied by full-scale experimental investigations. With the invention of sophisticated numerical tools for analysis like the finite element method (FEM), it has become possible to model the complex behavior of reinforced concrete members using Finite Element modelling. In the present paper models of reinforced concrete columns subjected to axial symmetric and eccentric loading are used. Nonlinear finite element analysis is used to analyse reinforced concrete columns up to failure with FEM software ANSYS. Reinforced concrete column subjected to the axial symmetric loading, are modeled considering the frequent use in the laboratory. **Finite Element Modelling of Pre-Stressed Hollow Core Slabs** Ali N. Deeb,(2015) carried out studied on an analytical model is presented to study the shear behavior of prestressed hollow core slabs. The study is conducted with respect to the shear considering a concentrated line load (in absence of concrete topping and filling of cores) using finite elements method adopted by the computer program ANSYS. The finite element models are developed using a smeared cracking approach for the concrete and three-dimensional link elements for the prestressing strands. The ANSYS finite element analysis results are compared with the experimental data of two prestressed hollow core slabs. The comparisons are made for load-deflection curves, failure load and crack pattern. The accuracy of the finite element models is assessed by comparison with the experimental results, which are to be in good agreement. **Behavior of Prefabricated Full-Depth Precast Concrete** Rajai Al-Rousana (2020) explained about reports on results and findings obtained from a nonlinear finite element analysis (NLFEA) of a prototype prefabricated full-depth precast concrete bridge deck panel system under different level of prestressing force. The NLFEA were validated with experimental results obtained from full-scale testing of the prototype bridge system. The benefits of the NLFEA can be highly appreciated when visualizing the substantial time and cost savings, the ability to change any parameter of interest, and the capability of demonstrating any interesting response at any load value and at any location in the system. The most attractive results were: (i) the system is capable of withstanding and maintaining its integrity under eight times the simulated AASHTO truck service load without considerable reduction in its ultimate strength capacity and stiffness, (ii) The NLFEA showed that the live load-induced bond stresses for 4 lines prestress level was almost 0.74 times the live load-induced bond stresses for 6 lines prestress level.

Finite Element Analysis of Composite Precast Roof Panel

S. Nalini's research focused on prefabricated composite roof panels, which offer economic and construction advantages. These panels have a high strength-to-weight ratio, reducing seismic forces and providing good thermal insulation. Using Finite Element Analysis (FEA) with ANSYS software, Nalini simulated the panels' static flexure behavior. The study emphasized the panels' structural performance under different loading conditions, aiming to optimize construction processes and improve efficiency. **To Study Impact of Prefabrication on Profitability Over Traditional Construction** Nitesh J. Ramchandani and Prof. Hemant Salunkhe's research delves into the impact of prefabrication on construction profitability compared to traditional methods. They highlight how prefabrication significantly affects time and cost in building projects, emphasizing its role in reducing construction waste. Their study proposes exploring the potential impact of prefabrication on waste reduction and handling activities. Prefabricated structures offer advantages like uniformity, strength, and thermal properties, contributing to cost-effectiveness in construction. The research aims to identify new technologies or methodologies for improving the construction industry, particularly focusing on the cost-effectiveness of precast concrete construction.

3. METHODOLOGY

Data Review and Collection to evaluate the performance of prefabricated slabs, relevant data collected and reviewed from scholarly articles and journal papers. This process involved identifying keywords related to prefabricated slab performance. The collected data then be incorporated into the presentation report to establish a comprehensive understanding of prefabricated slab performance. The focus on studies that employ the finite element method (FEM) to evaluate the performance of prefabricated slabs in building applications. **Primary and Secondary Data** for performance evaluation of prefabricated slab panel, we have collected primary data which come from a literature review of academic journals and research papers to understand established comprehensive studies of prefabricated slab panel. Secondary data can include industry articles, expert consultations, and consultant reports to gain insights into real-world applications and performance considerations. By combining these resources, we have established a well-rounded methodology for assessing prefabricated slab performance. **Prefabricated Slab Designing f**or Prefabricated slab design it’s all about defining load requirements, selecting appropriate slab dimensions and material properties, and detailing reinforcement layout and connection points, following relevant building codes and engineering software analysis using finite element modelling (ANSYS Workbench).



(a) Direct shear test (b) Beams in flexure

(c) Beams in shear (d) Slabs in flexure

Figure 3.1: *Finite Element Model*

Software Modelling While many finite element method (FEM) (Figure 3.1) software options exist, ANSYS stands out for evaluating prefabricated slab performance. ANSYS offers robust capabilities for simulating complex material behaviour, incorporating pre-existing connection details, and visualizing stress distribution. All crucial for accurate prefabricated slab analysis. **Discretization of Model** Ansys software was used to discretize the prefabricated slab model. This involved dividing the complex geometry of the slab into a mesh of smaller elements, such as squares or triangles. The element size and type were chosen based on the geometry of the slab and the desired level of accuracy in the analysis. The discretized model was then used to perform structural analysis, such as finite element analysis, to determine the stresses, strains, and deformations in the slab under various loading conditions.

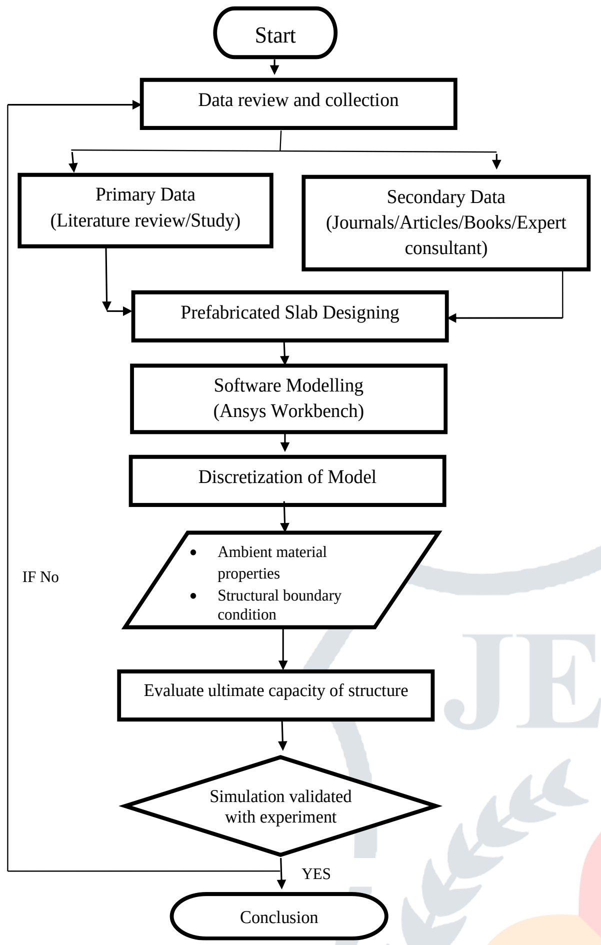


Figure 3.2: *Methodology*

Ambient Materials and structural Boundary Conditions ANSYS software will be used to define the material properties of the prefabricated slab and establish the structural boundary conditions. The software's material library will be employed to assign accurate material properties to the slab, considering factors like concrete strength and density. Additionally, ANSYS will be used to establish appropriate boundary conditions that simulate the real-world support conditions of the slab, ensuring a realistic representation of its structural behaviour. **Evaluate Ultimate capacity of structure** The ultimate capacity of the prefabricated slab will be evaluated using ANSYS finite element analysis software. The analysis will consider material properties, slab geometry, and support conditions to determine the maximum load the slab can withstand before failure. This will inform the design process to ensure the slab meets all structural requirements. **Comprehensive approach for designing prefabricated slabs** In conclusion, in this study investigated the design procedure for prefabricated slabs using ANSYS software for analysis we established a comprehensive approach for designing prefabricated slabs. This approach combines theoretical knowledge with practical finite element analysis, ensuring efficient and reliable design for prefabricated concrete slabs.

4. Manual Design

The Manual Design followed for regular reinforcement of slab to provided at mid span can be curtailed at $0.1l_x$ or $0.1l_y$ from support as IS 456-2000.

Table 4.1: *Span Depth Ratio of Slab*

l_y/l_x	1.0	1.1	1.2	1.3	1.4	1.5	1.7	2.0	2.5	3.0
α_x	0.06	0.07	0.08	0.09	0.10	0.10	0.11	0.12	0.12	0.12
α_y	0.06	0.06	0.06	0.06	0.05 1	0.05	0.04	0.03	0.02	0.01

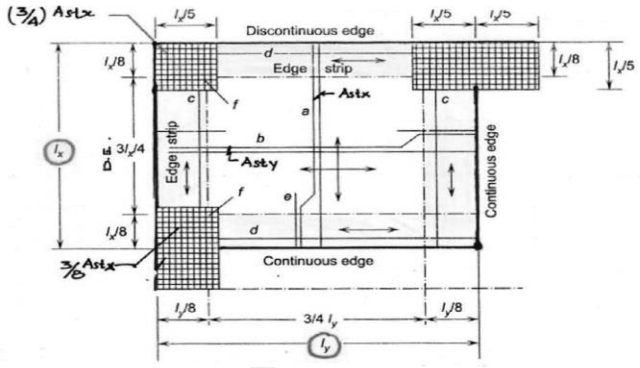


Figure 3.2: *Tow – way slab Design*

So, the minimum crack width prediction is approximately 1.81 mm. **Reinforcement details:** Reinforcement details for the slab include HYSD bars with a spacing of 110 mm. Centre to centre, provided at the shorter span only to resist bending and shear forces effectively. Additionally, 15 mm diameter pretension bars are incorporated into the slab design with a spacing of 600 mm centre to centre, also at the shorter span, to enhance tensile strength and minimize cracking. **Material Properties and Specifications:** ANSYS allows material specification through defining properties relevant to your simulation. This can involve mechanical properties (elasticity, strength), thermal properties (conductivity, capacity), or even user-defined material behavior for complex materials. You can either build your material from scratch or leverage built-in libraries and material databases for common materials.

Material properties for M40 Grade of steel concrete

i) Material Field Variables : Density - 0.024 Kg/m^3

ii) Isotropic Secant Coefficient of Thermal Expansion Coefficient of Thermal Expansion 1.4E-05 C^-1

iii) Isotropic elasticity Derive from – Young's modulus and poisson's ratio. Young's modulus - 2.5E+10 Pa, Poisson's Ratio - 0.18, Bulk modulus - 1.3021E+10 Pa, Shear modulus - 1.0593E+10 Pa, Compressive yield strength - 3.8E+07 Pa Compressive ultimate strength- 4E+07Pa

Material properties for structural steel: ANSYS requires defining material properties to accurately simulate real-world behavior. These properties encompass various aspects like elasticity, strength, density, thermal conductivity, and can even account for complex material behavior depending on the analysis type.

Material properties for Fe 500 Grade of Steel

Density: 7850 kg/m³

Thermal Conductivity: 60.5 W/m- C

Coefficient of Thermal Expansion: 1.2E-05C^-1

Isotropic Elasticity: Derived from Young's Modulus, which is 2.1E+05 MPa, Poisson's Ratio: 0.3, Bulk Modulus: 1.75E+11 Pa, Shear Modulus: 8.0769E+10 Pa, Tensile Yield Strength:

JETIR

500 MPa, Compressive Yield Strength: 500 MPa, Tensile Ultimate Strength: 560 MPa.

5. Results and discussion

The Solid65 element is used to model the concrete. This element has eight nodes with three degrees of freedom at each node – translations in the nodal x, y, and z directions. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. A schematic of the element is shown in Figure 5.1

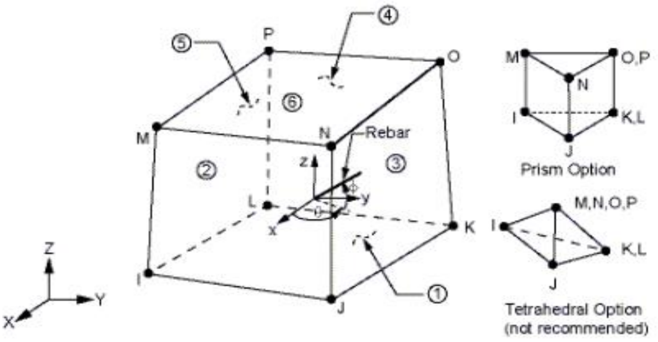


Figure 5.1 *FEA Solid 65 Element*

Steel Reinforcement: A Link180 element is used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degrees of freedom – translations in the nodal x, y, and z directions. This element is also capable of plastic deformation. This element is shown in Figure 5.2.

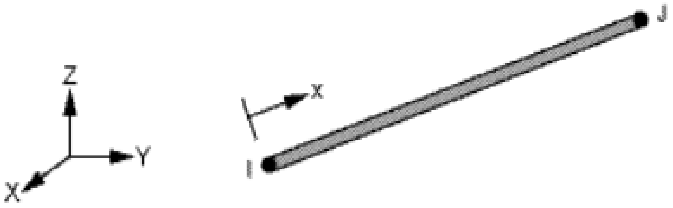


Figure 5.2 *Link 180 element*

Steel Plates and Supports: Steel plate and supports are modeled using element called Solid185. This element is defined by eight nodes having three degrees of freedom at each node translations in the nodal x, y, and z directions. The element is capable of plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. Solid185 is available in two forms: Homogeneous Structural Solid (default); and Layered Structural Solid. Homogeneous Structural Solid with simplified enhanced strain formulation is used to model steel plate for application of load.

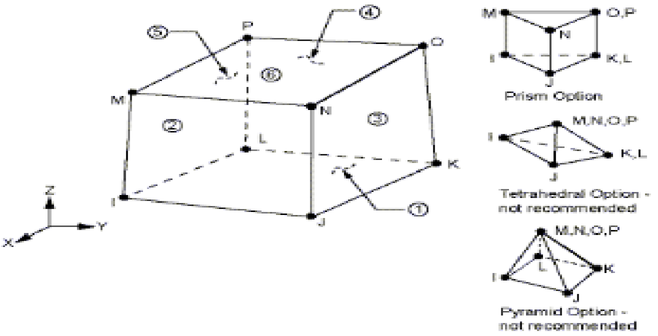


Figure 5.3 *Solid185 Element (Homogeneous Structural Solid) in ANSYS*

Procedure for FEA in Ansys software: A. Initialization: Commence the design process by accessing the "Static Structure" module within the ANSYS interface, the chosen platform for the structural analysis and simulation of the slab. **B. Engineering Data Input:** Navigate to the engineering data section to meticulously input essential material properties pertinent to the project, encompassing parameters for concrete, structural steel, and pre-tensioning steel, ensuring comprehensive representation of the project's requirements. **C. Geometry Definition:** Employ geometric drawing tools to meticulously delineate the slab's geometry, adhering to precise specifications. Configure a 2D representation of the precast slab, with dimensions of 4.3 by 5.5 meters in the XY direction, and establish a thickness of 170 millimetres using the exclude option for accurate depiction. **D. Reinforcement Incorporation:** Strategically integrate reinforcement elements to enhance structural robustness. Begin by delineating reinforcement for the shorter span, meticulously sketching out the layout to ensure comprehensive coverage. Subsequently, replicate this process for the longer span, adjusting parameters such as spacing and copy numbers to optimize reinforcement distribution. **E. Connection Establishment**: Facilitate cohesive integration by establishing connections between structural components, ensuring seamless load transfer and structural integrity. **F. Mesh Generation:** Generate a structured mesh for the slab panel to facilitate accurate structural analysis. This step is pivotal in ensuring precise simulation results reflective of real-world behavior. **G. Boundary Condition Application:** Apply boundary conditions, encompassing the application of pressure and designation of fixed supports, to emulate realistic structural responses. This step is imperative for obtaining accurate analytical results. **H. Solution Specification:** Define the desired analytical outcomes within the solution phase, encompassing parameters such as stresses, displacements, and reactions. These results are integral for evaluating the structural performance and informing subsequent design iterations. There are three techniques that exist to model steel reinforcement in finite element models for reinforced concrete: the discrete model, the embedded model, and the smeared model. In the work presented in this paper, discrete modeling technique is used for modeling the reinforcement. The reinforcement in the discrete model (Fig.19) uses bar or beam elements that are connected to concrete mesh nodes. Therefore, the concrete and the reinforcement mesh share the same nodes and concrete occupies the same regions occupied by the reinforcement.

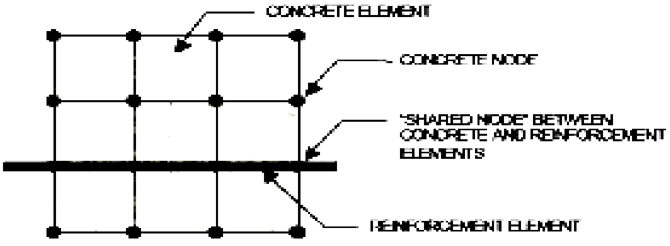


Figure 5.4 *Discrete Models for Reinforcement*

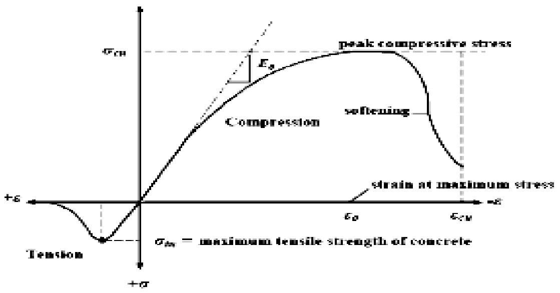


Figure 5.5 *Typical uniaxial compressive and tensile stress strain curve for concrete*

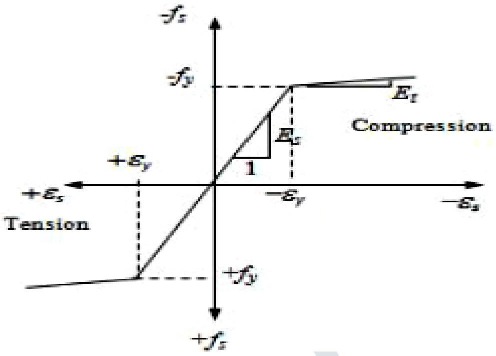


Figure 5.6 *Strain curve for the steel reinforcement*

Table 4.2: *Materials Properties*

Properties	Steel
Ultimate Tensile Stress (Mpa)	590
Ultimate Strain	0.003
Modulus of Elasticity (Gpa)	0.028
Poisson's Ratio	210
Density (kg/m3)	0.23

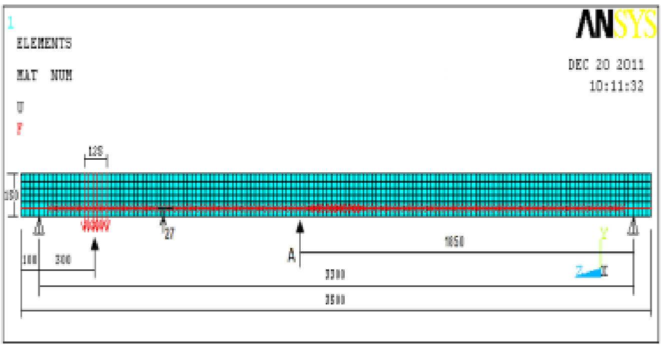
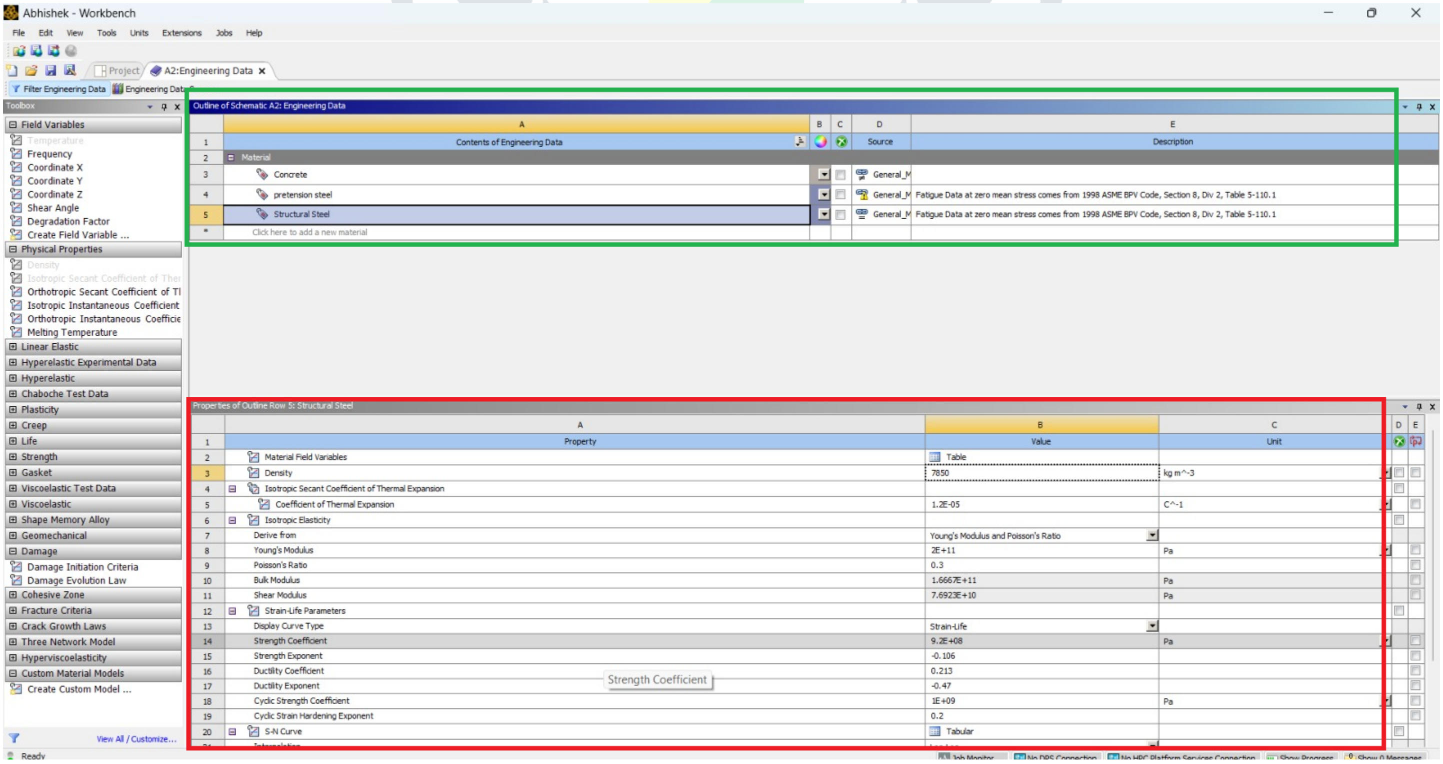


Figure 5.7 *Crack Paten due to flexural loading*

5. CONCLUSION

In conclusion, the evaluation of prefabricated slabs for this building project revealed key performance findings - positive or negative aspects related to strength, installation, cost, etc.. These findings suggest that prefabricated slabs can be a positive assessment - suitable, viable, efficient option for construction, depending on factors like e.g., building type, project scale. Further research on prefabricated slab evaluation could provide even more comprehensive insights into their effectiveness in the building industry. The finite element method (FEM) analysis successfully evaluated the performance of prefabricated slabs within the building model. This approach provided valuable insights into the slab's behavior under various loading conditions, allowing for assessment of its strength, deflection, and potential cracking patterns. The FEM analysis offers a reliable and efficient tool for optimizing prefabricated slab design in future building projects.

APPENDIX



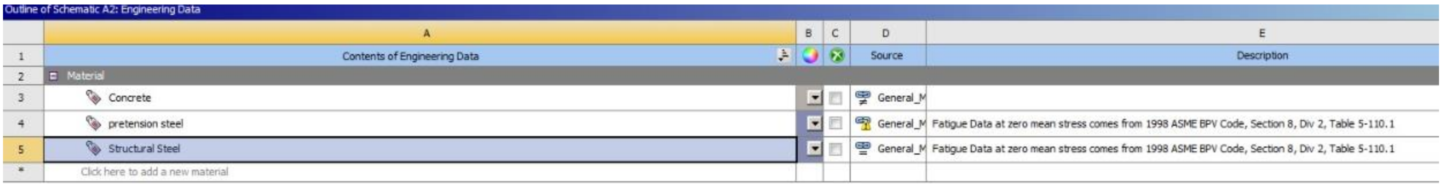
Outline of Schematic A2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Concrete		☐	General_M	
4	pretension steel		☐	General_M	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
5	Structural Steel		☐	General_M	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 5: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3	☐	☐
4	Isotropic Secant Coefficient of Thermal Expansion			☐	
5	Coefficient of Thermal Expansion	1.2E-05	C^-1		☐
6	Isotropic Elasticity			☐	
7	Derive from	Young's Modulus and Poisson's Ratio			
8	Young's Modulus	2E+11	Pa		☐
9	Poisson's Ratio	0.3			☐
10	Bulk Modulus	1.6667E+11	Pa		☐
11	Shear Modulus	7.6923E+10	Pa		☐
12	Strain-Life Parameters			☐	
13	Display Curve Type	Strain-Life			
14	Strength Coefficient	9.2E+08	Pa		☐
15	Strength Exponent	-0.106			☐
16	Ductility Coefficient	0.213			☐
17	Ductility Exponent	-0.47			☐
18	Cyclic Strength Coefficient	1E+09	Pa		☐
19	Cyclic Strain Hardening Exponent	0.2			☐
20	S-N Curve	Tabular		☐	

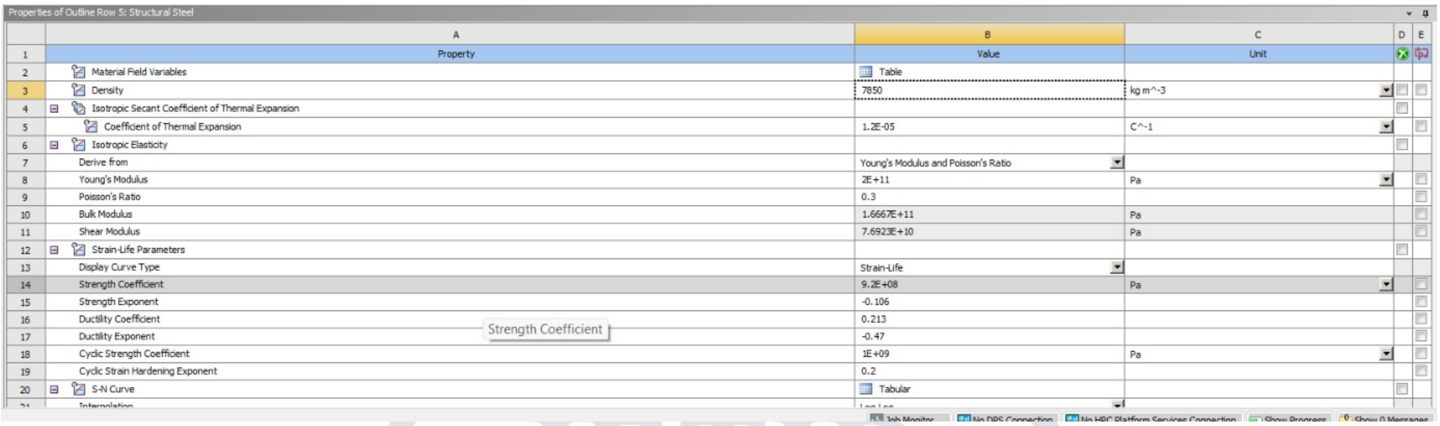
a. Ansys Workbench Interface *(Materials Assigning from library, material properties needed to be uploaded from experimental values)*



Outline of Schematic A2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	⊟ Material				
3	Concrete		☐	General_M	
4	pretension steel		☐	General_M	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
5	Structural Steel		☐	General_M	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

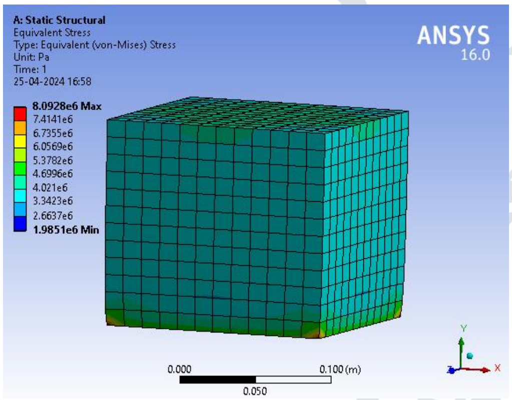
b. *Material Assigning from library*



Properties of Outline Row 5: Structural Steel

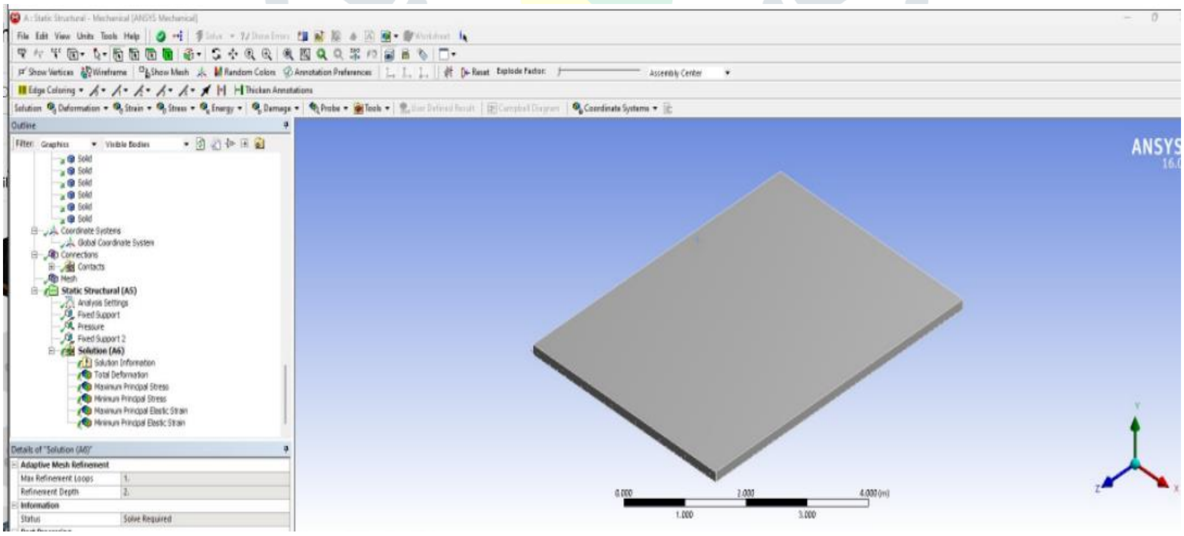
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3	☐	☐
4	⊟ Isotropic Secant Coefficient of Thermal Expansion			☐	
5	Coefficient of Thermal Expansion	1.2E-05	C^-1		☐
6	⊟ Isotropic Elasticity			☐	
7	Derive from	Young's Modulus and Poisson's Ratio			
8	Young's Modulus	2E+11	Pa		☐
9	Poisson's Ratio	0.3			☐
10	Bulk Modulus	1.6667E+11	Pa		☐
11	Shear Modulus	7.6923E+10	Pa		☐
12	⊟ Strain-Life Parameters			☐	
13	Display Curve Type	Strain-Life			
14	Strength Coefficient	9.2E+08	Pa		☐
15	Strength Exponent	-0.106			☐
16	Ductility Coefficient	0.213			☐
17	Ductility Exponent	-0.47			☐
18	Cyclic Strength Coefficient	1E+09	Pa		☐
19	Cyclic Strain Hardening Exponent	0.2			☐
20	⊟ S-N Curve	Tabular		☐	

c. *Experimetal Values uploaded to respective materials*



Sample analysis for Concrete Cube

Realtime simulation of crack patten in Ansys Workbench



Details of "Solution (A6)"

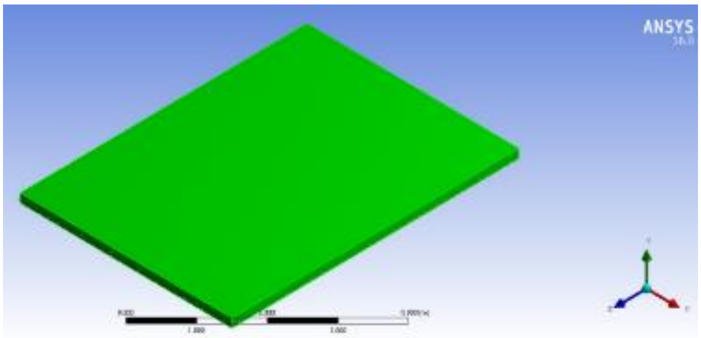
Adaptive Mesh Refinement

Max Refinement Loops	1.
Refinement Depth	2.

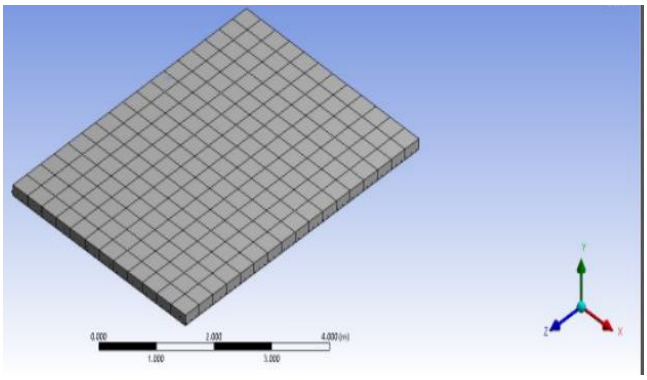
Information

Status	Solve Required

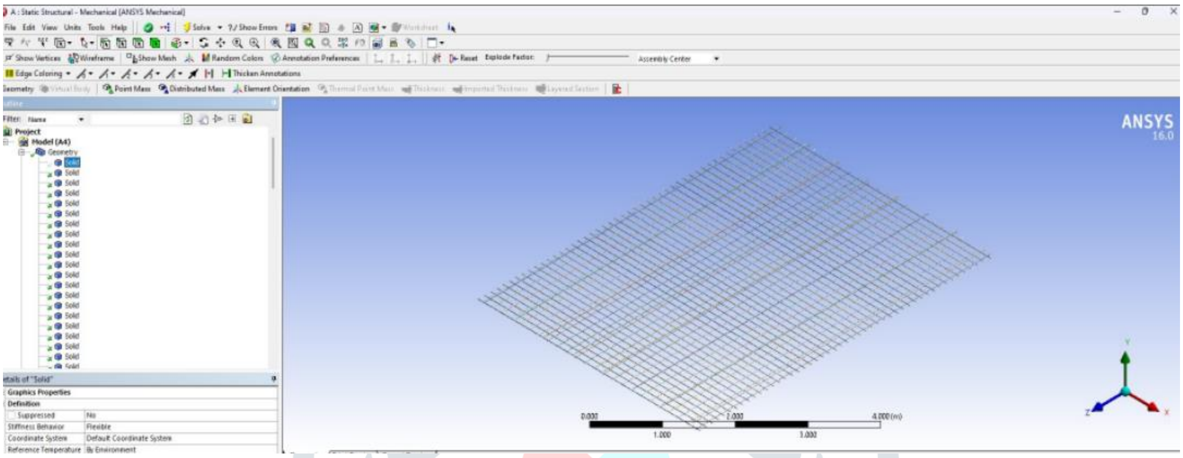
d. *Prefabricated Slab model*



e. Prefabricated Slab (FEA Solid) model



f. Discritized model



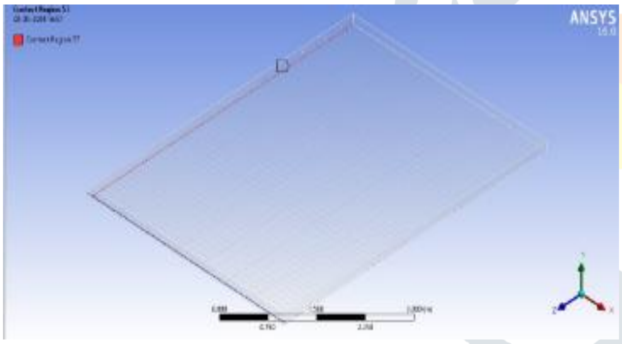
Details of "Solid"

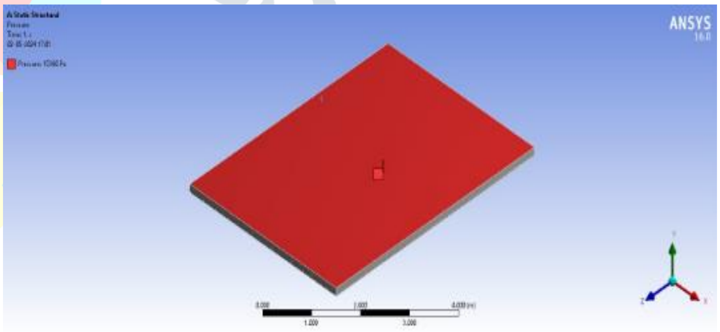
Graphics Properties

Definition

Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment

g. Ordinary Reinforcement and Prestress (Strand or Cable)





h. Supporting (boundary) Conditions *i. Loading by Pressure Conditions*

Aknowledgement

The authors would like to express their gratitude to all the faculty members from civil engineering department who have helped their research work in timely completion and valuable suggestions during the technical discussions.

REFERENCES

1. Nonlinear coupled thermal-structural analysis of precast concrete beam-to-column connections at high temperatures Noor Azim Mohd. Radzi, Muhaimim Kamal, Shareen Azuha, https://jresm.org/archive/resm2023.610ma1220.pdf

2. Nonlinear Analysis of Reinforced Concrete Column with ANSYS by V. S. Pawar1, P. M. Pawar2 (2016) https://www.irjet.net/archives/V3/i6/IRJET-V3I6419.pdf

3. Finite Element Analysis of Composite Precast Roof Panel Nalini (November 2014)https://www.ijert.org/research/finite-element-analysis-of-composite-precast-roof-panelunder- static-flexure-IJERTV3IS110506.pdf

4. To Study Impact of Prefabrication on Profitability Over Traditional Construction Nitesh j Ramchandani and Prof. Hemant Salunkhe (July 2019) https://www.ijeast.com/papers/533-540,Tesma403,IJEAST.pdf

5. F.A. Tavarez, .Simulation of Behavior of Composite Grid Rein-forced Concrete Beams Using Explicit Finite Element Methods., Master of Science Thesis, University of Wisconsin. Madison, Madison, Wisconsin, 2001.

6. Nonlinear Thermo-Structural Analysis of Lightweight Concrete and Steel Decking Composite Slabs. Juan Enrique Martínez-Martínez (2022) https://www.mdpi.com/1834308

7. Finite Element Modeling of Pre-Stressed Hollow Core Slabs Ali N. Deeb,(2015) https://iccae.journals.ekb.eg/article_44221.html

8. Issa, M.M., Salas, J.S., Shabila, H.I., and Alrousan, R.Z. Composite Behavior of Prefabricated Full-Depth Precast Concrete Panels Installed on Precast Prestressed Girders. PCI Journal 2006; 51(5): 132-145

9. PCI Manual for the design of hollow core slabs, second edition (1998) By Precast/Prestressed Concrete Institute, ISBN 0-937040-57-6, U.S.A ANSYS, 2004.

10. User’s Manual, Version, (11.0). Anthony, J., Wolanski, 2004.

11. Flexural behavior of reinforced and pre-stressed concrete beams using finite element analysis, Thesis submitted. Milwaukee, Wisconsin

12. Geraldine, C.S.; LEW, H.S. Performance of precast concrete beam-to-column connections subject to cyclic loading. PCI J. 1991, 36, 56–67.

13. Cai, J.; Feng, J.; Wang, Z.; Zhu, H. Research on seismic performance of prefabricated prestressed concrete assembled monolithic frame. J. Sun Yat Univ. (Nat. Sci. Ed.) 2009, 48, 136–140. (In Chinese)

14. Kaya, M.; Arslan, A.S. Analytical modeling of post-tensioned precast beam-to-column connections. Mater. Des. 2009, 30, 3802–3811. [CrossRef]

15. Fatema T. Study on Connection Between Precast Concrete Beam and Cast-in-Situ Column in Prefabricated Building Frames. J Eng Appl Sci. 2006;1(1):33-8.