INVESTIGATION OF SLAB-COLUMN CONNECTION BY FINITE ELEMENT ANALYSIS

By

Muhammad Ali Shaikh

(2004 – NUST – MS PhD – STR – 11)

A thesis submitted in partial fulfillment of
the requirements for the degree of
Master of Science

In

Department of Civil Engineering
National Institute of Transportation

National University of Sciences & Technology

Rawalpindi, Pakistan

(2006)

This is to certify that the

thesis titled

INVESTIGATION OF SLAB-COLUMN CONNECTION BY FINITE ELEMENT ANALYSIS

Submitted by

Muhammad Ali Shaikh

Has been accepted towards the partial fulfillment

of

the requirements

for

Master of Science in Civil Engineering

____

Brigadier Dr. Tayyeb Akram, Ph.D (USA)

National Institute of Transportation, Risalpur

National University of Sciences & Technology, Rawalpindi
Pakistan

INVESTIGATION OF SLAB-COLUMN CONNECTION BY FINITE ELEMENT ANALYSIS

By

Muhammad Ali Shaikh

(2004 – NUST – MS PhD – STR – 11)

A Thesis

of

Master of Science

Submitted to the

National Institute of Transportation

of

National University of Sciences & Technology

In partial fulfillment of the requirements

For the degree of

Master of Science in Civil Engineering

2006

DEDICATED TO

MY GRAND PARENTS, PARENTS AND

MY SISTERS

ACKNOWLEDGEMENT

I am thankful to All Mighty Allah who gave me the strength and courage to accomplish this research work.

I would like to express my foremost gratitude and sincere appreciation to Brig. Dr. Tayyeb Akram, Advisor and Committee Chairman, for providing invaluable support and dedicated encouragement to me in my research work. His helpful suggestions, comments and advice are the impetus behind the successful completion of this work. I am also profoundly grateful to Lecturer Shaukat Ali Khan for his valuable guidance, suggestions and advice.

I am also thankful to Lecturer Shazim Ali Memon for his continuous help and support during the course of this thesis research work.

In the end, I pay my earnest gratitude with sincere sense of respect to my parents for their unending support, encouragement, prayers and patience.

ABSTRACT

In the flat slab-on-column construction, high transverse stresses are concentrated at the slab-column connection, which lead to a non-ductile, sudden failure and results in the accidental collapse of flat slab buildings. The major parameters affecting the slab-column connection are the concrete strength, slab thickness, slab reinforcement and aspect ratio of column.

The application of numerical analysis methods based on the finite element theory for solving practical tasks allow to perform virtual testing of structures and explore their behavior under load and other effects in different conditions taking into account the elastic and plastic behavior of materials, appearance and development of cracks and other damages (disintegrations), and finally to simulate the failure mechanism and its consequences.

In this study, the models are developed to carry out the finite element analysis of slab-column connection by varying the slab thickness and slab confining reinforcement and to investigate their effect on the deflection and load carrying capacity.

The finite element analysis results indicate that by increasing the slab thickness, the deflection and the load carrying capacity of slab-column connection increases, more over, by increasing the slab confining reinforcement, the deflection decreases where as the load carrying capacity increases.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

Chapter 1

INTRODUCTION

1.1 GENERAL

A flat plate floor system is essentially a flat slab floor with the drop panels and column capitals omitted, so that a floor of uniform thickness is carried directly by prismatic columns. The flat slabs have been found to be economical and otherwise advantageous for such uses as apartment buildings, where the spans are moderate and relatively loads are light. The construction depth for each floor is held to the absolute minimum, with resultant savings in the overall height of the buildings. The smooth underside of the slab can be painted directly and left exposed for ceiling, or plaster can be applied to the concrete. Minimum construction time and low labor costs result from the very simple formwork. The problems associated with flat slab construction are shear stresses near the column and the transfer of moments from slab to column (Nilson et al. 2003).

The flat slab-on-column construction is subjected to high transverse stresses concentrated at the slab-column connection, which can lead to a non-ductile, sudden and brittle punching failure and results in the accidental collapse of flat slab buildings (Polak 2005).

Dilger (2000) has identified the most important parameters for the resistance of slab-column connection subjected to concentric loading as:

- Concrete strength
- Reinforcement ratio
- Aspect ratio of the supporting column
- Perimeter to thickness ratio
- Size effect, i.e., slab thickness

While the parameters of lesser importance are:

- Grade of reinforcement
- Arrangement of reinforcement
- Concrete cover

- Boundary conditions

The research methods of concrete structures are divided into three categories: experimental research, analytical empirical research, and analytical numerical research (computer simulation). The traditional practical design methods are generally based on the analytical empirical approach. Recently, the implementation of computer simulation methods in the field of analysis of building structures is becoming an effective tool used not only by scientific researchers but also by practicing engineers.

The application of numerical analysis methods based on the finite elements theory for solving practical tasks allow to perform virtual testing of structures and explore their behavior under load and other effects in different conditions taking into account the elastic and plastic behavior of materials, appearance and development of splints, cracks and other damages (disintegrations), and finally to simulate the failure mechanism and its consequences. It is very important that before practical application, finite elements analysis (FEA) results should be verified and validated comparing the analysis results with reliable experiment data (Vainiunas et al. 2004).

1.2 PROBLEM STATEMENT

The slab-column connection prematurely fails in the joint region, exhibiting a substantial reduction in column load carrying capacity. Such joint failure may entirely be prevented by properly considering the restraint to the joint by the surrounding slab, the amount of slab reinforcement, the aspect ratio (ratio of slab thickness to column dimensions), and the column and slab concrete strengths.

This research study is carried out to review the effect of varying slab thickness and slab confining reinforcement on slab-column connection by using the analytical numerical approach i.e., computer modeling and compare the obtained results to that of experimental work carried out earlier and further to study the effects of varying slab thickness and the size of slab confining reinforcement.

1.3 SCOPE

The scope of this research is to study the behavior and failure criterion of slab-column connection subjected to vertical loadings with variation in slab thickness and steel reinforcement in the slab region. This is done by varying slab thickness and slab confining reinforcement size; whereas, other parameters are kept constant.

For the purpose of analytical study of slab-column connection, the analysis is carried out using finite element method (FEM) program named "ADINA (Automatic Dynamic Incremental Nonlinear Analysis)" (ADINA R&D, Inc. 2001).

1.4 OBJECTIVES

The objectives of research are:

- To study the size effect, i.e., the varying slab size (thickness).
- To study the effect of slab confining reinforcement.
- To understand the behavior of slab-column connection for varying slab size and confining steel bar diameters.
- To study the cracking and crushing pattern of the slab-column connection.
- To compare the FEM analysis results with experimental work.

Chapter 2

LITERATURE REVIEW

2.1 GENERAL

Researchers have worked on various problems in slab-column joint, still it is quite new subject and there is lot of work needed to be done. The proceeding paragraphs mention the work done on various aspects of the slab-column joint, as well as the methods utilized to carry out the studies.

2.2 EFFECT OF DIFFERENT PARAMETERS

2.2.1 The Effect of Slab Concrete Strength

According to Marzouk and Hussein (1991), the catastrophic nature of the failure exhibited by the reinforced concrete flat plates when subjected to concentrated loads has concerned engineers for many years. This localized shear-type failure occurs in the immediate vicinity of the load is usually referred to as punching shear failure. A failure of this type is undesirable since, for most practical design cases, an over all yield mechanism will not develop before punching. The test results showed that the stiffness was increased with the increase in concrete strength but at a rate much less than the rate of $\sqrt{f_c^{'}}$ values. The high strength concrete slabs exhibited more brittle failure than the normal strength concrete.

According to Ngo (2001), the reinforced concrete flat slab system is widely used structural system. The catastrophic nature of the failure exhibited at the connection between the slab and column has concerned engineers. This area becomes the most critical area as far as the strength of slab is concerned due to the concentration of high bending moments and shear forces. The failure load may be considerably lower than the unrestrained flexure capacity of the slab. The use of high strength concrete improves the punching shear resistance allowing higher forces to be transferred through the slab-column connection.

2.2.2 The Effect of Surrounding Slab

Gamble and Klinar (1991) conducted the research on the edge and interior slab-column connection types. The results obtained by them showed that in case of edge connection, the concrete in the slab between the ends of the column attempted to fail by expanding horizontally, but this expansion was resisted along the three faces of column by the surrounding slab and its embedded reinforcement. The lateral expansion of slab caused vertical splitting cracks in the columns, starting at the slab surface and extending several inches to column. While in case of interior connection, since there was no free edge, the internal cracking development was not well known. Eventually several cracks formed in the slab as the concrete between the columns ends attempted to expand horizontally. Cracks radiated out ward from the column and the bottom of slab was pushed out significantly more than top, resulting in saucer like deflected shape. Although the top and bottom cracking pattern were very similar, the bottom cracks were generally much larger.

According to Shah et al. (2005), the effective strength of interior slab-column connection specimens having surrounding slab confinement was generally more than the specimens having no surrounding slab confinement and even with very high slab load that considerably reduces the joint strength and ductility in the presence of surrounding slab was substantial. The sandwich columns having no surrounding slab confinement have always lower effective strength than those with surrounding slab confinement with or without slab load.

2.2.3 The Effect of Slab Reinforcement Ratio/Amount

Gamble and Klinar (1991) conducted the test of interior slab-column connection by providing spiral steel in the slab in one of specimens and indicated that the failure load of the specimen increased as compared to the specimen having no spiral in slab portion.

Marzouk and Hussein (1991) tested four different series of specimen and indicated that within a given series, in spite of the increase in the stiffness as the steel reinforcement is increased, the ductility is decreased.

Shah et al. (2005) in the experimental investigation of slab-column connection revealed that the effective joint strength increased as slab steel ratio increased. The specimens with high slab reinforcement ratios failed at higher effective strengths than did the specimen with low slab reinforcement ratios. Moreover, due to the provision of lateral ties in the joint region of specimens, slightly higher effective strength was achieved as compared to the specimens having no lateral ties in the joint (Shah et al. 2005).

2.2.4 The Effect of Aspect Ratio

Aspect ratio (h/c) is the ratio of slab thickness (h) to column dimension (c). The test results by Marzouk and Hussein (1991) reveal that as the depth of slab increase ductility decrease, moreover, as the column size is increased both stiffness and ductility increase.

Gamble and Klinar (1991) indicated that the strength of column decreased as h/c increased and this reduction is higher for the specimens with larger ratio of column to slab concrete strength. In many cases the joint ratio will be less than 1/2 and often less than 1/3. However, joint aspect ratio of the order of the unity or more is not undesirable for slab with drop panels or for joints with rectangular columns. In case of rectangular columns, the smaller column dimensions govern the aspect ratio of the joint.

Shah et al. (2005) indicated that the interior slab-column effective strength decreased with an increase in aspect ratio h/c, moreover, the results showed that the ultimate slab load increased as the aspect ratio increased.

2.2.5 The Effect of Slab Load

Ospina and Alexander (1998) revealed that the effect of slab load was to reduce both maximum compressive stress and strain at peak stress and showed that when high slab load intensities were applied, the joint benefited from some confinement.

Shah et al. (2005) showed that the application of slab load on the slab-column connection reduced the effective strength and the effective strength also reduced considerably for very high joint strength ratios.

2.3 STRESSES IN FLAT SLAB NEAR COLUMNS

Mast (1970) described that the moment transfer between the flat slabs and columns, due to unbalanced gravity loads or due to lateral forces, caused substantial stresses in the vicinity of columns. Especially shear stresses became critical and often governed the design. The shear stresses were generally more critical at exterior columns than at interior, because at exterior columns the critical periphery did not extend all around the column shaft and, hence, was weaker than at interior columns. The portion of the total unbalanced moment 'M', which was transmitted by shear stresses, is a function of the distance from the column center line. In case of a two dimensional structure, like flat slab, a similar relation existed. Here it was both the distance from column center line and the width of the peripheral section which determined how much of 'M' will be transferred by shear stresses and how much by flexural and torsional moments. Also in transferring a moment between the column and flat plate, the participation of torsional, flexural, and shear stresses was a variable which depend upon the shape and size of column and on the dimensions and boundary conditions of the plate.

2.4 FINITE ELEMENT METHOD (FEM)

The finite element methods (FEM) are being widely used in engineering analysis through various general purpose commercial computer programs and many special purpose programs written for specific applications. The methods are employed extensively in the analysis of solids and structures and virtually in every field of engineering analysis. The FEM in engineering was initially developed on the physical basis for the analysis of problems in structural mechanics. However, it was soon recognized that the technique could be applied equally well to the solution of many other classes of problems (Bathe 1996).

The FEM is used to solve the physical problems in engineering analysis and design. The physical problem typically involves an actual structure or structural component subjected to certain loads. The idealization of physical problem to the mathematical problem requires certain assumptions that together lead to differential equations governing the mathematical model. The finite element analysis (FEA) solves this mathematical model. Since FEM technique is a numerical procedure, it is necessary to assess the solution accuracy. If the accuracy criteria are not met, the numerical, i.e., FEM solution has to be repeated with refined solution parameters such as finer meshes until a sufficient accuracy is reached. The choice of an appropriate mathematical model is crucial and completely determines the insight into the actual physical problem that is to be obtained by the analysis. Fig. 2.1 summarizes the process of FEA (Bathe 1996).

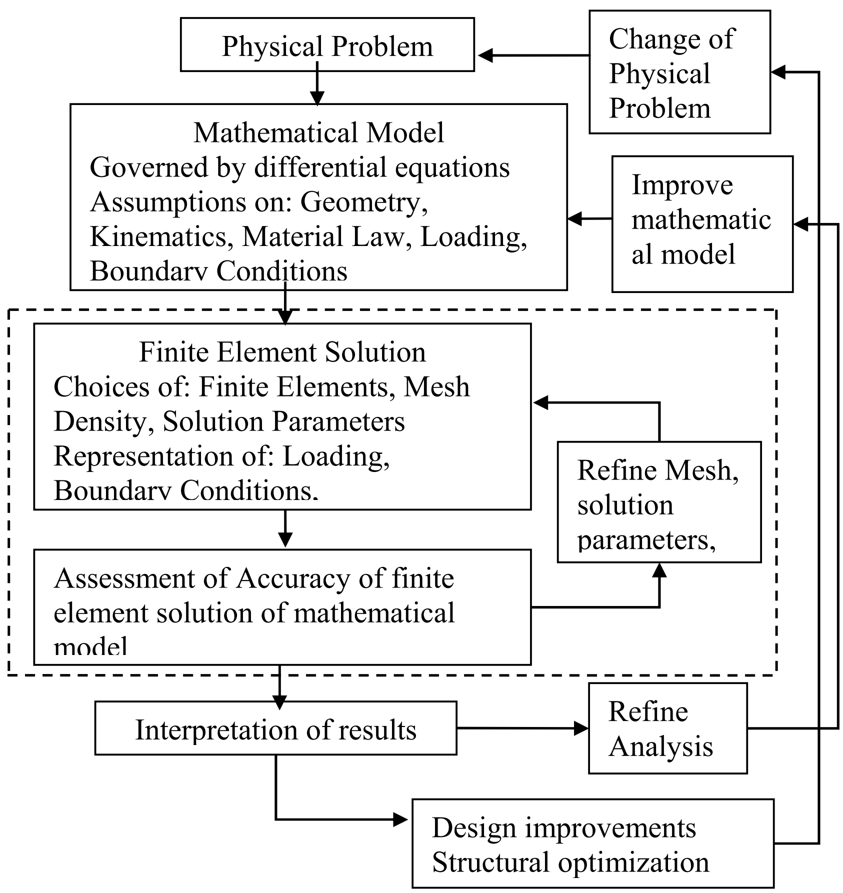


Fig. 2.1. The process of finite element analysis (Bathe 1996)

To define the reliability and effectiveness of a chosen model, a very comprehensive mathematical model of the physical problem and to measure the response of our chosen model against the response of the comprehensive model. In general, the very comprehensive mathematical model is a fully three dimensional description that also includes non linear effects.

- **Effectiveness of a Mathematical Model**

  The most effective mathematical model for the analysis is surely one which yields the required response to a sufficient accuracy and at least cost.

- **Reliability of a Mathematical Model**

  The chosen mathematical model is reliable if the required response is known to be predicted within a selected level of accuracy measured on the response of the very comprehensive mathematical model (Bathe 1996).

According to Cook et al. (2003), the finite element analysis (FEA) involves the following three steps:

- Pre-processing
- Numerical Analysis
- Post-processing

In the pre-processing phase, the input to the FEA software is given in terms of the geometry, material properties, loads, boundary conditions and meshing of the model. In the numerical analysis step, the FEA software solves the given input, generates the matrices that describe the behavior of each element, combines these matrices into a large matrix equation that represents the FE structure and solves this equation to determine the values of filed quantities. While in the post-processing phase, the FEA solution and quantities derived are listed or graphically displayed.

2.5 FINITE ELEMENT ANALYSIS OF RC STRUCTURES

Reinforced concrete (RC) structures are largely employed in engineering practice in a variety of situations. In most cases, these structures are designed following simplified procedures based on experimental data. Although traditional empirical methods remain adequate for ordinary design of RC members, the wide dissemination of computers and FEM have provided means for analysis of much more complex systems in a much more realistic way (Barbosa and Ribeiro 1998).

Blazic (1992) has carried out a finite element 3D analysis of the slab-column connection. He has investigated the shear strength and ductility of the interior slab-column connections with and without shear reinforcement and the results obtained by the FEA were compared with the previously done experimental test results. The results of the analysis indicated 10 per cent higher failure load (bending moment) than measured in the experiment. It was demonstrated that the numerical analysis based on the general non-local micro-plane model for concrete can predict the failure mode of the slab-column structure rather realistically. The numerical study as well as the experimental results indicated same type of diagonal shear failure mode.

Barbosa and Ribeiro (1998) analyzed the RC structure using nonlinear concrete model. The consequences of small changes in modeling were carried out in the study and satisfactory results were obtained from relatively simple and limited models. The general purpose FEM code ANSYS was used for the FEA and a series of analysis was carried out on the same structure with different aspects of material modeling. Due to the transverse and longitudinal symmetry, a quarter of the beam was taken for the study.

Vainiunas et al. (2004) carried out the non linear FEM analysis of RC floor slab-column connection. The model of the slab-column connection was presented and the features of the proposed formulation were discussed for non-linear 3D numerical analysis of punching shear behavior which accurately visualized the crack pattern and the strain-stress distribution inside the slab-column connection and finally the mechanism was able to be taken into account to evaluate design equation for punching shear strength capacity. The non-linear 3D analysis (3D-FEM analysis) was performed using the FEM software program MSC Marc. It was concluded that the non-linear FEA based on advanced 3D models can be effectively used for the simulation of a real behavior of the RC structures allow to perform virtual testing of structures and explore their behavior under load and other effects in different conditions. The choice of adequate material model for numerical simulation is the most important aspect in FEM analysis method to establish the right rules for structural behavior. It is obvious that the tensile reinforcement plays an important role in slab-column connection behavior subjected to punching force.

Chapter 3

METHODOLOGY

3.1 GENERAL

The primary purpose of this research is to carry out the finite element analysis (FEA) of slab-column connection and to study its behavior against variation in slab thickness and confining reinforcement provided in the slab portion.

3.2 ESTABLISHMENT OF VARIABLES

To carry out the finite element analysis of the slab-column connection, the selected variables are: the variation in slab thickness as 3.0, 3.5, 4.0, 4.5, and 5 inches (in.) and provision of steel bar in the slab portion as confining steel with cross sectional area as 0.00, 0.05, 0.11, and 0.20 square inch (in.^{2}).

3.3 MODEL DESIGNATION

The FEA model consists of 3.0, 3.5, 4.0, 4.5, and 5.0 in. thick slab with 0.00, 0.05, 0.11 and 0.20 in.^{2} area of confining steel in the slab portion. Depending upon these, the model designation symbols are summarized in table 3.1.

Table 3.1. Model Designation Symbols

Slab Column Joint	Slab Thickness (in.)					Steel Area in Slab (in.^{2})			
SCJ	3.0	3.5	4.0	4.5	5.0	0.00	0.05	0.11	0.20

The model designated as SCJ-3.0-0.00 describes slab-column connection/joint with 3.0 in. thick slab with no slab confining reinforcement. While SCJ-4.5-0.20 represents that the slab-column connection/joint with 4.5 in. thick slab and having 0.20 in.^{2} slab confining reinforcement area.

The model designations along with their geometry and area of slab confining reinforcement are tabulated in Table 3.2.

Table 3.2. Model Designation

Model Designation	Column			Slab			Steel Area in slab (in.^{2})
	Depth (in.)	Width (in.)	Height (in.)	Length (in.)	Width (in.)	Thickness (in.)	
SCJ-3.0-0.00	3	3	15	3	3	3.0	0.00
SCJ-3.5-0.00	3	3	15	3	3	3.5	0.00
SCJ-4.0-0.00	3	3	15	3	3	4.0	0.00
SCJ-4.5-0.00	3	3	15	3	3	4.5	0.00
SCJ-5.0-0.00	3	3	15	3	3	5.0	0.00
SCJ-3.0-0.05	3	3	15	3	3	3.0	0.05
SCJ-3.5-0.05	3	3	15	3	3	3.5	0.05
SCJ-4.0-0.05	3	3	15	3	3	4.0	0.05
SCJ-4.5-0.05	3	3	15	3	3	4.5	0.05
SCJ-5.0-0.05	3	3	15	3	3	5.0	0.05
SCJ-3.0-0.11	3	3	15	3	3	3.0	0.11
SCJ-3.5-0.11	3	3	15	3	3	3.5	0.11
SCJ-4.0-0.11	3	3	15	3	3	4.0	0.11
SCJ-4.5-0.11	3	3	15	3	3	4.5	0.11
SCJ-5.0-0.11	3	3	15	3	3	5.0	0.11
SCJ-3.0-0.20	3	3	15	3	3	3.0	0.20
SCJ-3.5-0.20	3	3	15	3	3	3.5	0.20
SCJ-4.0-0.20	3	3	15	3	3	4.0	0.20
SCJ-4.5-0.20	3	3	15	3	3	4.5	0.20
SCJ-5.0-0.20	3	3	15	3	3	5.0	0.20

3.4 SOFTWARE USED

In the study, Automatic Dynamic Incremental Nonlinear Analysis (ADINA) computer based finite element analysis (FEA) software was used. The

ADINA system is developed to provide the one finite element program system, which can be used to perform comprehensive finite element analysis of structures, fluids, and fluid-structure interaction. ADINA system is unique because it offers wide range of applicability and the ease of use, solution effectiveness and reliability of all program features are the key aspects of it. The structure can be modeled as linear or highly nonlinear, including material nonlinearities, large deformations and contact conditions. Static analysis, frequency solution or transient analysis using mode superposition, explicit or implicit time integration can be performed (ADINA R&D, Inc. 2001).

3.5 DEVELOPMENT OF FEA MODEL

The development of a FEA model consists of: defining the geometry, loading and boundary conditions of model, the finite element types used in the model, the material model used and the procedure of finite element analysis.

3.5.1 Geometry of Model

To analyze the structures by computer programs, when possible, the advantage of symmetry is taken so as to reduce the effort needed in data preparation and in the interpretation of results. When the structure has one or more planes of symmetry it is possible to perform the analysis on one-half, one-quarter or an even smaller part of the structure provided that the appropriate boundary conditions are applied at the nodes on the plane of symmetry. Also the elements situated on the plane of symmetry must have adjusted properties, i.e., if the FEM model is half of original model, the cross sectional properties to be used in the analysis must be equal to the half of the values in the actual structure. It is also possible, instead of changing the cross sectional properties, the material properties can also be reduced correspondingly and the forces applied at the nodes on a plane of symmetry must be reduced corresponding to the symmetry assumed from the actual structure (Ghali et al. 2004).

To develop the FEM model the advantage of symmetry was taken in this study. The slab-column joint model consisted of 15 in. upper column stub, 15 in. lower column stub and 3.0 in. to 5.0 in. thick slab and the column and slab were

6x6 in.^{2} in cross sectional area. It consists of four No. 5 longitudinal bars along each corner of cross section throughout the length of model and two No. 3 ring bars in each column stub and one confining reinforcement in slab region.

The reduced FEM model using the symmetry is obtained by cutting the whole model into three planes of symmetry; one horizontal plane passing through the slab at its mid height shown by plane "A" in Fig. 3.2, second vertical plane passing through the middle of column shown by plane "B" in Fig. 3.2 and the third vertical plane passing through the column corners shown by plane "C" in Fig. 3.1.

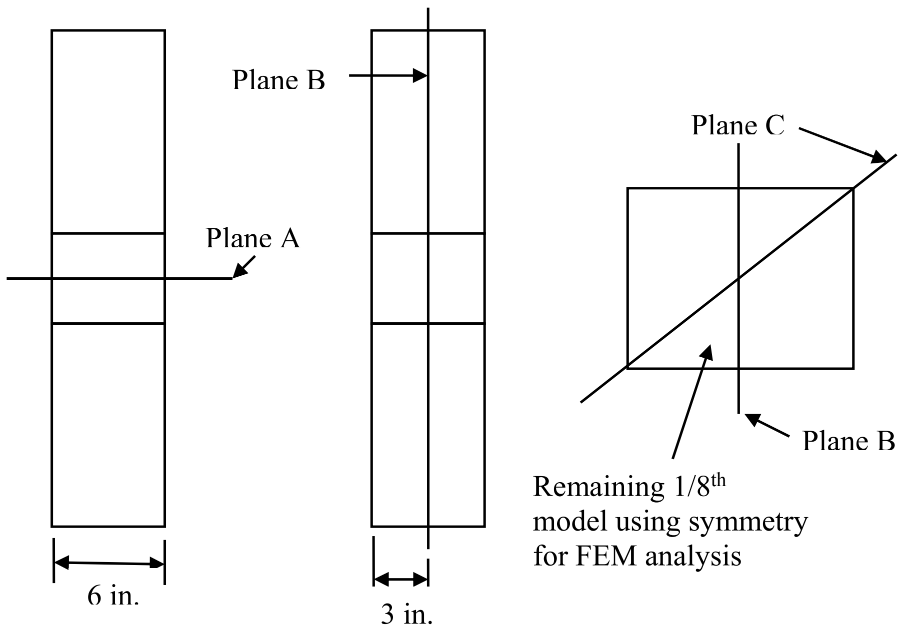


Fig. 3.1. Planes of symmetry

After carrying out the symmetry, the FEM model was reduced to one eighth (1/8th) of the original model containing 15 in. long upper column stub and 1.5 to 2.5 in. thick slab portion having triangular shape with 3 in. width and 3 in. length. It contains one No.5 longitudinal reinforcement bar at the corner, two No. 3 ring reinforcing bars in the upper column stub and one slab confining reinforcement bar of cross sectional area varying from 0.05 to 0.20 in^2 in the slab. Fig. 3.2 shows the developed FEM model using the symmetry with its dimension.

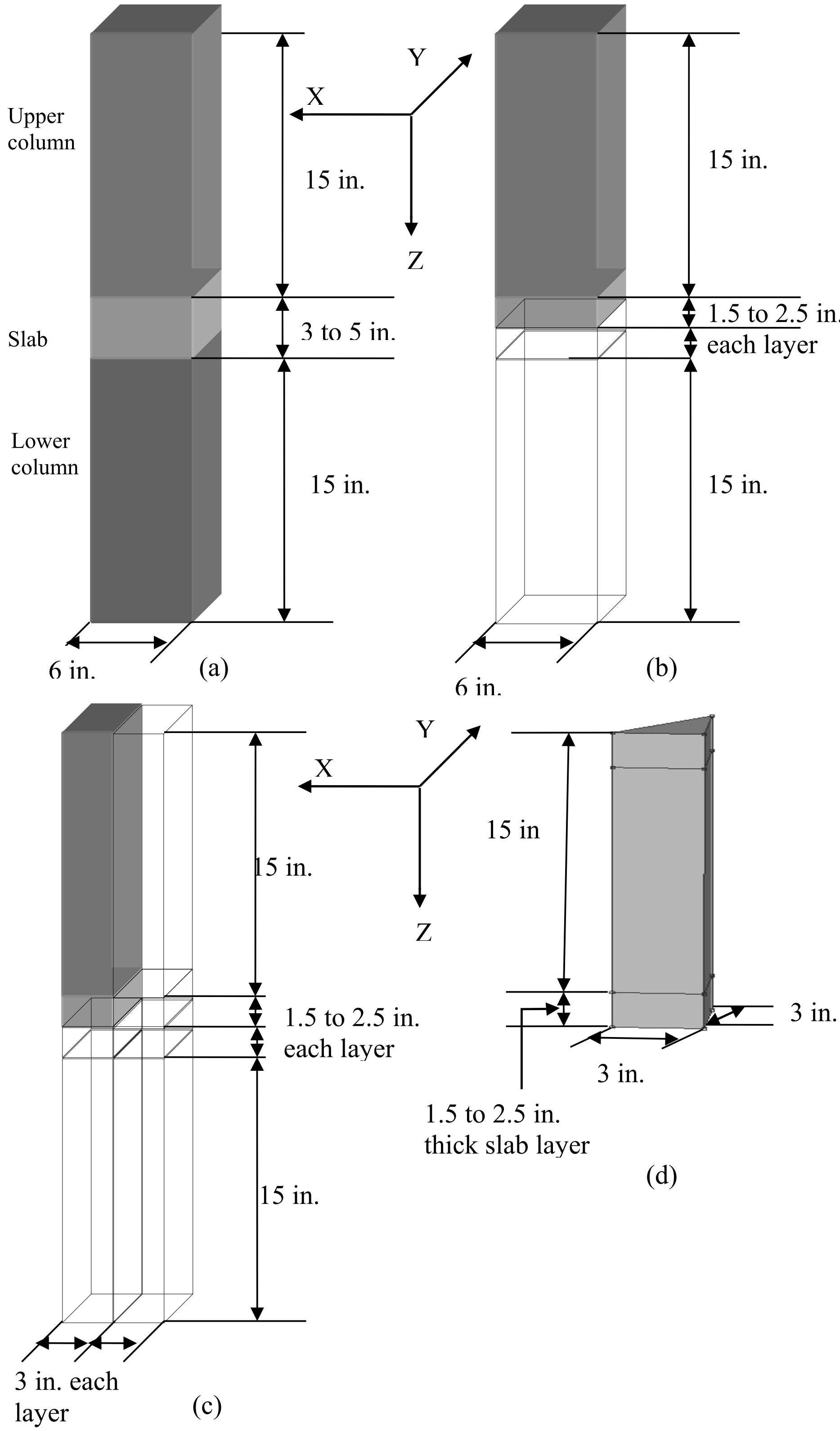


Fig. 3.2. Geometry of FEM models

After developing the geometry of model, the next step was the application of boundary conditions. All the boundary conditions were applied on the FEM model perpendicular to their plane of symmetry. In Fig 3.3, the arrows show the boundary conditions applied on the cut surfaces along their vertical planes.

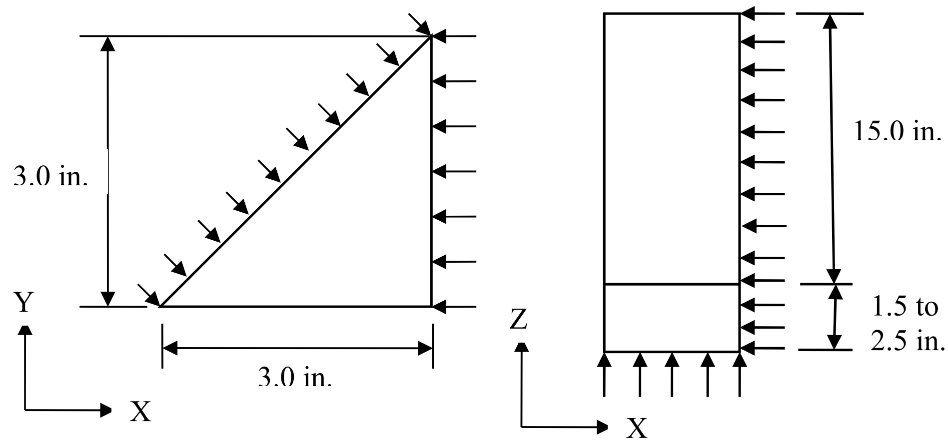


Fig. 3.3. Boundary conditions

After the application of boundary conditions, next step was to apply the load on the FEM models. The load applied on all the models were constant pressure of 0.1 ksi (kips per inch square) and also the time step was set constant.

3.5.2 Finite Elements Types

Since the FEM models developed are of reinforced concrete, hence, two classes of finite elements types were used from the element library of ADINA. The concrete was modeled by 20 node 3-D solid elements and steel with truss elements.

3.5.2.1 Three Dimensional (3-D) Solid Element

3-D solid element in the ADINA element library, used to model the concrete, is a variable from 4 to 27 nodes with 3 degrees of freedom on each node and tetrahedral, pyramid and prismatic shapes. It can be used with material as well as geometrical linearity and nonlinearity and is capable of providing the cracking and crushing patterns. The out put for concrete can be obtained in terms of stress, strain, cracks, and crushing. 20 node 3-D solid element with material non-linearity used in this study is shown in Fig. 3.4.

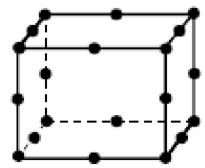


Fig. 3.4. 20 node 3-D solid element for concrete (ADINA 2001)

3.5.2.2 Truss Element

The truss element, used to model reinforcement, can be employed as 2 node, 3 node and 4 node element, or as a 1 node ring element. It can be used with linear as well as nonlinear material and geometrical models. The output can be obtained in terms of force in the steel, stress and strain for steel reinforcement from this truss element. 2 node truss element with material non-linearity used in this study is shown in Fig. 3.5.

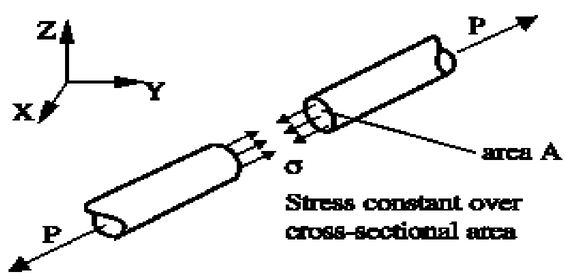


Fig. 3.5. Truss element for steel reinforcement (ADINA 2001).

3.5.3 Material Model

The concrete material model defines the properties of concrete types used in the FEM models and plastic bi-linear material model defines the properties of steel reinforcement. The concrete material model was defined separately for column concrete and slab concrete because these two contain different concrete strength properties.

3.5.3.1 Concrete Material Model

The concrete material model available in ADINA library can be used with 2-D and 3-D solid elements with linear as well as non-linear properties of concrete. Three basic features used in the concrete material model are:

- A nonlinear stress-strain relationship to allow for weakening of the material under increasing compressive stresses.
- Failure envelopes that define failure in tension and compression.
- A strategy to model post-cracking and crushing behavior of material.

Fig. 3.6 shows the stress-strain curve for concrete.

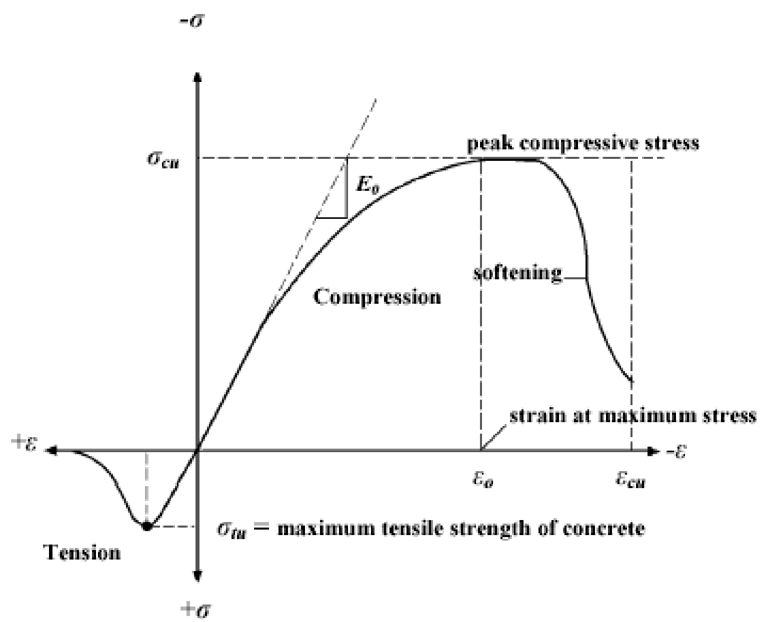


Fig. 3.6. Stress-Strain curve for concrete

For concrete, ADINA requires material properties in terms of:

- Poisson's ratio (ν)
- Tangent modulus of concrete (E_c)
- Ultimate uni-axial tensile strength (modulus of rupture, f_r)
- Ultimate maximum compressive stress (f'_c)

Refer Table A-3.1 (Appendix I) for the material properties of column and slab concrete. Depending upon the ultimate maximum compressive strength of concrete (f'_c), the modulus of elasticity (E_c) and modulus of rupture (f_r) were calculated by equations given by ACI 318-02 (ACI 2002) as:

$$E_c = 57000\sqrt{f'_c} \quad (1)$$

$$f_r = 7.5\sqrt{f'_c} \quad (2)$$

3.5.3.2 Plastic Bi-linear Material Model

The plastic bi-linear material model available in ADINA can be used with 2 node truss elements for defining the properties of reinforcement used in FEM models. The steel for FEM models was assumed to be elastic-perfectly plastic material and identical in tension and compression. Fig. 3.7 shows the typical stress-strain curve for steel.

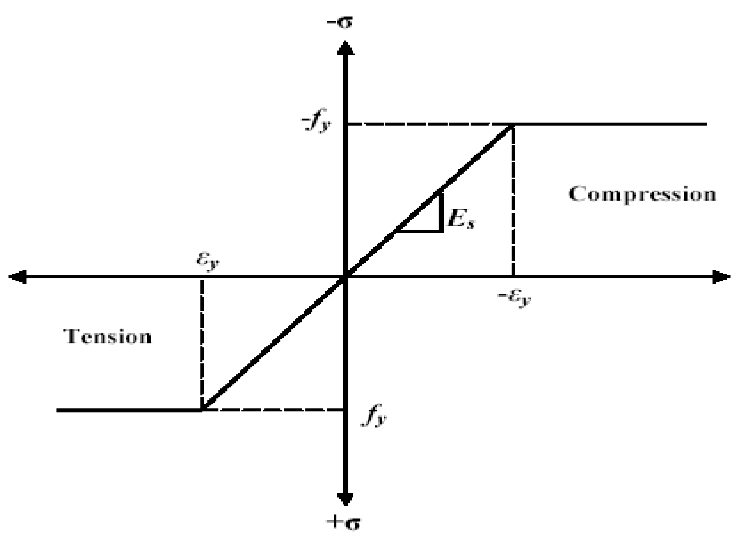


Fig. 3.7. Stress-Strain curve for steel

For steel, ADINA requires the material properties in terms of:

- Poisson's ratio (ν)
- Tangent modulus of Steel (E_s)
- Yield Stress of steel (f_y)

Refer Table A-3.1 (Appendix I) for the properties of steel used in the development of FEM model.

3.5.4 Finite Element Analysis (FEA)

After defining the geometry, loading and boundary conditions, FEM elements types, and the material models, the first step in finite element analysis (FEA) is the discretization of the model, i.e., the meshing of the model. An important step in finite element modeling is the selection of the mesh density. The convergence of results is obtained when an adequate number of elements are used in a model. This is practically achieved when an increase in the mesh density has a negligible effect on the results. Fig. 3.8 shows the meshing and applied load on the FEM slab-column joint model used in the study.

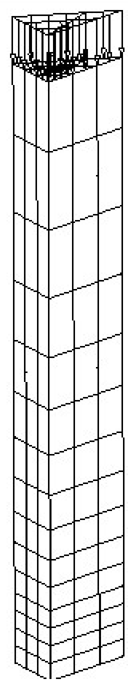


Fig. 3.8. FEM discretization of model

In FEM analysis, the total load applied to a finite element model is divided into a series of load increments called load steps. At the completion of each incremental solution, the stiffness matrix of the model is adjusted to reflect nonlinear changes in structural stiffness before proceeding to the next load increment.

The ADINA program (ADINA 2001) uses different techniques of analysis in order to achieve the solution of the FEM model. In this study, Newton-Raphson equilibrium iteration method was used keeping the convergence criteria for displacement. This method is an incremental analysis performed with time (or load) step size and the iterations are continued until appropriate convergence criteria are satisfied. The characteristic of this method is that a new tangent stiffness matrix is calculated for each iteration (Bathe 1996; and ADINA 2001).

3.6 MATERIAL PROPERTIES

In the study two types of materials are used, i.e., concrete and steel. The concrete in column and slab have different properties while the reinforcement has the same properties only the bar size is changed. Table A-1 (Appendix I) shows the detail of material properties used for slab-column joint.

Chapter 4

COMPUTATIONAL MODELING RESULTS AND ANALYSIS

4.1 GENERAL

The FEM analyses are performed using ADINA system software. This chapter presents the discussion and analysis of FEM results obtained by carrying the FEM analysis on the models explained in Chapter 3. The FEM analysis results are graphically represented in Appendix I.

4.2 ANALYSIS OF FEA RESULTS

This section presents the analysis and discussion on the results obtained from FEM analyses of models. Generally, similar behavior was obtained from all the models subjected to the variations in slab confining steel reinforcement and slab thickness.

4.2.1 Comparison between FEM and Experimental Analysis Results

4.2.1.1 Comparison of Stress-Strain Behavior

The experimental analysis of slab-column connection was carried out at National Institute of Transportation, (NIT), Risalpur by Ehsan Ullah Khan in 2001 (Khan 2001). In the experimental analysis program only three types of model were tested. All the experimental specimen were 6x6 in.^{2} in cross section containing each 15 in. long upper and bottom column stub and 3 in. and 4.5 in. thick slab with four No. 5 longitudinal bars and four No. 3 ring bars in two specimen situated in the column at 3 in. and 9 in. from end in upper and lower column stubs and one model with five No. 3 ring bars, four situated as in two models where as one ring bar was provided at the center of the slab portion as the confining steel in the slab. The

FEM models developed in this study in accordance with the laboratory tested models are SCJ-3.0-0.00, SCJ-4.5-0.00 and SCJ-4.5-0.11. Fig. A-1 to Fig. A-6 (Appendix I) show the comparison of steel stress-strain curve for experimental and FEM analyses models.

Fig. 4.1 shows similar trend between the FEM and experimental analysis results. The strain increases as the stress increased in both the models. In both the curves, the slight variation in slope is observed after the cracking has started.

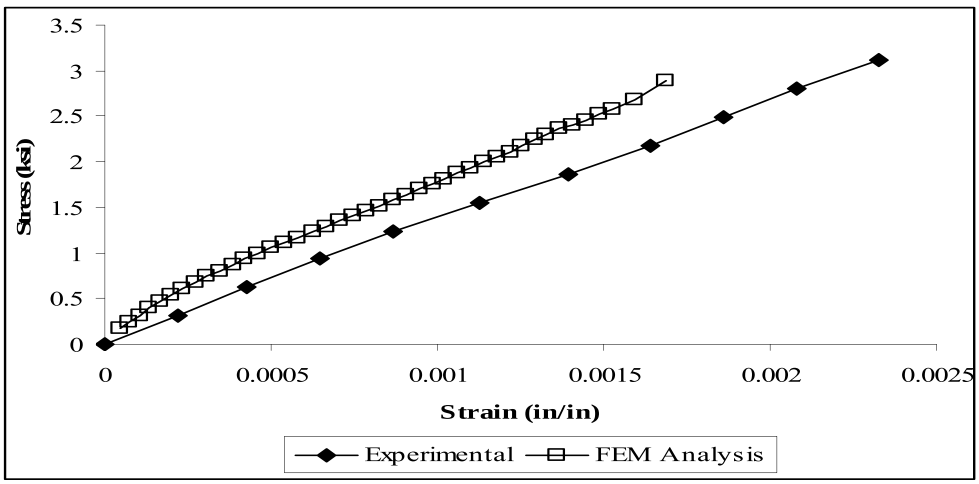
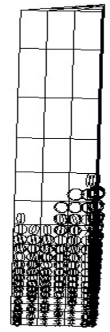


Fig. 4.1. Stress-strain curve for slab steel for 4.5 in slab and 0.11 in^2 confining steel

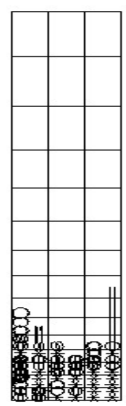
4.2.1.2 Comparison of Crack Pattern

Generally, similar type of crack pattern is observed in FEM and experimental analyses. The cracks initiate from the slab exceeding towards the column edges and causing the failure of specimen/model. The cracks start in the slab portion because of the reason that the slab concrete is of lower strength as compared to column concrete hence the failure of slab-column connection starts from slab and it exceeds towards the column. Fig. 4.2 and Fig 4.3 show the crack and failure pattern for experimental tested specimen and there corresponding FEM models.



(a) Experimental failure pattern (b) FEM analysis failure pattern

Fig. 4.2. Crack pattern in experimental and FEM analyses for SCJ-4.5-0.00



(a) Experimental failure patters (b) FEM analysis failure patters

Fig. 4.3. Crack pattern in experimental and FEM analyses for SCJ-4.5-0.11

4.2.2 General Behavior of Slab-Column Connection

4.2.2.1 Stress-Strain Behavior of Concrete

The general behavior of all the models remained approximately similar as expected. The strains routinely kept increasing as the load over the column increased. Fig. A-7 to Fig A-26 (Appendix I) shows the concrete compressive stress-strain curve for all models.

Fig. 4.4 shows that the column concrete has steep slope while the slab concrete has moderate slope for model SCJ-3.0-0.00. It is because of the reason that the column is of high strength concrete as compared to the slab concrete, hence the modulus of elasticity (E_c) of column is high as compared to the modulus of elasticity (E_c) of slab.

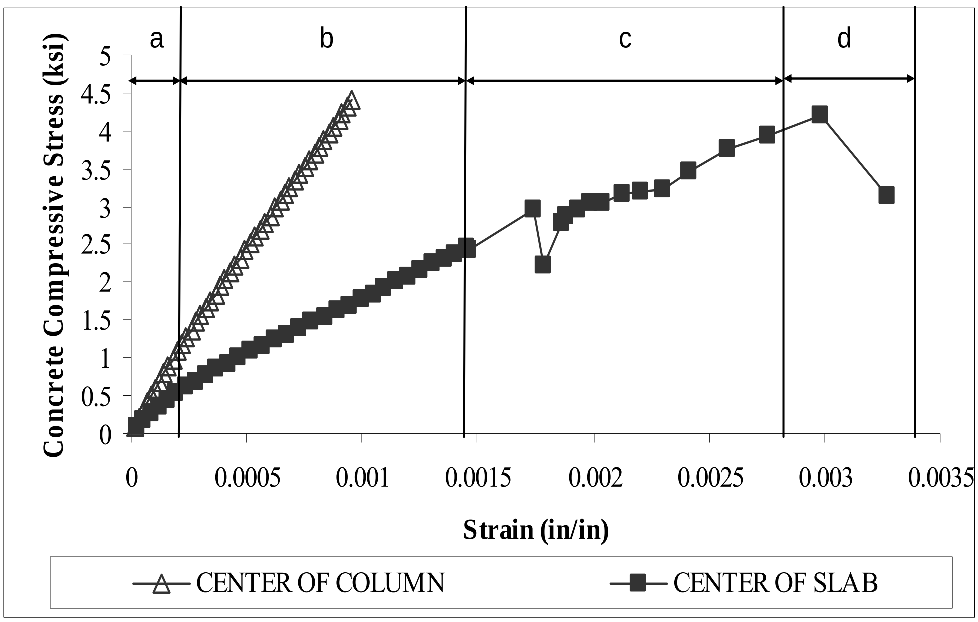


Fig. 4.4. Concrete compressive stress-strain curve for model SCJ-3.0-0.00

The column concrete is linear elastic while the slab concrete has non-linear stress-strain curve. The stress-strain curve of concrete shows four different regions. The stress-strain curve is linear for region “a”, and however there is slight decrease in the slope of the stress-strain curve in region “b” than region “a”. The cracking in slab concrete starts at the beginning of region “b”. At the initiation of cracking in the concrete, the slope of stress-strain curve is decreased, similar behavior was observed by Macgregor (1997). The cracks in the slab portion are small in region "b" and there is not so much redistribution of stresses in this region. After region "b", the crushing of concrete is started and the stress-strain curve enters in next region "c", in this region the non-linear behavior can be seen. The cracks are open and there is the redistribution of stresses in the slab concrete, the concrete is still capable to take the stresses. The last region "d" is also the non-linear portion of the stress-strain curve and the variation in the stress-strain curve in this region is significantly high.

4.2.2.2 Stress-Strain Behavior of longitudinal Steel

Generally the stress-strain behavior of longitudinal steel was observed to be same as that for concrete. The strains kept increasing as the load over the column increased. Fig. A-27 to Fig A-46 (Appendix I) shows the concrete compressive stress-strain curve for all models.

Fig. 4.5 for model SCJ-3.0-0.00 shows similar behavior as Fig. 4.21, the column steel stress-strain curve is steep than that for slab steel. The column steel curve is linearly elastic and slab steel curve is linear and non-linear and divided in four regions. Region "a" linear, region "b" also linear with start of cracks, region "c" non-linear combined with crushing of concrete and redistribution of stresses and the last region "d" non-linear with significant redistribution of stresses.

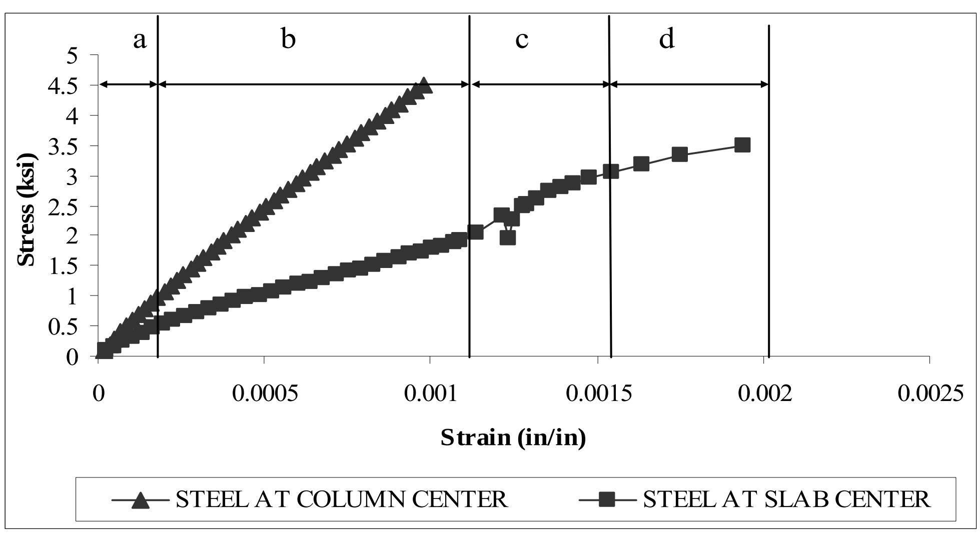


Fig. 4.5. Steel stress-strain curve for model SCJ-3.0-0.00

4.2.2.3 Stress-Strain Behavior of Slab Confining Steel

Generally, similar type of stress-strain response was observed in all the models by slab confining reinforcement. Fig. A-47 to Fig A-61 (Appendix I) shows the slab confining reinforcement stress-strain curve for all models.

The stress-strain curve of slab confining steel (Fig. 4.6) for model SCJ-3.0-0.05 shows three different regions, named as a, b, and c. In region "a", the behavior is linear-elastic as the stress increases the strain is also increased. In region "b" the curve is non-linear and the cracking in the slab has initiated and due to cracking, there is redistribution of stresses from concrete to steel. The last region "c" is also the non-linear region of the stress-strain curve of confining steel. In this region the crushing of concrete has started and ultimately model fails.

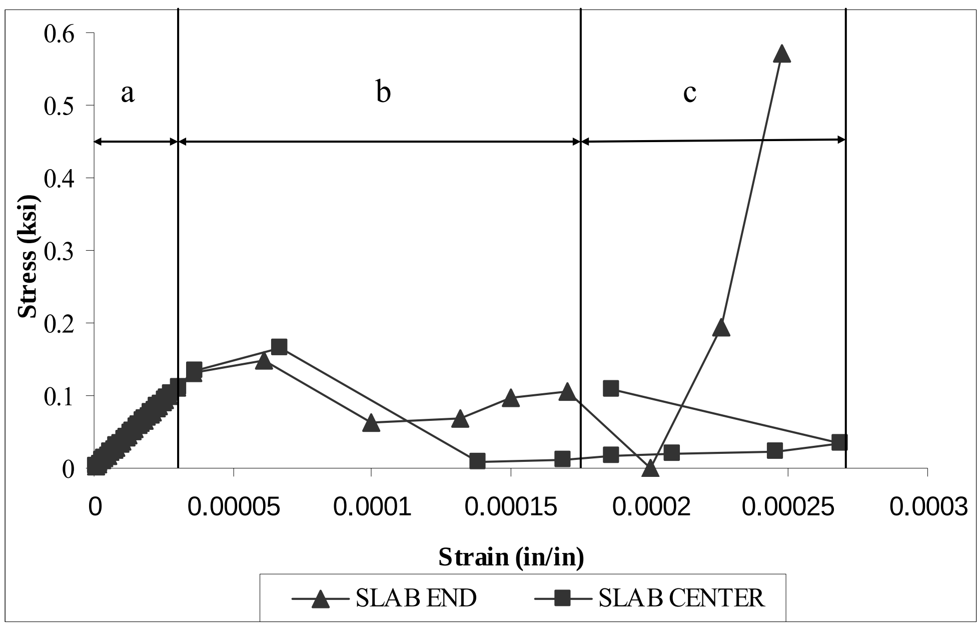


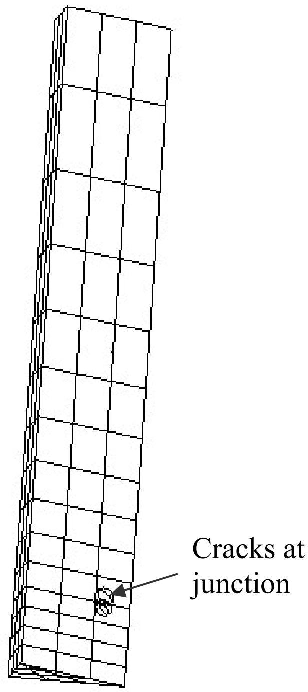
Fig. 4.6. Stress-strain curve of confining steel of 0.05 in^2 area for SCJ-3.0-0.05

4.2.3 Cracking and Crushing Pattern

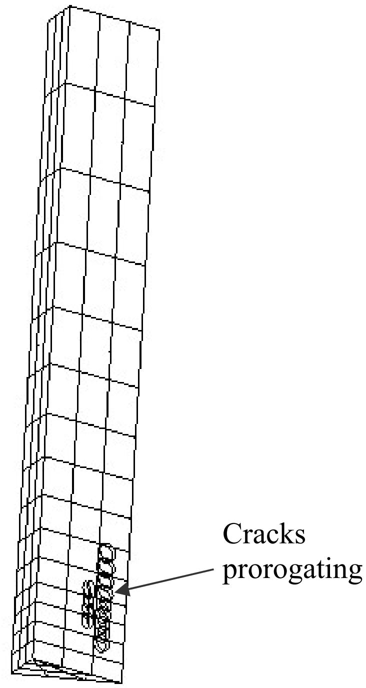
Generally, similar type of cracking is observed in all the slab-column joint FEM models. The cracks are originated from the junction of slab-column joint and then they propagate throughout the entire slab exceeding towards the edges of column and resulting in the crushing of slab portion causing total failure of models.

A typical crack pattern and failure due to crushing obtained by the FEM analysis of slab-column joint model SCJ-4.5-0.11 at different loading stages is shown in Fig. 4.7.

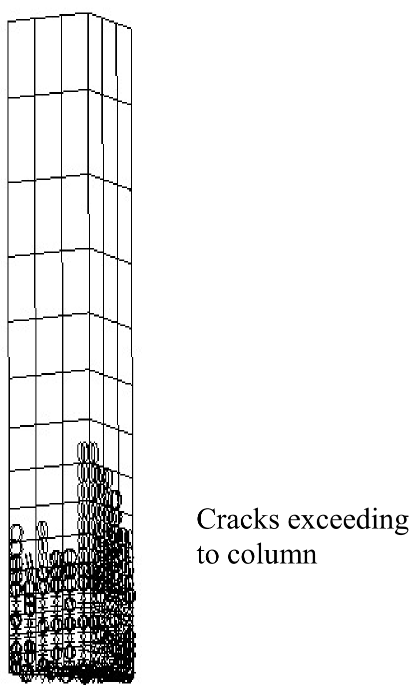
The first cracks appear at the slab-column joint junction at a load of about 48 kips and as the load is increased, the cracks also increase and cause the failure of slab portion at a load of about 186 kips. The crushing of the slab concrete starts at about 178 kips. Similar, cracking pattern was observed by Gamble and Klinar (1991); Ospina and Alexander (1998) and Shah et al. (2005). Table 4.1 shows the first cracking, first crushing and failure load for all models. Fig. 4.8 shows the load deflection curve for all the models.



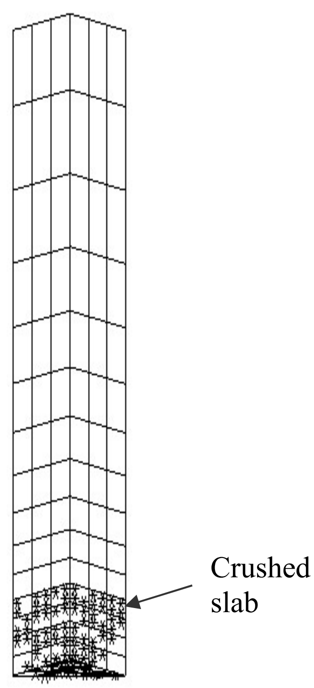
(a) First loading cracks



(b) Cracks at 180 kips load



(c) Cracks at failure load



(d) Crushing at failure

Fig. 4.7. Crack patters and crushing of model SCJ-4.5-0.11

Table 4.1. First Cracking. Crushing and Failure Load for all FEM Models

MODEL	First cracking load (kips)	First crushing load (kips)	Failure load (kips)
SCJ-3.0-0.00	43.2432	138.2832	150
SCJ-3.5-0.00	51.84	155.52	145
SCJ-4.0-0.00	51.84	159.84	177.12
SCJ-4.5-0.00	47.52	159.84	164.16
SCJ-5.0-0.00	47.52	155.52	176.4
SCJ-3.0-0.05	43.2432	142.6032	146.9232
SCJ-3.5-0.05	51.84	177.12	216
SCJ-4.0-0.05	51.84	172.8	198.72
SCJ-4.5-0.05	47.52	177.12	185.76
SCJ-5.0-0.05	47.52	172.8	190.08
SCJ-3.0-0.11	43.2432	146.9232	155.5632
SCJ-3.5-0.11	51.84	172.8	185.76
SCJ-4.0-0.11	51.84	172.8	190.08
SCJ-4.5-0.11	47.52	177.12	185.76
SCJ-5.0-0.11	47.52	164.16	216
SCJ-3.0-0.20	43.2432	146.9232	151.2432
SCJ-3.5-0.20	51.84	172.8	177.12
SCJ-4.0-0.20	51.84	177.12	216
SCJ-4.5-0.20	47.52	168.48	172.8
SCJ-5.0-0.20	47.52	164.16	177.12

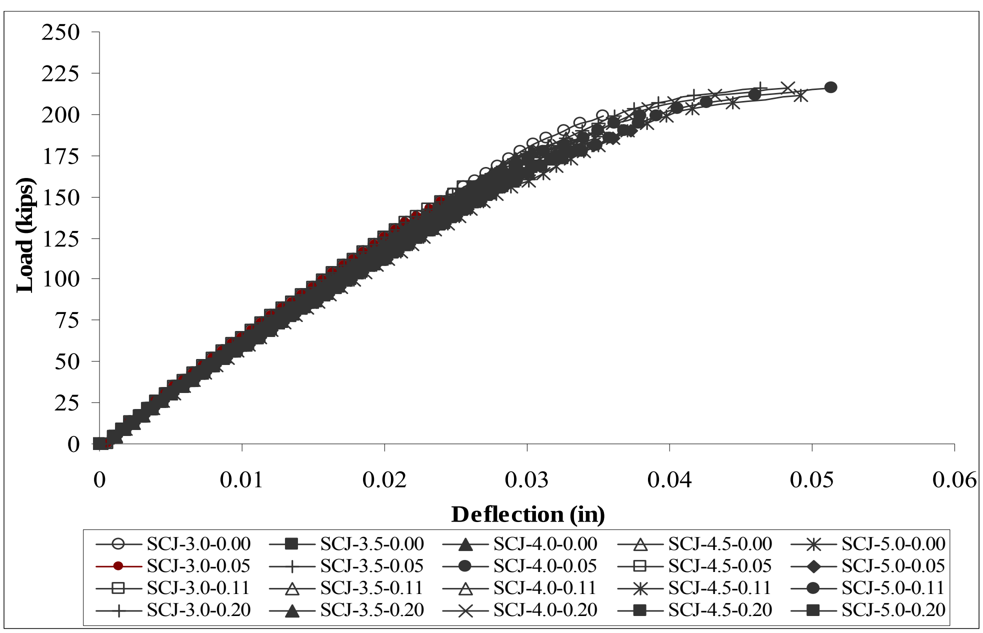


Fig. 4.8. Load-deflection curve for all FEM models

4.2.4 Effect of Slab Confining Reinforcement

4.2.4.1 Effect on Deflection

The analysis of FEM results indicates that as the area of confining reinforcement in the form of ring/tie in slab portion is increased from 0.05 to 0.20 inch square, the deflection decreases as shown in Fig. 4.9.

For 4.0 in. thick slab the deflection decreases from 0.0246 in. to 0.0244 in. as the slab confining reinforcement is increased from 0.05 to 0.20 inch square. With an increase of 400 per cent the decrease in deflection is about 1 per cent. This concludes that with increase in slab confining reinforcement, the reduction in deflection is insignificant. Similar trends are seen in all the models.

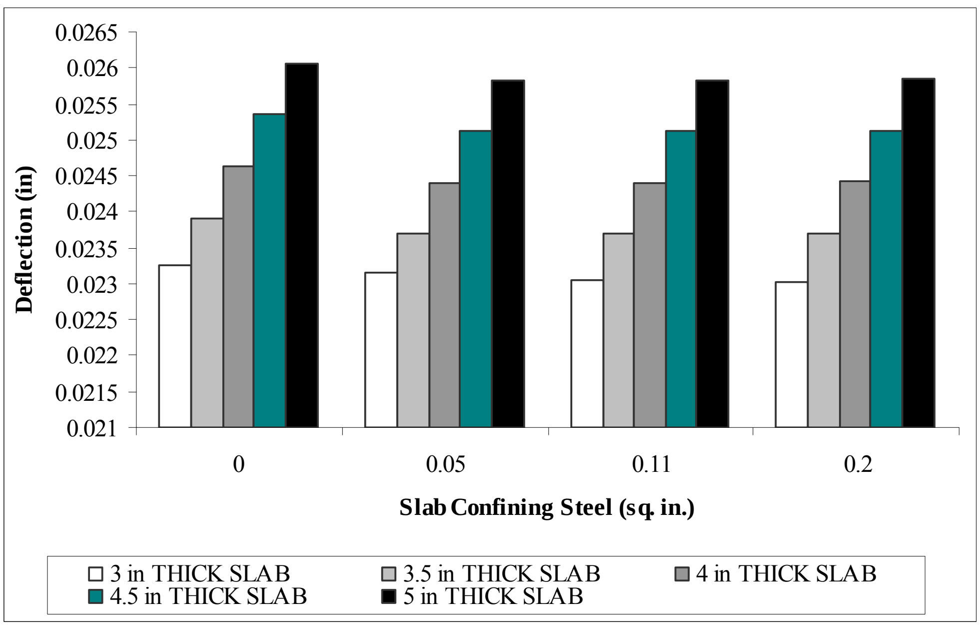


Fig. 4.9. Effect of slab confining reinforcement on deflection at 142.6 (kips) load

4.2.4.2 Effect on Failure Load

In order to understand the effect of slab confining reinforcement on the failure load, linear regression analysis of the results was done and a trend line was developed. Fig. A-62 to Fig. A-66 (Appendix I) show the trend lines observed for failure load of model with respect to slab confining reinforcement.

It can be seen from Fig. 4.10, the failure load increases with increase in the slab confining reinforcement area. Generally, similar trend was observed in all the FEM models.

For 4 in. thick slab, the failure load increases from 177.12 (kips) to 216 (kips) as the slab confining reinforcement increases from 0.00 to 0.20 in^2, the increase in failure load is approximately 22 per cent. This concludes that the load bearing capacity increases significantly with increase in slab confining reinforcement.

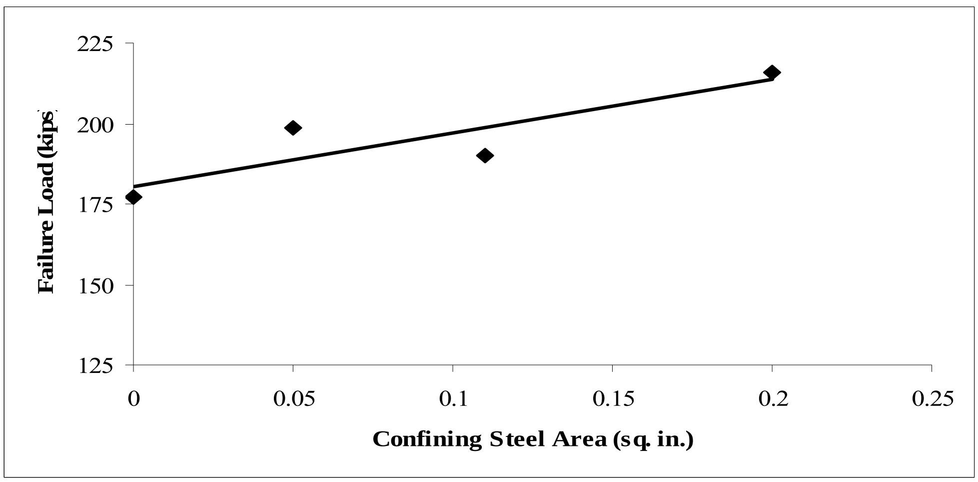


Fig. 4.10. Effect of slab confining reinforcement on failure load for 4.0 in thick slab

4.2.5 Effect of Slab Thickness

4.2.5.1 Effect on Deflection

It is observed that as the slab thickness is increased, the deflection also increases shown in Fig. 4.11.

For 0.11 in.^{2} slab confining steel, the deflection increases from 0.023 in. to 0.026 in. as the slab thickness is increased from 3 in. to 5 in., the increase in deflection is approximately 12 per cent. The increase in deflection with increase in slab thickness is found to be significant. Similar trends are observed in all models.

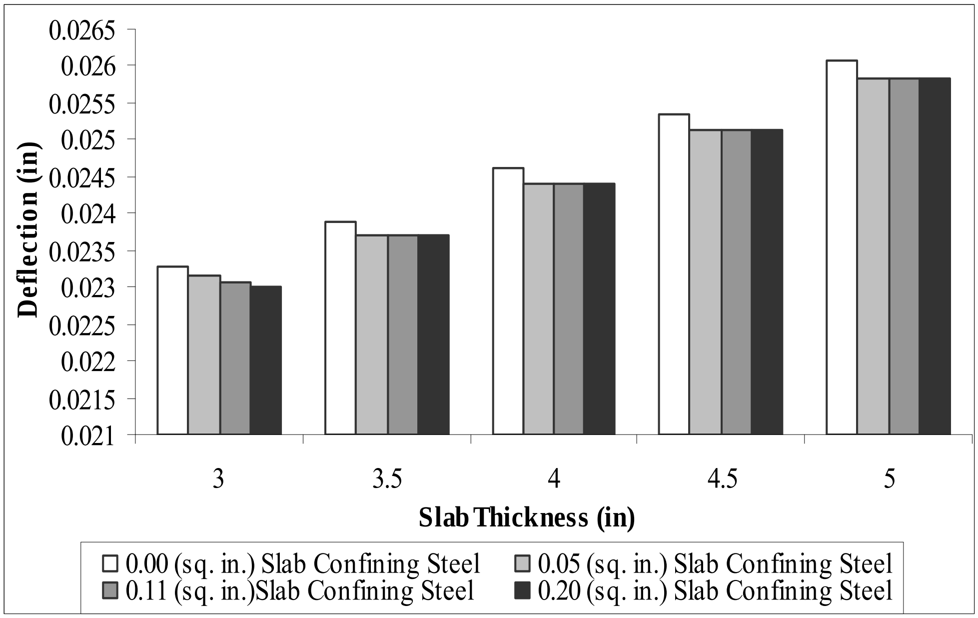


Fig. 4.11. Effect of slab thickness on deflection at 142.6 (kips) load

4.2.5.2 Effect on Failure Load

To evaluate the effect of varying slab thickness on the load bearing capacity of the model, linear regression was done and trend lines obtained from regression analysis show that with increase in slab thickness, the load carrying capacity of model is also increased. Fig. A-67 to Fig. A-70 (Appendix I) show the trend lines observed for failure load of model with respect to slab thickness.

From Fig. 4.12, it is clear that the load carrying capacity increases with increase in the slab thickness.

From the analysis of Fig. 4.12, it is clear that for 0.11 in^2 slab confining steel, the failure load increases from 155 (kips) to 216 (kips) as the slab thickness is increased from 3 in to 5 in, the increase in failure load is approximately 39 per cent. The increase in load carrying capacity with increase in slab thickness is found to be significant. Similar trends are observed in all models.

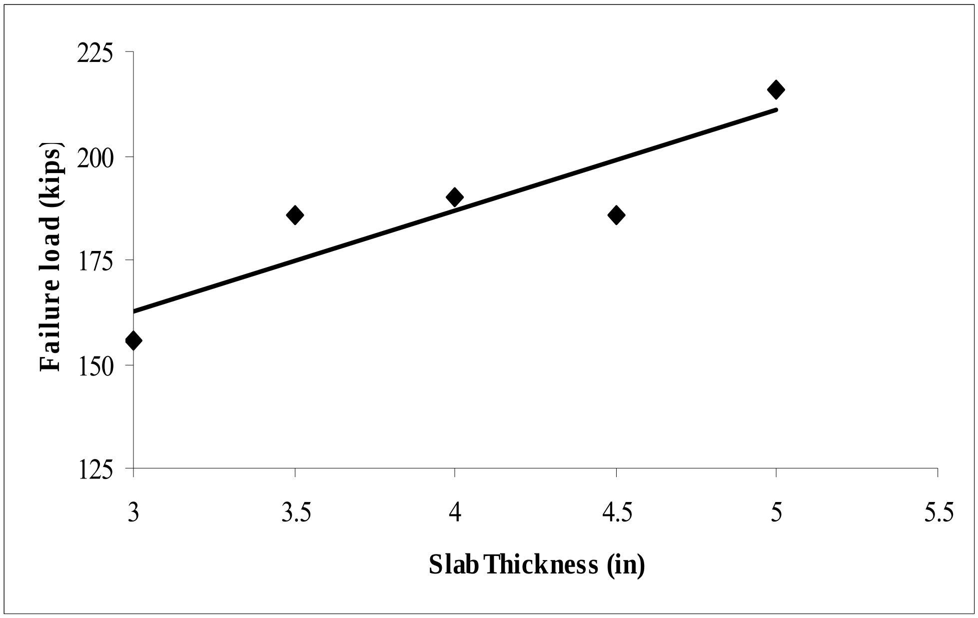


Fig. 4.12. Effect of slab thickness on failure load for 0.11 in^2 slab confining steel

Chapter 5

CONCLUSION AND RECOMMONDATIONS

5.1 CONCLUSIONS

Based upon the FEM analysis of slab-column connection in this research study, following conclusions are drawn:

- With increase in slab confining reinforcement area, the decrease in deflection is observed.
- It is observed that with the increase in slab confining reinforcement area the load carrying capacity of the slab-column connection increases.
- It is observed that the deflection increases with increases in slab thickness.
- By increasing the thickness of slab, the load carrying capacity of slab-column connection increases.

5.2 RECOMMONDATIONS

Following is recommended for future studies and research in this area;

- Using the FEM analyses, further study may be done by varying the concrete compressive strengths in both column and slab.
- FEM analysis with varying column dimensions along with varying slab thickness may be studied to see the effect of column aspect ratio.
- The FEM models generated in this study may be checked against lateral loading in addition to vertical loads.

REFERENCES

ADINA R&D, Inc., (2001). “Automatic dynamic incremental nonlinear analysis.” Watertown, MA.

ADINA user’s manual, Version 7.5. (2001). ADINA R &D, Inc., Watertown, MA.

American Concrete Institute (ACI). (2002). “Building code requirements for structural concrete and commentary.” *ACI 318-02/318R-02,* Detroit.

Barbosa, A. F., and Ribeiro, G. O. (1998). “Analysis of reinforced concrete structures using ANSYS nonlinear concrete model.” *Engineering Mechanics New Trends and Application,* 1-7.

Bathe, K. J. (1996). *Finite element procedures.* Prentice-Hall, Upper Saddle River, N.J., 1-4.

Blazic, S. (1992). “Finite element 3D analysis of the slab-column connection.” *Engineering Modelling*, 5(3-4), 99-106.

Cook, R. D., Malkus, D. S., Plesha, M. E., and Witt, R. J. (2003). *Concepts and applications of finite element analysis.* John Wiley and Sons, Inc., New York, 13.

Dilger, W. H. (2000). “Flat-slab column connections.” *Prog. Struct. Engng Mater.*, 2, 386-399.

Gamble, W. L., and Klinar, J. D. (1991). “Tests on high-strength concrete columns with intervening floor slabs.” *J. Struct. Eng.*, 117(5), 1462-1476.

Ghali, A., Neville, A. M., and Brown, T. G. (2004). *Structural analysis a unified classical and matrix approach.* Taylor and Francis Group, New York. 689-691.

Khan, E. (2001). “Load transfer from high strength concrete columns through ordinary strength concrete slabs.” MSc dissertation, NIT, Risalpur,

Macgregor, J. G. (1997). *Reinforced concrete mechanics and design*, Prentice-Hall, Upper Saddle River, N.J.

Marzouk, H., and Hussein, A. (1991). ”Experimental investigation on the behavior of high-strength concrete slabs.” *ACI Strcut. J.*, 88(6), 701-713.

Mast, P. E. (1970). “Stresses in flat plates near columns.” *ACI Journal*, 67(10), 761-768.

Ngo, D. T. (2001). “Punching shear resistance of high-strength concrete slabs.” *Electronic J. Struct.l Eng.*, 1(2001), 52-59.

Nilson, A. H., Darwin, D., and Dolan, C. W. (2003). *Design of concrete structures.* Thirteenth Edition. McGraw-Hill Companies, Inc, New York. 609-610.

Ospina, C. E., and Alexander, A. D. B. (1998). “Transmission of interior concrete column loads through floors.” *J. Strcut. Eng.*, 124(6), 602-610.

Polak, M. A. (2005). “Ductility of reinforced concrete flat slab-column connection.” *Blackwell Publishing Comput. Aided Civ. Infrastruct. Eng.*, 20(2005), 184-193.

Shah, A. A., Dietz, J., Tue, N. V., and Koenig, G. (2005) “Experimental investigation of column-slab joints.” *ACI Struct. J.*, 102(1), 103-113.

Vainiunas, P., Popovas, V., and Vilnius, A. J. (2004). “Non linear analysis of RC floor slab-column joint connection.” *Theoratical Foundation of Civil Engineering*, XII, 527-538.