Two Dimensional Linear Finite Element Analysis Of Post-tensioned Beams

Rodolfo Hutchinson

University of Central Florida

Part of the Civil Engineering Commons

Find similar works at: https://stars.library.ucf.edu/etd

University of Central Florida Libraries http://library.ucf.edu

This Masters Thesis (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations, 2004-2019 by an authorized administrator of STARS. For more information, please contact STARS@ucf.edu.

STARS Citation
https://stars.library.ucf.edu/etd/198
TWO DIMENSIONAL LINEAR FINITE ELEMENT ANALYSIS OF POST-TENSIONED BEAMS WITH EMBEDDED ELEMENTS USING MATLAB

By

RODOLFO ANTONIO HUTCHINSON MARIN
B.S. in Civil Engineering
Universidad Santa Mara La Antigua, 2000
Panama City, Panama

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in the Department of Civil and Environmental Engineering in the College of Engineering and Computer Science at the University of Central Florida Orlando, Florida

Fall Term
2004
The objective of this research project was to create a Finite Element Routine for
the Linear Analysis of Post-Tensioned beams using the program CALFEM® [20]
developed at the division of Structural Mechanics in Lund University, Sweden. The
program CALFEM and our own made files were written in MATLAB, an easy to learn
and user-friendly computer language.

The approach used in this thesis for analyzing the composite beam consists in
embedding the steel tendons at the exact location where they intersect the concrete parent
elements, without moving the concrete parent element nodes. The steel tendons are
represented as one dimensional bar elements inserted into the concrete parent elements,
which at the same time are represented as 8 node Iso-parametric plane elements.

The theory presented in Ref. [4] served as basis for the modeling of the post-
tensioned beams; however it only explained the procedure for modeling simple
reinforced concrete beams, due to this we needed to make the appropriate adjustments so
we could model post-tensioned beams.

Assembly of the tendon stiffness into the concrete elements will depend on the
bond interface between the steel and concrete, this bonding effect will be modeled using
link elements; the stiffness of this link element used in the concrete-tendon interface will
be the change in cohesion (between the grout or duct and the steel tendon) at the interface
due to the relative slip between the concrete and the steel elements nodes. Loads (Distributed, Concentrated or Post-Tensioning) are applied directly into the concrete parent elements, and then from their resultant displacement the displacements and forces of all the steel tendon elements are obtained, this is done consecutively for all the post-tensioned tendons at every load increment.

Four examples from different references and software programs are solved and compared with our results: (1) A simply reinforced cantilever plate. (2) A reinforced concrete beam, under the effect of a vertical concentrated load at mid-span. For this problem the force distribution along the steel reinforcement is obtained for two conditions, perfectly bonded and perfectly un-bonded, our results are compared with the ones obtained with the program SEGNID. (3) Consists of a continuous un-bonded post-tensioned beam with two spans, without stress losses on the tendon. The reactions at the supports and the concrete stress distribution at the location of the mid-support are obtained after the post-tensioning force is applied at both ends. (4) Consist on a un-bonded post-tensioned beam with stress losses on the tendons due to friction, wobbling and anchorage loss, under gradual loading and consecutive post-tensioning of two tendons, the results are compared with the ones reported using the program BEFE [5] developed at the University of Technology Graz, Austria. The results obtained using our program are very similar to the ones obtained with the other programs, including the more powerful curved embedded approach used by BEFE [5].
Dedicado a la memoria de aquellas personas que han dejado una profunda huella en mi vida: mi madre Bertha Marin de Hutchinson, mi hermano Fernando Ivan Gonzales Marin, mis abuelos Rodolfo Hutchinson Sr. y Leonardo Marin Sr., mi tía Dona Romelia Marin y mis abuelas Bertha Marin de Medrano y Cecilia Almillategui de Guerra.
ACKNOWLEDGEMENTS

I would like to thank the following:

Dr. Okey Onyemelukwe, for the opportunity of doing this research under his guidance.

Dr. Lei Zhao, for his suggestions and recommendations about the modeling of posttensioned beams when we discussed the nature of my research. His teachings inspired my renewed interest in the theory of Engineering Mechanics.

My Supervisor at MECAA, Dr. Jacquelyn Smith for giving me the opportunity to work as tutor at her department, without doubt I could not have finished my studies without her invaluable help.

Dr. Manoj Chopra, for showing me the utility and power of other numerical methods in Engineering like the BEM, it completely changed the course of my career.

Edward Severino for suggesting me to make a finite element program for my thesis.

My Father and best friend, Rodolfo A. Hutchinson Sr., who has always been my support at very difficult times, always being there when I needed council and advice.

My good friend Ramon R. Fernandez for the help he has provided me since we know each other from elementary school. I wish you the best success in your engineering career, you really deserve it.
All my classmates Swapnil Chogle, Brian Glassman, Sanjay Shahji, Chris O’Riordan, Gahda Moussa; and Professors in the College of Engineering Dr. Shio-San Kuo, Dr. Belgin Erel, Dr. Sashi Kunnath, Dr. Sheriff El-Tawil, Dr Ayman Okeyl, Dr. David Nicholson and Dr. Faissal Moslehy; with whom I had the honor to share a classroom, I learned a lot from each and every one of you.

Also I would like to thank Dr. Mohammed Arafa and Dr. Sparowitz L Hartl for their help during the testing of the program when I used their examples to validate my results. The other examples that I employed were obtained from previous research done by Dr. Ergin Citipitioglu and Dr. Nasreddin El-Mezaini, for both of them my eternal gratitude.

Finally I would just like to thank Ana Ferreras, Katherine Meza, Jorge Arevalo, Cesar Casanas, Andrea Rios, and Hillary Sample for their friendship while I was couring my Master’s studies here at the department of Civil Engineering in UCF.
# TABLE OF CONTENTS

LIST OF FIGURES ........................................................................................................... xi

LIST OF TABLES ............................................................................................................. xv

1 INTRODUCTION ........................................................................................................... 1

1.1 Finite Element Analysis of Reinforced and Pre-Stressed Concrete...................... 3

1.2 Pre-stressed Concrete ............................................................................................ 7

1.3 Pre-stress Losses .................................................................................................... 8

1.3.1 Elastic Shortening .............................................................................................. 8

1.3.2 Frictional Losses ................................................................................................ 9

1.3.3 Anchorage Loss .................................................................................................. 9

1.3.4 Creep ................................................................................................................ 10

1.3.5 Steel Relaxation ................................................................................................ 10

1.3.6 Shrinkage of Concrete ....................................................................................... 11

1.4 Chapters Content .................................................................................................. 11

1.4.1 Chapter Two ..................................................................................................... 12

1.4.2 Chapter Three .................................................................................................. 12

1.4.3 Chapter Four .................................................................................................... 12

1.4.4 Chapter Five .................................................................................................... 12

1.4.5 Chapter Six ....................................................................................................... 13
2 LITERATURE REVIEW ........................................................................................................ 14
  2.1 Introduction .................................................................................................................. 14
  2.2 Previous Research ...................................................................................................... 15
    2.2.1 Discrete Modeling by Using Correction Technique for NMP ......................... 17
    2.2.2 Embedded Modeling by Using Curved Embedded Elements ......................... 20
    2.2.3 Reinforcement or Tendon Modeling by Using the Smeared Approach .......... 29
3 FINITE ELEMENT FORMULATION ............................................................................. 30
  3.1 Behavior of Concrete .................................................................................................. 30
    3.1.1 Concrete Material .............................................................................................. 35
  3.2 Reinforcing Steel ......................................................................................................... 44
    3.2.1 Steel Material Matrix ....................................................................................... 48
  3.3 Bond-Slip Model .......................................................................................................... 56
4 COMPUTER PROGRAM AND FLOWCHART ................................................................ 71
  4.1 CALFEM files ............................................................................................................. 71
    4.1.1 Assem ................................................................................................................ 71
    4.1.2 Extract ............................................................................................................... 73
    4.1.3 Coordxtr ......................................................................................................... 74
    4.1.4 Eldraw2 ............................................................................................................ 76
    4.1.5 Hooke ............................................................................................................... 77
    4.1.6 Plani8e .............................................................................................................. 79
    4.1.7 Plani8s .............................................................................................................. 81
    4.1.8 Plani8f .............................................................................................................. 82
    4.1.9 Solveq .............................................................................................................. 83
4.2 Computer Flowchart ........................................................................................................ 84
   4.2.1 IsoInputData ............................................................................................................. 84
   4.2.2 IsoDof .................................................................................................................... 88
   4.2.3 IsoCoord ................................................................................................................. 88
   4.2.4 IsoEdof ................................................................................................................... 89
   4.2.5 IsoGraph ................................................................................................................ 89
   4.2.6 IsoE ........................................................................................................................ 91
   4.2.7 IsoCoordTrussNodes ............................................................................................ 92
   4.2.8 IsoDofTruss .......................................................................................................... 93
   4.2.9 IsoEdofTruss ......................................................................................................... 93
   4.2.10 IsoGraph2 .......................................................................................................... 94
   4.2.11 IsoBoundaryCond .............................................................................................. 96
   4.2.12 IsoInitialForce2 .................................................................................................. 96

5 EXAMPLES ..................................................................................................................... 116

   5.1 Square Plate Problem with Straight Reinforcing Layers (Full Bonded Interface) 116
      5.1.1 Normal Stress on Concrete ............................................................................... 118
      5.1.2 Shear Stress on Concrete ................................................................................. 120
      5.1.3 Stresses along the Steel Reinforcement Layers .............................................. 122

   5.2 Simply Supported Reinforced Concrete Beam (Bonded or Un-bonded Interface)
      ..................................................................................................................................... 124
      5.2.1 Full Bond Condition: ....................................................................................... 127
      5.2.2 Un-Bonded Condition ....................................................................................... 128

   5.3 Two-Span Continuous Beam .................................................................................. 129
LIST OF FIGURES

Figure 2.1 One-dimensional Parent Element.............................................................. 18
Figure 2.2 Parabolic Steel Element Represented in Global Coordinates................. 23
Figure 2.3 Parabolic Steel Element Represented in Natural Coordinates ................. 23
Figure 2.4 Location of the Steel Element Nodes in Global Coordinates .................. 26
Figure 2.5 Location of the Steel Element Nodes in Natural Coordinates .................. 26
Figure 3.1 Compressive Stress-strain Response of Concrete and its Constituents .... 31
Figure 3.2 Strength Failure Envelope of Concrete (ref. [4])..................................... 33
Figure 3.3 Quadratic Plane Element used for Representing the Concrete Parent Elements.................................................................................................................. 37
Figure 3.4 Shape Functions for the Concrete Parent Elements (ref. [24])................. 39
Figure 3.5 Stress-Strain Relationship for non Pre-stressed Reinforcement (ref.[4]).... 44
Figure 3.6 Stress-Strain Relation for Post-tensioned Steel....................................... 46
Figure 3.7 Stress-Strain Relation Employed for the Steel Elements ....................... 47
Figure 3.8 Representation of the Steel Tendon or Reinforcement Embedded into the Concrete Parent Element (ref. [4])................................................................. 50
Figure 3.9 Steel Bar Element Represented in Natural Coordinates......................... 51
Figure 3.10 Steel Element Intersecting the Left Side of a Concrete Element (ref. [4])... 52
Figure 3.11 Representation in Natural Coordinates of the Parent Element’s Side being Intersected by the Steel Bar Element ................................................................. 53

Figure 3.12 Bond-link Element (ref. [4]) .......................................................................................................................... 56

Figure 3.13 Bond-slip Relation Between the Grout and the Steel Rebar (or Tendon) .... 57

Figure 3.14 Bond Stiffness Incorporated into the Stiffness of the Steel Bar Element Embedded (ref. [4]) .................................................................................................................................... 59

Figure 3.15 Distributed Load on a Concrete Parent Element ........................................ 64

Figure 3.16 Transfer of Forces from Two Steel Bar Elements (Sharing an Intersecting Point) into the Concrete Parent Elements ................................................................. 65

Figure 3.17 Steel Bar Elements Embedded into Concrete Parent Elements ................. 68

Figure 4.1 Nodes Numbering on the Concrete Parent Elements ........................................ 74

Figure 4.2 (Case 1.1) ............................................................................................................................................... 101

Figure 4.3 (Case 1.2) ............................................................................................................................................... 102

Figure 4.4 (Case 1.3) ............................................................................................................................................... 102

Figure 4.5 (Case 1.4) ............................................................................................................................................... 103

Figure 4.6 (Case 2.1) ............................................................................................................................................... 103

Figure 4.7 (Case 2.2) ............................................................................................................................................... 104

Figure 4.8 (Case 2.3) ............................................................................................................................................... 104

Figure 4.9 (Case 2.4) ............................................................................................................................................... 105

Figure 4.10 (Case 3.1) ............................................................................................................................................... 105

Figure 4.11 (Case 3.2) ............................................................................................................................................... 106

Figure 4.12 (Case 3.3) ............................................................................................................................................... 106

Figure 4.13 (Case 3.4) ............................................................................................................................................... 107
LIST OF TABLES

Table 3.1 Ultimate Stress - Yield Stress Relation for Post-tensioned Steel ................... 46

Table 3.2 Slip Values and Maximum Shear $\tau_{\text{max}}$ between Smooth Bars and Grout

According to the Model Code 90 (ref. [23]))............................................................... 58

Table 5.1 Comparison Between the Results Obtained Using MATLAB and the Results

Presented in ref. [1] for the Stresses along the Lower Steel Reinforcement Layer 128
1 INTRODUCTION

The profession of structural engineering is the oldest of all the branches of engineering, one with great reputation and responsibility. At the beginning of civilization the design of houses, temples and bridges were made based in pure empirical and experimental knowledge. Few documents and treaties about construction recommendations were available. Engineers’ knowledge was obtained from trial and error experience or transmitted from father to son. Even with all these adverse conditions, Engineers provided practical solutions to challenging structural problems, examples are the pyramids in Egypt or the temple of Hagia Sophia in Istanbul, this one is a peculiar case because it’s a structure of more that 1400 years located in one of the regions in the planet with highest seismic activity and still stands firmly today.

Huge advances have been made in the profession since ancient times; today design codes created by local officials exist in every country as guidelines to be followed by professional engineers, examples are the American Concrete Institute (ACI) code, the International Building Code (IBC) and the Comite Euro-International Du Beton (CEB-FIP) Model Code 1990. There is also available an enormous reference of past projects for which engineers can refer to for consulting reasons. It may look like everything is almost known in this branch of structural engineering, but even today structures still collapse due to negligence, natural disasters or just lack of knowledge.
The structural engineer’s principal responsibility is to provide the contractor the most cost-effective design for a structure under specific load cases, based on its location, function, etc.

The structural engineer’s work can be divided in two major parts: the Analysis and the Design of the structure. The design which is based on different approaches like the empirical Allowable Stress Design (ASD), Load and Factor Design (LFD), Load and Resistance Factor Design (LRFD), etc., will follow the local code requirement’s that were created not to restrain the designer creativity but to protect the public.

Before the design phase the Engineer needs to obtain the most critical load combination of Dead Load, Live Load, Wind Load, etc based on the function the structure will provide (Prison, Hospital, Dormitory, Parking Lot, etc.) and location (Near the Coast line, Seismic Risk zone, etc) to be able to design the structural elements.

Until the mid 1900’s the Analysis of indeterminate Reinforced Concrete Structures was made using nowadays obsolete methods, based for example on the Slope Deflection Method developed by Axel Bendixen [7] or the Moment Distribution Method developed by Hardy Cross [7]. For tall buildings this was a very arduous job, due to this, approximate methods like the Portal Method or the Cantilever Method were used for the analysis of large structures, and even these methods were very time consuming (and not completely accurate). Also, all these methods were only valid for the linear analysis of the structure. Nowadays these analytical methods might be obsolete but without doubt their principles form the basis of the theory of structures.

At the end of the 19th and beginning of the 20th century the theory of structures advanced far ahead compared with the practical tools available in those days for solving
practical problems on beams and columns. Due to the absence of computers for the analysis of structural elements in bridges or buildings, the hand calculation using i.e. Theory’s of Elasticity principles on beams and columns was definitely not even an option, you could imagine the frustration of engineers before the appearance of the computers, due to this the design of bridges and buildings was almost empirical, meaning that the results were amplified by a factor of “safety” to account for the effect of several uncertainties.

1.1 Finite Element Analysis of Reinforced and Pre-Stressed Concrete

In the 1950’s engineers were writing stiffness equations in matrix format, with the help of digital computers, these advances were made for solving design problems in the aeronautical Industry, it had to pass several years before it was made public due to the company’s policies.

The term “Finite Element” appears to be first used by Clough in 1960, it had to pass a couple of years before the FEA acquired recognition in 1963 as a form of the Rayleigh-Ritz method, and from its beginning this branch of Computational Mechanics has evolved at an exponential rate, new elements have been developed from the first three-node triangular element used by Turner to model the skin of a wing. There are elements nowadays for solving problems in 2 dimensional Plane Stress and Plane Strain analysis, 3-Dimensional analyses, Axial Symmetry, etc.

But even with the appearance of the personal computer, due to its cost, these methods of analysis were reserved for special cases such as high sensitive structures like
nuclear power plants. Thanks to the upgrade and accessibility of personal computers, these days almost any professional can perform a Finite Element Analysis on any type of structure with the right knowledge for the modeling of the structure, meaning the Engineer-Analyst know what kind of elements he needs to use for the structure modeling, how to apply the loads into the model, and most importantly, how to fix or adjust the input data files in case the program interface doesn’t allow the user to make the adjustment on the meshing. Thanks to the pioneering work of several individuals in this branch of engineering (Most of the pioneering finite element work was initiated at the University of California at Berkeley) like R.L. Taylor, O.C. Zienkiewicz, K.J. Bathe, Robert D. Cook, T. Belytschko, Alex Scordelis, Christian Meyer and many others, the Finite Element Method has been established as a solid procedure in solving analysis of structures; and even its use has not been just restricted into structural mechanics, it has found its way into fields like medicine and Bio-Engineering where I think its most useful potential lies ahead in improving common peoples life.

For now the finite element method has come into the point where it can provide even more accurate solutions, like coupling it with the Boundary Element Method, which also is a very powerful approach of analysis. We cannot say which one is better but what we can assure is that one complements the other. New methods are being researched at this moment like the Mesh-less Method, which also has a very promising future.

The modeling of reinforced concrete and Steel structures has been maybe one of the areas of structural engineering where a huge amount of effort (and resources) from many researchers has been focused, especially for modeling the effect of dynamic loads due to earthquakes, which causes havoc every year worldwide, or the effect of fatigue
loads due to traffic on bridges. With Reinforced concrete it is even more complex because its composite nature consists of two completely different materials working together at the same time. Steel is a homogeneous material and its properties are very well defined, on the other hand, concrete is a heterogeneous material made of mortar, cement and aggregates; because of this its properties cannot be defined easily.

Each one has its own behavior under the effect of static or dynamic loads, their own elastic and plastic range, etc. Also between both materials, secondary effects like tension stiffening and doweling effect do occur.

Between the end of the 1970’s and the end of the 1980’s several papers have been written about the modeling of Post-tensioned Concrete; state of the art research made by Scordelis [6] and Elwi [2] are some examples.

It is easy to model the behavior of each material separately, but the problem is how to model their composite nature, including all the effects that occur at their interface (surface of contact between the steel reinforcement and the concrete). With post-tensioned concrete we need to add the issue of modeling the tendon’s parabolic profile for example.

At the beginning there were two methods available for modeling the steel reinforcement: the smeared approach and the discrete approach. The first one is suitable for homogenously distributed steel reinforcement, for example a reinforced shear wall. With this smeared approach the quantities of the reinforcement are smeared uniformly over the element. For the discrete approach, the steel tendon elements are connected to the mesh at the concrete element nodes. On both cases the mesh becomes dependant of the reinforcement layout. Also the latter approach turns out to be expensive
computationally due to the increased number of small elements and the loss of accuracy due to the element aspect ratios, so a new approach had to be created for modeling the steel tendon own local axis independently of the global orientation of the concrete mesh, this being called the embedded approach.

Scordelis [6] proposed embedding the steel element as a 1 dimensional bar inside a concrete frame element (2 nodes, 3 D.O.F per node), depending on the local coordinates of the steel element it would transfer its axial forces and moments (due to its eccentricity with respect to the frame parent element cross sectional center of gravity) into the frame concrete parent element. About the inclusion of the steel segment stiffness into the frame concrete element, it was incorporated from its local axis into the frame element global axis of reference.

Elwi [2] proposed a very interesting approach for modeling the steel tendon, by inverse mapping it could be modeled exactly as parabolic, embedded inside the concrete parent element, this is the method used by the program BEFE for modeling Post-Tensioned concrete. In my opinion, this is the most accurate way of modeling a post-tensioned beam, by using this parabolic embedded approach.

Filippou [4] created a model (the one used in this research) for which the steel reinforcement is modeled as a 1 dimensional bar element embedded into an 8 node iso-parametric concrete parent element, and at the sides of the concrete parent element where the steel intersects the concrete element, link elements are used to connect the steel and the concrete elements to represent the bonding effect between both materials. In this way the steel is incorporated into the concrete element and from the concrete element displacement, the displacement and forces on the steel elements are obtained. The effect
of friction at the interface between the steel and the duct (or the steel and the grout) is governed by the change of cohesion on the interface due to the relative slip of the steel element nodes with respect to the concrete parent element nodes.

In the last example of this research we will see the difference in the results when the tendon is modeled exactly as parabolic embedded inside the concrete element, compared to the steel tendon being modeled as a straight embedded 1-D bar element.

1.2 Pre-stressed Concrete

Due to the fast growth of population in urban-areas the necessity of designing buildings and bridges with a considerable span length at an affordable cost, the construction industry found a perfect solution for its needs in using pre-stressed concrete. The use of pre-stressed concrete became very popular since its introduction by Freysenet in the 1930’s. It is very practical, provides the designer the opportunity to increase the span of beams and also reduces the cost of the work by allowing the members to be prefabricated, hence reducing the construction time (and costs) by a considerable amount. Even for seismic areas where it still hasn’t been widely used by the construction industry due to code restrictions, the use of post-tensioned concrete have the potential of being used for columns, as a solution for reducing the drift of a building under seismic forces.

The process and science of post-tensioning is very well known. It consists of concrete cast around un-tensioned steel. After the concrete reaches an acceptable strength the steel is tensioned with a jack, then the duct containing the steel tendon is grouted, this provides bond between the concrete and the steel tendon, increasing the capacity of the
structural member and at the same time providing protection for the steel tendon against corrosion. It differs from the normal reinforced concrete in the concept that high strength tendons transfer stresses into the concrete, compressing it before any static superimposed load is applied, at the same time the eccentricity of the tendons can be shaped before the casting of concrete as parabolic along the span of the beam, allowing the designer to use larger span beams (a great advantage).

### 1.3 Pre-stress Losses

From the instant that the initial jacking force is applied to the tendon and then transferred into the concrete there will be immediate post-tensioned losses (consisting of Elastic shortening which occurs only when all tendons are not jacked simultaneously, anchorage losses and losses due to friction between the tendon and the duct), and other time dependant losses (Creep, Shrinkage and Steel Relaxation).

#### 1.3.1 Elastic Shortening

When the initial forces are transferred from the tendons into the concrete, elastic shortening may occur; as it depends on the method used to jack the tendons. If all the tendons are jacked simultaneously there will not be a loss due to elastic shortening, on the other hand if the tendons are jacked in sequence, the last tendon being jacked will not have any loss due to elastic shortening, but the first one will have the cumulative elastic shortening from all the other tendons jacked after it. This loss is saved in our program by
reading the contraction of all the concrete parent elements after each tendon is jacked, and then from their displacement (of the concrete parent elements) the strain change in the steel elements is obtained.

1.3.2 Frictional Losses

The frictional loss between the duct and the tendons has two main components: the curvature and the wobble frictional losses. The first one is due to the change of angle of the tendon profile, the second one is due to the unintended angle changes of the tendon along its length, it depends on the rigidity of the sheathing, the diameter of the sheathing, the sheath type, the spacing of the sheath supports and the form of construction. In our routine from the initial forces at both ends of the beam (having the maximum force located at the live anchor and the minimum at the other end of the beam) the force at every intersection point between the concrete parent element and the steel tendon is obtained by interpolation.

1.3.3 Anchorage Loss

After the jacking force is applied into the tendon, it’s necessary to anchor the tendons; this often results in additional pre-stress losses due to the setting of the anchor wedges. The length of the tendon inside the beam affected by this anchorage pre-stress loss is a function of the frictional losses obtained previously, for un-bonded tendons for example, this length may be very large. In our program this pre-stress loss is obtained by
asking the user the wedge setting (usually should be around 0.25 inches), from this setting using a subroutine we obtain the length of the tendon affected by this anchorage loss, and assuming a constant frictional loss per unit length we obtain the new post-tensioned force at the live anchor, and the new distributed force along the tendon.

1.3.4 Creep

The response of concrete depends on the rate and the time history of loading. If we maintain a constant sustained stress upon concrete for some time, the strain will increase. This increase in strain on the concrete from long-term loads causes the modulus of elasticity of the concrete to be modified. Obtaining the amount of creep of a particular concrete takes more of an empirical approach and without specific tests accuracy better than 30% should not be expected, but there exist approximate procedures to estimate the creep deformations.

1.3.5 Steel Relaxation

The required force to hold a steel tendon at a constant elongation will decrease with time, this phenomenon is called relaxation. If the initial stress applied on the steel is less than 0.55Fpy it can be neglected. Its effect on steel is analogous with the effect of creep on concrete, and can only be predicted like creep only if information for the specific material under specific conditions is available; there exist also approximate procedures like creep to calculate the elastic relaxation on steel.
1.3.6 Shrinkage of Concrete

Concrete looses its moisture with time; hence it will decrease in volume unless kept under water or in air at 100 % humidity. It depends on the composition of the concrete, the amount of water in the mix and the quality of the aggregate (hard and dense aggregates absorb less water resulting in less shrinkage). Approximate expressions for obtaining the shrinkage of concrete are available for moist-cured concrete or for steam-cured concrete.

As we explained earlier only the anchorage loss and the frictional forces are accounted for a tendon at the instant it is being jacked, latter we need to add also the loss due to the elastic shortening caused by the rest of the tendons been jacked subsequently, unless of course all tendons are jacked simultaneously.

Time dependent losses due to shrinkage, creep and steel relaxation; need also to be taken into consideration when we want to obtain the actual state of stresses along the tendon at any instant.

1.4 Chapters Content

The content of each chapter is briefly explained in the following pages.
1.4.1 Chapter Two

In this chapter the basic theory of the different methodologies used for modeling Reinforced or Pre-stressed concrete is explained, this include the smeared approach, the discrete approach used on references [1] and [3], and the special embedded approach used on reference [5].

1.4.2 Chapter Three

In this chapter we discuss the finite element formulation employed in the program for the modeling of reinforced or post-tensioned concrete beam using the embedding approach.

1.4.3 Chapter Four

This chapter contains the description of every single file written in MATLAB for this thesis, including a quick description of the CALFEM [20] files used by the program.

1.4.4 Chapter Five

Four different problems are presented on this chapter, these problems were obtained from three different references and their results are compared with ours. The first one was a cantilever reinforced plate, the second problem consisted of a simple
supported reinforced concrete beam, the third problem consisted of a two span post-tensioned beam and the fourth problem of a simple supported post-tensioned beam.

1.4.5 Chapter Six

Here we discuss our findings and conclusions after comparing our results with the different references. Also we provide some recommendations for future research.
2 LITERATURE REVIEW

A compilation of the applicable literature showing the basic theory of the different methodologies used for modeling Reinforced or Pre-stressed concrete is presented in the present section.

2.1 Introduction

The tendon pre-stress loses that occur immediately after the jacking force is applied are due to anchorage setting, elastic shortening of the concrete and the frictional forces from the tendon profile. After these initial losses there will be additional time dependant losses like steel relaxation, creep and shrinkage. By adding all these possible immediate or time dependant losses, the engineer can obtain the total pre-stress loss that the post-tensioned member is going to suffer on a determinate lapse of time.

The modeling procedure that was employed in the program written in MATLAB consisted practically in using the concrete element as a governing element, where the forces are been transferred from the steel tendons (oriented on any direction) into the concrete parent element. Then, from the displacement of the concrete parent element nodes the displacement of the steel elements nodes that are in contact with any of the four
sides of the concrete parent element is obtained, taking into account also the effect of bonding at the interface between both materials.

2.2 Previous Research

One of the first researchers to model reinforced concrete was Scordelis. From the three different approaches for modeling the reinforced concrete, in 1967 Ngo and Scordelis [8] were the first to propose the discrete model, also Nilson [9] in 1968 (with small modifications and using a different bond model). The reinforcing bars were modeled using special elements (2-D triangular Elements or axial bar elements) that were connected to the concrete using fictitious springs that represent the bond effect between the steel and the concrete. This method allows the representation of the different material properties very precisely, but the problem is that the finite element mesh pattern will be restricted by the location of the reinforcement.

The second approach for modeling the steel reinforcement into the concrete parent element is the smeared approach. What does this method of analysis consist? Basically, the steel is modeled as homogenously distributed throughout the concrete element to form a composite stiffness; this method is simple and perfect for concrete elements like shear walls that have a uniform distribution of reinforcement. This approach also has an advantage when we want to model the appearance of cracks in the concrete, compared with modeling the cracks discretely, because it distributes the cracking over the entire concrete element or at the integration points within the concrete element. This provides the opportunity of using the same structural nodal point topology
throughout the nonlinear solution without having to define a new node topology for each analysis step (discrete crack modeling). Several papers using this approach have been published in different journals of engineering. Barzegar [10] in 1989 presented a formulation for the analysis of Reinforced Concrete membrane elements with anisotropic reinforcement, also Vechio [11] in 1990 presented a formulation for Reinforced Concrete Membranes based on the modified compression field theory, another paper written using smeared approach was written by Hu and Schnorbrich [12] in 1990 for the nonlinear analysis of cracked Reinforced Concrete.

A number of different formulations to model the reinforcement into the concrete using the third approach (embedding) were developed afterwards to overcome the problem of mesh dependency in the discrete model. Phillips and Zienkiewcz [13] in 1976 developed an embedded approach as long as the reinforcement layer was aligned with one of the concrete Iso-parametric element local axes. In other words, with this approach of reinforcement embedding, the steel element is considered as an axial member built into the Iso-Parametric Element such that its displacements are consistent with those of the concrete element, in this case the steel element and the concrete element are perfectly bonded.

Basically, for modeling a post-tensioned concrete beam a method will be needed that will allow an independent choice of concrete mesh and at the same time allow the post-tensioned tendon to intersect the parent element without restrictions. That is why the embedded approach was used in this research with the inclusion of bonding between the steel and the concrete material. The results obtained with the program developed in this study were compared with those from 4 examples. These problems were originally done
with different approaches. The first three examples were obtained from 2 different references El-Mezani [3] and Mehlhorn [1], and the method used for modeling the reinforcement or the steel tendon was the discrete approach. On these three examples the nodes of the concrete element are moved to the location of the coordinates were the tendon or steel reinforcement intersect the concrete element. The last example had used the embedding approach by embedment of curved steel elements.

The theories are presented next.

### 2.2.1 Discrete Modeling by Using Correction Technique for NMP

By moving the concrete nodes into the coordinate location where the steel element intersects the concrete element the analyst will face a problem, which is called node-mapping distortion (NMD). If the change of location of the node from its original position is too large a singular Jacobian matrix will be obtained in the processing of the stiffness matrix. This can be eliminated by the following technique developed by Citipitioglu and Nicolas [15], which modifies the shape function of the concrete parent element. This technique can be used for 1, 2 and 3 dimensional parent elements, but for practical purposes, it will be explained for one and two dimensional parent elements only.

The shape functions for one, two or three-dimensional elements can be generated by superimposing the appropriate functions that are derived for one-dimensional interpolations. These functions, which are derived for one dimensional parent elements with \( n \) interior nodes of arbitrary positions \( \xi = \xi_1, \xi_2, ..., \xi_n \), as shown in Fig.2.1, are presented next:
L \left( \xi_i \right) = 0.5 \left( 1 + \xi_i \xi \right) \quad \text{(Eq. 2.1)}

R \left( \xi_i \right) = \prod_{m=1}^{n} \frac{\xi_i - \xi_m}{\xi_i - \xi_m} \quad \text{(Eq. 2.2)}

Q \left( \xi_k \right) = \frac{\xi_k^2 - 1}{\xi_k^2 - 1} \quad \text{(Eq. 2.3)}

S \left( \xi_k \right) = \prod_{m=1, m \neq k}^{n} \frac{\xi_k - \xi_m}{\xi_k - \xi_m} \quad \text{(Eq. 2.4)}

Where k = 1, 2... n

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure2.1.png}
\caption{One-dimensional Parent Element}
\end{figure}

\section*{2.2.1.1 One-Dimensional Parent Elements}

For a one dimensional parent element with "n" number of interior nodes (by interior nodes we mean excluding the exterior end nodes: i.e. the first and the last node)
the shape function of both the exterior and interior nodes can be obtained by the following equations:

**End nodes shape functions**

\[ N(\xi_i) = L(\xi_i) \cdot R(\xi_i) \quad \text{(Eq.2.5)} \]

**Interior node shape functions**

\[ N(\xi_k) = Q(\xi_k) \cdot S(\xi_k) \quad \text{(Eq.2.6)} \]

### 2.2.1.2 Two-Dimensional Parent Elements

In the case of two dimensional parent elements the modified shape function of both the corner nodes and the side nodes can be obtained with the following expressions:

**Corner node Shape functions**

\[ N(\xi_i, \eta_i) = L(\xi_i) \cdot L(\eta_i) \left[ R(\xi_i) + R(\eta_i) - 1 \right] \quad \text{(Eq.2.7)} \]

**Side node shape functions**

\[ N(\xi_i, \eta_k) = L(\xi_i) \cdot Q(\eta_k) \cdot S(\eta_k) \quad \text{(Eq.2.8)} \]
\[ N(\xi_k, \eta_l) = L(\eta_l) \cdot Q(\xi_k) \cdot S(\xi_k) \]  \hspace{1cm} (Eq.2.9)

With the previous approach the adjustment in the shape function of the concrete parent element nodes can be made, without worrying about node mapping distortion when its side nodes are moved at the location where the concrete parent element is being intersected by the steel element. Then after this procedure the assembly of the stiffness of both materials can be easily performed.

2.2.2 Embedded Modeling by Using Curved Embedded Elements

The last example presented in this thesis consisted of a post-tensioned beam modeled with the program BEFE, this beam uses embedded curved elements. The theory is based on a paper published by Elwi and Hrudey [2] in 1989.

The employment of embedded parabolic elements will allow the analyst to upgrade the accuracy of the analysis as much as it is possible. It will be included not only nodes from the steel elements that are in contact with one of the sides of the concrete parent element, nodes from the steel element that are inside the concrete parent element will also be employed, now the only problem is that for solving the analytical integration the natural coordinates of those steel nodes inside the concrete parent element are needed, for this reason it will be necessary to do an inverse mapping operation. The procedure for mapping the concrete parent element from global to natural coordinates is straightforward.

By using a bilinear field like the following expression
\[
X = A_0 + A_1 \xi + A_2 \eta + A_3 \xi \eta \\
Y = A_0 + A_1 \xi + A_2 \eta + A_3 \xi \eta
\]  
(Eq.2.10)

The shape function of the Concrete parent element in the X and Y global coordinates will be represented as:

\[
X = \Phi^* \{x\} \\
Y = \Phi^* \{y\}
\]  
(Eq.2.12)

(Eq.2.13)

Where \(\{x\}\) and \(\{y\}\) are vectors with the global coordinates of the concrete parent element nodes and the vector \(\Phi^*\) contains the shape function of the concrete parent element nodes. For the conversion of the concrete parent element global coordinates into natural coordinates the Jacobian will be employed, hence the relation between both coordinate systems will be:

\[
\begin{bmatrix}
dx \\
dy
\end{bmatrix} = [J] \cdot \begin{bmatrix}
d\xi \\
d\eta
\end{bmatrix}
\]  
(Eq.2.14)

Where:
Due to its parabolic shape, the steel element shape function will be obtained using an independent normalized coordinate “ζ” from a linear field (i.e. 3 nodes):

\[
\begin{align*}
X &= A_0 + A_1 \zeta + A_2 \zeta^2 \\
Y &= A_0 + A_4 \zeta + A_5 \zeta^2
\end{align*}
\] (Eq.2.15)

These two equations (2.15 and 2.16) will provide the shape function of the steel element

\[
\begin{pmatrix}
X \\
Y
\end{pmatrix} = \begin{bmatrix}
\langle \psi \rangle & 0 \\
0 & \langle \psi \rangle
\end{bmatrix} \cdot \begin{pmatrix}
x^* \\
y^*
\end{pmatrix}
\] (Eq.2.17)

Where \(x^*\) and \(y^*\) are vectors containing the global coordinates of the steel element associated with the concrete parent element. The vector \(\langle \psi \rangle\) contains the shape function of the steel element nodes.
Figure 2.2 Parabolic Steel Element Represented in Global Coordinates

Figure 2.3 Parabolic Steel Element Represented in Natural Coordinates
The length “S” in global coordinates is related to X and Y by the following

\[ ds = \sqrt{(dx)^2 + (dy)^2} \]  

(Eq.2.18)

Dividing Eq.2.18 by \( d\zeta \) will give:

\[ \frac{ds}{d\zeta} = \sqrt{\left(\frac{dx}{d\zeta}\right)^2 + \left(\frac{dy}{d\zeta}\right)^2} \]  

(Eq.2.19)

Eq.2.19 can be solved after obtaining the derivative of Eq.2.17 with respect to \( d\zeta \)

\[
\begin{bmatrix}
\frac{dx}{d\zeta} \\
\frac{dy}{d\zeta}
\end{bmatrix} =
\begin{bmatrix}
\frac{dy}{d\zeta} & 0 \\
0 & \frac{dy}{d\zeta}
\end{bmatrix}
\begin{bmatrix}
x^* \\
y^*
\end{bmatrix}
\]  

(Eq.2.20)

By integration along the natural coordinate length \( d\zeta \), the volume or the surface area along the steel layer inside the concrete parent element can be obtained.

For the steel volume will be:

\[ dV_s = t \cdot A_s \cdot ds \]  

(Eq.2.21)

Or
\[ dV_s = \int_{\zeta} t \cdot A_s \left( \frac{ds}{d\zeta} \right) d\zeta \]  \hspace{1cm} (Eq.2.22)

And for the surface area:

\[ dS_s = t \cdot O_s \cdot ds \]  \hspace{1cm} (Eq.2.23)

Or

\[ dS_s = \int_{\zeta} t \cdot O_s \left( \frac{ds}{d\zeta} \right) d\zeta \]  \hspace{1cm} (Eq.2.24)

Where \( dV_s \) and \( dS_s \) represent a differential element of volume and surface area respectively of the reinforcement layer, \( A_s \) represents the cross sectional area of the layer per unit thickness and \( O_s \) the perimeter of the layer per unit thickness.

The natural coordinates of all the steel nodes inside the concrete parent element are needed before doing the integration along the embedded steel element. The steel reinforcement layer geometry is defined by the location of the layer nodes. The integration of the incremental virtual work in the reinforcing layer is made using the strain in the parent element at the location of the points of the reinforcement layer inside the concrete parent element. The integration path for the embedded steel reinforcement is presented in Fig.2.4. Once more the mapping between local and global coordinates of the concrete parent element is presented in Eq.2.25.
\[
\begin{bmatrix}
X \\
Y
\end{bmatrix} =
\begin{bmatrix}
<\Phi(\xi,\eta)> & 0 \\
0 & <\Phi(\xi,\eta)>
\end{bmatrix}
\begin{bmatrix}
x \\
y
\end{bmatrix}
\]

(Eq. 2.25)

Figure 2.4 Location of the Steel Element Nodes in Global Coordinates

Figure 2.5 Location of the Steel Element Nodes in Natural Coordinates
By analytical integration we start at point “O”, for which the global coordinates 
\((xo, yo)\) are known and end at point “P” for which the global coordinates 
\((xp, yp)\) are also known. The natural coordinates of point “P”  \((\xi_p, \eta_p)\) is to be found and for convenience the natural coordinates of point “O” is taken to be the origin of the natural coordinate system.

Assuming that the mapping from \((xp, yp)\) to \((\xi_p, \eta_p)\) exists and is unique, the choice of the integration path from “O” to “P” is arbitrary; a convenient choice will be a straight line between “O” and “P”.

By defining “S” a normalized distance along this line with S=0 at “O” and S=1 at “P” the path in the global coordinates (Fig.2.4) can be expressed in parametric form as:

\[
\begin{cases}
X(S) = X_o + S(X_p - X_o) \\
Y(S) = Y_o + S(Y_p - Y_o)
\end{cases}
\]  

(Eq.2.26)

Thus

\[
\begin{cases}
\frac{dx}{dy} = \frac{X_p - X_o}{Y_p - Y_o} 
\end{cases}
\]

(Eq.2.27)

And from Eq.2.14

\[
\begin{cases}
\frac{d\xi}{d\eta} = [J(\xi, \eta)]^{-1} \begin{bmatrix} dx \\ dy \end{bmatrix}
\end{cases}
\]

(Eq.2.28)
Then

\[
\begin{align*}
\begin{bmatrix}
\frac{d\xi}{d\eta} \\
\frac{d\eta}{d\xi}
\end{bmatrix} &= [J(\xi,\eta)]^{-1} \cdot \begin{bmatrix}
X_p - X_o \\
Y_p - Y_o
\end{bmatrix} ds \\
\end{align*}
\]  

(Eq.2.29)

That is equal to:

\[
\begin{align*}
\begin{bmatrix}
\frac{d\xi}{ds} \\
\frac{d\eta}{ds}
\end{bmatrix} &= [J(\xi,\eta)]^{-1} \begin{bmatrix}
X_p - X_o \\
Y_p - Y_o
\end{bmatrix} \\
\end{align*}
\]  

(Eq.2.30)

This is a system of two first order differential equations. Solving the system as an initial value problem with the initial condition:

\[
\begin{align*}
\begin{bmatrix}
\xi \\
\eta
\end{bmatrix} \bigg|_{S=0} &= \begin{bmatrix} 0 \\ 0 \end{bmatrix}
\end{align*}
\]

And integrating from \(S=0\) to \(S=1\) the coordinates \((\xi_p, \eta_p)\) are obtained. It can be solved using i.e. the Runge-Kutta schemes as a standard algorithm for the integration of initial value problem. This was the approach used for obtaining the local coordinates of the steel element nodes inside the concrete parent element. After the location of this node is obtained, the stiffness of the steel element can be assembled into the concrete element by analytical integration. This assembly assumes perfect bond between the steel and the concrete elements.
2.2.3 Reinforcement or Tendon Modeling by Using the Smeared Approach

The employment of this method for modeling reinforced concrete members is used mostly for modeling cracks developed at the concrete parent element Gaussian points, or for modeling reinforced concrete slabs. For the latter, the steel and the concrete on the slab are represented using plate elements. The slab is divided into several layers of plate elements, and then the steel and the concrete material properties are distributed all along the plates located at the level of the steel and concrete respectively (Because in this thesis and in the references used to compare our results like ref [1], ref [3] and ref [5] the smeared approach was not used, the reader is referred to references [4] and [14] for a detailed discussion of the smeared modeling approach on concrete slabs).
3 FINITE ELEMENT FORMULATION

On this chapter the theory employed for modeling a post-tensioned beam of one or two continuous spans is explained. The modeling of the steel elements into the concrete parent elements was done using the embedding approach.

3.1 Behavior of Concrete

While the compressive stress-strain response of the constituents of concrete (the aggregates and the cement paste) are linear, the stress-strain response of the resulting concrete is nonlinear (the aggregates are stiffer and stronger than the paste). The nonlinearity of the concrete stress-strain response is caused by the interaction between the paste and the aggregate, as such the initial tangent stiffness of the concrete $E_c$ lies between the stiffness of the aggregate and the stiffness of the paste.
Figure 3.1 Compressive Stress-strain Response of Concrete and its Constituents

The value of the tangent stiffness of the concrete $E_c$ can be estimated from the stiffness of the aggregates and the paste using composite material modeling laws, but if only the strength and the unit weight of the concrete are known, $E_c$ can be estimated from the equation recommended by the ACI code:

$$E_c = w_c^{1.33}33\sqrt{f_c} \quad \text{Psi}$$  \hspace{1cm} (Eq.3.1)

For normal weight concretes this equation gives:

$$E_c = 57000\sqrt{f_c} \quad \text{Psi}$$  \hspace{1cm} (Eq.3.2)
However some researchers (Ref. [16]) point out that Eq.3.2 overestimates the stiffness of the concrete with strength greater than 6000 psi. They recommend that the stiffness of normal weight concrete be calculated as:

\[ E_c = 40000\sqrt{f'_c} + 1000000 \text{ Psi} \]  

(Eq.3.3)

This equation was employed in the program written in MATLAB, to obtain the modulus of elasticity of the concrete material when \( f'_c \) was greater than 6000 psi, if it is smaller then Eq. 3.2 will be used.

The response of any structure under static or dynamic loads depends on the stress-strain relation of the constituent materials. As it is well known, the concrete is excellent in compression, that’s why the stress-strain relation of concrete in compression is of primary interest, this stress-strain relation can be obtained from a cylinder test.

The stress-strain relation is linear up to around 30% of the concrete’s compressive strength, after this there is a gradual softening of the concrete up to it’s maximum compressive strength (at this point the material stiffness drops to zero), beyond the strain at the maximum compressive strength there is a strain softening on the stress-strain relation up to the point that failure occurs due to concrete crushing (see Fig. 3.1).

The strength failure envelope of concrete, under combinations of biaxial stress is different to that under uni-axial loading conditions (Fig. 3.2).
Figure 3.2 Strength Failure Envelope of Concrete (ref. [4])

The biaxial strength envelope of concrete under proportional loading in Fig.3.2 (Kupfer et al. 1969; Tatsuji et al. 1978) shows that (i.e. under biaxial compression) concrete exhibits an increase of compressive strength of about 25% of the uni-axial strength when the stress ratio \( \frac{\sigma_1}{\sigma_2} \) is 0.5. On the other hand, under biaxial tension concrete exhibits a constant or maybe slightly increased strength compared with that under uni-axial loading. With a combination of tension and compression, the concrete strength decreases linearly with increasing the tensile stress.

The principal stress ratio has an influence on the stiffness and the strain ductility of the concrete. Thus, under biaxial compression, concrete exhibits an increase in the initial stiffness attributed to Poisson’s effect, and also an increase in strain ductility.
meaning that less internal damage takes place under biaxial compression than under uni-axial loading.

Most of the beams or slabs subjected to bending moments experience biaxial stress combinations in the tension-tension or compression-compression region, because of this, a different degree of approximation must be used in each region: one for the compression-compression region, and another for the tension-tension or compression-tension region. The behavior of the model depends on the location of the present stress state in the principal stress space (Fig. 3.2).

In the biaxial compression region, the model remains linear elastic for stress combination inside the initial yield surface. There are two surfaces on Fig. 3.2; both (initial yield and ultimate load surface) are described by the expression proposed by Kupfer et al. (1969).

\[ F = \frac{(\sigma_1 + \sigma_2)^2}{\sigma_2 + 3.65\sigma_1} - A \cdot f_c = 0 \]  
(Eq.3.4)

Both Stresses \( \sigma_1 \) and \( \sigma_2 \) are the principal stresses, \( f_c \) is the uni-axial concrete Strength and \( A \) is a parameter. This parameter \( A \) is equal to 0.6 when it defines the initial yield surface while \( A=1.0 \) defines the ultimate load surface under biaxial compression. Stress Combinations outside the initial yield surface but inside the ultimate failure envelope must be described by a nonlinear model, i.e. an orthotropic model.

In this thesis, concrete is analyzed only in the linear elastic range for both regions of biaxial tension and biaxial compression.
3.1.1 Concrete Material

The study presented in this thesis was done using plane stress analysis (instead of the three dimensional stress analysis), which well matches the reality of the four beam examples contained in chapter five due to the absence of in-plane stresses on these examples.

For stress combinations inside the initial yield surface in Fig. 2, concrete behaves as homogeneous and linear isotropic. Then the stress-strain relation for plane stress problems will have the simple form:

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}
= \frac{E_c}{1-\nu^2}
\begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & \frac{1-\nu}{2}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\]  
(Eq.3.5)

Where \(\nu=\) Poisson’s ratio

From Eq. 3.5 we get the concrete material matrix in global coordinates

\[
[D_{ci}] = \frac{E_c}{1-\nu^2}
\begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & \frac{1-\nu}{2}
\end{bmatrix}
\]  
(Eq.3.6)

To obtain the element stiffness matrix \([K]\) of a particular concrete element, it will be defined as a two dimensional element which lie in the x-y plane of the beam
elevation. All the examples contained on chapter five were analyzed using plane stress analysis, the assumption that the concrete parent element is loaded only on its own plane (the x-y plane) is followed. The stresses will be constant through the thickness of each concrete element, and due to the absence of any restrain on the z-direction (in or out of plane x-y) the stresses $\sigma_z$, $\tau_{xz}$ and $\tau_{yz}$ will be equal to zero. All these assumptions lead to a plane stress field.

Interpolation is the cornerstone of the finite element method. The shape function matrix $[N]$ serves as a basis from which a finite element can be formulated.

$$f = [N]{f} = \sum_{i=1}^{n} N_i f_i$$  \hspace{1cm} (Eq.3.7)

Where: $n$ represent the number of degrees of freedom in the element, $f$ is a dependent field variable and the terms $N_i$ are interpolation functions.

Each interpolation function $N_i$ defines how $f$ varies within the element when the corresponding degree of freedom has unit value, while the other dofs are equal to zero.

A field $f$ is said to have $C^m$ continuity if derivatives of the field through order $m$ are continuous. The concrete parent element has $C^0$ continuity, then if $f = f(x)$, $f$ is $C^0$ continuous if $f$ is continuous, but $f_x$ is not.

For the 8 node iso-parametric Concrete Parent Element there are two dependent field variables, each one is in function of the parent element natural coordinates $(\xi, \eta)$: $U = U(\xi, \eta)$ and $V = V(\xi, \eta)$. 

36
The 8-node concrete parent element (Figure 3.3) has 16 degrees of freedom. The displacements $U$ and $V$ are interpolated from 8 nodal values, that is:

$$ U = \sum_{i=1}^{n} N_i \cdot U_i \quad \text{(Eq.3.8)} $$

And

$$ V = \sum_{i=1}^{n} N_i \cdot V_i \quad \text{(Eq.3.9)} $$

The shape functions contained in Eq. 3.8 and Eq. 3.9 are equal to:
\[ N_1(\zeta, \eta) = \frac{1}{4}(1 - \xi)(1 - \eta)(-1 - \xi - \eta) \]  
(Eq.3.10a)

\[ N_2(\zeta, \eta) = \frac{1}{4}(1 + \xi)(1 - \eta)(-1 + \xi - \eta) \]  
(Eq.3.10b)

\[ N_3(\zeta, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta)(-1 + \xi + \eta) \]  
(Eq.3.10c)

\[ N_4(\zeta, \eta) = \frac{1}{4}(1 - \xi)(1 + \eta)(-1 - \xi + \eta) \]  
(Eq.3.10d)

\[ N_5(\zeta, \eta) = \frac{1}{2}(1 - \eta)(1 - \xi^2) \]  
(Eq.3.10e)

\[ N_6(\zeta, \eta) = \frac{1}{2}(1 + \xi)(1 - \eta^2) \]  
(Eq.3.10f)

\[ N_7(\zeta, \eta) = \frac{1}{2}(1 + \eta)(1 - \xi^2) \]  
(Eq.3.10g)

\[ N_8(\zeta, \eta) = \frac{1}{2}(1 - \xi)(1 - \eta^2) \]  
(Eq.3.10h)
The concrete parent element is iso-parametric, meaning that the shape functions in global coordinates and natural coordinates have the same degree. Hence:

If \[ U = [N] \begin{bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{bmatrix} \]  \hspace{1cm} (Eq.3.11)

Then \[ X = [N] \begin{bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{bmatrix} \]  \hspace{1cm} (Eq.3.12)
Both U and V are in function of $\xi$ and $\eta$. The discretization of the displacement field $\{d\}$ is:

$$\{d\} = \begin{bmatrix} U \\ V \end{bmatrix} = \sum_{j=1}^{n} N_j \begin{bmatrix} 0 \\ N_j \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix}_j$$

(Eq.3.13)

Where: $n$ is the number of nodes of the concrete element, and $N_j$ is the shape function for the j-th node. The strain-displacement relation becomes:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial N}{\partial x} & 0 \\ 0 & \frac{\partial N}{\partial y} \\ \frac{\partial N}{\partial y} & \frac{\partial N}{\partial x} \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix}^{ele}$$

(Eq.3.14)

Or

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 1 & 1 & 0 \end{bmatrix} \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial y} \end{bmatrix} = [X_1]$$

(Eq.3.15)

Relating both coordinate systems (Global and Natural):
\[
\begin{align*}
\begin{bmatrix}
\frac{\partial u}{\partial \xi} \\
\frac{\partial v}{\partial \xi} \\
\frac{\partial u}{\partial \eta} \\
\frac{\partial v}{\partial \eta}
\end{bmatrix} &= 
\begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & 0 & 0 \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\
0 & 0 & \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}
\end{bmatrix} \\
& \times 
\begin{bmatrix}
\frac{\partial u}{\partial \xi} \\
\frac{\partial v}{\partial \xi} \\
\frac{\partial u}{\partial \eta} \\
\frac{\partial v}{\partial \eta}
\end{bmatrix}
\end{align*}
\] (Eq.3.16)

Or

\[
\begin{align*}
\begin{bmatrix}
\frac{\partial u}{\partial \xi} \\
\frac{\partial v}{\partial \xi} \\
\frac{\partial u}{\partial \eta} \\
\frac{\partial v}{\partial \eta}
\end{bmatrix} &= 
\begin{bmatrix}
J_{11} & J_{12} & 0 & 0 \\
J_{21} & J_{22} & 0 & 0 \\
0 & 0 & J_{11} & J_{12} \\
0 & 0 & J_{21} & J_{22}
\end{bmatrix} \\
& \times 
\begin{bmatrix}
\frac{\partial u}{\partial \xi} \\
\frac{\partial v}{\partial \xi} \\
\frac{\partial u}{\partial \eta} \\
\frac{\partial v}{\partial \eta}
\end{bmatrix}
\end{align*}
\] (Eq.3.17)

From Eq. 3.17, Matrix [X2] is obtained using the inverse matrix operation

\[
\begin{align*}
\begin{bmatrix}
\frac{\partial u}{\partial \xi} \\
\frac{\partial v}{\partial \xi} \\
\frac{\partial u}{\partial \eta} \\
\frac{\partial v}{\partial \eta}
\end{bmatrix} &= 
\begin{bmatrix}
\frac{\partial u}{\partial \xi} \\
\frac{\partial v}{\partial \xi} \\
\frac{\partial u}{\partial \eta} \\
\frac{\partial v}{\partial \eta}
\end{bmatrix}
\end{align*}
\] (Eq.3.18)

Where
\[
[X2] = \begin{bmatrix} J_{11}^* & J_{12}^* & 0 & 0 \\ J_{21}^* & J_{22}^* & 0 & 0 \\ 0 & 0 & J_{11}^* & J_{12}^* \\ 0 & 0 & J_{21}^* & J_{22}^* \end{bmatrix}
\]

Eq.3.18 is then incorporated into Eq.3.15

\[
\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = [X1][X2]^* \begin{bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial \xi}{\partial u} \\ \frac{\partial \eta}{\partial \xi} \\ \frac{\partial \xi}{\partial \eta} \end{bmatrix}
\]

(Eq.3.19)

Where

\[
\begin{bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial \xi}{\partial u} \\ \frac{\partial \eta}{\partial \xi} \\ \frac{\partial \xi}{\partial \eta} \end{bmatrix} = [X3] \{X4\}
\]

At the same time Matrix [X3] and Vector \{X4\} are equal to
\[
\begin{bmatrix}
\frac{\partial N_1}{\partial \xi} & \frac{\partial N_1}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_1}{\partial \xi} & \frac{\partial N_1}{\partial \eta} \\
\frac{\partial N_2}{\partial \xi} & \frac{\partial N_2}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_2}{\partial \xi} & \frac{\partial N_2}{\partial \eta} \\
\frac{\partial N_3}{\partial \xi} & \frac{\partial N_3}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_3}{\partial \xi} & \frac{\partial N_3}{\partial \eta} \\
\frac{\partial N_4}{\partial \xi} & \frac{\partial N_4}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_4}{\partial \xi} & \frac{\partial N_4}{\partial \eta} \\
\frac{\partial N_5}{\partial \xi} & \frac{\partial N_5}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_5}{\partial \xi} & \frac{\partial N_5}{\partial \eta} \\
\frac{\partial N_6}{\partial \xi} & \frac{\partial N_6}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_6}{\partial \xi} & \frac{\partial N_6}{\partial \eta} \\
\frac{\partial N_7}{\partial \xi} & \frac{\partial N_7}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_7}{\partial \xi} & \frac{\partial N_7}{\partial \eta} \\
\frac{\partial N_8}{\partial \xi} & \frac{\partial N_8}{\partial \eta} & 0 & 0 \\
0 & 0 & \frac{\partial N_8}{\partial \xi} & \frac{\partial N_8}{\partial \eta}
\end{bmatrix}^T
\]

\{X_4\} = \{U_1, V_1, U_2, V_2, U_3, V_3, U_4, V_4, U_5, V_5, U_6, V_6, U_7, V_7, U_8, V_8\}^T

Finally Eq. 3.15 will be equal to:

\[
\begin{aligned}
\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} &= [X1]*[X2]*[X3]*[X4] \\
\text{(Eq.3.20)}
\end{aligned}
\]

Equation 3.20 can also be expressed in the form:

\[
\begin{aligned}
\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} &= [B]*[X4] \\
\text{(Eq.3.21)}
\end{aligned}
\]
Where Matrix \([B] = [X1] [X2] [X3]\).

Matrix \([B]\) is used for creating the concrete parent element stiffness matrix. The concrete element stiffness matrix \([K]\) is obtained from:

\[
[K]_c = \int_x [B] ^T \cdot [D_{gl}]_x \cdot [B] dV
\]  
(Eq.3.22)

### 3.2 Reinforcing Steel

A single stress-strain relation usually is used to define the material property of regular steel.

![Stress-Strain Relationship](image)

Figure 3.5 Stress-Strain Relationship for non Pre-stressed Reinforcement (ref.[4])
Where $E_{s1}$ is the steel modulus before yielding and $E_{s2}$ is the steel modulus after yielding, it is assumed that after the steel reaches yielding, the strength of the steel keeps constant so $E_{s2}$ is zero.

The kind of stress-strain curves for reinforcing steel bars like the one in Fig.3.5 are obtained from coupon test of bars loaded monotonically in tension, and for practical purposes the stress-strain curve in compression is the same as in tension.

The steel-strain relation exhibits an initial linear elastic portion, then after the elastic limit the stress drops off until fracture occurs. The extension of this yield plateau is a function of the tensile strength of steel, for high-strength steel the yield plateau is shorter than for relatively low-strength steel.

For post-tensioning tendons, due to their high strength the stress-strain relation is quite different to that of non pre-stressed normal steel reinforcement. The following relation can approximate the stress-strain relationship for post-tensioned steel tendons:

\[ f_p = E_p \epsilon_{pf} \leq f_{py} \]  

(Eq.3.23)

There are some strands or tendons that do not exhibit a yield plateau, for these tendons an equivalent “yield stress” can be defined as the stress at a strain of 0.01. Some tables are available like Table 3.1, which relates the ultimate stress $f_{pu}$ and the yield stress $f_{py}$ for different types of pre-stressing steel.
Table 3.1 Ultimate Stress - Yield Stress Relation for Post-tensioned Steel

<table>
<thead>
<tr>
<th>Tendon Type</th>
<th>$f_{py} / f_{pu}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low-relaxation Strand</td>
<td>0.90</td>
</tr>
<tr>
<td>Stress-relieved Strand</td>
<td>0.85</td>
</tr>
<tr>
<td>Plain Pre-stressing Bars</td>
<td>0.85</td>
</tr>
<tr>
<td>Deformed Pre-stressing Bars</td>
<td>0.80</td>
</tr>
</tbody>
</table>

Also, a more accurate representation of the stress-strain response of pre-stressing strands can be obtained with the modified Ramberg-Osgood function recommended by Mattock [18] for low-relaxation strand and stress-relieved strands.

For low-relaxation strands:
\[ f_p = E_p \varepsilon_{pf} \left[ 0.025 + \frac{0.975}{1 + (118 \varepsilon_{pf})^{10}} \right] \leq f_{pu} \]  
(Eq.3.24)

For Stress-relieved strands:

\[ f_p = E_p \varepsilon_{pf} \left[ 0.03 + \frac{0.97}{1 + (121 \varepsilon_{pf})^{167}} \right] \leq f_{pu} \]  
(Eq.3.25)

In this research, the stress-strain relationship is divided into two phases (both the same for non-pre-stressed steel bars under tension or compression). The stress increases linearly up to the yield stress \( f_y \) (\( f_{py} \) in the case of pre-stressed steel tendons), and after it reaches yielding the stress keeps constant up until the point that the steel reinforcement fails by rupture.

Figure 3.7 Stress-Strain Relation Employed for the Steel Elements
Because the steel elements have the form of reinforcing bars or wires, it is not necessary to introduce three dimensional complexities, then it will be enough to use the idealized one dimensional stress-strain relation for the steel elements presented in Fig.3.7. The reason for using this approximation is the convenience for the computational model.

### 3.2.1 Steel Material Matrix

In this thesis, we used the embedded model. This model is useful in connection with higher order Iso-Parametric concrete elements. The reinforcing bar or steel tendon is considered as an axial member built into the Iso-Parametric element such that the displacements are consistent with those of the concrete element. This model implies perfect bond between the concrete material and the steel, hence it will be necessary to make some adjustments for the inclusion of the bond effect in this model. The reinforced or post-tensioned concrete was modeled as an 8 node Iso-Parametric Element with a steel truss element embedded. For the truss element the stiffness matrix is given by:

\[
\begin{bmatrix}
P_1 \\ P_2
\end{bmatrix} = \frac{A*E}{L} \begin{bmatrix}
1 & -1 \\
-1 & 1
\end{bmatrix} \begin{bmatrix}
d_1 \\ d_2
\end{bmatrix}
\]  

(Eq.3.26)

Or:

\[
\{P\} = [K_{LO}] \bullet \{d\}
\]  

(Eq.3.27)
In the Global Axis the steel element is represented by:

\[
\begin{bmatrix}
P_{1x} \\
P_{1y} \\
P_{2x} \\
P_{2y}
\end{bmatrix} = \begin{bmatrix} T_1 \end{bmatrix}^T \cdot [K_{LO}] \cdot \begin{bmatrix} T_1 \end{bmatrix} \cdot 
\begin{bmatrix}
d_{1x} \\
d_{1y} \\
d_{2x} \\
d_{2y}
\end{bmatrix}
\]  
(Eq.3.28)

Where:

\[
[T_1] = \begin{bmatrix} \cos \nu & \sin \nu & 0 & 0 \\
0 & 0 & \cos \nu & \sin \nu \end{bmatrix}
\]  
(Eq.3.29)

By the above transformation, the end points of the steel bar do not necessarily coincide with the parent (concrete) nodes; hence the global stiffness matrix of the steel element will need to undergo another transformation before we can assemble the steel stiffness into the concrete element stiffness matrix. The global steel stiffness matrix will have the form:

\[
[K_{gr}] = [T_2]^T \cdot [T_1]^T \cdot [K_{LO}] \cdot [T_1] \cdot [T_2]
\]  
(Eq.3.30)

In Fig.3.8, the embedding of a steel element into a particular concrete parent element is presented.
The eight node concrete element will be analyzed with the following mechanism: Taking one of its four sides, i.e. side $4$; and representing this concrete element side as a 3 node $C_0$ element with two degrees of freedom at each node; the shape function of this side of the concrete parent element will be:

$$U = a_0 + (a_1 \xi) + (a_2 \xi^2)$$  \hspace{1cm} (Eq.3.31)

$$V = a_0 + (a_1 \xi) + (a_2 \xi^2)$$  \hspace{1cm} (Eq.3.32)
Figure 3.9 Steel Bar Element Represented in Natural Coordinates

\[
\begin{bmatrix}
U_1 \\
U_2 \\
U_3
\end{bmatrix} = \begin{bmatrix}
1 & -1 & 1 \\
1 & 0 & 0 \\
1 & 1 & 1
\end{bmatrix} \begin{bmatrix}
a_0 \\
a_1 \\
a_2
\end{bmatrix}
\]

(Eq.3.33)

\[
\begin{bmatrix}
a_0 \\
a_1 \\
a_2
\end{bmatrix} = \begin{bmatrix}
0 & 1 & 0 \\
-1/2 & 0 & 1/2 \\
1/2 & -1 & 1/2
\end{bmatrix} \begin{bmatrix}
U_1 \\
U_2 \\
U_3
\end{bmatrix}
\]

(Eq.3.34)

Substituting Eq. 3.34 into Eq. 3.31:

\[
U = \begin{bmatrix} 1 & \xi & \xi^2 \end{bmatrix} \cdot \begin{bmatrix}
0 & 1 & 0 \\
-1/2 & 0 & 1/2 \\
1/2 & -1 & 1/2
\end{bmatrix} \begin{bmatrix}
U_1 \\
U_2 \\
U_3
\end{bmatrix}
\]

(Eq.3.35)

\[
U = \begin{bmatrix}
-\frac{\xi}{2} + \frac{\xi^2}{2} & 1 - \xi^2 & \frac{\xi}{2} + \frac{\xi^2}{2}
\end{bmatrix} \cdot \begin{bmatrix}
U_1 \\
U_2 \\
U_3
\end{bmatrix}
\]

(Eq.3.36)

Or
\[ U = [N] \cdot \begin{bmatrix} U_1 \\ U_2 \\ U_3 \end{bmatrix} \]  
(Eq.3.37)

Where \([N]\): Shape Function Matrix of Side 4

\[ N_1 = \frac{\xi}{2} \cdot (\xi - 1) ; N_2 = (1 + \xi) \cdot (1 - \xi) ; N_3 = \frac{\xi}{2} \cdot (1 + \xi) \]

It is now necessary to locate the position of the steel nodes relative to the concrete element boundaries. Once again we analyze side 4.

Figure 3.10 Steel Element Intersecting the Left Side of a Concrete Element (ref. [4])
\[ N_i = \frac{\xi}{2} \cdot (\xi - 1) \quad (Eq.3.38) \]

\[ N_j = (1 + \xi) \cdot (1 - \xi) \quad (Eq.3.39) \]

\[ N_k = \frac{\xi}{2} \cdot (1 + \xi) \quad (Eq.3.40) \]

*The Node J is the Origin of the Natural Coordinate System*

Figure 3.11 Representation in Natural Coordinates of the Parent Element’s Side being Intersected by the Steel Bar Element
Where

\[ X = \left[ \frac{\left( \frac{li}{2} \right) - ci}{li/2} \right] = \left[ 1 - \frac{2*ci}{li} \right] = \frac{2ci}{li} - 1 \]  
(Eq.3.41)

After obtaining the distance \( X \), this value is then substituted into the shape functions of the concrete element side being intersected.

i.e. \( N_i = \frac{1}{2} \left( \frac{2ci}{li} - 1 \right) \left( \frac{2ci}{li} - 2 \right) \)  
(Eq.3.42)

and setting \( \frac{ci}{li} = ri \)

\[ N_i = \frac{1}{2} (2ri - 1)(2ri - 2) = (2ri - 1)(ri - 1) \]  
(Eq.3.43)

\[ N_j = \left( 1 - \left( \frac{2ci}{li} - 1 \right) \right) \left( 1 + \frac{2ci}{li} - 1 \right) = 2ri(2 - 2ri) = 4ri(1 - ri) \]  
(Eq.3.44)

\[ N_k = \frac{1}{2} \left( \frac{2ci}{li} - 1 \right) \left( \frac{2ci}{li} - 1 + 1 \right) = \frac{1}{2} (2ri - 1)(2ri) = ri(2ri - 1) \]  
(Eq.3.45)

Since the concrete elements have four faces and the steel intersects the concrete element on faces 2 & 4 for this demonstration, we proceed to assemble the global matrix of the steel contribution on the concrete element.
\[
[T_2] = \begin{bmatrix}
A_1 & 0 & 0 & 0 & 0 & A_3 & A_2 \\
0 & 0 & B_1 & B_2 & B_3 & 0 & 0
\end{bmatrix} \quad \text{(Eq.3.46)}
\]

\(A_1\) is the Sub-Matrix on node 1, \(A_2\) on node 8 and \(A_3\) on node 7.

\(B_1\) is the Sub-Matrix on node 3, \(B_2\) on node 4 and \(B_3\) on node 5.

Where:

\[
A_1 = \begin{bmatrix}
2p^2 - 3p + 1 & 0 \\
0 & 2p^2 - 3p + 1
\end{bmatrix} \quad B_1 = \begin{bmatrix}
2q^2 - 3q + 1 & 0 \\
0 & 2q^2 - 3q + 1
\end{bmatrix}
\]

\[
A_2 = \begin{bmatrix}
-4p^2 + 4p & 0 \\
0 & -4p^2 + 4p
\end{bmatrix} \quad B_2 = \begin{bmatrix}
-4q^2 + 4q & 0 \\
0 & -4q^2 + 4q
\end{bmatrix}
\]

\[
A_3 = \begin{bmatrix}
2p^2 - 2p & 0 \\
0 & 2p^2 - 2p
\end{bmatrix} \quad B_3 = \begin{bmatrix}
2q^2 - 2q & 0 \\
0 & 2q^2 - 2q
\end{bmatrix}
\]

\(p = \frac{c_1}{l_1}, \quad q = \frac{c_2}{l_2}\) and 0 is a 2X2 null matrix.
3.3 Bond-Slip Model

The effect due to bond-slip will be included using a link element. Using the Classical bond link element developed by Ngo and Scordelis [8] shown in Fig.3.12.

![Bond-link Element](image)

Figure 3.12 Bond-link Element (ref. [4])

Using the element presented in Fig.3.12, the shear and normal force along the interface will be represented by the relation (ref. [21]):

\[
\begin{bmatrix}
F_r \\
F_s
\end{bmatrix} =
\begin{bmatrix}
K_r & 0 \\
0 & K_s
\end{bmatrix} \begin{bmatrix}
dr \\
ds
\end{bmatrix}
\]

(3.47)

Where:
$K_r$: Represents the dowel action (it can be assigned a low value or just ignored, on this research we didn’t took into account the dowel effect on the steel bar).

$K_y$: Shear stiffness of the interface

The program written in MATLAB use the Analytical Bond stress-slip relationship for monotonic loading based on the CEB-FIP Model Code 90 (Fig.3.13) for the tendon-grout interface.

Figure 3.13 Bond-slip Relation Between the Grout and the Steel Rebar (or Tendon)

Between 0 and S1 the value of shear or cohesion is controlled by:

$$\tau = \tau_{\text{max}} \left(\frac{S}{S1}\right)^\psi$$  \hspace{2cm}  (Eq.3.48)
The Model Code 90 (ref. [23]) states that for defining the bond stress-slip relationship of smooth bars the analyst can refer to Table 3.2.

Table 3.2 Slip Values and Maximum Shear $\tau_{\max}$ between Smooth Bars and Grout

According to the Model Code 90 (ref. [23])

<table>
<thead>
<tr>
<th></th>
<th>Cold Drawn Wire</th>
<th>Hot Roller Bars</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Good Bond</td>
<td>All Other Bond</td>
</tr>
<tr>
<td>Conditions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>S1=S2=S3</td>
<td>0.01 mm</td>
<td>0.01 mm</td>
</tr>
<tr>
<td>A</td>
<td>0.5</td>
<td>0.5</td>
</tr>
</tbody>
</table>

$\tau_{\max} = \tau_f = 0.1\sqrt{f_{ck}} \quad 0.05\sqrt{f_{ck}} \quad 0.3\sqrt{f_{ck}} \quad 0.15\sqrt{f_{ck}}$

Where $f_{ck}$ ksi (MPa), the characteristic compressive strength, is defined as that strength below which 5% of all possible strength measurements for the specified concrete may be expected to fall.

The value of $E_b$ (Bond Modulus at the interface of the tendon and the grout) between 0 and S1 is going to be equal to the (slope) value of the shear at the interface ($\tau$) divided by the relative slip (S), and for slip values greater than S1 $E_b=0$ because the shear stress keeps constant.

The Bonding Stiffness will be equal to:

$$K_b = E_b * A$$

(Eq.3.49)
Where $A$, the bar circumferential tributary area to the bond link element is:

$$A = \frac{m \cdot \pi \cdot d_b \cdot l}{2 \cdot b}$$  \hspace{1cm} (Eq.3.50)

$m$=number of steel bars at the same level of the cross-section

$l$=spacing of bond links

$b$=width of the member cross-section

To include this bonding stiffness into the steel element, it will be needed to make some adjustments. The model that will be used to represent the stiffness of the bonding between the steel and the concrete will have the following shape:

Figure 3.14 Bond Stiffness Incorporated into the Stiffness of the Steel Bar Element
Embedded (ref. [4])
The local Stiffness Matrix of the steel bar with bonding included, using the link elements will be:

\[
\begin{bmatrix}
P_1 \\
P_2 \\
P_3 \\
P_4
\end{bmatrix} =
\begin{bmatrix}
K_{bi} & -K_{bi} & 0 & 0 \\
-K_{bi} & K_s + K_{bi} & 0 & -K_{bj} \\
0 & 0 & K_{bj} & -K_s \\
0 & -K_{bj} & -K_s & K_s + K_{bj}
\end{bmatrix}
\begin{bmatrix}
d_1 \\
d_2 \\
d_3 \\
d_4
\end{bmatrix}
\]  
(Eq.3.51)

Where:

\[
K_s = \frac{E_s A_{steel}}{L} \quad \text{(Eq.3.52)}
\]

\[
K_{bi} = K_{bj} = E_b \cdot A \quad \text{(Eq.3.53)}
\]

Using Static condensation, the steel element degrees of freedom will be in function of the concrete parent element degrees of freedom.

Rearranging Eq.3.51

\[
\begin{bmatrix}
P_c \\
P_s
\end{bmatrix} =
\begin{bmatrix}
K_{cc} & K_{cs} \\
K_{cs} & K_{ss}
\end{bmatrix}
\begin{bmatrix}
d_c \\
d_s
\end{bmatrix}
\]  
(Eq.3.54)

Or
\[ P_c = K_{cc} d_c + K_{cs} d_s \]  \hspace{1cm} (Eq.3.55)

and

\[ P_s = K_{cs} d_c + K_{ss} d_s \]  \hspace{1cm} (Eq.3.56)

Substituting for \( d_s \) in Eq. 3.56:

\[ d_s = [P_s - K_{cs} d_c] \bullet [K_{ss}]^{-1} \]  \hspace{1cm} (Eq.3.57)

Now Substituting Eq. 3.57 into Eq.3.55

\[ P_c = K_{cc} d_c + K_{cs} [P_s - K_{cs} d_c] [K_{ss}]^{-1} \]  \hspace{1cm} (Eq.3.58)

Arranging again we get

\[ P_c - K_{cs} K_{ss}^{-1} P_s = [K_{cc} - K_{cs}^T K_{ss}^{-1} K_{cs}] [d_c] \]  \hspace{1cm} (Eq.3.59)

That will be represented as

\[ \{P_c^*\} = [K_{cc}^*] \bullet \{d_c\} \]  \hspace{1cm} (Eq.3.60)
Where:

$$K'_{ec} = \frac{K_s}{1 + K_s \left( \frac{1}{K_{bi}} + \frac{1}{K_{bj}} \right)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = K_{eq}$$

Matrix $[K_{eq}]$ represents the local stiffness of the steel element, including the bonding effect. It will be necessary to convert this matrix into the parent element global coordinate.

$$[K_{Gl,k}] = [T_2]^T \cdot [T_1]^T \cdot [K_{eq}] \cdot [T_1] \cdot [T_2]$$  \hspace{2cm} (Eq.3.61)

Where $[T_1]$ and $[T_2]$ were previously defined (Eq.3.29 and Eq.3.46 respectively)

Now the steel stiffness matrix can be assembled with the concrete parent element stiffness Matrix $[K]$ (Eq.3.22).

$$[K]_{de} = [K]_k + \sum_{i=1}^{n} ([K_{Gl,k}]_k)$$  \hspace{2cm} (Eq.3.62)

After the element stiffness matrix of the composite concrete parent element with the steel element embedded is obtained, the global stiffness matrix $[K]$ of the post-tensioned beam is assembled. The stiffness of the post-tensioning strands is added into
the concrete only after the complete transfer of post-tensioning forces (complete transfer of post-tensioning forces includes all the losses due to friction, elastic shortening and wedge-pull-in) has occurred between the steel and the concrete.

After the global stiffness matrix \([K]\) of the post-tensioned beam is assembled, the global force vector \(\{R\}\) containing all static imposed loads on the beam is created (including the forces that result from the post-tensioning of the steel tendons).

For the static loads imposed, if this load is due to self weight of the concrete it will be equal to:

\[
\{R\}_{\text{self weight}} = - \sum_{\text{ele}} \int [N]^T \cdot \left\{ \frac{0}{w} \right\} dV = - \sum_{\text{ele}} t \int [N]^T \cdot \left\{ \frac{0}{w} \right\} dA \quad \text{(Eq.3.63)}
\]

Where:

\( t \) : thickness of the concrete element

\([N]^T\) : shape function of the concrete parent element

\( w \) : self weight of the concrete, usually around 150 lb/ft\(^3\)

If there is a distributed load; i.e. on top of a concrete element, the distribution of the load on the concrete element nodes will be equal to:

\[
\{R\}_{\text{distribute}} = \left[ \begin{array}{ccccccccc} 0 & 0 & 0 & N_k & 0 & N_i & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & N_k & 0 & N_i & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & N_j & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & N_j & 0 & 0 \end{array} \right] \begin{bmatrix} 0 \\
0 \\
0 \\
0 \end{bmatrix} dx \quad \text{(Eq.3.64)}
\]
Or in natural coordinates:

\[
\{R\}_{\text{load}}^{\text{distributed}} = \begin{bmatrix} 0 & 0 & 0 & N_k & 0 & N_i & 0 & 0 & 0 & 0 & N_j & 0 & 0 & 0 & 0 \end{bmatrix}^T \begin{bmatrix} 0 \end{bmatrix} J d\bar{\xi} \tag{Eq. 3.65}
\]

Where \( N_i = \frac{1}{2} \xi (\xi - 1); N_j = (1 - \xi)(1 + \xi); N_k = \frac{1}{2} \xi (\xi + 1) \) and \([J]\): Jacobian.

Figure 3.15 Distributed Load on a Concrete Parent Element

Then the force vector \( \{R\}_{\text{load}}^{\text{distributed}} \) in the global axis of reference of the concrete element is assembled into the global force vector \( \{R\} \).

With respect to the forces on the beam due to the post-tensioning of the tendon, each of the intersecting points between the concrete parent element and the steel tendon
elements is being shared by two steel elements at the same time (except of course the intersecting points at both tendon ends).

To obtain the resultant force on a concrete element due to the forces of a steel element embedded within it, it is necessary to multiply the local forces of the steel element by two transformation matrices to transfer the forces from the steel local axis into the concrete element global axis.

Figure 3.16 Transfer of Forces from Two Steel Bar Elements (Sharing an Intersecting Point) into the Concrete Parent Elements

\[
\begin{align*}
\begin{bmatrix}
F_{b_H} \\
F_{b_V} \\
F_{b_{2H}} \\
F_{b_{2v}}
\end{bmatrix} &=
\begin{bmatrix}
\cos \theta_b & 0 \\
\sin \theta_b & 0 \\
0 & \cos \theta_b \\
0 & \sin \theta_b
\end{bmatrix}
\begin{bmatrix}
F_{b_1} \\
F_{b_2}
\end{bmatrix} = [T]^{T}
\begin{bmatrix}
F_{b_1} \\
F_{b_2}
\end{bmatrix}
\end{align*}
\]

(Eq.3.66)
These are the forces of the steel tendon with respect to the global axis, but still we need another transformation to input these forces into the concrete parent element.

\[
\begin{bmatrix}
P_{ah} \\
P_{av} \\
P_{bh} \\
P_{bv} \\
P_{ch} \\
P_{cv} \\
P_{dh} \\
P_{dv} \\
P_{eh} \\
P_{ev} \\
P_{fh} \\
P_{fv} \\
P_{gh} \\
P_{gv} \\
P_{hh} \\
P_{hv}
\end{bmatrix} = \begin{bmatrix} T2 \end{bmatrix}^T \cdot \begin{bmatrix} F_{b\ h} \\
F_{b\ v} \\
F_{b\ 2h} \\
F_{b\ 2v} \end{bmatrix} = \begin{bmatrix} T2 \end{bmatrix}^T \cdot \begin{bmatrix} T1 \end{bmatrix}^T \cdot \begin{bmatrix} F_{b\ h} \\
F_{b\ v} \end{bmatrix}
\]  
(Eq.3.67)

Where \([T2]\) was previously defined in Eq.3.46

Eq. 3.67 contains the vector with the equivalent global forces on the concrete parent element obtained from the original local forces of the steel element. These forces in the global axis are then assembled into the global force vector \(\{R\}\).

Now with the value of the global force vector \(\{R\}\) and Global stiffness matrix \([K]\) known the displacement \(\{d\}\) in the global axis of reference of all the concrete nodes of
the post-tensioned beam can be obtained. This is done with the function \textit{Solveq} from CALFEM.

\[
[K]^{*} \{d\} = \{R\} \quad \text{(Eq.3.68)}
\]

From the concrete element nodes displacements the displacement of the steel element nodes at each intersecting point can be obtained. Without any adjustments, the displacements of the steel element nodes obtained from Equation 3.69 would assume perfect bonding between both materials. By extracting the node displacements \(\{\Delta d\}_{ste}\) of a particular concrete parent element (with a steel element embedded within) from the global beam displacement vector \(\{d\}\), we can obtain the displacement increase (or decrease) \(\{\Delta d_s\}\) of the steel element embedded inside the concrete parent element.

\[
\{\Delta d_s\} = [T_1] \cdot [T_2] \cdot \{\Delta d\}_{ste} \quad \text{(Eq.3.69)}
\]

Again, this change of displacement \(\{\Delta d_s\}\) of the steel element by using Eq.3.69 will assume perfect bonding between the steel element and the concrete parent element. It will be necessary to make some adjustments to include the bonding effect at the interface between the steel element and the concrete parent element. First the displacement of the concrete parent element parallel to the local axis of the steel element embedded is extracted for each and every concrete parent element with a steel element embedded.
\[ \{\Delta d_c\} = [T_1] \cdot [T_2] \cdot \{\Delta d\}_{ele} \]  

(Eq. 3.70)

Where \(\{\Delta d\}_{ele}\) is the change of displacement of the concrete parent element.

Figure 3.17- Steel Bar Elements Embedded into Concrete Parent Elements

Using Figure 3.17 as an example, Eq. 3.70 is needed to obtain the displacement increment of each of the five concrete elements with steel elements embedded within.

Assuming a perfect continuity on each intersecting point shared by two steel bar elements, a single global displacement vector for the concrete all along the interface between the steel tendon and the concrete material will be assembled. This global concrete vector \(\{\Delta d_c\}\) contains the displacement of the concrete with respect to the local...
axis of the steel elements at every intersecting point along the entire interface between both materials.

Using Eq. 3.56:

$$\Delta P_s = K_{cs} \{\Delta d_c\} + K_{ss} \{\Delta d_s\}$$

Eq. 3.56 will be in function of $\Delta d_s$; this single global vector will contain the displacement (about their respective steel elements local axis) of all the steel elements nodes along the entire tendon or steel reinforcement interface.

$$\{\Delta d_s\} = [K_{ss}]^{-1}(\{\Delta P_s\} - [K_{cs}][\Delta d_c])$$

(Eq.3.71)

Where:

$$[K_{cs}] = \begin{bmatrix} -K_{bi} & 0 \\ 0 & -K_{bj} \end{bmatrix} ; \quad [K_{ss}] = \begin{bmatrix} -K_{s} & -K_{s} \\ -K_{s} & -K_{s} \end{bmatrix} ; \quad \{\Delta P_s\} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

The reason vector $\Delta P_s$ (applied loads into the steel elements nodes) is equal to zero is because the pre-stress force at the intersecting points is applied directly into the concrete parent elements.

After obtaining the steel displacement at every intersecting point, the user will be able to know the amount of elongation (or contraction) of the displacement on both nodes
at every steel element. From this change of displacement on the steel elements nodes, the increase (or decrease) of the actual force on each steel element will be obtained.
Some of the files used on this research were obtained from CALFEM (ref. [20]), the rest of the files were created using MATLAB. We will give first a brief description of the files from CALFEM employed in the analysis, and then we will present a flowchart of the entire program written in MATLAB with a brief explanation of the new files created.

4.1 CALFEM files

The software CALFEM [20] (Computer Aided Learning of the Finite Element Method) was developed at the division of Structural Mechanics in Lund University, Sweden. The files from CALFEM used in the routine were:

4.1.1 Assem

Purpose: Assemble element matrices

Syntax:

\[ K = \text{assem} \left( \text{edof}, K, Ke \right) \] or \[ \begin{bmatrix} K \end{bmatrix}, f = \text{assem} \left( \text{edof}, K, Ke, f, fe \right) \]
This file adds the stiffness matrix $Ke$ of a single parent concrete element into the beam global stiffness matrix $K$, according to the topology matrix $edof$. The element topology matrix $edof$ is defined as:

$$edof = [el \; dof1 \; dof2 \; ... \; dof_{ned}]$$

Where $ned$ represents the amount of degrees of freedom on a single element. The first column contains the element number, and columns 2 to ($ned + 1$) contain the corresponding global degrees of freedom of a particular element.

In the case where the matrix $Ke$ is identical for several elements, assembling of these can be carried out simultaneously. Each row in $edof$ then represents one element, i.e. $nel$ is the total number of considered elements.

$$edof = \begin{bmatrix}
  el_1 & dof_1 & dof_2 & \ldots & dof_{ned} \\
  el_2 & dof_1 & dof_2 & \ldots & dof_{ned} \\
  \vdots & \vdots & \vdots & \ddots & \vdots \\
  el_{nel} & dof_1 & dof_2 & \ldots & dof_{ned}
\end{bmatrix}$$

This matrix has one row for each element. If $f_e$ and $f$ are also given in the function, then the element load vector $f_e$ (i.e. self weight of a single element) is added to the global force vector $f$ of the entire beam.
4.1.2 Extract

Purpose: To extract element nodal quantities (i.e. displacements) from a global solution vector $a$.

Syntax:

$$ed = \text{extract}(edof, a)$$

This function extracts element displacements $a^e$ from the global solution vector $a$. Input variables are the topology matrix $edof$ defined in section 4.1.1, and the global solution vector $a$. The output variable:

$$ed = (a^e)^T$$

This vector contains the element displacement vector. If $edof$ contains more than one element, $ed$ will be a matrix equal to:

$$ed = \begin{bmatrix}
(a^e)^T_1 \\
(a^e)^T_2 \\
\vdots \\
(a^e)^T_{nel}
\end{bmatrix}$$

Where row $i$ gives the element displacements for the element defined in row $i$ of $edof$, and $nel$ is the total number of considered elements. For a two-dimensional iso-
parametric 8 node element, the extract function will extract 16 nodal displacements (each one for each degree of freedom of the concrete element) for every element given in edof, and create a matrix ed of size (nel x 16).

\[
ed = \begin{bmatrix}
v_1 & v_2 & v_3 & v_4 & v_5 & v_6 & v_7 & v_8 & v_9 & v_{10} & v_{11} & v_{12} & v_{13} & v_{14} & v_{15} & v_{16} \\
v_1 & v_2 & v_3 & v_4 & v_5 & v_6 & v_7 & v_8 & v_9 & v_{10} & v_{11} & v_{12} & v_{13} & v_{14} & v_{15} & v_{16} \\
\ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots & \ldots \\
v_1 & v_2 & v_3 & v_4 & v_5 & v_6 & v_7 & v_8 & v_9 & v_{10} & v_{11} & v_{12} & v_{13} & v_{14} & v_{15} & v_{16}
\end{bmatrix}
\]

4.1.3 Coordxtr

Purpose: Extract the element coordinates from a global coordinate matrix

Figure 4.1 Nodes Numbering on the Concrete Parent Elements
Where i.e.

\[ Ex = \begin{bmatrix} x_1 & x_3 & x_{17} & x_{15} & x_2 & x_{11} & x_{16} & x_{10} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \end{bmatrix} \]

Syntax:

\[ [Ex, Ey] = \text{coordxtr}(\text{Edof}, \text{Coord}, \text{Dof}, \text{nen}) \]

This function extracts the element coordinates from the global coordinate matrix \( \text{Coord} \), for elements with equal number of nodes and degrees of freedom. Input variables are the element topology matrix \( \text{Edof} \), the global coordinate matrix \( \text{Coord} \), the global topology matrix \( \text{DOF} \), and the number of element nodes \( \text{nen} \) on each element.

\[
\text{Coord} = \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ \vdots & \vdots \\ x_n & y_n \end{bmatrix}
\]

\[
\text{Dof} = \begin{bmatrix} k_1 & l_1 & \ldots & m_1 \\ k_2 & l_2 & \ldots & m_2 \\ k_3 & l_3 & \ldots & m_3 \\ \vdots & \vdots & \ddots & \vdots \\ k_n & l_n & \ldots & m_n \end{bmatrix}
\]

\[
\text{nen} = [\text{nen}]\]

The nodal coordinates in row \( i \) of \( \text{Coord} \) correspond to the degrees of freedom of row \( i \) in \( \text{Dof} \). Components \( k_i, l_i \) and \( m_i \) define the degrees of freedom of node \( i \), and \( n \) is the total number of nodes on the structure.
The output variables $Ex$ and $Ey$ are matrices defined i.e. by:

$$Ex = \begin{bmatrix}
    x_1^1 & x_2^1 & x_3^1 & \cdots & x_{nel}^1 \\
    x_1^2 & x_2^2 & x_3^2 & \cdots & x_{nel}^2 \\
    \vdots & \vdots & \vdots & \ddots & \vdots \\
    x_1^{nel} & x_2^{nel} & x_3^{nel} & \cdots & x_{nel}^{nel}
\end{bmatrix}$$

Where row $i$ gives the x-coordinates of the element defined in row $i$ of $Edof$, and where $nel$ is the total number of considered elements.

### 4.1.4 Eldraw2

**Purpose:** Draw the undeformed mesh for a two dimensional structure

**Syntax:**

- `eldraw2 (Ex,Ey)`
- `eldraw2 (Ex,Ey,plotpar)`
- `eldraw2 (Ex,Ey,plotpar,elnum)`

This function displays the undeformed mesh for the two dimensional beam. The input variables are the coordinate matrices $Ex$ and $Ey$ obtained from the function `coordxtr`. The variable `plotpar` sets plot parameters for `linetype`, `linecolor` and `node marker`.
Plotpar = [linetype linecolor nodemark]

Linetype = 1 solid line    Linecolor = 1 black    Nodemark = 1 circle

2 dashed line                        2 blue                        2 star
3 dotted line                        3 magenta                      0 no mark

4 red

Default is solid black lines with circles at nodes. Also the element numbers can be
displayed at the center of the concrete element if a column vector elnum with the element
numbers is supplied. This column vector can be derived from the element topology
matrix Edof:

elnum = Edof(:, 1)

As it was explained on section 4.1.1, the number of rows in matrix Edof is equal
to the number of concrete parent elements, each row contains in the first column the
number assigned to that specific concrete parent element, the rest of the columns contain
the number assigned to each of the concrete parent element degrees of freedom.

4.1.5 Hooke

Purpose: Create the material matrix for a linear elastic and isotropic material. This
function is used to create the material matrix of the concrete parent elements.
Syntax:

\[ D = \text{hooke}(p\text{type}, E, \nu) \]

This file computes the material matrix \( D \) for the concrete parent elements, which are treated as linear elastic and Isotropic. The variable \( p\text{type} \) defines the type of analysis

\( \text{ptype} = 1 \) Plane Stress

\( \text{ptype} = 2 \) Plane Strain

\( \text{ptype} = 3 \) Axisymmetry

\( \text{ptype} = 4 \) Three dimensional analysis

The material parameters \( E \) and \( \nu \) define the modulus of elasticity (\( E_c \)) and the Poisson’s ratio of the concrete material respectively.

For \( \text{ptype} = 1 \) (Plane Stress), \( D \) is equal to:

\[
D = \frac{E}{1-\nu^2} \begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & 1-\nu
\end{bmatrix}
\]

(Eq.4.1)

For \( \text{ptype}=2 \) (Plane Strain) and \( \text{ptype}=3 \) (Axisymmetry), \( D \) is obtained as:
\[
D = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix}
1 - \nu & \nu & \nu & 0 \\
\nu & 1 - \nu & \nu & 0 \\
\nu & \nu & 1 - \nu & 0 \\
0 & 0 & 0 & \frac{1}{2}(1 - 2\nu)
\end{bmatrix}
\]  
(Eq.4.2)

For the three dimensional case, ptype=4, \(D\) is equal to:

\[
D = \frac{E}{(1 + \nu)(1 - 2\nu)} \begin{bmatrix}
1 - \nu & \nu & \nu & 0 & 0 & 0 \\
\nu & 1 - \nu & \nu & 0 & 0 & 0 \\
\nu & \nu & 1 - \nu & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{2}(1 - 2\nu) & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{2}(1 - 2\nu) & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1}{2}(1 - 2\nu)
\end{bmatrix}
\]  
(Eq.4.3)

### 4.1.6 Plani8e

Purpose: Compute the stiffness matrix for an 8 node iso-parametric concrete parent element in plane strain or plane stress.

Syntax:

\([Ke, fe]=\text{plani8e} \ (ex, ey, ep, D, eq)\) Or \(Ke=\text{plani8e} \ (ex, ey, ep, D)\)
This file provides the element stiffness $Ke$ and the element load vector $fe$ (self-weight of the element) for an 8 node iso-parametric concrete parent element in plane strain or plane stress.

The element nodal coordinates are provided to the function by $ex$ and $ey$. The type of analysis $ptype$ (1 for plane stress and 2 for plane strain), the element thickness $t$ and the number of Gaussian points $n$ ($n \times n$) are supplied by $ep$.

\[
\begin{align*}
ex &= [x_1 \ x_2 \ x_3 \ \ldots \ x_8] \\
ey &= [y_1 \ y_2 \ y_3 \ \ldots \ y_8] \\
ep &= [ptype \ t \ n]
\end{align*}
\]

The material properties are provided by the constitutive matrix $D$ (section 4.1.5). If different $Di$ matrices are required in the gauss points these $Di$ matrices are stored in a global vector $D$.

\[
D = \begin{bmatrix}
D_1 \\
D_2 \\
. \\
. \\
D_{n^2}
\end{bmatrix}
\]

If the self-weight of the element per unit volume is included, the element load vector $fe$ is computed, the input vector will be:
Where \( bx \) and \( by \) are loads per unit volume (in the \( x \) and \( y \) global direction), most of time they are equal to (but not always):

\[
bx=0 \text{ and } by=150 \frac{lb}{ft^3} \left( \frac{23.6 KN}{M^3} \right)
\]

### 4.1.7 Plani8s

Purpose: Compute the stresses and strains in an 8 node iso-parametric concrete parent element in plane strain or plane stress.

Syntax:

\[
[es, et, eci]=\text{plani8s (ex, ey, ep, D, ed)}
\]

This file computes the stresses \( es \) and the strains \( et \) in an 8 node iso-parametric concrete parent element in plane stress or plane strain. The input variables are \( ex, ey, ep \) and the matrix \( D \) (see section 4.1.5). The vector \( ed \) contains the nodal displacements \( a^e \) of the element (see section 4.1.2 for the procedure about how to obtain \( a^e \)). The output variables will have the following form:
These are the stress, strain and the coordinates of the integration points. The index $n$ denotes the number of integration points used in the concrete parent element.

### 4.1.8 Plani8f

Purpose: Compute the internal element force vector in an 8 node iso-parametric concrete parent element in plane strain or plane stress.

Syntax:

$$ef = \text{plani8f}(ex, ey, ep, es)$$

This file is used to obtain the element forces $ef$ in an 8 node iso-parametric concrete parent element in plane stress or plain strain. The input variable $ex$, $ey$ and $ep$ are defined in section 4.1.6, the input variable $es$ is defined in section 4.1.7.

The output variable:

$$ef = [f_{i1} f_{i2} \ldots f_{i16}]$$
This variable contains the components of the internal force vector (the internal force on each degree of freedom of the element).

### 4.1.9 Solveq

**Purpose:** Solve equation system.

**Syntax:**

\[ a = \text{solveq}(K,f) \quad \text{Or} \quad a = \text{solveq}(K,f,bc) \quad \text{Or} \quad [a,Q] = \text{solveq}(K,f,bc) \]

Solves the equation system

\[ K \cdot a = f \quad (\text{Eq.4.4}) \]

Where \( K \) is the Global Stiffness Matrix of the concrete parent element with the steel bar element embedded into it (including the bonding effect). The global force vector is defined as \( f \). The solution of the system of equations is stored in a vector \( a \), which is created by the function. The vector \( bc \) defines the boundary conditions or supports of the beam as is explained on section 4.2.11. If \( Q \) is given in the function, the reaction forces are then computed according to:

\[ Q = (K \cdot a) - f \quad (\text{Eq.4.5}) \]
4.2 Computer Flowchart

The next files were all written in MatLab. By typing *IsoMain* at the MatLab command interface, the analysis of the post-tensioned beam initiates. This file, the principal file of the entire program, contains at the same time 12 other files. Each of these files and their function within the program are now explained:

4.2.1 IsoInputData

This file is used to ask from the user and save the Material Properties of the beam and its physical dimensions:

- \( L \) Length of the Beam
- \( \rho \) Specific Weight of the concrete been used in the beam
- \( f'c \) Concrete Strength Value
- \( \nu \) Poisson Module for the concrete material.
- \( StrainCo \) Corresponding Uni-axial Strain of concrete for the concrete strength value of \( f'c \).
- \( ptype \) Type of two dimensional analysis to be performed \((1=\text{Plane-Stress},2=\text{Plane-Strain})\).
- \( Gpoints \) Amount of Gaussian points that are going to be used in the Finite element analysis \((1X1 = 1 \text{ Gaussian Point}, 2X2 = 4 \text{ Gaussian Points}, 3X3 = 9 \text{ Gaussian Points})\).
**Nvd**  
Amount of vertical divisions that the height of the beam will be sub-divided into.

**Pp**  
Beam’s Cross Section shape (1 = Rectangular, 2 = Introduce manually each vertical division thickness, 3 = Inverted “T”).

**Kb**  
Cohesion value at the duct-tendon interface.

At the end of this file two more files are attached:

### 4.2.1.1 IsoCrossSection

This file creates the matrix $nvdm$ containing the height and thickness of all the concrete parent elements contained on each vertical division (each row of elements along the beam length). For practical reasons and to keep the results the most accurate as possible, on a particular vertical division (single row of elements along the beam length) all the concrete elements have the same height and thickness dimensions with the length-height ratio of each concrete element equal or almost equal to 1. Because the thickness of two separate vertical divisions can be different, this will be useful, specially when dealing with a beam that has a non regular cross-section.

The first column of the $nvdm$ matrix represents the number assigned to that particular vertical division (the order of the vertical division goes from the bottom towards the top of the beam), the second column contains the height and the third column the thickness of the concrete elements on that vertical division (single row of elements along the beam).
Syntax:

\[ nvdm = size(n, 3) \]

Where: \( n \) is the Number of vertical divisions the beam’s depth is subdivided.

### 4.2.1.2 IsoPrestressType

Used to obtain from the user the eccentricity values at both ends and mid-length of the beam, this eccentricities are stored in the matrix \( NtMatrix \), this matrix contains 4 columns, the first column contains the number assigned to a particular tendon, the second column, third column and fourth column contain respectively the eccentricities at left end, mid-span and right end of the beam for that specific tendon in inches, the eccentricity will be negative if the tendon is above the neutral axis of the cross section, the number of rows for this matrix will depend on the number of tendons at different eccentricities along the beam.

Syntax:

\[ NtMatrix=size(n, 4) \]

Where: \( n \) represents the number of tendons with different eccentricities along the beam.
About the Yield Tensile Strength, the program will ask for one of the following: The ultimate tensile stress (fpu) or the yield tensile Stress (fpy) of the tendon, both in psi units.

According to the Tensioning Technique, the program will ask the user about the type of technique chosen by the designer: (1) Pre-Stressed or (2) Post-Tensioned. In the case the user chose Pre-Stressed approach (1), it will ask the tendon diameter in inches to obtain the tendons cross-sectional area, also it will ask the steel stress at nominal transfer length (fps) and the steel stress at the effective transfer length (fs) both in psi, after this a sub-routine contained in the file IsoPrestressCase will obtain the transfer length and the development length for the pre-stressed tendon, this based on the PCI Manual.

On the other hand if the user selected the Post-Tensioned option (2), a menu with the different types of ducts available will appear:

a) Flexible metal sheathing
b) Rigid metal duct
c) Unbonded pregreased tendon
d) Unbonded Mastic-Coated tendon

The curvature coefficient and the wobbling coefficient will be selected according to the type of duct used, to obtain the Post-Tension loss due to friction between both ends of the beam.
4.2.2 IsoDof

This file assigns a number for each of the two degrees of freedom of every node contained in a concrete parent element; each concrete parent element has 8 nodes (16 degrees of freedom in total).

The number assigned to each degree of freedom is stored in the matrix $Dof$ which has 2 columns, each row representing a single node, the first column contains the number assigned to the degree of freedom in the global X axis and the second column the number assigned to the degree of freedom in the global Y axis of that specific node.

Syntax:

$$Dof = \text{size}(n, 2)$$

Where: $n$ represents the total number of nodes contained in all the concrete parent elements.

4.2.3 IsoCoord

This file determines the coordinates in the “X” and “Y” Global axis for each one of the nodes of all concrete parent elements. It creates the matrix $Coord$ which has the number of rows equal to the number of nodes, with the first column representing the “X” coordinate in the global “X” axis and the second column the “Y” coordinate in the “Y” global axis of the same particular node.
Syntax:

\[ Coord = \text{size} \left( n, 2 \right) \]

Where \( n \) represents the total number of nodes contained in all the concrete parent elements.

### 4.2.4 IsoEdof

This file creates the matrix \( Edof \), containing the number assigned to all the degrees of freedom contained on each concrete parent element. The first column contains the number assigned to a specific concrete element, from column 2 to 17 we have the numbers that were assigned to each of the degrees of freedom contained in that particular concrete element.

Syntax:

\[ Edof = \text{size} \left( n, 17 \right) \]

Where \( n \) represents the number of concrete parent elements present in the mesh.

### 4.2.5 IsoGraph

This file creates the graphic of the undeformed concrete elements using the CALFEM© [20] functions \( \text{Coordxtr} \) (section 4.1.3) and \( \text{Eldraw2} \) (section 4.1.4).
Syntax:

\[ [Ex, Ey] = \text{coordxtr} (Edof, Coord, Dof, nen) \]

Input variables are the element topology matrix \( Edof \), the global coordinate matrix \( Coord \), the matrix \( Dof \) and the number of element nodes in each element \( nen \) (8 nodes per element). The output variables are the matrices \( Ex \) and \( Ey \), these matrices contain the coordinates in X and Y respectively of every node contained in a concrete parent element.

\( Ex = \text{size} (n,8) \)

\( Ey = \text{size} (n,8) \)

Where: \( n \) is the total number of concrete parent elements.

Syntax:

\( \text{eldraw2}[Ex, Ey, Plotpar, elnum] \)

Input variables are the matrices \( Ex \) and \( Ey \) formed by the function \( \text{coordxtr} \). The variable \( plotpar \) sets plots parameters for linetype, linecolor and node marker. Element numbers will be displayed at the center of the element with a column vector \( elnum \).

\( elnum = Edof (:, 1) \)
Where the first column of the matrix $Edof$ (see section 4.1.1) contains the number assigned to each concrete parent element

### 4.2.6 IsoE

This file detects the tendon eccentricity at each intersection point between the concrete parent elements and the steel tendon. The vector $JJ$ and $KK$ contains the coordinates on the global “X” and “Y” axis respectively of those intersection points between the concrete parent elements and the steel tendon. The coordinates in “Y” which represent the actual eccentricity of the tendon at each intersection point are obtained using the following equations:

When $0 < x < L/2$

$$KK(n, 1) = \left[ \frac{Bh}{2} \right] - e_{\text{mid\text{-}span}} - \left( e_{\text{mid\text{-}span}} - e_{\text{left\text{-}anchor}} \right) \bullet \left[ \frac{L - x}{L/2} \right]^2 \right]$$

Eq. 43

When $L/2 < x < L$

$$KK(n, 1) = \left[ \frac{Bh}{2} \right] - e_{\text{mid\text{-}span}} - \left( e_{\text{mid\text{-}span}} - e_{\text{right\text{-}anchor}} \right) \bullet \left[ \frac{x - L}{L/2} \right]^2 \right]$$

Eq. 44
Where

\[ L \]: Total Length of the Concrete Beam

\[ e_{\text{mid-span}} \]: Tendon eccentricity at mid-span

\[ e_{\text{left-anchor}} \]: Tendon eccentricity at left end

\[ e_{\text{right-anchor}} \]: Tendon eccentricity at right end

\[ x \]: Location in the “X” axis of the intersection point, taking the beam’s left end as the origin.

\[ Bh \]: Beam’s Height

The eccentricity of the tendon will be positive if the location of the intersection point is below the beam’s cross-section neutral axis and negative if is above the beam’s cross-section neutral axis.

### 4.2.7 IsoCoordTrussNodes

This file obtains the Global coordinates in “X” and “Y” at both ends (intersecting points) of each bar element. These one dimensional bar elements will be used to represent the tendon or steel reinforcement inside the concrete parent elements. This file creates the matrix \[ CoordTrussNodes3 \] that contains two columns; the first column contains the Coordinates in the Global “X” axis and the second column the coordinates in the Global
“Y” axis of each intersecting point, where the steel tendon elements and the concrete parent elements intersect. This matrix is created from the $JJ$ and $KK$ vectors.

Syntax:

$\text{CoordTrussNodes3} = \text{size} \left( n, 2 \right)$

Where: $n$ represents the total number of intersecting points.

### 4.2.8 IsoDofTruss

This file creates the matrix $DofTruss$. This matrix has two columns, and one row for each intersecting point. The first and second columns contain the number assigned for each of the two Global degrees of freedom (in the X and Y axis respectively) present at every intersecting point.

Syntax:

$\text{DofTruss} = \text{size} \left( n, 2 \right)$

Where: $n$ is the total number of intersecting points.

### 4.2.9 IsoEdofTruss

Creates the Matrix $EdofTruss$, this matrix has five columns and one row for each steel bar element. The first column contains the number assigned to each bar element
between two intersecting points, the other 4 columns contain the numbers assigned to each of the two Global degrees of Freedom present at both nodes (intersecting points) of a particular bar element.

Syntax:

\[ EdofTruss = size \left( n, 5 \right) \]

Where: \( n \) is the total number of bar elements.

4.2.10 IsoGraph2

This file plots the undeformed bar elements inside the concrete parent elements. Once more the function \( coordxtr \) is employed to obtain the coordinates in the “X” and “Y” global axes of both nodes on each bar element.

Syntax:

\[
\begin{align*}
[Ex2, Ey2] &= coordxtr \left( EdofTruss, CoordTrussNodes3, DofTruss, nen \right)
\end{align*}
\]

Input variables are the bar element matrix \( EdofTruss \) (4.2.9), the bar element global coordinate matrix \( CoordTrussNodes3 \) (4.2.7), the bar element matrix \( DofTruss \) (4.2.8) and the number of nodes on each bar element \( nen \) (2 nodes per bar element).

The output variables are the matrices \( Ex2 \) and \( Ey2 \), these matrices contain the coordinates in X and Y respectively of both nodes contained on each bar element.
\[ Ex2 = \text{size}(n,8) \]
\[ Ey2 = \text{size}(n,8) \]

Where: \( n \) represents the total number of bar elements.

Again the function \textit{eldraw2} from the program CALFEM\textsuperscript{©} [20] is used, this time to plot the Global coordinates in the “X” and “Y” global axes of each bar element nodes.

Syntax:

\textit{eldraw2} [Ex2, Ey2, Plotpar, elnum]

Input variables are the matrices \( Ex2 \) and \( Ey2 \) created by the function \textit{coordxtr}. Again the bar elements number can be displayed at the center of the bar element providing the column vector \( elnum \).

\[ elnum = \text{EdofTruss}(:,1) \]

Where the first column of the matrix \textit{EdofTruss} contains the number assigned to each steel one dimensional bar element.
4.2.11 IsoBoundaryCond

This file is used to apply the corresponding restrictions to those degrees of freedom of a particular node of any concrete parent element where the supports of the beam will be allocated.

With this file, the matrix $bc$ is created. Consisting of two columns, the first column contains the number assigned for the degree of freedom (with prescribed displacement or restricted against displacement) of a particular node of a concrete parent element where the support is going to be located, the second column contains the prescribed displacement of those degrees of freedom, in this case if their displacement are restricted their value will be zero.

Syntax:

$$bc = \text{size} (n,2)$$

Where: $n$ is the total number of degrees of freedom with prescribed or restricted displacement.

4.2.12 IsoInitialForce2

The objective of this file is to:

a) Obtain the Initial Stress distribution along each steel tendon (in the case of post-tensioned beams) including the loss of pre-stress due to wobbling and friction.
b) Transfer the initial pre-stress from the steel tendons into the concrete.

c) Apply any static load on the concrete beam (concentrated or distributed).

d) Obtain the displacement of every node of the concrete parent elements due to the tendon’s pre-stress or any applied static load, and from it obtain the relative displacement of the steel element nodes taking into account the bonding interface between the steel elements and the concrete parent elements.

e) From the displacement of the steel element nodes, the increase (or decrease) of the tendons pre-stress force is obtained.

f) Assemble the stiffness of the steel tendons or rebars into the concrete parent elements after the transfer of forces has occurred.

4.2.12.1 Initial Stress Distribution along each Steel Tendon (Including the Loss of Pre-stress due to Wobbling and Friction)

In the case a post-tensioned beam is being analyzed the program asks the user the initial post-tension force applied at one side of the beam while the other side stays fixed. From the end of the beam where the jacking force is applied (live anchor) to the other end there will be a pre-stress loss due to wobbling and friction.

4.2.12.1.1 Curvature Frictional Loss:

\[ dp = \mu \cdot P \cdot d\alpha \]  

(Eq.4.6)
4.2.12.1.2 Wobble Loss:

\[ dp = k*P*dx \]  \hspace{1cm} (Eq.4.7)

Adding both losses:

\[ dp_{total} = \mu*P*d\alpha + k*P*dx \]  \hspace{1cm} (Eq.4.8)

\[ \int_{Pa}^{Pb} \frac{dp}{P} = \mu \int_{0}^{\alpha} d\alpha + k \int_{0}^{x} dx \]  \hspace{1cm} (Eq.4.9)

Equals to:

\[ Pb = Pa* e^{(\mu x + k\alpha)} \]  \hspace{1cm} (Eq.4.10)

Where:

\( Pa = \text{Tendon Force at location A} \)

\( Pb = \text{Tendon Force at location B} \)

\( \mu = \text{Friction Coefficient} \)

\( \alpha = \text{Total intended cumulative angle change between A and B (in radians)} \)

\( x = \text{Tendon length between A and B} \)
\[ \theta_1 = \frac{2 \cdot (e_0 - e_1)}{L/2} \]  \hspace{1cm} (Eq.4.11)

\[ R1 = \frac{\left\{ \frac{L}{2} \right\}^2}{2 \cdot (e_0 - e_1)} \]  \hspace{1cm} (Eq.4.12)

\[ \theta_2 = \frac{2 \cdot (e_0 - e_2)}{L/2} \]  \hspace{1cm} (Eq.4.13)

\[ R2 = \frac{\left\{ \frac{L}{2} \right\}^2}{2 \cdot (e_0 - e_2)} \]  \hspace{1cm} (Eq.4.14)

Where

\[ e_0 = \text{eccentricity at mid-span} \]

\[ e_1 = \text{eccentricity at left end} \]

\[ e_2 = \text{eccentricity at right end} \]

Then:

\[ x = (\theta_1 \times R1) + (\theta_2 \times R2) \]  \hspace{1cm} Eq.4.15

\[ \alpha = \theta_1 + \theta_2 \]  \hspace{1cm} Eq.4.16
When the program has the force at both ends of the beam, then by interpolation it proceeds to obtain the initial pre-stress force at every intersecting point between the steel bar elements and the concrete parent elements.

When $0 < X < L/2$

$$P_x = \left( \frac{L}{2} - X \right) \times \left( \frac{P_a - P_{\text{mid-span}}}{L/2} \right) + P_{\text{mid-span}}$$  \hspace{1cm} \text{Eq.4.17}$$

When $L/2 < X < L$

$$P_x = \left[ \frac{P_{\text{mid-span}} - P_h}{L/2} \right] \times (L - X) + P_h$$  \hspace{1cm} \text{Eq.4.18}$$

Where

$X$ = Coordinate in the Global “X” axis of the intersecting point,

with the origin located at the left end of the beam

$P_{\text{mid-span}}$ = Initial Pre-stress Force at mid-span

$L$ = Length of the concrete Beam.

$P_x$ = Initial Pre-Stress force at the intersecting point.

The forces obtained by interpolation are the initial steel bar element forces just at transfer before any static load is applied. For every additional static load or tendon...
jacked, the resultant forces on the steel bar elements due to these static loads applied on the concrete elements have to be added or subtracted to the Initial Pre-stress forces at transfer obtained at each intersecting point by interpolation.

4.2.12.2 Transfer of the Initial Pre-stress from the Steel Tendons into the Concrete Elements.

This is done by the file *IsoComposite1*, this file first detects at which mode the steel bar element intersects the concrete parent element. There are 20 different ways the steel tendon or rebar can intersect a concrete parent element.

4.2.12.2.1 Steel Bar Element Intersects Concrete Parent Element at Sides 1 and 3

![Figure 4.2 (Case 1.1)](image)

Figure 4.2 (Case 1.1)
Figure 4.3 (Case 1.2)

Figure 4.4 (Case 1.3)
4.2.12.2.2 Steel bar Element Intersects Concrete Parent Element at Sides 1 and 2
Figure 4.7 (Case 2.2)

Figure 4.8 (Case 2.3)
4.2.12.2.3 Steel bar Element Intersects Concrete Parent Element at Sides 2 and 3
Figure 4.11 (Case 3.2)

Figure 4.12 (Case 3.3)
4.2.12.2.4 Steel bar Element Intersects Concrete Parent Element at Sides 1 and 4
Figure 4.15 (Case 4.2)

Figure 4.16 (Case 4.3)
4.2.12.2.5 Steel bar Element Intersects Concrete Parent Element at Sides 4 and 3
Figure 4.19 (Case 5.2)

Figure 4.20 (Case 5.3)
The file will run a routine for every steel element to see which parent elements it intersects, and by which case of the 20 different situations.

After the case is selected for a particular steel bar element, then the file obtains the 2 transformation matrices \([T1]\) and \([T2]\), so then by using Eq.3.69 it will proceed to calculate the distribution of the pre-stress force from the steel bar element into the concrete parent element and then this forces on the concrete parent element are assembled into the global force vector of the beam by using the CALFEM function \(assem\) (section 4.1.1). This routine is done for every single steel bar element (in the case of post-tensioned beams), it will be seen that i.e. if the tendon has constant eccentricity then the forces will cancel at the intersecting points were two steel bar element share a single intersecting point, with the exception of the post-tensioning forces at both end anchorages of the beam.
4.2.12.3 Static Loads Applied on the Beam (Due to Self Weight, Distributed Loads or Concentrated Loads)

By using the file *IsoPlani8edl* the self-weight of the beam is incorporated into the global force vector $f$, the user will be asked for the percentage of the dead load that he or she wants to include into the global force vector at the time a particular steel tendon is jacked, using Eq.3.65 the self-weight of the beam is assembled into the global force vector of the beam.

The file *IsoLoads* is used to include any point load or distributed load into the global force vector. Using Eq.3.66 any distributed load on the beam can be assembled into the global force vector $f$. For the point loads (concentrated loads) the program asks the user the number of concentrated loads, their respective value and their node location on the beam. After this the program assembles each and every one of these concentrated loads into the global force vector $f$. Loads are considered negative if they are directed towards the left or downward.

4.2.12.4 Resultant Displacement of the Concrete Parent Elements and Steel Elements Nodes

The displacement of every node of the concrete parent elements due to the tendon’s pre-stress or any applied static load is obtained, and from their value the program extracts the relative displacement of the rebar or steel tendons nodes taking into account the bonding interface between the steel elements and the concrete parent elements.
The concrete parent element nodes displacement is obtained as it is explained on
the section 4.1.9 using Eq.4.4. After the global displacement vector \( a \) (which contains the
displacement of all the concrete parent element nodes) is obtained, then the function
\( IsoComposite4 \) is used to obtain the concrete displacement at the level (parallel to the
local axis of the steel bar element) of every single steel bar element embedded. This is
done by using the transformation matrices \([T1]\) and \([T2]\) once again, by multiplying the
displacement of all nodes of a particular concrete parent element by these two matrices
we can obtain the concrete displacement parallel to the local axis of the steel element
embedded into that concrete element (Eq.3.72). After the concrete displacement parallel
to the steel bar element local axis is obtained, then the file \( ConcreteVector3 \)
condense from the concrete nodes the displacement of the steel bar elements nodes with the
inclusion of the bonding effect (using Eq.3.73).

4.2.12.5 Increase (or Decrease) of the Tendon Pre-stress Force, Obtained from the
Displacement of the Steel elements Nodes

This is done with the file \( IsoDispTendon \), after having the amount of displacement
of every steel element nodes (section 4.2.12.4), the change of forces in every single steel
bar element \( \{ \Delta f_1 \} \) is obtained by:

\[
\begin{bmatrix}
\Delta f_1 \\
\Delta f_2
\end{bmatrix} = \frac{A_{steel}E_{steel}}{L} \begin{bmatrix}
1 & -1 \\
-1 & 1
\end{bmatrix} \begin{bmatrix}
\Delta d_1 \\
\Delta d_2
\end{bmatrix}
\]  

(Eq.4.19)
Where \( \frac{A_{\text{steel}}E_{\text{steel}}}{L} \) is the axial stiffness of a single steel bar element, \( \begin{bmatrix} \Delta d1 \\ \Delta d2 \end{bmatrix} \) contains the increase in elongation (or contraction) obtained from section 4.2.12.4 of the steel element due to transfer of forces by bond from the concrete parent element.

This increase (or decrease) in the forces on the steel bar elements are updated each time a tendon is jacked or a static load is superimposed on the beam. This increase or decrease of forces are added or subtracted to the previous state of forces on the steel bar elements, just prior to the jacking or the application of the point or distributed load (or any other static load).

4.2.12.6 Assembling of the Steel Element’s Stiffness into the Concrete Elements after the Transfer of Forces.

The assembly of the steel tendon stiffness only occurs after transfer (in the case of post-tensioned beams); it is done as it was explained in Eq.3.62, Eq.3.63 and Eq.3.64. After the stiffness of the tendon is included into the concrete beam, then the program proceeds to apply the new forces on the beam due to the next tendon being jacked or any other new superimposed static load applied on the beam, the displacements of the concrete elements are again obtained and steps 4.2.12.4, 4.2.12.5 and 4.2.12.6 are repeated once more for the actual steel tendon being jacked and any other tendon previously jacked.
Assemble all concrete parent elements stiffness matrices into the total stiffness matrix of the beam.

Simply Reinforced Beam

Assemble the Rebar’s stiffness into the beam’s stiffness matrix according to it’s location along the structure.

Post-Tensioned Beam

Obtain all the existing tendon’s profile along the beam

Assemble the pre-stress force into the global force vector according to the tendon profile.

Introduce the beam’s self weight or any other applied static forces into the beam’s global force vector.

Using the existing bond along the rebar or tendon interface, the displacement of the steel elements is obtained from the displacement of the concrete parent element nodes.

From the displacement of the steel elements nodes the resultant forces that have been transmitted through bond from the concrete into the steel elements are obtained.

In the case of analyzing a Simply reinforced beam, the resultant forces along each reinforcement layer are plotted.

In the post-tensioned beam, due to the elastic shortening product of the actual tendon been jacked the state of forces along all the tendons that were previously jacked are updated and plotted. After this the stiffness of the actual tendon been jacked is incorporated into the beam’s global stiffness matrix.

Additional tendon jacked

Yes

No

End of Routine

Figure 4.22 Computer Program Flow Chart
This chapter deals with four examples solved with the program developed in this thesis. These examples were obtained from different references. The first example consists of a square plate problem without bond slip (full bonded) under an axial compressive distributed load. The second example deals with a simple supported beam with two bottom layers of steel reinforcement under the effect of a vertical concentrated load at mid-span. The third and fourth examples deal with a 2 span post-tensioned beam and a single span post-tensioned beam respectively. The software has the ability to analyze both Reinforced Concrete Beams and Post-Tensioned Concrete members. Each example is explained in the following sections.

### 5.1 Square Plate Problem with Straight Reinforcing Layers (Full Bonded Interface)

The square plate is under a uniform compression “P”. The side length of the plate is equal to “L”, $E_c = 4053.232$ ksi, $\nu=0.25$, thickness is 1.0, and the percentage of steel for each layer is $\rho = \left(\frac{A_s}{L \times 1}\right) = 0.025$. For convenience we will use numerical values for “P” and “L”. Lets assume $L=130$ inches, thickness=1 in and $P=225$ lb/in, then the cross-sectional area of each steel layer will be $A_s = 3.25\ in^2$. Both steel layers (top and bottom)
had a concrete cover equal to $0.25L = 32.5$ in. The self weight of the plate was ignored, and to account for perfect bonding between both materials, a very high value for the bond stiffness $\left(1 \times 10^{10} \text{ lb/in} \right)$ was employed.

Figure 5.1 Problem #1 Mesh (ref. [3])

Figure 5.2 Problem #1 Mesh (MATLAB)
In Fig. 5.2 we have the mesh obtained using MATLAB; all the supports on the left face of the plate were modeled as rollers restricted in the global $X$ direction with the exception of the one at mid-height, which was modeled as pinned.

The results obtained by El-Mezaini and Citipitioglu, presented in ref. [3], for a square plate of length “L” under a distributed stress “P” compared with the results obtained using MATLAB are as follows:

5.1.1 Normal Stress on Concrete

Fig. 5.3 and Fig. 5.4 contain respectively the results from ref. [3] and MATLAB, for the normal stresses in concrete at different locations along the plate length (0.2L, 0.6L and 0.9L) measured from the left supports.

Figure 5.3 Concrete Normal Stress at 0.2 L, 0.6 L and 0.9L for Problem # 1 (ref. [3])

The normal stress in the concrete at 0.2 L (26 in) from the left supports obtained from ref. [3] was equal to .71P all along the cross section of the plate, where P was assumed equal to 225 lb/in, hence the stress in concrete at 0.2L is equal to 159.75 lb/in. Employing MATLAB, as it is shown in Fig.5.4, the value of stress in concrete was also constant along the plate height equal to 0.71P this is equal to 159.75 lb/in. The percentage difference between both was 0 %.

The normal stress in the concrete at 0.6L (78 in) from the left supports according to ref. [3] was also almost constant along the plate height, approximately equal to 0.818P or 184.05 lb/in at the bottom and top of the plate elevation, at mid-height the normal stress was equal to .72P or 162 lb/in and at the level of the steel bars the normal stress was equal to .66P or 148.5 lb/in. Employing MATLAB, at the top and bottom of the plate
cross section the normal stress was \(0.8P = 180 \text{ lb/in} (2.2\% \text{ difference})\), at the level of the steel bars the normal stress obtained was \(0.65P = 146.25 \text{ lb/in} (1.5\% \text{ difference})\) and at mid-height \(0.75P = 168.75 \text{ lb/in} (4.2\% \text{ difference})\).

At a distance equal to \(0.9L (117 \text{ in})\) from the left supports the stress in concrete according to ref. [3] was \(0.7P = 157.5 \text{ lb/in} \) at the top and bottom reinforcement level, \(1P = 225 \text{ lb/in} \) at the plate’s Mid-Height and \(1.0P = 225 \text{ lb/in} \) at the top and bottom of the plate’s cross section. Using MATLAB the normal stress on the concrete at Mid-Height were \(0.95P = 213.75 \text{ lb/in} (5 \% \text{ difference})\), at the top and bottom reinforcement level \(0.72P = 162.0 \text{ lb/in} (2.8 \% \text{ difference})\) and at the top and bottom of the plate’s cross section \(1.018P = 229.05 \text{ lb/in} (1.8 \% \text{ difference})\). Both sets of results were almost the same for the stresses on the concrete at the Top, Mid-Height and Bottom level of the Plate’s cross section.

### 5.1.2 Shear Stress on Concrete

Figure 5.5 and Figure 5.6 contain the results from ref. [3] and MATLAB respectively, for the shear stresses on the concrete at a distance equal to \(0.9 L \) from the left supports.
Figure 5.5 Shear Stress along the Plate Height at 0.9 L for Problem # 1 (ref. [3])

Figure 5.6 Shear Stress along the Plate Height at 0.9 L for Problem # 1 (MATLAB)
At the Top and Bottom reinforcement level the shear stresses were equal to zero according to ref. [3] and MATLAB (difference 0%); the exact same results were obtained at mid-height. At a distance of 0.3 L from the top and bottom of the plate cross section the shear stress on the concrete obtained from ref. [3] was .24P=54 lb/in, employing MATLAB the shear stress in the concrete at the same location was .25P=56.25 lb/in (4.2% difference).

5.1.3 Stresses along the Steel Reinforcement Layers

Figure 5.7 and Figure 5.8 contain the stresses along the reinforcement bars obtained from ref. [3] and MATLAB respectively. The vertical axis is in function of the resultant stress along the reinforcement bars divided by the applied pressure P (P=225 lb/in).

\[ \frac{\sigma_s}{P} \]

Figure 5.7 Stresses along the Steel Bars for Problem # 1 (ref. [3])
From ref. [3] the Steel Resultant Stress-P ratio was equal to 5.5 at the left end supports, 5.5 at 26 inches from the left supports, 5.35 at 52 inches from the left support, 5.3 at 78 inches from the left supports, 4.55 at 104 inches from the left supports and 0 at 130 inches from the left supports.

Employing MATLAB, the Steel Stress-P ratio was 5.69 at the left end supports (3.4 % difference), 5.67 at 26 inches from the left supports (3.1 % difference), 5.58 at 52 inches from the left supports (4.3 % difference), 5.27 at 78 inches from the left supports (0.6 % difference), 4.27 at 104 inches from the left supports (6.1 % difference) and 0 at 130 inches from the left supports (0% difference).

With this example we proved the validity of the routine created in MATLAB for the transmission of displacements from the deformed concrete into the steel embedded elements, in this case assuming perfect bonding between both materials.

Figure 5.8 Stresses along the Steel Bars for Problem # 1 (MATLAB)
5.2 Simply Supported Reinforced Concrete Beam (Bonded or Un-bonded Interface)

This second example consisted of a simple supported beam with a concentrated load applied at mid-span. The results obtained using MATLAB for the steel stresses along the reinforcement were compared with the results obtained by Dr. Mohammed Arafa and Dr. Gerhard Mehlhorn from ref. [1] obtained using the program SEGNID.

The Simply Supported Reinforced concrete beam has a span length of 12 ft, with cross sectional dimension 21.75” X 9”. The beam is subjected to a concentrated load at mid-span. In ref. [1] only half of the beam was modeled to take advantage of symmetry.

The material properties were the following: For the concrete material $E_c = 3300$ ksi and $\varepsilon_{cu} = 0.003$; the bottom reinforcement consist of two layers of #9 steel bars (two bars per layer) with $F_{y,steel} = 90.6$ ksi and $E_s = 27800$ ksi. The first layer is located 5.875 inches below the cross section center of gravity; the second layer is located 8.375 inches below the cross section center of gravity. The results obtained from both programs were compared for the second steel layer only.
Figure 5.9 and Figure 5.10 contain the mesh configuration for the concrete beam from ref. [1] and MATLAB respectively. This beam has a concentrated load at the right end (72 in) of 3.5 kips. The left support is located 9 inches from the left end (modeled as...
a roller). The supports on the beam’s cross-section at the right end were all rollers restrained in the $X$ global direction. Results were compared at the lower bottom steel reinforcement layer (8.375 inches below the cross section center of gravity), for two conditions: Fully bonded and perfectly un-bonded. The next figure (Fig.5.11) contains the results for the steel stress distribution on the lower bottom steel reinforcement from ref. [1] for different cases of bond stiffness. $Cr$ represents the Bonding Stiffness between the steel reinforcement and the concrete. When $Cr$ has a high value (i.e. 3000), the reinforcement is assumed as bonded and when it has a very low value (i.e. 0) is assumed as un-bonded.

Figure 5.11 Lower Bottom Reinforcement Stress Distribution (ref [1])
5.2.1 Full Bond Condition:

For the lower steel layer (8.375 in. below the cross section center of gravity), modeling half beam only and assuming all supports (Vertical and Horizontal) as rollers:

![Stress Distribution along the Lower Steel Layer Fully Bonded (MATLAB)](image)

Figure 5.12 Stress Distribution along the Lower Steel Layer Fully Bonded (MATLAB)

The stress values obtained using MATLAB along the lower steel layer (Fig.5.12) are compared in table 5.1 with the ones presented on ref. [1] (Fig. 5.11).
Table 5.1 Comparison Between the Results Obtained Using MATLAB and the Results Presented in ref. [1] for the Stresses along the Lower Steel Reinforcement Layer

<table>
<thead>
<tr>
<th>Stress location along the lower steel layer, measured from the left end of the beam (in)</th>
<th>Results obtained from our program (ksi)</th>
<th>Results according to ref.[1] (ksi)</th>
<th>Difference in %</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>-0.005</td>
<td>-0.005</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>0.142</td>
<td>0.15</td>
<td>5.3 %</td>
</tr>
<tr>
<td>15</td>
<td>0.198</td>
<td>0.18</td>
<td>9.0 %</td>
</tr>
<tr>
<td>28</td>
<td>0.587</td>
<td>0.63</td>
<td>7.0 %</td>
</tr>
<tr>
<td>36</td>
<td>0.808</td>
<td>0.86</td>
<td>6.0 %</td>
</tr>
<tr>
<td>55</td>
<td>1.310</td>
<td>1.40</td>
<td>6.4 %</td>
</tr>
<tr>
<td>72</td>
<td>1.603</td>
<td>1.78</td>
<td>9.9 %</td>
</tr>
</tbody>
</table>

5.2.2 Un-Bonded Condition

For the lower steel layer (8.375 in. below the cross section center of gravity), modeling half beam only and assuming all supports (Vertical and Horizontal) as rollers:
For the un-bonded condition along the lower steel layer (Fig. 5.13), the stress distribution obtained using MATLAB was constant along the reinforcement, equal to 1.06 ksi. According to ref. [1] (Fig. 5.11), the stress on the lower steel layer was also constant along the reinforcement, equal to .96 ksi; the difference between both results was 9.4%.

5.3 Two-Span Continuous Beam

The third example consists of a two span, 30 ft each, continuous beam. The steel tendons are jacked at both ends of the beam with a pre-stressing force of 199.600 kips. Self weight and pre-stress losses due to friction were neglected. On this example the post-tensioned beam was modeled twice using different mesh sizes. The reactions at the three
supports and the normal stresses on the concrete located at the middle support are computed and compared with the results presented in ref. [3].

Figure 5.14 Mesh Configuration for Problem # 3 (ref. [3])

Figure 5.15 Mesh Configuration A for Problem # 3 (MATLAB)
The reactions obtained from ref. [3] (Fig.5.14) for the left, middle and right supports were 1666.25 lb (upward), 3332.52 lb (downward) and 1666.25 lb (upward) respectively. Employing MATLAB the reactions obtained for mesh A at the left, middle
and right supports were 1858 lbs (upward), 3716 lb (downward) and 1858 lb (upward) respectively. In the case of mesh B the reactions obtained at the left, middle and right supports were 1858 lb (upward), 3716 lb (downward) and 1858 lb (upward) respectively. The reactions obtained for both mesh configurations (A and B) were equal, with a difference of 10% with respect to the results presented in ref. [3].

The normal stresses obtained on the concrete located at the middle support (Fig. 5.17) using MATLAB were -2.748 ksi on the top fiber and 1.052 ksi on the bottom fiber for mesh A. For mesh B the stresses obtained were -2.81 ksi on the top fiber and 1.083 ksi on the bottom fiber. The top and bottom fiber stresses obtained from ref. [3] (Fig. 5.14 and Fig. 5.17) were -2.64907 ksi and 1.129 ksi respectively. For mesh A, the difference was 3.6 % on the top fiber and 7.3 % on the bottom fiber when the results were compared with ref. [3]. On mesh B, the difference was 5.7 % on the top fiber and 4.2 % on the bottom fiber when the results were compared with ref. [3].

5.4 Simply Supported Post-tensioned Beam

The fourth and last problem consisted of a simple supported post-tensioned beam with two single tendons located at different eccentricities.

The first tendon had eccentricities of 7.048 in (0.18 m) above the cross section center of gravity at the left end of the beam, 7.872 in (0.2 m) below the cross section center of gravity at mid-span and 7.048 in (0.18 m) above the cross section center of gravity at the right end of the beam. The second tendon had eccentricities of 7.048 in (0.18 m) below the cross section center of gravity at the left end of the beam, 11.808 in
(0.3 m) below the cross section center of gravity at mid-span and 7.048 in (0.18 m) below the cross section center of gravity at the right end of the beam.

The material properties are as follows:

\[ E_c = 4250 ksi (32000 MPa) \] (Concrete behaves as perfectly linear elastic)
\[ \nu = 0.2 \]
\[ E_{tendon} = 28227.869 ksi (195000 MPa) \]
\[ F_{yield} = 200.49 ksi (1385 MPa) \]
\[ A_{tendon} = .93 \text{in}^2 (6.00 \text{cm}^2) \]

The dimensions of the beam are 590.4 in (15 m) long, 12.59 in (0.32 m) width and 31.5 in (0.8 m) high.

Figure 5.18 Beam’s Mesh and Geometric Dimensions for Problem # 4 (ref. [5])
The load cases that are applied successively to this beam are the following:

**5.4.1 Load Case 1**

A pre-stress of 189.000 ksi is applied on the first tendon, plus 50 % of the beam’s dead load. Due to friction and wobbling, the stress on the tendon along the beam is going to vary between both ends. After obtaining the distribution of the steel stress along the tendons, the force the steel transmits into the concrete parent elements at every intersection point between both materials is obtained. The effect of these steel forces on the concrete parent element nodes are then added to the forces on the concrete parent element nodes due to the self weight of the beam.

It is important to make clear that the initial forces on the steel tendons with the inclusion of losses due to friction (in case of anchorage setting) are the forces of the steel elements just at transfer, meaning that to keep this assumed initial condition on every steel tendon after applying the initial pre-stress, the contraction of the beam due to this initial pre-stress force must be recorded, this contraction of the concrete beam is due to the elastic shortening effect of the concrete under the applied initial pre-stress. This
contraction or elongation will be subtracted (or added) to the resultant displacement of the steel due to every new load applied on the beam to obtain the actual strain state of the tendon.

For the load case 1, the results employing MATLAB were compared with the results obtained from ref. [5] and the exact theoretical value for the stress distribution along the first tendon, the comparison is presented in figure 5.20. The exact theoretical value was obtained from ref. [5] and consists in obtaining the results by hand using Euler’s cable friction theory.

![Figure 5.20 Stress Distribution along the First Tendon under Load Case 1, Comparing the Results from MATLAB, ref. [5] and the Theoretical Solution](image)

Figure 5.20 Stress Distribution along the First Tendon under Load Case 1, Comparing the Results from MATLAB, ref. [5] and the Theoretical Solution
5.4.2 Load Case 2

This second load case consisted in applying a wedge-pull-in to the live anchor of the first tendon, this will cause further losses on the first steel tendon due to the anchorage setting. The new force distribution along the first steel tendon due to this anchorage loss will be related to the linear loss of stress due to the friction along the first tendon and the duct interface that was presented in load case 1.

The initial pre-stress force that was applied on the beam in the previous load case (without anchorage loss) is removed from the global force vector by clearing the global force vector and applying the new initial pre-stress force (with the loss due to friction and anchorage setting included) and also again applying the 50% of the beam Dead Load (One more time it is needed to take into account the contraction of the beam due to this new force distribution on the steel tendon and record it as the initial displacement of tendon 1 just after transfer).

The new force distribution on the steel tendon will be obtained by asking the user the amount of wedge-pull-in on the anchor, from this value and by trial and error (assuming the initial length of the tendon affected by the wedge-pull-in of the anchor equal to half-length of the beam) the pre-stress loss \( \Delta P \) at the live anchor and the new distribution of forces on the steel tendon just at transfer are obtained.

\[
\Delta P = 2pl_{set} \tag{Eq.5.1}
\]
Where $\Delta P$ represents the change of stress at the live anchor, $l_{set}$ is the length of the tendon affected by the anchorage set, which is related to the anchorage setting by:

$$l_{set} = \sqrt[3]{\frac{\Delta_{set} A_s E_s}{P}}$$  \hspace{1cm} \text{(Eq.5.2)}$$

From the new steel force at the live anchor and the length of the tendon affected by the anchorage setting, by interpolation the force of the tendon at each intersecting point between the steel tendon and the concrete parent elements is obtained. These forces are then applied to the beam in conjunction with the self weight of the beam (50%). The results for the first tendon under load case 2, compared with ref. [5] and the exact theoretical answer are presented in Fig. 5.21.
Figure 5.21 Stress Distribution along the First Tendon Due to Load case 2, the Results from MATLAB are Compared with ref. [5] and the Theoretical Solution

5.4.3 Load Case 3

In this load case a pre-stress of 188.050 ksi is applied on the second tendon and the remaining 50% of the beam’s dead load is also included in the global force vector, the procedure followed was the same as the one explained for the load case 1.

The results obtained using MATLAB for the force distribution on the second tendon under load case 3 compared with the results from ref. [5] and the exact theoretical answers were the following:
5.4.4 Load Case 4

Load case 4 consists in applying a wedge-pull-in, but this time on the second tendon. The results of the force distribution along the second tendon due to load case 4 obtained using MATLAB, compared with ref. [5] and the theoretical exact answer are presented in Fig. 5.23. After applying the initial pre-stress on the second tendon (friction and anchorage loss included) plus the reminding 50% of the beam’s dead load, in addition to all the previous loads applied before on the beam (Pre-Stress with anchorage loss of the first tendon plus the initial 50% dead load) we record the contraction of the beam due to all the loads applied on the beam.
Figure 5.23 Stress Distribution along the Second Tendon Due to Load Case 4, the Results from MATLAB are Compared with ref. [5] and the Theoretical Solution

The contraction or elongation of the second tendon due to all the loads already mentioned will be recorded as the initial state of displacement at transfer for the second tendon.

With respect to the distribution of forces along the first tendon due to the tensioning and anchorage setting of the second tendon:
Figure 5.24 Stress Distribution along the First Tendon for Load Cases 3 and 4, the Results from MATLAB are Compared with ref. [5] and the Theoretical Solution.

On Figure 5.24, our results are compared with the results from ref. [5] and with the exact theoretical solution obtained by hand using Euler’s cable friction theory copied from ref. [5]. For the Euler cable friction theory the concrete and the tendon are assumed to be rigid and the interface is assumed to behave rigid-plastic.

The resultant stresses along the first tendon were obtained using the following procedure: the total displacement along the first tendon due to the entire beam’s self-weight and the pre-stress force on the first and second tendons is obtained (load case 4), after this the initial displacement along the first tendon due to the beam’s contraction at load case 2 is recalled. Then the displacement difference between load case 4 and load
case 2 is converted into forces, which will then be added (or subtracted) to the initial pre-stress force along the first tendon just after transfer of load case 2. By doing this it will be obtained how much the steel force along the first tendon increased (or decreased) between the load history. It may be a little confusing but then it has logic from the fact that the forces along the tendons will change for each new load applied on the concrete beam.

Finally, the results obtained with MATLAB for all four load cases were very similar to the results from ref. [5].
6 CONCLUSION AND RECOMMENDATIONS

6.1 Introduction

With the four examples presented in the previous chapter the capability of this program for the elastic analysis of post-tensioned beams or simple reinforced concrete beams has been proved. The program used on our thesis was written using the computer language known as MATLAB. These days the majority of programs available in the market are still being written in FORTRAN.

It has been proven over many years the power and utility of FORTRAN as a computer language, but sometimes users complain about the difficulty of programming using FORTRAN due to the fact that it is not very user-friendly. Because it is necessary to have a solid base in programming to be able to model any type of structure under any service loads, the skills or knowledge in programming will be crucial for a valid modeling. This is what makes attractive the use of MATLAB, it is very easy to learn, and at the same time is very powerful as a tool for programming.
6.2 Modeling Approach

In this research we presented the fundamental theory on modeling reinforced or Post-Tensioned beams using the Finite Element Method. The modeling of the steel tendon or steel reinforcement inside the concrete beam was made using the embedding approach. Our results were verified by comparing four examples with the results from different references (ref [1], ref [3] and ref [5]). Two of the three references (ref [3] and ref [4]) used the discrete approach, where the concrete nodes are moved to the locations where the concrete parent element is being intersected by the steel bar element. For the last example, our result was compared with the result stated in ref [5], in this reference work the steel bar element was modeled as curved and embedded into the concrete element in any particular direction.

The reason why we chose to use in our program the embedding approach for the modeling of reinforced or post-tensioned beams, is that it allows the steel element to be modeled inside the concrete beam in any direction, allowing the concrete meshing to be independent of the steel tendon layout.

The embedded modeling approach we used consisted in representing both materials (Steel and Concrete) separately with different elements. An 8 node Iso-Parametric 2-Dimensional element was used in representing the concrete and a one dimensional bar element was used for modeling the steel bar element.

After the properties of the two materials (steel and concrete) and the mesh description of the beam are specified, the existing loads (due to pre-stress or self-weight of the beam) are applied into the concrete parent elements, and from friction transmitted
into the steel. To do this, after we obtained the concrete parent elements displacement (due to the applied Pre-Stress or static superimposed forces on the beam) we model the entire interface between both materials like a continuous truss, where the supports of the truss have a predefined displacement (equal to the displacement of the concrete parent element at the points where it is intersected by the steel bar element). Both nodes (from the steel and from the concrete parent element) at the intersecting point are connected by a spring element, with its stiffness obtained from the Bond Stress-Slip relationship at the interface between both materials.

In our examples the stiffness at the interface was simulated as perfectly bonded (With a high bond stiffness value) or perfectly un-bonded (Bond stiffness almost equal to zero).

From the resultant displacement of the steel bar elements we obtain the increase (or decrease) of the steel tendon (or reinforcement) forces along the beam, which will be added (or subtracted) to the initial pre-stress force along the tendon. For practical reasons this initial pre-stress force along the steel tendon was assumed to be equal to the stress distribution depicted in the PCI MANUAL [19], which is linearly decreasing from the live anchor towards the other end of the beam due to the wobbling and frictional effects on the concrete-tendon interface. In case we include the anchorage loss (or wedge-pull-in), then the initial tendon Pre-Stress will be modified along an approximate tendon length ($l_{\text{est}}$) obtained by trial and error to account for the loss of stress on the tendon due to this anchorage setting.

Also, we could suggest to the reader or the user of our program that with the inclusion of some additional files into the structure of the computer program explained in
detail on chapter four, the non-linear analysis of the beam (in two or three dimensions) can be performed. For the non-linear modeling of the concrete material we would recommend the rotational orthotropic smeared crack model to obtain the stresses on the concrete parent elements Gaussian points.

Using the embedding approach almost any kind of post-tensioned or reinforced structure can be modeled in two or three dimensions. This modeling approach of the steel embedded into a concrete parent element can also be applied on shell elements or three-dimensional solid elements.

The most useful aspect of this program is that it can quickly provide the stresses on the concrete at any location along the beam or the stress distribution along the tendon under a specific load history (i.e. during the beam’s construction process), so the steel and concrete stresses can be updated for every new additional static load applied on the beam. It will also save time by giving the analyst a first hand result of stresses on the concrete material at any location along the beam for a particular beam’s cross section, so by inspecting the stresses on the concrete the analyst will be able to choose quickly the appropriate beam’s cross section dimensions.

### 6.3 Conclusions

It is necessary to increase the use and employment of the finite element method as an engineering tool in the construction field. Sometimes this method is under-estimated because these days the code offers enough margin of security to the designer. The designers often prefer to use a well established design method that has been tested for
many years, instead of a finite element program to model in more detail a particular structure.

As it is common in this field, not all the engineering problems are the same, there will be some occasions were the engineer will need a more accurate analysis on a specific section or member of a structure, and for this the Finite Element method can be a very valuable tool to obtain results with a high degree of accuracy. At the end the results presented in chapter five for the four different problems were in good accordance with the results presented in the different references, meaning that by using the discrete or the embedded modeling approach the difference between both results will be small.

6.4 Recommendations for Further Research

The steel bar elements embedded inside the concrete parent elements were modeled as straight one dimensional bar elements, due to this a couple of difficulties were encountered. The first and most important one is that in some situations when two adjacent steel bar elements at any intersecting point had a large length difference between them, the stresses on the steel reinforcement or tendon could vary significantly at that particular intersecting point (see FIG.6.1). Also, we remind the reader that because we are using straight bar elements for the steel elements, when we have a parabolic tendon profile, there will be a small angle difference between the local axes of two adjacent steel bar elements, but the effect of this difference of local angles between two adjacent steel elements on the tendon stresses at that particular intersecting point is almost negligible.
If this kind of discontinuity appears at the graph interface, it can be corrected by running the routine again with more concrete elements on the mesh. Also this discontinuity problem could be permanently corrected if instead of a one dimensional two node steel bar element we used a three or four node steel bar element, or even going further with using the model used in reference [5], because if a quadratic 2 dimensional element is used for modeling the concrete parent element and a linear one dimensional element is employed for modeling the tendon or reinforcement, the continuity between both materials can not be fully guaranteed.

By using i.e. a parabolic reinforcement element or more nodes for the one dimensional steel bar elements, a smooth transition of the displacement between two adjacent steel bar elements at a particular intersecting point will be guaranteed, regardless of their size ratio. The only disadvantage will be the increase of computational time due to the additional steel bar elements nodes.
If the user wishes to develop a nonlinear analysis, several adjustments will be necessary. The most important will be taking into account the change of location of the steel element nodes during the load history. For each new load case on the beam, the original position of the steel element nodes with respect the concrete parent element faces being intersected will change, so it will be necessary to update their location inside the concrete mesh.

About the main computer language (MATLAB) it is proven to be a very powerful tool, but sometimes when we ran the sub-routines to identify the specific case on which the steel bar element intersected the concrete parent element (Sections 4.2.12.2.1, 4.2.12.2.2, 4.2.12.2.3, 4.2.12.2.4 and 4.2.12.2.5) the program could not identify this intersecting point. This occurred a couple of times when the coordinates of one of the two steel bar element nodes had too many precision numbers, for this we had to use the MATLAB function fix or round for rounding the value of the coordinates of the steel bar element node. The sub-routine could then detect the intersecting point between the steel bar element and the concrete parent element.

Besides adding more nodes into the steel bar elements embedded into the concrete parent elements, another detail that can improve the results of the program is by including another Bond Stress-Slip relationship, but this one at the live anchorage of the beam. It will represent more accurately the change or loss of pre-stress on the tendon due to the anchorage set. Our program assumes no bonding at the tendon-duct interface when the initial pre-stress forces are transferred from the tendon into the concrete elements, at the beginning there is full compatibility between the tendon and the duct only at the location of the beam anchorages. After the initial transfer, the grout is added (incorporating the
stiffness of the tendon into the concrete elements), and then the interface changes between the duct and the tendon, with a high value for the bond stiffness to simulate the bond effect between both materials.
LIST OF REFERENCES


[16] ACI Committee 318 Building Code Requirements for Reinforced Concrete (ACI 318-89) and Commentary (ACI 318 R-89). American Concrete Institute, Detroit,


