UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO
PROGRAMA DE MAESTRÍA Y DOCTORADO EN INGENIERÍA
INGENIERÍA CIVIL – ESTRUCTURAS

HYBRID SIMULATION IN THE MULTI-SCALE ANALYSIS OF STRUCTURES

TESIS
QUE PARA OPTAR POR EL GRADO DE:
MAESTRO EN INGENIERÍA

PRESENTA:
CESAR PANIAGUA LOVERA

TUTOR PRINCIPAL:
DR. A. GUSTAVO AYALA MILIÁN, INSTITUTO DE INGENIERÍA

Ciudad Universitaria, Cd. Mx. AGOSTO 2016

Universidad Nacional Autónoma de México
UNAM

Biblioteca Central
Dirección General de Bibliotecas de la UNAM

UNAM – Dirección General de Bibliotecas
Tesis Digitales
Restricciones de uso

DERECHOS RESERVADOS ©
PROHIBIDA SU REPRODUCCIÓN TOTAL O PARCIAL

Todo el material contenido en esta tesis esta protegido por la Ley Federal del Derecho de Autor (LFDA) de los Estados Unidos Mexicanos (México).

El uso de imágenes, fragmentos de videos, y demás material que sea objeto de protección de los derechos de autor, será exclusivamente para fines educativos e informativos y deberá citar la fuente donde la obtuvo mencionando el autor o autores. Cualquier uso distinto como el lucro, reproducción, edición o modificación, será perseguido y sancionado por el respectivo titular de los Derechos de Autor.

JURADO ASIGNADO:

Presidente:	Dr. Pérez Gavilán Escalante Juan J.
Secretario:	Dr. Retama Velasco Jaime
1er. Vocal:	Dr. Ayala Milián Amado Gustavo
2do. Vocal:	Dr. Rivera Vargas Dario
3er. Vocal:	Dr. Juárez Luna Gelacio

Lugar o lugares donde se realizó la tesis: INSTITUTO DE INGENIERÍA, UNAM

TUTOR DE TESIS:

DR. A. GUSTAVO AYALA MILIÁN

FIRMA

A mis padres y hermanos.
A Dios.

Reconocimientos

Quiero expresar mi más sincero agradecimiento al Dr. Gustavo Ayala por el tiempo, conocimientos y guía que me ha brindado durante toda mi estancia bajo su tutela. A Dr. Jaime Retama por sus valiosas y continuas aportaciones. A los miembros de mi jurado por el tiempo invertido en revisar y enriquecer el contenido de este trabajo.

Reconozco y agradezco al Consejo Nacional de Ciencia y Tecnología (CONACyT) por el apoyo brindado por medio de su programa de becas. Así mismo al Instituto de Ingeniería de la Universidad Nacional Autónoma de México por permitirme desarrollar mis actividades dentro de su proyecto *"Análisis Sísmico de Estructuras Multi-Escala de Estructuras de Concreto Reforzado"*.

Declaración de autenticidad

Por la presente declaro que, salvo cuando se haga referencia específica al trabajo de otras personas, el contenido de esta tesis es original y no se ha presentado total o parcialmente para su consideración para cualquier otro título o grado en esta o cualquier otra Universidad. Esta tesis es resultado de mi propio trabajo y no incluye nada que sea el resultado de algún trabajo realizado en colaboración, salvo que se indique específicamente en el texto.

César Paniagua Lovera. México, D.F., agosto de 2016

Resumen

Actualmente en nuestro país, el diseño de un sistema estructural se basa principalmente en las disposiciones del Reglamento de Construcciones para el Distrito Federal y en el Manual de Obras Civiles de la CFE. En dichos documentos, generalmente se parte de los resultados del análisis de las estructuras reticulares para su dimensionamiento y/o revisión. Empleando esta clase de modelos, se presentan algunas situaciones que no deben pasarse por alto. La primera de ellas es que, para analizar una estructura, se considera que todos sus elementos estructurales son modeladas como macro-elementos, incluso en zonas donde no es adecuada esta consideración; además de no considerase el daño como un un fenómeno progresivo, sino uno que se manifiesta de forma instantánea, mediante la inclusión de zonas de daño concentrado como es el caso de las articulaciones plásticas. La segunda es que, el diseño de las conexiones y juntas entre elementos se hace de acuerdo a la normatividad vigente, lo cual es eficiente, sin embargo es necesario buscar alternativas que den mayor certidumbre de lo que realmente sucede en las zonas donde se localiza el daño.

En este trabajo se presenta una propuesta de análisis que ayuda a aminorar las limitantes mencionadas anteriormente, particularmente aquellas derivadas de emplear los métodos convencionales de análisis estructural, buscando abrir la discusión a la posibilidad de realizar análisis de detalle, y posteriormente diseños, para estas zonas, basados en estos resultados, sin necesariamente recurrir a recomendaciones de diseño derivadas de estudios experimentales.

La tesis se desarrolla tomando 2 ejes de estudio principales que son: una propuesta funcional de acoplamiento entre un sistema general de macro elementos y subsistemas (micro) donde se modelen las zonas donde se requiera un mayor detalle empleando elementos finitos sólidos más elaborados y leyes de comportamiento más generales y principalmente; un procedimiento que permita mediante un solo proceso la relación armoniosa de dos o más modelos de análisis estructural para con ello tener las ventajas de cada uno de ellos.

Abstract

Currently in Mexico, the design of a structural system is primarily based on the provisions of the Building Regulations for the Federal District and in the Manual of Civil Works of the CFE. These documents, often generalize part of the results of analysis of reticular structures for sizing and / or revision. Using this class of models, some situations that should not be overlooked present. The first is that in analyzing a structure, it is considered that all structural elements are modeled as macro-elements, even in areas where it is not appropriate in this regard; also, the structural damage in not studied as a progressive behavior, but as an instant, punctual manifestation, with the appearance of plastic hinges. The second is that the design of connections and joints between elements is done according to regulations, which is efficient, however it's necessary to look for alternatives that provide greater certainty of what really happens in areas where it is located the damage.

This work presents an analysis proposal that helps to reduce the limitations mentioned above, particularly those derived from using conventional methods of structural analysis, seeking to open the discussion to the possibility of detailed analysis and subsequently designs for these areas, without necessarily resorting to design recommendations derived from experimental studies.

The thesis is developed along two main lines of study which are: a functional coupling proposal between a general system of macro elements (frames) and subsystems (micro) which are areas where more detail is required, modeled using more elaborate finite element solids and more general laws to describe their behavior; a procedure which allows a single process by the harmonious relationship of two or more models of structural analysis to thereby have the advantages of each.

Contents

List of Figures **xiii**

List of Tables **xv**

1 Introduction **1**
- 1.1 Problem Definition . . . 1
- 1.2 General Objective . . . 3
- 1.3 Specific Objectives . . . 3
- 1.4 Motivation . . . 4
- 1.5 Research Scope . . . 4
- 1.6 Outline . . . 5

2 Background **7**

3 Theoretical Framework **15**
- 3.1 The Finite Element Method . . . 15
  - 3.1.1 Non Linear Analysis . . . 16
  - 3.1.2 Sub-structuring . . . 17
- 3.2 Multi-Scale Analysis . . . 18
- 3.3 Surface Coupling Techniques . . . 20
  - 3.3.1 Transition Elements . . . 20
  - 3.3.2 Connecting dissimilar elements. Rigid Elements. . . . 21
  - 3.3.3 Degrees of Freedom Subordination . . . 22
    - 3.3.3.1 Lagrange Multipliers . . . 22
    - 3.3.3.2 Penalty Methods . . . 24
    - 3.3.3.3 Transformation equations . . . 24
  - 3.3.4 Internal Forces Transmission . . . 26
- 3.4 Experimental Testing . . . 26
  - 3.4.1 Hybrid Simulation . . . 28
    - 3.4.1.1 Assessment of Hybrid Simulation . . . 28
    - 3.4.1.2 OpenFresco as a Framework for Hybrid Simulation . . . 30
  - 3.4.2 Coupled Numerical Simulation . . . 31

4 Validation Tests and Method Proposed **35**
4.1 Element Coupling Tests 35
4.1.1 Beam-Plane Solid Transition Element 35
4.1.1.1 Formulation 35
4.1.1.2 Axial Load 36
4.1.1.3 Bending Load 37
4.1.1.4 Fixed beam 39
4.1.1.5 Evaluation. 40
4.1.2 Dissimilar Elements 40
4.1.2.1 Static Case 41
4.1.2.2 Dynamic Case 44
4.1.2.3 Evaluation 45
4.2 Numerical Coupling Tests 46
4.2.1 Beam-Column 47
4.2.1.1 Static Load 47
4.2.1.2 Dynamic Load 48
4.2.2 Cantilever 49
4.2.2.1 Dynamic Load 49
4.3 Analysis Method 50

5 Application Examples and Comparative **53**
5.1 One Bay-One Story Frame 53
5.2 Two Bay-Three Story Frame 55
5.3 Evaluation of Results 59

6 Conclusions **61**
6.1 Future studies 62

Bibliography **63**

List of Figures

1.1 Plastic hinges model for a plane frame 1
1.2 Strain-stress relation graphics for concrete under longitudinal strain. . . 2
1.3 Resistant forces integration for bending in RC section. 2

2.1 Detailed model for local analysis of a truss connection. 8
2.2 Modeling of a three-dimensional frame with the Element Coupling Method. 8
2.3 Transition elements applied in the modeling of different types of multi-scale problems. 9
2.4 Computational homogenization of a shell-solid system in the Gauss integration points. 10
2.5 Applications of static experimental tests. 11
2.6 Configuration for HS of a three pier bridge. 12
2.7 Configuration for HS of a structure-foundation-soil system. 12

3.1 Common hysteresis models. 16
3.2 Incremental solution procedure. 17
3.3 Different scales of hierarchy to represent the damage of a solid. 19
3.4 Transition element connecting beam and plane stress solid elements. . . 20
3.5 Mesh transition methods for plane problems. 21
3.6 Connection of a 2D beam with a mesh of 4 node plane-stress elements. a) Hinge mechanism (No moment transfered at point A). b), c) Moment is transfered between the beam and the plane mesh A. 22
3.7 Complex analysis using Lagrange tie elements. 23
3.8 Physical interpretation of the penalty method, a fictitious stiff bar to enforce equal displacements. 24
3.9 Force distribution for a local analysis model, according to a linear stress theory. 26
3.10 Actuators configuration for a V-Brace testing. 27
3.11 Classic Hybrid Simulation testing procedure flowchart. 29
3.12 OpenFresco framework components to conduct an HS. 31

4.1 Plane frame-solid continuum transition element degrees of freedom and measures. 35

4.2 Plane transition element dimensions, with an unitary base. Subjected to longitudinal displacements in its corner nodes. 37
4.3 Displacement imposition configuration for the bending test and internal forces diagrams for its analogue frame (bending moment and constant shear). 38
4.4 Fixed beam with an applied uniform distributed load, modeled with Bernoulli and transition elements at its center. 39
4.5 Coupling application on a region of a plane frame. 41
4.6 Stress field representations in the solid plane elements, for the static load case 43
4.7 Accelerogram of Mexico City earthquake, 1985, component East-West, registered in SCT station. 44
4.8 Horizontal response of the coupled one bay frame under dynamic excitation, reported at Nodes 2 and 4. 45
4.9 Beam-Column structural model for numerical coupled analysis. 47
4.10 Base acceleration input for the Beam-Column numerically coupled analysis and its horizontal displacement response 48
4.11 Cantilever structural model for numerical coupled analysis. 49
4.12 Base acceleration input for the Cantilever numerically coupled analysis and its horizontal displacement response 50

5.1 Detailed portion of the model assigned to the slave program and its interface degrees of freedom. 53
5.2 Comparison between the complete and multi-scale coupled structure horizontal displacements in the One Bay Frame under SCT E-W 1985 earthquake. 54
5.3 Geometry and characteristics of the two bay-three story frame example. 55
5.4 Analysis models for the two bay-three story frame example. 56
5.5 Load pattern proportions applied for the incremental static analysis applied to the models of the two bay- three story frame. 56
5.6 Capacity curves obtained from the incremental analysis of the three different structural models 57
5.7 Decay of the lateral stiffness of the 3 models for the two bay- three story steel frame, during the pushover analysis. 58

List of Tables

3.1 Comparison of experimental testing methods for structures 27

4.1 Analysis results from longitudinal strain tests in transition elements. . . 37
4.2 Analysis results from bending tests in transition elements. 38
4.3 Comparison of transition model with analytical solution of a fixed beam. 39
4.4 Stress state in the center of span for a fixed beam modeled with transition elements and the analytical solution. 40
4.5 Reported displacements and moments from the vertical load analysis for different models of the one bay frame. 42
4.6 Reported stresses in extreme fibers of the coupling surface from the vertical load analysis for different models of the one bay frame. 42

5.1 Results from the pushover analysis of the 3 models for the two bay- three story steel frame. 58

Chapter 1

Introduction

1.1 Problem Definition

In the modeling of a structure to study its behavior, normally, the most efficient methods are employed in terms of the results quality and difficulty of application. This conventional solution is to represent beams and columns with frame (bending) elements connected by a rigid diaphragm. This model is subjected to vertical loads, produced by the building occupation, and lateral loads, mostly effect of the natural phenomena on the structure.

The general structural behavior under low demand levels is usually studied with linear-elastic models, while the approach followed to simulate material non-linearity under high demand levels is to introduce in the elements with high demands (shear or bending) the possibility of developing plastic hinges, aimed to reproduce the mechanism of redistribution of the internal forces in the elements [2, 23] ,this approach is known as concentrated plasticity. This kind of models has been really useful as it allows the engineers to establish general design criteria such as that of strong column-weak beam.

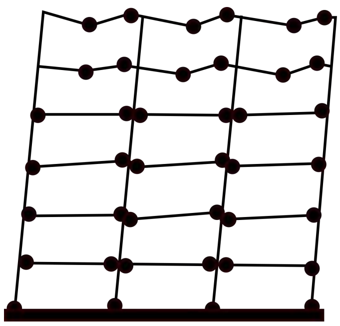
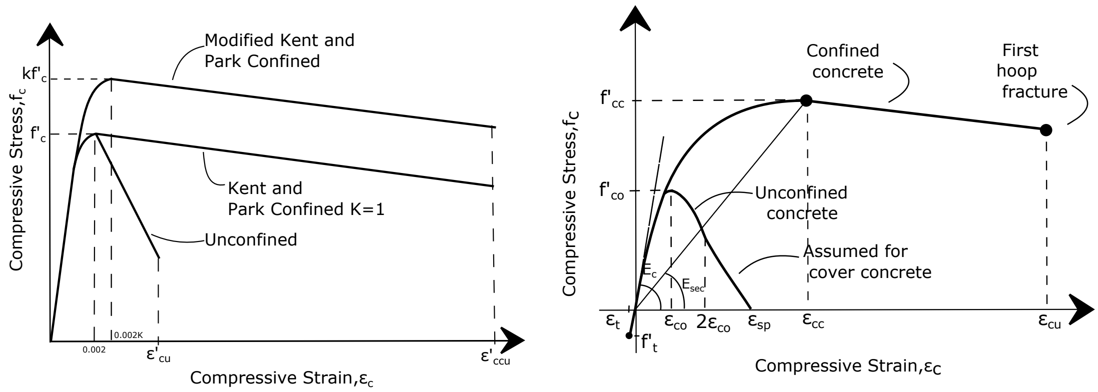


Figure 1.1: Plastic hinges model for a plane frame

The reinforced concrete is one of the most widely materials in construction, has been extensively studied, but most design engineers still using, to analyze structures, simplifying behavior hypotheses that do not follow real behavior, particularly under high demand levels. The RC behavior, as most of materials, may be expressed in the strain-stress curves, that clearly show a non linear relationship, Kent and Park (1971) propose a parabola with linear fall, and Mander et al. (1988) a completely non-linear continuous expression, both of them applicable to confined or unconfined concrete.



(a) Park & Kent **(b)** Priestley & Mander

Figure 1.2: Strain-stress relation graphics for concrete under longitudinal strain.

To determine the strength of a frame section under load actions , an integration of internal forces is done, often idealizing the distribution of stresses and strains to make possible the calculations.

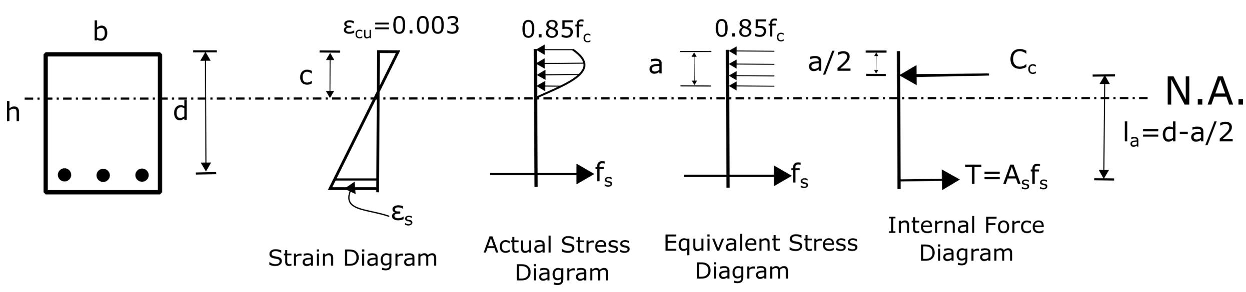


Figure 1.3: Resistant forces integration for bending in RC section.

The last two ideas, non-linear analysis and strength calculation, have something in common, a problem of high heterogeneous nature or conditions in a small portion has been tried to be solved by considerations that attack it in a bold manner.

Even though these solutions are satisfactory in general, there are other points of view and solutions for the non-linearity problem, one of them is to increase the precision of the analysis in the portions of the model where more information is needed and non-linear behavior presents. Doing it this way, proportionates advantages as: the mechanism of energy dissipation and internal forces redistribution would no longer be concentrated in points, instead these forces would be redistributed in a well defined zone of the model, adding certainty to the analysis results and, consequently, to the design of these parts of the structure.

This approach is not commonly followed, as there are still some limitations, such as the lack of analysis tools in programs to combine different numerical and experimental portions and methods; the selection of element characteristics in lower scale models, type and behavior (expressed in constitutive models for materials) and the integration of response of these models for the transfer of effects and the coupling with higher scale parts of the structure. The three situations mentioned, has been studied by different research groups, therefore this work aims to use their work in an integral manner to apply it to the non-linear analysis of complex structural systems.

1.2 General Objective

The general objective of this thesis is to propose and validate an integral procedure that allows the detailed study of the local effects in joints and selected portions of a complex structure taking into consideration the participation of rest of the structure where this detail of results are not required. All this exemplified for the analysis of plane frames under seismic loads.

1.3 Specific Objectives

The goals required to achieve the previously stated objective are:

• Formulate a transition to efficiently connect the degrees of freedom of a beam-column element to those of a plane solid finite element.

• Take advantage of the concepts of hybrid simulation to formulate a coupled analysis, establishing concurrent communication between two or more different models within a single global analysis.

• Perform the structural analysis of two dimensional frames that consider for different parts of the structure different modeling scales, considering non-linear behavior.

1.4 Motivation

Conventional structural analysis procedures have showed their effectiveness through the time producing the large number of well-behaved buildings designed with their results. Unfortunately, there are still instances where complex structures cannot be correctly analyzed in a feasible and efficient way. These situations have motivated diverse research efforts to seek more general and efficient analysis alternatives, overcoming the limitations of the most conventional methods:

• Using linear analysis in zones where non-linearity is evident and have important contributions for the representation of the general structural damage..

• Use of simplified non-linear analysis models e.g., those using plastic hinges at element sections to concentrate non-linear effects, to model gross non-linear behavior.

• Using more refined analysis procedures such as non-linear Finite Elements are computationally expensive to use as they may take long time to produce results which are often harder to interpret.

Due to the inherent limitation of a single analysis approach for a complex structure, this thesis proposes to use different approximations with different levels of computational efficiency and quality of results. The method proposed uses elastic and/or inelastic macro-element models for the parts of the general structure where detailed results are not needed and/or important, and a refined non-linear finite element model allowing the analysis of damaged prone zones where detailed results are needed. All this, using single analysis tools to produce detailed results of use for design purposes. The application of such analysis approach may have a great impact on the design and detailing of connections and other critical zones where damage may occur, all this leading to more economical and safe designs.

1.5 Research Scope

• This thesis only considers the analysis of plane frames, thus, all the employed or implemented elements have the same nature.

• In the examples, the plane-stress elements used in the detailed zones of the steel frame model have a behavior characterized by an elasto-plastic constitutive model.

• The coupled simulation processes presented considers multiple numerical models that as a whole represent the complete structure to be analized

• The modeling and analysis procedure proposed may be used for structural evaluation porpuses.

1.6 Outline

The present work is divided in 6 chapters. This first chapter contains the objective and the goals of this investigation and a short outline of the analysis procedure proposed. The second chapter is a short summary of the past works related to this thesis theme. The third contains a theoretical framework, divided in concepts of dynamics and substructuring, non-linear behavior of the materials, coupling of different finite element types in a single analysis and the computational treatment of sub-assemblies to do the coupled and hybrid simulations.

The fourth chapter corresponds to the tests, necessary to validate the sub-structuring methodology proposed to perform in a single analysis the analysis of multiple models. The fifth chapter contains the numerical illustrative examples applying the concepts presented in the previous chapters. Finally, the last chapter presents the conclusions of this thesis and some suggestions for future research related to this work.

Chapter 2

Background

Multiple options to perform the analysis of structures have emerged in the past few decades, some of them focus on consider the small details of selected parts or portions of a structure, to record their internal stress distribution or project their non-linear effects to the global behavior. This chapter are exposed the progresses of two of these options both related to the topic of this thesis.

The **multi-scale analysis** defined in 3.2 is a relatively recent numerical tool in which the global structural behavior and non-linear features of local details in a large complex structure may be concurrently analyzed in order to meet the needs of structural state evaluation as well structural deteriorating, solving the model in different scales. In structural engineering the most popular solution correspond to a the application of constraints equations to impose conditions of one system into the other.

Li et al. [17] applied the sub-structures concept to condensate the local degrees of freedom of connections in bridge trusses (figure 2.1), coupling the different scale elements with constraint equations, updating the reduced super element stiffness matrix based on the lower scale non linear effects, this implementation was done through multiple user routines .

Yue et al. [38] proposed an element coupling procedure for the analysis of reinforced concrete (RC) buildings, based on linear constrain equations in the boundary surfaces. To maintain the linear and constant relation equations, the model is divided in three types of elements : frame elements, linear elastic solids and non-linear solids (figure 2.2). While in 2014, [36] presented a mixed-dimensional FE coupling method that achieves both displacement and stress compatibility at the interface between the different element types; in this proposal, an iterative procedure to update constraint equation at each analysis step is executed to consider material and geometric non-linear effects of the lower scale of analysis in the global scale. This is an alternative to not include the linear portions in local analysis.

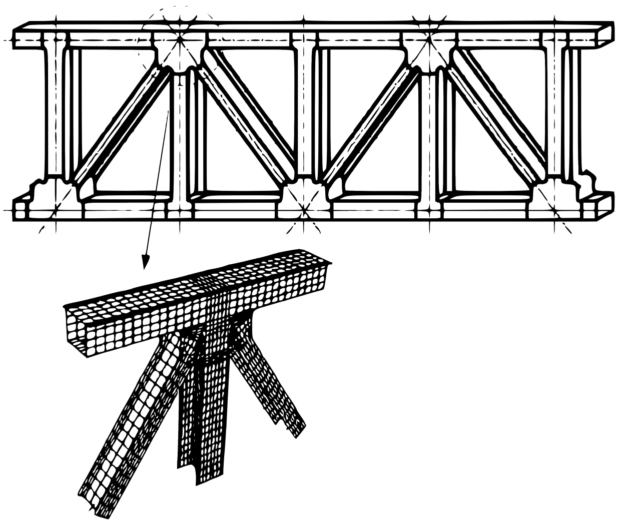


Figure 2.1: Detailed model for local analysis of a truss connection.

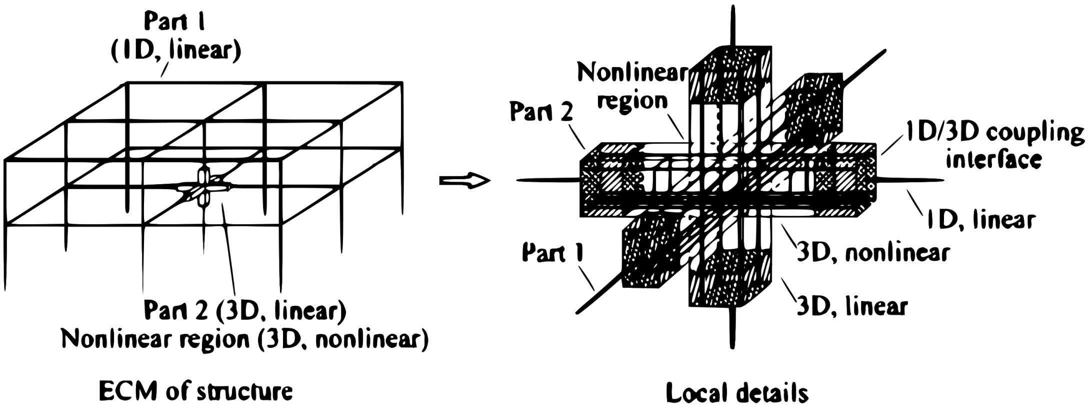
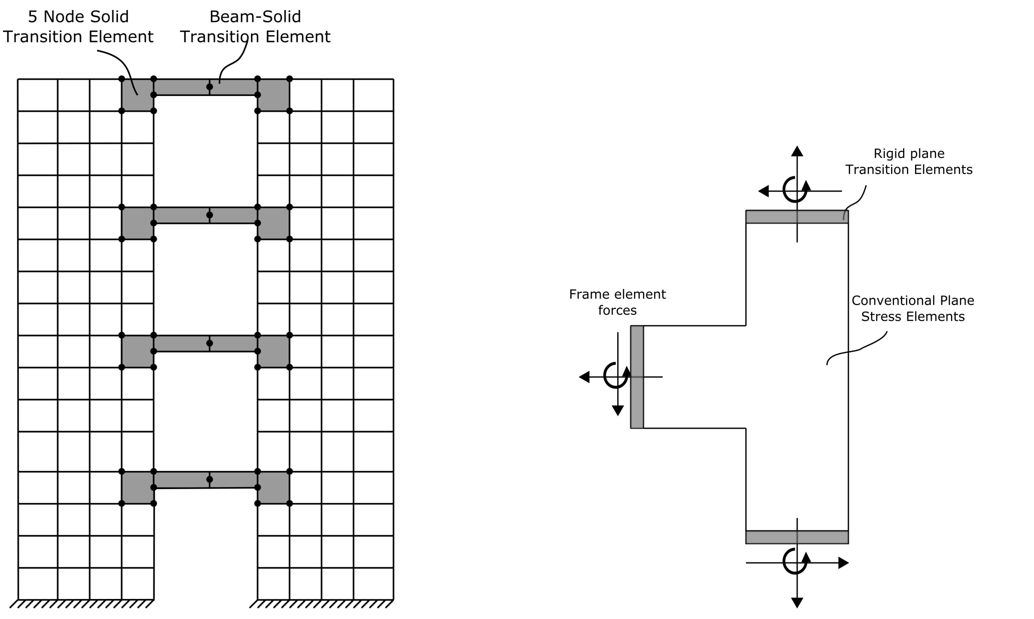


Figure 2.2: Modeling of a three-dimensional frame with the Element Coupling Method.

A different method for multi-scale analysis of RC buildings was proposed by [21], using a parallelized scheme called *master-slave*. In this approach, a structure is divided to be solved in different FE programs in an iterative procedure, in which the master imposes displacements and receives resisting forces from the slaves. The coupling of the different scales is carried out by the imposition of the kinematics hypothesis of the beam model on the surfaces of the 3D models, achieving displacement and forces compatibility between the models. This approach required routines for information transfer between the programs and for kinematic impositions, mainly.

In the presented approaches to the multi-scale analysis, the models contain conventional elements in both end of the coupling surface, where, each of them is subjected to conditions imposed by the kinematics of the other end. The transition elements are other option, where the behavior hypothesis of the elements to be coupled are introduced in the formulation of a new one, with degenerated degrees of freedom and different nature at each end.

Kim and Hong, [16], formulated special transition elements for the numerical modeling of shear walls, solving the incompatibility related to the degrees of freedom by the introduction of a transformation matrix, that operated with the original stiffness matrix of an element, modify its DOFs. Another application of transition elements in multi-scale analysis of plane frames is proposed in [26], where the rigid body kinematic assumptions are imposed in a plane stress triangle element, by the application of the transformation matrix process. These special elements are placed in the coupling surface to transmit the effects of the global scale into the local scale.



(a) Shear wall with beams **(b)** Frame elements with plane solids

Figure 2.3: Transition elements applied in the modeling of different types of multi-scale problems.

In the previously exposed methods, the coupling is associated to a surface or element where the relations between the different scale systems are established, nevertheless, multiple authors have proposed approaches with overlapping regions. The volume coupling deals with the situation where two or more models are overlapped in a common zone.

Specially relevant in problems in which it is not possible to define a coupling surface in the micro-scale model. In these problems the Representative Volume Elements (RVE) approach may be used, defining a model in the neighborhood of a material point in the macro-scale, used to calculate the representative properties of the micro-scale model at that point [10]. A proposal based on this method was presented by [11], where a shell structure is studied in two scales with solid brick elements, in a FE^2 iterative scheme.

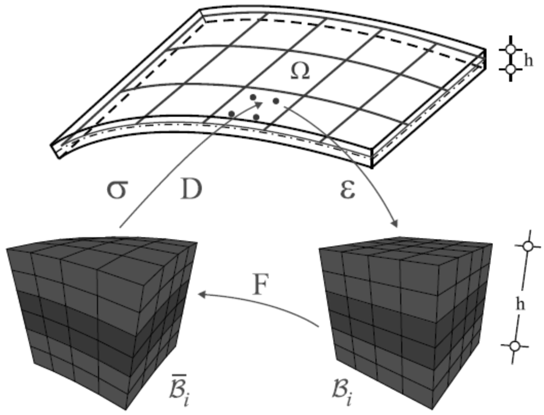


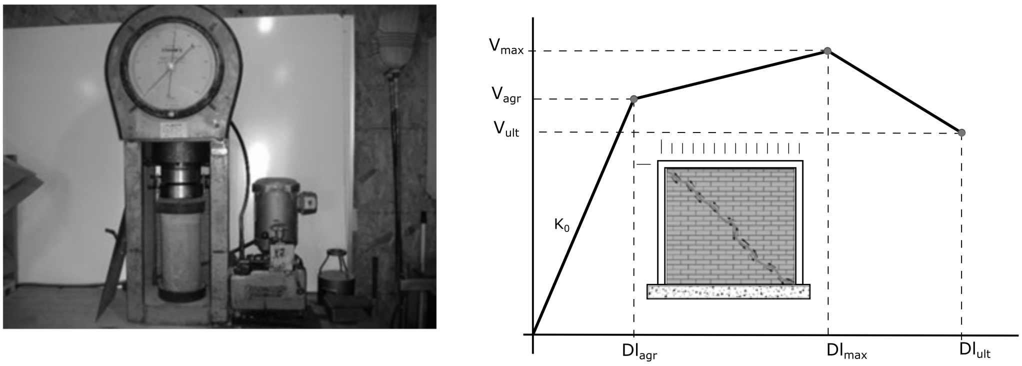
Figure 2.4: Computational homogenization of a shell-solid system in the Gauss integration points.

The **experimental testing** described in section 3.4, is one of the most extensively used method to evaluate the structural behavior, and several variations of testing have been proposed through the years.

Static tests are the first established experimental testing method, specially suitable to determine most of the basic properties of materials and complex behavior structure components such as connections. The building codes [4] specify the use of experimental static test as a normalized practice, e.g. compressive strength of concrete ,figure 2.5a, is an obligatory test to supervise the quality of this material.

In research, this approach is used to determine the general behavior of structures under load patterns, e.g. masonry walls under lateral load on their plane (figure 2.5b) which present fragile failure induced by tensile stress presented along their diagonal.

This type of tests have been principally useful to describe diverse non-linear phenomena or obtain information for analysis, but with the inconvenient that they do not capture behaviors associated to inertial effects or load application rate, since the force/displacement is imposed slowly.



(a) Compressive strength test **(b)** Lateral load test on masonry walls

Figure 2.5: Applications of static experimental tests.

In order to carry out the most realistic evaluation of the seismic performance of structures, the shaking tables appeared in 1893 (Tokyo), allowing the research groups to consider all the types of effects involved in the behavior, such as non-linearity of any source, inertial forces and load rate effects.

Even when the results of shaking table experimentation may be considered the closest to the exact description of the structural response under a given base excitation, this method presents a series of issues in its application as evaluation tool. Among these: the model size is limited to the capacity of the table, it is very expensive to use and the results obtained are only associated to one structure subjected to one input.

The limitations of the previously described tests gave rise to a new type of test, hybrid simulation, also known as pseudo-static and proposed at mid of 1970 decade. It uses on-line computer calculation and control together with experimental measurements of the actual resisting forces of the structure to provide a realistic simulation of the dynamic response. For simulating the earthquake response of a structure, the record of a real or artificially generated earthquake ground acceleration history is given as input data to the computer running the PSD algorithm. The horizontal displacements of the building floors (where the mass of the structure can be considered to be concentrated) are calculated for a small time step using a suitable time integration algorithm. These displacements are then applied to the tested structure by servo-controlled hydraulic actuators connected to the reaction wall. Load cells on the actuators measure the forces necessary to achieve the required story displacements and these structural restoring forces are returned to the computer for use in the next time-step calculation. Because the inertia forces are numerically modeled, there is no need to perform the test on the real time-scale, thus allowing very large models of structures to be tested with a relatively modest hydraulic power requirement [24].

Pegon and Pinto [24], described an application of the pseudo-dynamic test, oriented to the modeling of large scale irregular bridges, structures not suitable for a shaking table test, nevertheless, sub-structuring concepts allowed to separate their base piers from the analytical model in order to couple physical portions to it (figure 2.6).

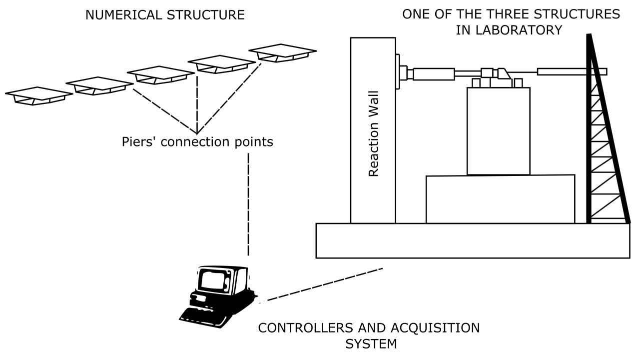


Figure 2.6: Configuration for HS of a three pier bridge.

Hybrid simulation is a highly versatile method for testing, extensible to other factors involved in the structural behavior, in 2012, [33] presented a pseudo-dynamic test for structure-soil-foundation systems (figure 2.7), where the super-structure was modeled numerically by finite elements while the foundation and the soil was a physical test. The excitation was an acceleration history at the soil. The results found of this works show how approximate the hybrid simulation is, compared to a shaking table test.

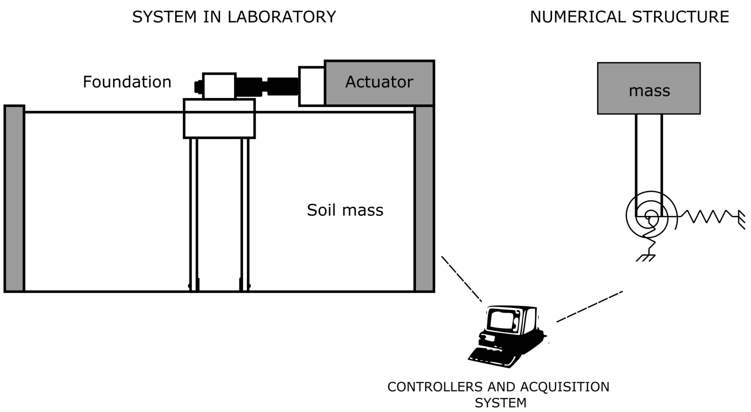


Figure 2.7: Configuration for HS of a structure-foundation-soil system.

The on-line hybrid simulation distributed in multiple laboratories is a difficult task due the lack of a common framework to relate different numerical and experimental tools employed in each laboratory. To solve this problem [32] created an environment independent application, modular and open source, to act as software framework for deploying hybrid simulation in a robust, transparent, scalable and easily extensible fashion.

Currently, multiple research groups are focused in the extension of the capacities of OpenFresco adding useful modules as the developed by Huang et al. in 2008 which allows to the users run concurrent numerical simulations in different FE-softwares located in different laboratories by the interaction of their interface degrees of freedom.

One example of this framework application is the presented by [18] where a coupled simulation and an hybrid simulation test were conducted to study the behavior of moment resisting steel frames with infill panels.

Chapter 3

Theoretical Framework

3.1 The Finite Element Method

The Finite Element Method is a numerical solution for field problems that can be expressed as differential equations or a integral expression [5]. The method seeks to divide the domain in small portions or elements with a well-known behavior to solve and subsequently reintegrate the system, in an approach limited by the formulation assumptions and the capacity of the computer that executes the analysis [39].

To illustrate the method, a conventional formulation of an elastic solid mechanics problem can be expressed as follows:

The Potential Energy is defined as:

$$\pi_P = U_{Int} - W_{Ext} \tag{3.1a}$$

$$\pi_P = \int \frac{1}{2}\sigma_{ij}\epsilon_{ij}dV - \ (b_i + \frac{\partial \sigma_{ij}}{\partial X_j})U_i dV \tag{3.1b}$$

Applying the divergence theorem and Cauchy's relation for the surface forces:

$$\pi_P = \frac{1}{2}\int \sigma_{ij}\epsilon_{ij}dV - \int_V b_i U_i dV - \int_S t_i U_i dS \tag{3.1c}$$

Introducing the Strain-Displacement [B] and material constitutive relations [C], the Potential Energy may be written in its matrix form:

$$\pi_P = \frac{1}{2}\int_V [u]^T[B]^T[C][B][u]dV - \int_V [u]^T[b]dV - \int_S [u]^T[t]dS \tag{3.1d}$$

Deriving with respect to displacements and equating to zero:

$$\frac{\partial \pi_P}{\partial u} = \frac{1}{2}\int_V [B]^T[C][B][u]dV - \int_V [b]dV - \int_S [t]dS = 0 \tag{3.1e}$$

The last equation, results from the application of the Principle of Minimum Total Potential Energy representing the equilibrium of a system.

3.1.1 Non Linear Analysis

The classic non-linear analysis procedures for reticular structures, involve the non linearity by the introduction of plastic hinges, these appear when a point (or points) of the frame reach a demand level, usually in bending, after this an element responds according to a defined hysteresis model that describes the force-displacement behavior of the section.

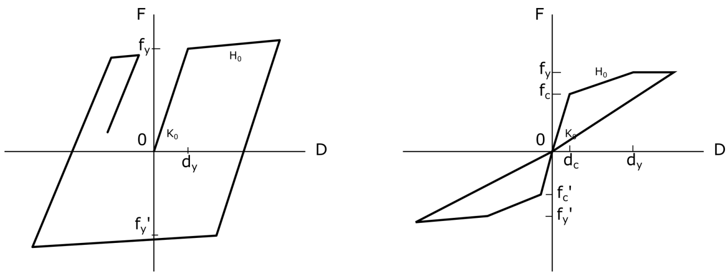


Figure 3.1: Common hysteresis models.

A response obtained from experimental tests or conventional models of a structure, subjected to external actions, is a general characterization its behavior. For numerical simulation, an insight to the source of non-linearity must be done, in order to incorporate it to the model. In structural analysis there are 4 sources of non-linear behavior [8]:

• Material: Material behavior depends on current deformation state and possibly past history of the deformation. Other constitutive variables may be involved.

• Geometric: Takes into account changes in geometry as the structure deforms in the setting up of strain-displacement and equilibrium equations. Greatly used in slender structures and stability analysis.

• Force Boundary Conditions: Applied forces depend on deformation, e.g. pressure loads of fluids over solid structures.

• Displacement Boundary Conditions: Displacement boundary conditions depend on the deformation of the structure. The most important application are the "contact" problems, where the prescribed displacements depend of the internal.

The numerical solution of non-linear problems (focusing in material and geometric NL) is through the Incremental-Iterative Analysis, process in which the equilibrium equations of the problem are expressed in a residual form, and linearized obtaining a "tangent stiffness matrix" [3], then a solution technique is applied iteratively until a convergence criteria is satisfied.

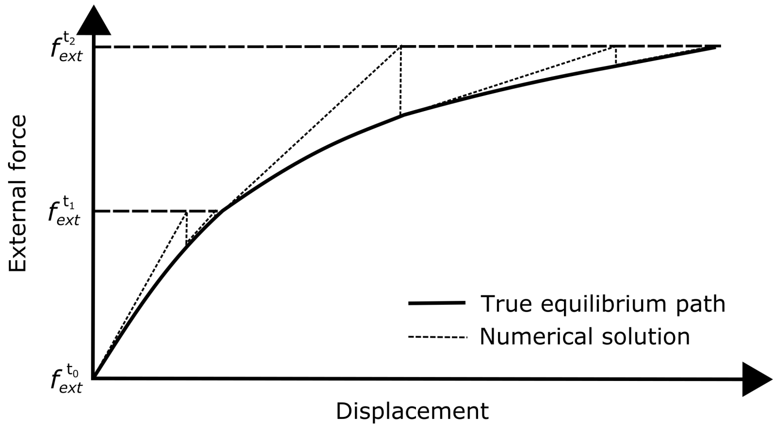


Figure 3.2: Incremental solution procedure.

3.1.2 Sub-structuring

The idea was proposed originally in aerospace engineering [7] with the purpose of carry out a first level breakdown, dividing the complete model to be studied by specialized groups of experts, taking advantage of the repetition of global components and overcoming the computer limitations. The last motivation has evolved (with the computers' processing power) to improve the efficiency in the solution of a complex system, by incorporating parallel processing and sub-domains concepts.

A super-element is an assemblage of finite elements that, upon assembly, may be regarded as an individual element for computational purposes. These elements must satisfy the mathematical condition of being rank-sufficient, thus, the super-element does not possess spurious kinematic mechanisms (The rigid body patterns are the only zero energy displacement configurations) [7, 9], this property guarantees that the static-condensation process works properly.

The macro-elements and sub-structures are super-elements, the difference resides in the selection of the elements to create the super-element, to call it sub-structure the group of elements are a distinguishable portion of the structure, regardless of the given name, the computational processing of them is the same.

The process of sub-structuring requires to classify the degrees of freedom in internal and boundary, the internal DOFs are eliminated from the stiffness matrix by Static Condensation and only the boundary DOFs are related with other super-elements to calculate the system solution. After the solution of the global system, the internal

DOFs values can be recovered from the previously obtained boundary ones.

3.2 Multi-Scale Analysis

In traditional approaches of modeling, even in FEM, the analyst tends to focus on one particular scale, the effects of other scales are neglected by assuming that the system is homogeneous in them. The philosophy of multiscale, multi-physics modeling is the opposite [37]. It is based on the viewpoint that:

1. Any system of interest can always be described by a hierarchy of models of different complexity. This allows to introduce about more detailed models when a coarse model is no longer adequate. It also gives us a basis for understanding coarse models from more detailed ones. In particular, when empirical coarse models are inadequate, one might still be able to capture the macroscale behavior of the system with the help of the microscale models.

2. In many situations, the system of interest can be described adequately by a coarse model, except in some small regions where more detailed models are needed. These small regions may contain singularities, defects, chemical reactions, or some other events of interest. In such cases, by coupling models of different complexity in different regions, we may be able develop modeling strategies that have an efficiency that is comparable to the coarse models, as well as an accuracy that is comparable to that of the more detailed models.

3. The introduction of multiple scales contribute to relax coarse model hypotheses, like classical continuum mechanics ones in the neighborhood of some critical points, in order to capture the physical phenomena better.

For a multi-scale analysis, there are different levels of detail, ordered in a hierarchy, where each level is a refinement of the higher levels, but, since they all describe the same physical system, the different models have to produce consistent results, this lead us to two major tasks [37, 38] : The first is a basic understanding of the different levels of analysis, their formulations, assumptions and properties. The second is, planting an analytical relation between different models that couples them smoothly, if there is a big cap between levels, a intermediate level, meso-scale, must be planted and formulated.

According to [10], three levels of analysis can be considered (each one with its respective sub-hierarchies):

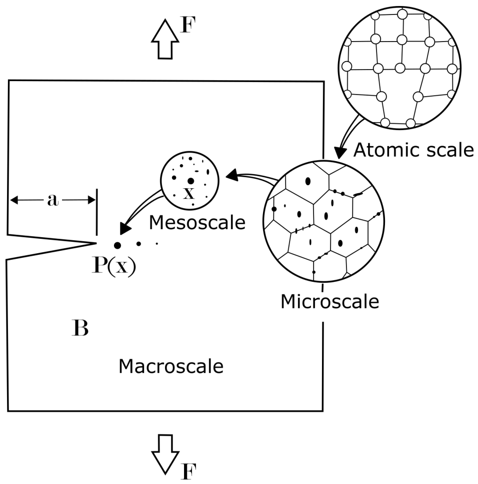


Figure 3.3: Different scales of hierarchy to represent the damage of a solid.

• **Macro-scale:** This scale has sizes or dimensions distinguishable by the observer, for mechanic and thermal problems it's associated to continuum mechanics.

• **Micro-scale:** The objects which can be seen with a microscope are included in this level, their dimensions vary between millimeters and microns. Some examples of the processes studied in this level are: the formation of vacuums, porosities, micro-fissures, etc.

• **Nano-scale:** This last level refers to all the objects with dimensions smaller than the micro-scale, suitable to study atomic phenomena.

In the context of structural engineering, the coupled objects normally are different types of continuum elements (beam,plate,shell and solid), the multi-scale FE simulation needs a rational method to combine mixed dimensional FE elements at their interfaces in a single structural model [36]. The principal issue in continuum elements coupling is to guarantee displacement continuity and force equilibrium in the region of interface between different element types.

There are two major methods to achieve the mentioned objectives:

In **volume coupling** exists a region where different models co-exist and usually planted using the Arlequin method, where the models aren't added but crossed and glued to each other, namely, there is a superposition of mechanical states in the *glued* zone, is a weighted sum of all the crossed models in the refined regions. One of the main technical difficulties to apply this method is the lack of flexibility of classic numerical tools to introduce the necessary procedures to apply the method [6].

While in **surface coupling** there is no overlapping region of different models where they are coupled, some of the most used techniques to realize it are discussed in next section.

3.3 Surface Coupling Techniques

When two or more different types or elements will be employed in a structural model, a transition between them is necessary, in order to achieve an appropriate transmission of the mechanical behavior between two or more different analysis level models without any excessive skip in the fields under study.

In this section a group of possibilities to achieve this objective are described, presenting their theory and doing an assessment of them.

3.3.1 Transition Elements

Initially based on the idea of introduce assumptions in the formulation of an element, the transition elements can act as a gate between different types of classic elements if the differences in the hypothesis of both elements are recognized and included [1].

For example, in most analysis, the beams and solid plane elements are considered separately, even though there exists a very close relationship between these elements, relation that may be established identifying the differences. The greatest differences between these elements are: the kinematic assumption that plane sections initially normal to the neutral axis remain plane (and normal to the neutral axis in Bernoulli's beam theory), and the stress assumption that normal stress in the neutral axis is equal to zero.

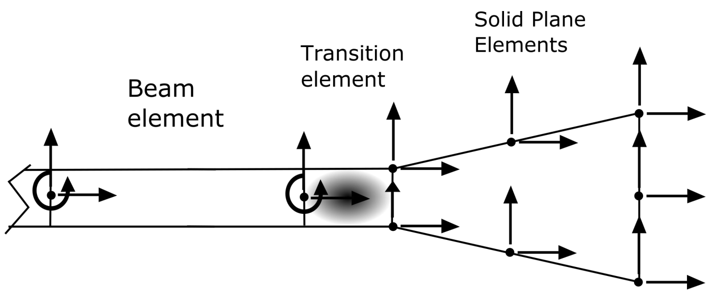


Figure 3.4: Transition element connecting beam and plane stress solid elements.

To formulate a transition element is necessary to rewrite the displacement field, adding the kinematic assumptions that represent the behavior to be imposed, leading to a new interpolation matrix and strain-displacement relationships involving all the degrees of freedom in the transition. After realizing these steps, the element formulation becomes the same of a conventional finite element.

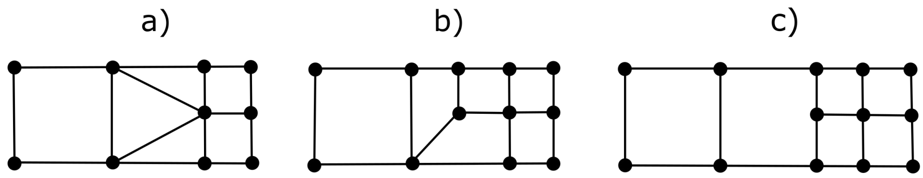


Figure 3.5: Mesh transition methods for plane problems.

The transition elements can be used to relate different types of finite elements but, they present the issue that impose certain mesh conditions after the element, this problem is solved partially by using mesh transition methods between fine and coarse meshes. In the figure 3.5 three different transitions are presented, the first ones do not need any special elements, while to use C) it's necessary to implement in the FE code a special 5 node element that ensures compatibility [14]

3.3.2 Connecting dissimilar elements. Rigid Elements.

The dissimilar elements are whose DOF are of different nature, for example, in the connection of a beam element (3 degrees of freedom, one rotational and 2 traslational per node) with plane elements (2 degrees of freedom, only traslational DOFs per node), the dissimilarity exists [5].

A manner of dealing with these situations is to impose constrains for the DOFs (exposed in the next section) , but in some cases this procedures may not be needed if the two different nature portions converge in a series of nodes.

For a beam-plane relation one of the options is to extend the beam element to pass through the plane elements by adding one or two more beam type elements embedded in the plane mesh, this will simulate the conditions of a rigid node. If we only connect the beam to the plane elements the corner node, the mechanism acts as a hinge and no "moment" is transferred.

This technique has the benefit that it is not always necessary to implement a special element in the employed FE code; however, there are some authors that develop special elements in conjunction with this technique to improve their results [26].

One of its disadvantages is the case in which rigid elements are necessary to model, this can lead to numerical problems due to the disproportion between their stiffness values and those of the other elements of the model. Another problem is that the concept might be used improperly, thus, leading to erroneous results. For example, a

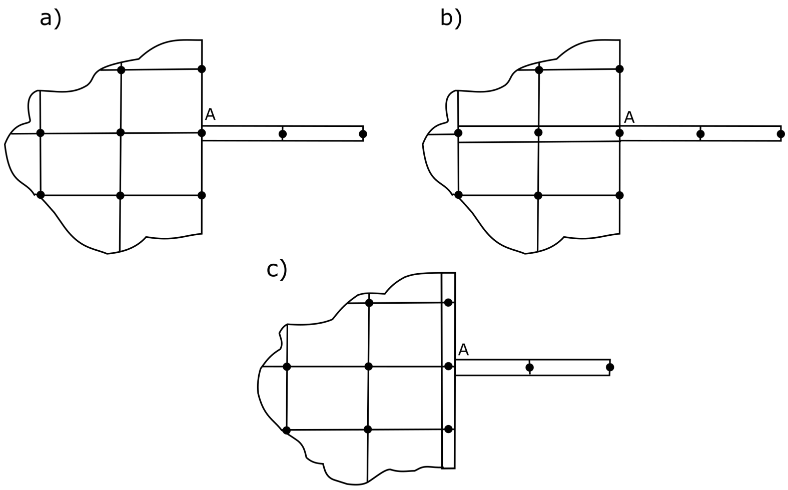


Figure 3.6: Connection of a 2D beam with a mesh of 4 node plane-stress elements. a) Hinge mechanism (No moment transfered at point A). b), c) Moment is transfered between the beam and the plane mesh A.

column fixed to a slab only subjected to a lateral load on the top of the column. If in the structural model the column is connected to the plate elements that represent the slab only at one point, the reported displacement at beam's end decreases as the mesh becomes denser, contradicting the FEM principles [25].

3.3.3 Degrees of Freedom Subordination

The coupling of two substructures can be done through the imposition of relationships between groups of degrees of freedom, in order to simulate the behavior of one substructure in the other one. In the following some of the methods, which are based in operating the mathematical expressions to impose the mentioned constrains, are presented.

3.3.3.1 Lagrange Multipliers

A widely used method for optimization problems to determine the maximum or minimum values of a constrained function, the restrictions are introduced in the expression with multipliers, and then obtain the stationary value of the new expression.

Applying this method to the Potential Energy finite element formulation, it takes the following form:

$$\pi_P = \frac{1}{2}\mathbf{d^T K d} - \mathbf{d^T f} + \lambda^{\mathbf{T}}(\mathbf{Cd} - \mathbf{Q}) \tag{3.2a}$$

$$\frac{\partial \pi_P}{\partial \mathbf{d}} = \mathbf{Kd} - \mathbf{f} + \mathbf{C^T}\lambda = \mathbf{0} \tag{3.2b}$$

$$\frac{\partial \pi_P}{\partial \lambda} = \mathbf{Cd} - \mathbf{Q} = \mathbf{0} \tag{3.2c}$$

$$\begin{bmatrix} \mathbf{K} & \mathbf{C^T} \\ \mathbf{C} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{d} \\ \lambda \end{bmatrix} = \begin{bmatrix} \mathbf{f} \\ \mathbf{Q} \end{bmatrix} \tag{3.2d}$$

The involved terms in 3.2 d for the restriction are a **C** matrix which represents a linear combination of the displacements to obtain a constants vector **Q**. While **K** and **d** are the usual structure stiffness matrix and displacement vector, respectively. The obtained multiplier values λ can be interpreted as the forces that impose the defined constraints [7, 12].

Houlsby et al. [12] implemented a variant of the method in which takes all the constraint conditions as special tie elements that relates two or more different types of element, these elements are threated for the program like conventional ones, so there is no need of extra operations with the global stiffness matrix.

The general advantages of the method are the following:

-Is theoretically exact, the only errors are product of the numerical solution of the equation system.
-Gives the imposition forces directly, which may be useful in certain applications.
-Is included in various finite element analysis programs to manage different types of constrains to be used [22, 35], if the necessary imposing conditions for the studied problem are available there is no need to be implemented in the used FE code.

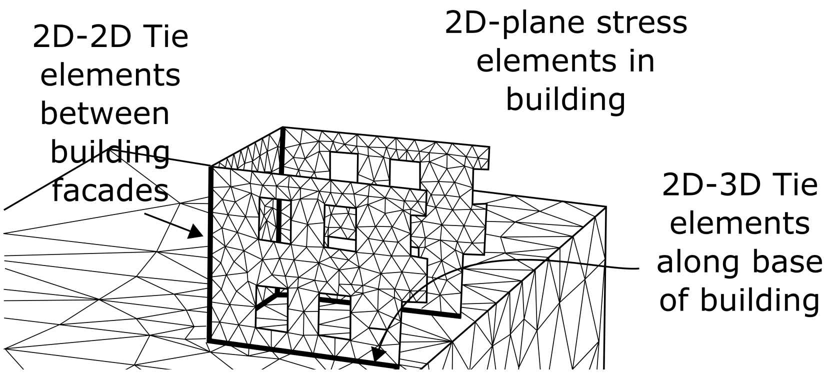


Figure 3.7: Complex analysis using Lagrange tie elements.

Is not a disadvantage free method, applying it introduces additional variables to the system, extending the original stiffness matrix, and modifies it to lose its positive definiteness, which might be a problem for the equation solvers.

3.3.3.2 Penalty Methods

In this method the constraint expressions are operated to obtain the called "Penalty Functions" [5], with a corresponding weight, and basically any configuration that goes against the constraints need a excessive amount of force to present, making them unfeasible.

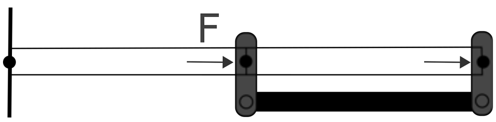


Figure 3.8: Physical interpretation of the penalty method, a fictitious stiff bar to enforce equal displacements.

This method has the advantage that for its implementation do not need any transformation of the original stiffness matrix, i.e. the displacement vector variables stay the same. Also there is no difference in the operation of constrained equations and those which are not, thus, all the post-assembly procedures and system solving are calculated in the habitual way. An important advantage is that the method is not greatly affected if a constraint is repeated, only doubles its weight, but no causing undue harm. Finally the positive definiteness is not lost, whence, doesn't affect certain numerical processes.

The method present the main difficulty of choosing the penalty weight, for a simple problems it is not a issue, but for complex ones it might take long time and finding the correct weight becomes a task itself, although "Square Root Rule" establish a guide to select the penalty weight, it is not always definitive and forces the user to do numerical experimentation looking to define the value of weight for each problem[7].

Another disadvantage is that the addition of great numbers to the stiffness matrix may ill-conditioning it, causing numerical difficulties in the solution of the system.

3.3.3.3 Transformation equations

In this method the original degrees of freedom are modified by transformation matrices, which contain the imposed constrains equations, modifying the system to be a function of the modified degrees of freedom and then solving it [5].
The advantages of applying the Transformation Equations method are that is exact and

reduces the number of DOFs to be solved, also it's easy to learn and apply it by hand [7].

The drawbacks appear in the general case implementation as routine in a FE software and the selection of DOFs to be reduced, also the method adds an auxiliary system to be solved and then pass the results to the original one, increasing the quantity of processes and stored matrices, which may slow the process of obtaining the solution.

3.3.4 Internal Forces Transmission

A procedure with a different focus has been developed by [19] for plane frames , in which two different scale models are coupled by a interface program, responsible of transforming the global model forces (3 per node for a plane frames) to a group of forces applicable to the local models (2 per each node of the plane stress elements) according to a linear stress distribution in the cross section, figure 3.9.

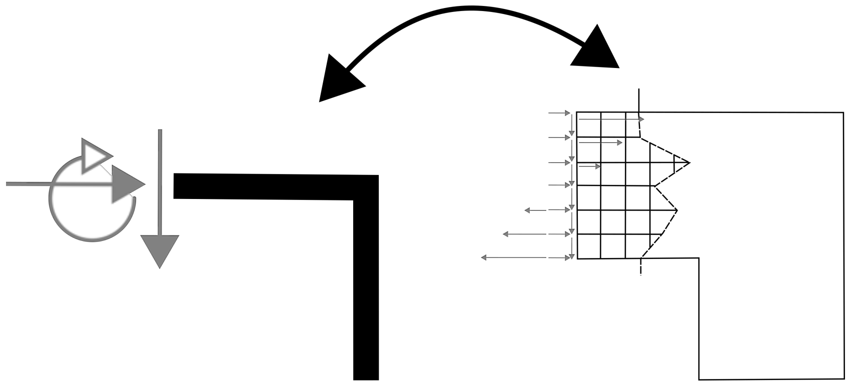


Figure 3.9: Force distribution for a local analysis model, according to a linear stress theory.

The advantages of this procedure are that the complete model is divided in multiple substructures to be analyzed separately, not involving all the DOFs in the global analysis and giving detailed information in the sub-structured zones. Finally, the problems with this procedure are that the user must input a force distribution in the communication algorithm, thing that is apparently simple when the substructures are conformed of a elastic-linear material, but in the non-linear case may vary considerably with the stress distribution. Another problem is that the proposed procedure uses a file system [13], adding a "save and read" subprocess to restart the local analysis each iteration of global process, increasing the computation time.

3.4 Experimental Testing

Currently several well-established methods to conduct laboratory tests exist, allowing to evaluate, in a reliable way, the seismic behavior of a complete structure or some of its components.

The first is the most common technique, called the *Quasi-static test*, where the studied structural component(s) is(are) subjected to a predefined history of loads or displacements on a series of specimens. This test can quantify the effects of : the changes in materials properties, detailing, load rates, boundary conditions, and other factors in a relatively easy and economical manner. The issue with this experimentation is that the applied load or displacement patterns aren't realistic (not obtained from interaction with the rest of a structural system), raising questions about whether the specimens were under or over tested for real project situations.

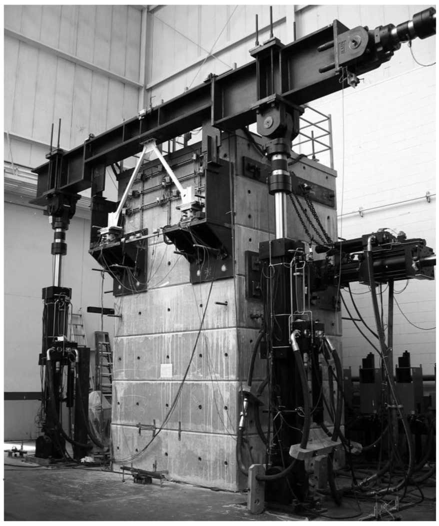


Figure 3.10: Actuators configuration for a V-Brace testing.

The second form of laboratory testing is the *Shaking Table*, based in the dynamic similitude principles, they are able to simulate conditions that resemble those occurring during a particular earthquake for a complete structure. These tests are excellent to evaluate complex structures, since the inertial effects, non-linearity, damage, damping, are completely involved without any calculation. On the other hand, only a few shaking tables exist with the means to apply accurate base motions to full-scale structures. The limited capacity and size of most available shaking tables place significant restrictions on the size, weight, and strength of a specimen that can be tested. As a result, reduced-scale or highly simplified specimens are commonly necessitated, which call into question the realism of many shaking table tests.

	Quasi-static	Shaking Table	Hybrid Simulation
Dynamic Effects	NO	YES	YES
Strain Rate Effects	NO	YES	YES (if real-time test)
Large or Full Scale Tests	YES	NO (limited by table)	YES

Table 3.1: Comparison of experimental testing methods for structures.

Finally, the third method is Hybrid Simulation, since it's the employeed in this work will be exposed in what follows.

3.4.1 Hybrid Simulation

Proposed by [34], the *Hybrid Simulation* is an experimental test where a model is constructed considering numerical and physical portions, to simulate the complete structural system [30]. This type of test is executed with a step-by-step numerical solution of the motion governing equations and involving directly the physical elements' resistance as forces.

The hybrid simulation is typically implemented as a displacement-based test, where the mass and viscous damping characteristics of the physical portions are modeled numerically, and the incremental displacement response under a dynamic excitation is computed at each step in function of the current state of the numerical and physical portions of the structure. Since the dynamic aspects of the model are treated numerically, the physical part of the test can be conducted quasi-statically, this type of test are also known as *Pseudo-dynamic test*.

The general process for a typical hybrid simulation is shown in figure 3.11.

3.4.1.1 Assessment of Hybrid Simulation

Advantages

- The applied load is determined by analytical processes, thus, the hybrid simulation provides means to study the structure under a wide variety of loading sources, without modifying the physical portions of the model.
- The HS gives the possibility of sub-dividing complex structures into sub-assemblies with a well known modeling by the FEM and physical tests in laboratory that represent the highly non-linear portions of the structure.
- The size limitations of shaking table test may be overcome, making possible full scale dynamic tests.
- Almost any test may be conducted on different time scales (quasi-static conditions, real time or rapid test), proving the researcher multiple benefits, such as: the common equipment in testing facilities is enough for most HS; slow tests allow inspection and tracking of damage; the time-rate can vary to satisfy dynamic similitude requirements or capture load rate effects, etc.
- Experimental and analytical sub-assemblies can be geographically distributed allowing researchers to take advantage of the different capabilities available in different laboratories.
- The user selects the location of each part of structure, based on difficulty of analysis and results quality.

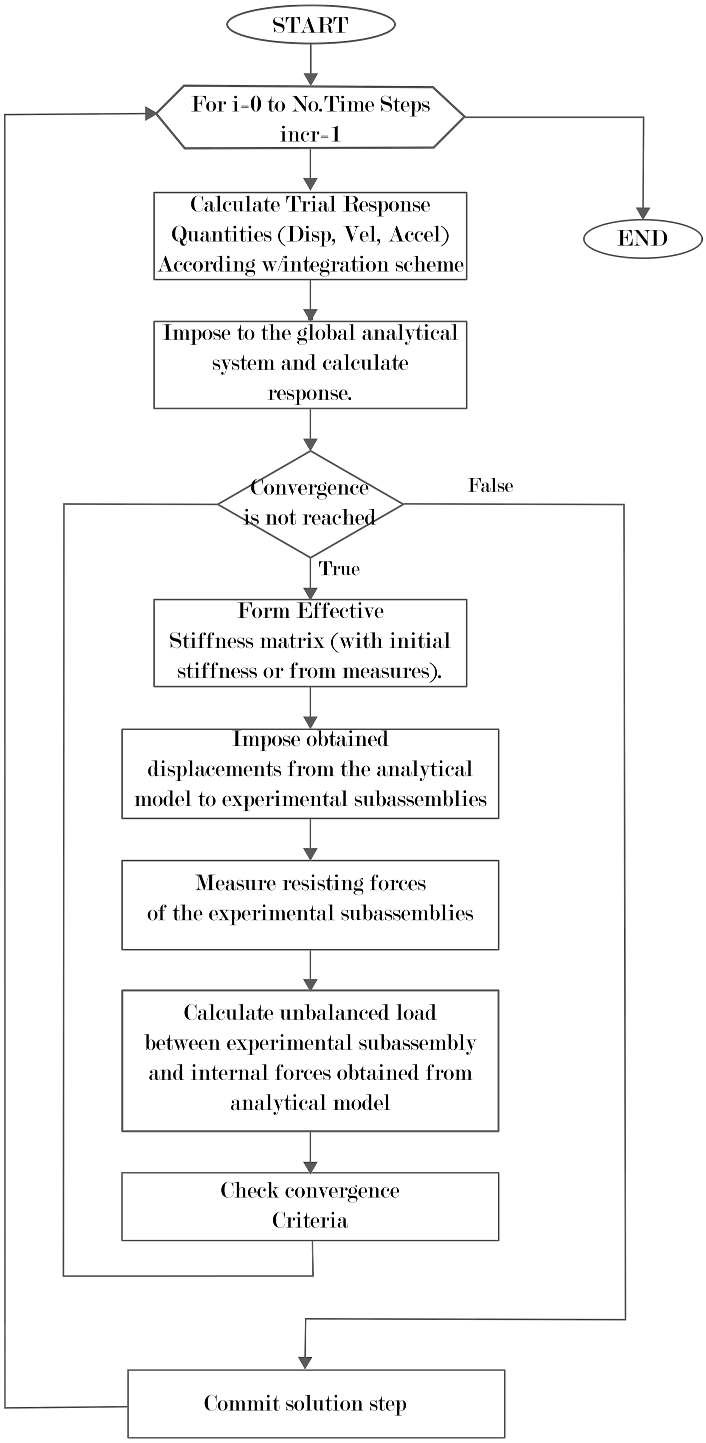


Figure 3.11: Classic Hybrid Simulation testing procedure flowchart.

Difficulties

• Implementation of an HS is highly dependent of testing site and its data control and acquisition systems. This situation creates the necessity of a great number of software customizations to adapt different structure types and laboratory equipment.

• There are multiple errors that may occur at different stages of a hybrid simulation test, affecting the solution process: (1) modeling errors due to the discretization process, and assumptions about energy-dissipation and analysis; (2) numerical errors introduced by the integration and equilibrium solution algorithms; (3) experimental errors generated by the control and transfer systems; and (4) experimental errors introduced by the instrumentation devices and the data-acquisition system.

• The simulations that include non-linear effects in analytical sub-assemblies and inelastic experimental portions, generally require iterative integration methods, which can lead to a convergence criteria issues, making the process slow in most cases. The non-linear analysis use to make difficult or impossible real time and tests.

• The distributed simulations require communications between test sites, opening the possibility of network delaying and breakdowns.

• Force/mixed control and similitude simulations need to be investigated much more.

• For large and more complex models high performance computing, that utilizes multi-processor and parallel computing, in order to distribute the operation loads.

• Real and fast time test induce inertial forces in the experimental sub-assemblies that cannot be ignored.

3.4.1.2 OpenFresco as a Framework for Hybrid Simulation

Open source Framework for Experimental Setup and Control (OpenFresco) is an environment independent, modular, and open source software framework for deploying hybrid simulation worldwide in a robust, transparent, scalable and easily extensible fashion [31]. OpenFresco provides a series of common operations and services, necessary to implement local and globally distributed hybrid simulations.

OpenFresco is independent of the FE-software used, however for its ideal realization, the software must allow the addition of new 'user' elements, and if it's necessary for the type of hybrid simulation planned, new integration operators.

Three main, and relatively common, set of tasks are handled by OpenFresco, in order to ease implementations for computer-controlled tests. The first involves the transformation of response quantities in the numerical model at the boundary nodes, from the used by the FE-software to those used in laboratory (or sub-assemblies). These tasks are managed by the *ExperimentalSetup* class (transformations) and the *ExperimentalElement* class (Representation of the physical sub-assemblies in the mas-

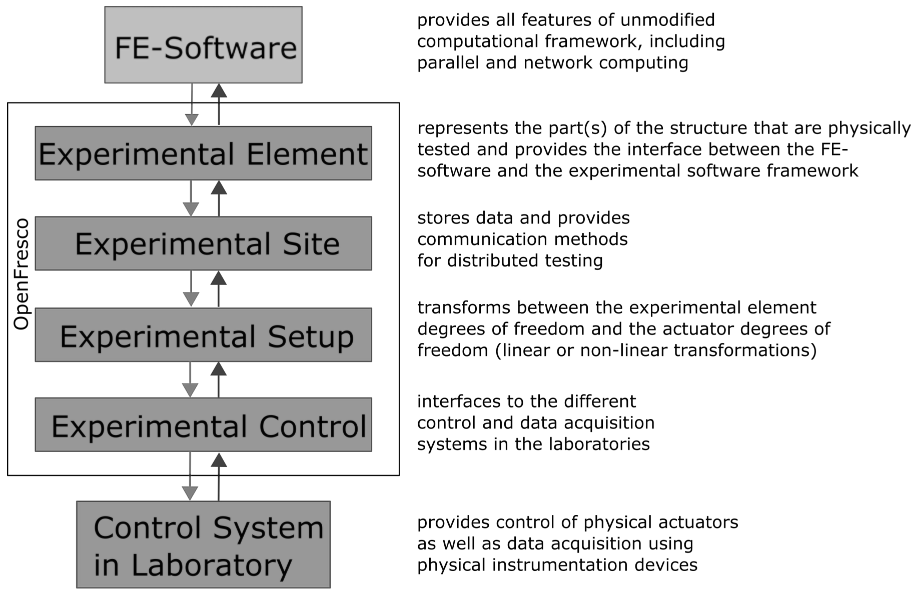


Figure 3.12: OpenFresco framework components to conduct an HS.

ter integration process). The second task is establishing communication with the laboratory control and data acquisition systems , performed by *ExperimentalControl* class. The third task, associated with geographically distributed testing, the *ExperimentalSite* class provides the services and coordination for communicating the different experimental sites (servers) and the software that integrates the complete system (as client). The organization of this classes is shown in figure 3.12.

3.4.2 Coupled Numerical Simulation

With the same philosophy of Hybrid Simulation of embed experimental portions into a numerical analysis. The FE coupled simulation is a technique where multiple numerical models are *glued* to construct a complete structural model.

This technique is suitable for the recent engineering tendencies, where the structural systems are analyzed from a multidisciplinary focus, allowing to construct a complex model, exploiting different softwares modeling capacities in a concurrent process, avoiding the data file systems.

When multiple displacement based FE codes are coupled together, according with [28], one is selected to act as a *master*, solving the complete structural system, while the other programs model and analyze multiple structural sub-assemblies, acting as *slaves*. The master program can model sub-assemblies in its own environment, but this is not a requirement for the proper function of the coupled simulation. Each slaved subassembly acts as a super-element and is connected to the master program via its interface degrees-of-freedom.

The coupled programs are communicated by a coordinator or middleware program, used to accommodate all the overhead, data storage, communication methods, system control, transformations, etc.

The theoretical scheme of the method proposed by [] is based on the penalty method, imposing the prescribed interface displacements as a constraint function in the following manner:

The Potential Energy expression:

$$\pi_P = \frac{1}{2}\int_V [u]^T[B]^T[C][B][u]dV - \int_V [u]^T[b]dV - \int_S [u]^T[t]dS \tag{3.3a}$$

Rewriting it in function of stiffness matrix and nodal forces:

$$\pi_P = \frac{1}{2}[u]^T[K][u] - [u]^T[F] \tag{3.3b}$$

To couple the slaved sub-assemblies to the master program, the displacements at the interface degrees of freedom are prescribed as constrains equations:

$$g(d) = Qd - \bar{d}_g \tag{3.3c}$$

Where Q ($\mathrm{N_{adpt}xN_{DOF}}$) and k ($\mathrm{N_{adpt}xN_{adpt}}$) matrices are defined as follows:

$$Q_{ij} = \begin{cases} 1 \quad if \quad i = j \in N_{adpt} \\ 0 \qquad\quad otherwise \end{cases} \tag{3.3d}$$

$$k_{ij} = \begin{cases} k \quad if \qquad i = j \\ 0 \qquad\quad otherwise \end{cases} \tag{3.3e}$$

Applying the penalty method to introduce the constraint equations

$$\pi_P = \frac{1}{2}[u]^T[K][u] - [u]^T[F] + \frac{1}{2}g^T kg \tag{3.3f}$$

Deriving with respect to displacements **d** and equating to zero, in order to minimize:

$$\frac{\pi_P}{\partial d} = Kd_k - F + Q^T kg(d_k) = Kd_k - F + Q^T k(Qd_k - \bar{d}_g) = 0 \tag{3.3g}$$

Equivalently:

$$[K + Q^T kQ]d_k = F + Q^T k\bar{d}_g \quad (3.3h)$$

$$Kd_k - F = Q^T [K\bar{d}_g - kQd_k] \quad (3.3i)$$

This last expression can be only satisfied if the imposed displacements by the sub-assemblies are equal to the reported by the master program, this condition must be satisfied while the internal equilibrium of each sub-assembly is satisfied too, therefore all the components of the system are in force equilibrium and displacement compatibility.

The general operation process, starting on the side of the master program, the super-elements (represent the portions modeled in the slave program) receive the vector of global displacements from the master integration numerical method. Later, it sends these displacements (via TCP/IP socket) to the OpenFresco simulation application server. The experimental site and setup modules are responsible of storing and transforming the received quantities if it's necessary. Then, the trial displacements are next passed to the "SimFEAdapter" experimental control that provides the connection to the adapter element (utilizing a TCP/IP socket). Next, the adapter element combines the received displacements with its own, and impose them in the slave program. Once the equilibrium and convergence has been reached, the force vector is returned to the SimFEAdapter experimental control (across the TCP/IP socket). The experimental site and setup are again responsible of storing and transforming the response quantities. After that, the simulation application server returns the force vector through the TCP/IP socket to the super-element in the master program. Finally, the super-element saves them as element forces and returns them to the master integration method, which is capable to determine the new trial displacements and proceed to the next time step.

It is important to notice that both the super-element in the master program and the adapter element(s) in the slave program(s) must be implemented as user-defined elements into each of their published programming interfaces. These elements are the only necessary implementations to carry a coupled analysis utilizing OpenFresco as middle-ware.

Chapter 4

Validation Tests and Method Proposed

4.1 Element Coupling Tests

4.1.1 Beam-Plane Solid Transition Element

According with the ideas of section 3.3.1, a plane solid-frame transition element has been formulated, this procedure and the results of models containing this element under diverse loading conditions are presented below.

4.1.1.1 Formulation

The next element formulation corresponds to a plane stress solid to frame transition, for this procedure, suppose a classic 4-node plane element, that will be degenerated to obtain the element shown in figure 4.1.

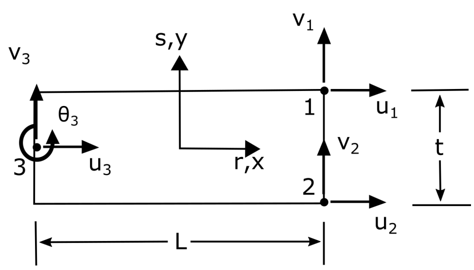


Figure 4.1: Plane frame-solid continuum transition element degrees of freedom and measures.

The nodal displacement vector for is defined as:

$$\mathbf{u^T} = \begin{bmatrix} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 & \theta_3 \end{bmatrix} \tag{4.1a}$$

The interpolation functions and matrix are:

$$h_1 = \frac{1}{4}(1+r)(1+s), \;\; h_2 = \frac{1}{4}(1+r)(1-s), \;\; h_3 = \frac{1}{3}(1+r) \tag{4.1b}$$

$$\mathbf{H} = \begin{bmatrix} h_1 & 0 & h_2 & 0 & h_3 & 0 & -\frac{t}{2}sh_3 \\ 0 & h_1 & 0 & h_2 & 0 & h_3 & 0 \end{bmatrix} \tag{4.1c}$$

Then, the displacement field can be expressed by the product of the matrix **H** and the vector **u**. It's necessary to introduce a Jacobian operator that relates the natural and local coordinates of the element:

$$\frac{\partial}{\partial \mathbf{x}} = \mathbf{J^{-1}} \frac{\partial}{\partial \mathbf{r}} \tag{4.1d}$$

$$\mathbf{J} = \begin{bmatrix} L/2 & 0 \\ 0 & t/2 \end{bmatrix}, \; \mathbf{J^{-1}} = \begin{bmatrix} 2/L & 0 \\ 0 & 2/t \end{bmatrix} \tag{4.1e}$$

Additionally, the differential operator for plane problems is:

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \tag{4.1f}$$

Finally, the strain-displacement matrix is obtained from the previous expressions:

$$\mathbf{B} = \mathbf{LH} \tag{4.1g}$$

$$\mathbf{B} = \begin{bmatrix} \frac{(1+s)}{2L} & 0 & \frac{(1-s)}{2L} & 0 & -1/L & 0 & \frac{t}{2L}s \\ 0 & \frac{(1+r)}{2t} & 0 & -\frac{(1+r)}{2t} & 0 & 0 & 0 \\ \frac{(1+r)}{2t} & \frac{(1+s)}{2L} & -\frac{(1+r)}{2t} & \frac{(1-s)}{2L} & 0 & -1/L & -\frac{(1-r)}{2} \end{bmatrix} \tag{4.1h}$$

To calculate the stiffness matrix, the constitutive relation is introduced and the product is integrated numerically:

$$\mathbf{K} = \int_V \mathbf{B^T} \mathbf{C} \mathbf{B} dV = b \sum_{i=1}^{m} \sum_{j=1}^{n} \mathbf{B^T}(x_i, y_j) \mathbf{C} \mathbf{B}(x_i, y_j) \tag{4.1i}$$

4.1.1.2 Axial Load

The first analysis model for the transition element is a displacement imposition, simulating a longitudinal strain condition in a bar. The geometry of the model is expressed in figure 4.2 , and only consist in a straight frame with a constant depth of 10 cm, unitary base and a variable length. The supposed material is structural steel, with a Young's modulus E =2x10^6 kgf/cm^2 and a Poisson's modulus ν of 0 or 0.3, for different tests.

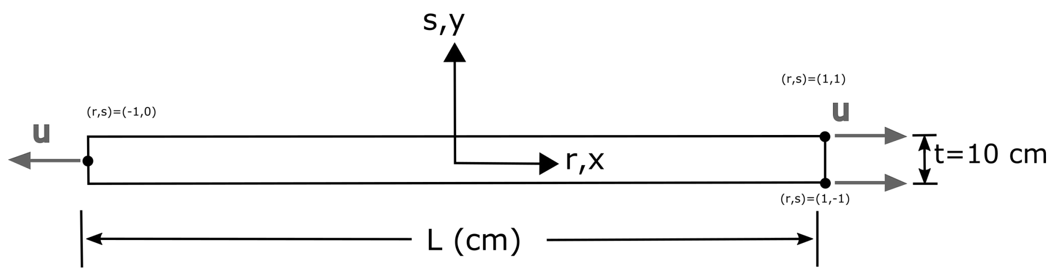


Figure 4.2: Plane transition element dimensions, with an unitary base. Subjected to longitudinal displacements in its corner nodes.

Parameter	Test 1	Test 2	Test 3	Test 4
ν	0	0.3	0	0.3
$L(cm)$	150	150	1.5	1.5
$\Delta u\ (cm/cm)$	2	2	0.02	0.02
ϵ_{11}	0.0133	0.0133	0.0133	0.0133
ϵ_{33}	0	0	0	0
ϵ_{13}	0	0	0	0
$\sigma_{11}\ (kgf/cm^2)$	$2.667x10^4$	$2.446x10^4$	$2.667x10^4$	$2.446x10^4$
$\sigma_{33}\ (kgf/cm^2)$	0	$7.339x10^3$	0	$7.339x10^3$
$\sigma_{13}\ (kgf/cm^2)$	0	0	0	0

Table 4.1: Analysis results from longitudinal strain tests in transition elements.

The reported results in table 4.1 resemble to the obtained from a classic Bernoulli's beam in tests 1 and 3, in which a 0 was assigned to Poisson's moduli. For the case 2 and 4, a stress component, normal to the 'neutral axis' of the frame, appear and is associated to the Poisson effect (the displacement imposition do not allows the deformation in direction y, producing the stress component σ_{33}).

4.1.1.3 Bending Load

The validation test for the bending load case is a displacement imposition, the constrained DOFs are the rotational and vertical ones in the Node 3. The geometry of this model is the same of the axial load case, the difference consists in the boundary conditions, that are, full displacement restrain at nodes 1 and 2, and two imposed displacements at 3, which are described in figure 4.3.

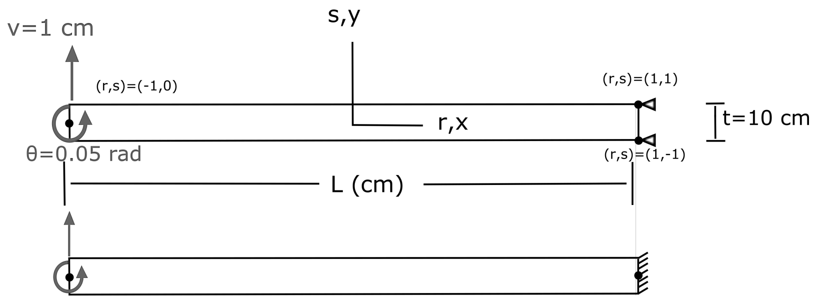


Figure 4.3: Displacement imposition configuration for the bending test and internal forces diagrams for its analogue frame (bending moment and constant shear).

Parameter	Test 1	Test 2
ν	0	0.3
$L(cm)$	150	150
$u\ (cm)$	0	0
$v\ (cm)$	1	1
$\theta\ (RAD)$	0.05	0.05
ϵ_{11}	$0.001666s$	$0.001666s$
ϵ_{33}	0	0
ϵ_{13}	$0.025r - 0.03166$	$0.02r - 0.03166$
$\sigma_{11}\ (kgf/cm^2)$	$3333.34s$	$3058.1s$
$\sigma_{33}\ (kgf/cm^2)$	0	$917s$
$\sigma_{13}\ (kgf/cm^2)$	$25000.0r - 31666.7$	$16055r - 20336$

Table 4.2: Analysis results from bending tests in transition elements.

The reported values in table 4.2 cannot be compared directly with the analogue frame results, since the finite elements average the structural behavior in their interior. To establish a comparison, the bending moment average for the analogue beam is employed $\bar{M} = 55555.55\ kg \cdot cm$ (the bending moment equation is $M(x) = 263779.3 - 2776.32x$), and the constant shear force $V = 2776.3\ kg$. Applying the principles of resistance of materials, the normal stress is determined by $\sigma = \frac{M*t/2}{1*t^3/12}$ resulting equal to 3333.33 kg/cm^2, which is very similar to the obtained in test 1. While the maximum shear stress for the beam may be determined with $\tau_{max} = \frac{3V}{2A} = \frac{3V}{2(1)(t)}$, substituting the measures and forces, the resulting valor is 416.5 kg/cm^2. This last value shows that the distribution and magnitude of the shear stress in transition element may be very different to the assumed in beam theory.

4.1.1.4 Fixed beam

This last test corresponds to a fixed beam subjected to distributed uniform load, in order to monitor the behavior of short transition elements as part of a small structural system. The geometry of the model is expressed in figure 4.4, the total length of the beam is 150 cm, divided in two beams of 74 cm and two central transition elements of length equal to 1 cm, the depth of the beam is 10 cm , and unitary base. The material keeps linear-elastic with a Young's modulus E =2x10^6 kgf/cm^2 and a Poisson's modulus ν of 0.3 The applied load magnitudes are $\omega = 15\, kgf/cm$s and $P = 15\, kgf$

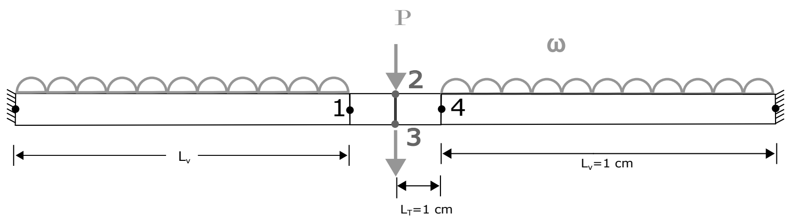


Figure 4.4: Fixed beam with an applied uniform distributed load, modeled with Bernoulli and transition elements at its center.

Parameter	Transition	Bernoulli
$u_1\,(cm)$	0	0
$v_1\,(cm)$	-0.1186	-0.11927
$\theta_1\,(RAD)$	-8.40E-05	-8.44E-05
$u_{2,3}\,(cm)$	0	0
$v_{2,3}\,(cm)$	-0.11864	-0.11931
$u_4\,(cm)$	0	0
$v_4\,(cm)$	-0.1186	-0.11927
$\theta_4\,(RAD)$	8.40E-05	8.44E-05

Table 4.3: Comparison of transition model with analytical solution of a fixed beam.

Parameter	Transition	Bernoulli
σ_{11} (kgf/cm^2)	$-839.6s$	$-/+843.75(max)$
σ_{33} (kgf/cm^2)	0	0
σ_{13} (kgf/cm^2)	$-41.98r+1.98$	$168.75(max)$

Table 4.4: Stress state in the center of span for a fixed beam modeled with transition elements and the analytical solution.

This last analysis results show that the introduction of transition elements do not affect significantly the general behavior of structures. And corroborate that the normal stress is described adequately in the transition, while the shear distribution maintains different to the expected (solution based in Euler-Bernoulli theory).

4.1.1.5 Evaluation.

The transition elements offer a solution to couple different structural systems (e.g frame-wall) and levels of approximation in a defined region, maintaining the expected behavior of the general model and the neighborhood of the transition. Nevertheless the issues with their application are evident:

• Inside their domain, the normal (longitudinal) stress is described adequately, but the shear distribution is questionable.

• The introduction of the transition elements difficult the result interpretation.

• To be used, each transition type must be formulated and added to the finite element software as a user routine.

• When a transition element is introduced in a structure imposes certain mesh conditions, requiring a mesh transition after each of these elements.

4.1.2 Dissimilar Elements

In what follows, the concepts discussed in section 3.3.2 are applied in a reticular structure,in order to impose the kinematic conditions of the Euler-Bernoulli beam theory on selected portions of the structure.

The test model is a one bay frame, with a span of 500 cm, measured from the center of the columns, height of 300 cm, the section of columns is constant with 20 cm in the base, and 30 cm of depth. The material is assumed linear-elastic with a Young's modulus E

$=2\text{x}10^5$ kgf/cm^2 and a Poisson's modulus ν of 0.18, volumetric weight $\gamma = 2400\, kg/m^3$, these properties magnitudes were selected to resemble concrete.

The proposed configurations to impose the conditions of frames on the solid elements, are 2. The first consists in placing orthogonal bar elements at the coupling zone, these elements have an Elasticity Modulus much higher than the rest of the frames, around 1000 times, the modulus must be high enough to impose the frame conditions on the plane stress elements, but not excessively, to maintain the stiffness matrix well conditioned. In the second proposal the frame is extended and embedded into the plane stress elements, this arrangement equilibrate de frame bending moment through the solid elements resisting forces.

Figure 4.5: Coupling application on a region of a plane frame.

4.1.2.1 Static Case

This load case corresponds to a self weight analysis ($\omega = 1.44\, kgf/cm$) and overload with value $\omega = 2\, kgf/cm$, assigned a uniform distributed load. The three models analyzed are the following:

A)Only frame elements
B)Frame with perpendicular frames in the coupling surfaces
C)Embedded frames in the plane stress elements.

The internal forces, displacements and stresses are monitored, principally, near of the coupling zone, to verify if there is no excessive gap between the two different types of elements related.

The results of table 4.5 show that both proposals to introduce plane stress elements into the model, describe the general structural behavior in a very similar manner, and

	Model A	Model B	Model C
$M_4\,(kgf-cm)$	24472	26098	23932
$M_{32}\,(kgf-cm)$	42890	41732	42283
$u_4\,(cm)$	$-7.22x10^{-4}$	0.00359	$-7.22x10^{-4}$
$v_4\,(cm)$	-0.0245	-0.02269	-0.02265
$\theta_4\,(RAD)$	$6.26x10^{-4}$	$6.00x10^{-4}$	$6.39x10^{-4}$
$u_{32}\,(cm)$	-0.01404	-0.01145	-0.01657
$v_{32}\,(cm)$	-0.00158	-0.00158	-0.00157
$\theta_{32}\,(RAD)$	$2.38x10^{-4}$	$2.65x10^{-4}$	$2.19x10^{-4}$

Table 4.5: Reported displacements and moments from the vertical load analysis for different models of the one bay frame.

in concordance with the obtained from the model C, this results represent the coupling capacity of integrating the detailed model forces to equilibrate the coarse ones at the boundary.

	Model A	Model B	Model C
$\sigma_{11}@4\,(kgf/cm^2)$	$+/-8.15$	$8.42/-8.52$	$2.81/-2.06$
$\sigma_{33}@4\,(kgf/cm^2)$	0	$1.32/-1.38$	$3.29/-2.70$
$\sigma_{13}@4\,(kgf/cm^2)$	1.7	$0.65/0.45\,,1.73(MAX)$	$2.27/1.88\,,8.74(MAX)$
$\sigma_{11}@32\,(kgf/cm^2)$	$+/-14.30$	$14.11/-15.98$	$4.24/-4.13$
$\sigma_{33}@32\,(kgf/cm^2)$	0	$2.07/-2.43$	$4.93/-5.21$
$\sigma_{13}@32\,(kgf/cm^2)$	$0.7(MAX)$	$0.01/0.29\,,0.93(MAX)$	$3.35/3.51\,,11.99(MAX)$

Table 4.6: Reported stresses in extreme fibers of the coupling surface from the vertical load analysis for different models of the one bay frame.

The results, expressed in figure 4.6 and table 4.6, show that the best coupling option is the used in model B, since there is not much gap between the reported stress fields by the solid plane elements mesh and the calculated with beam theory. The proposal of Model C presents an excessive stress gap in the first two rows, depth of the embedded frames, but after these rows, the reported stress fields are very similar to the obtained in Model B, thus, is applicable if the analyst accepts loosing that portion of the results.

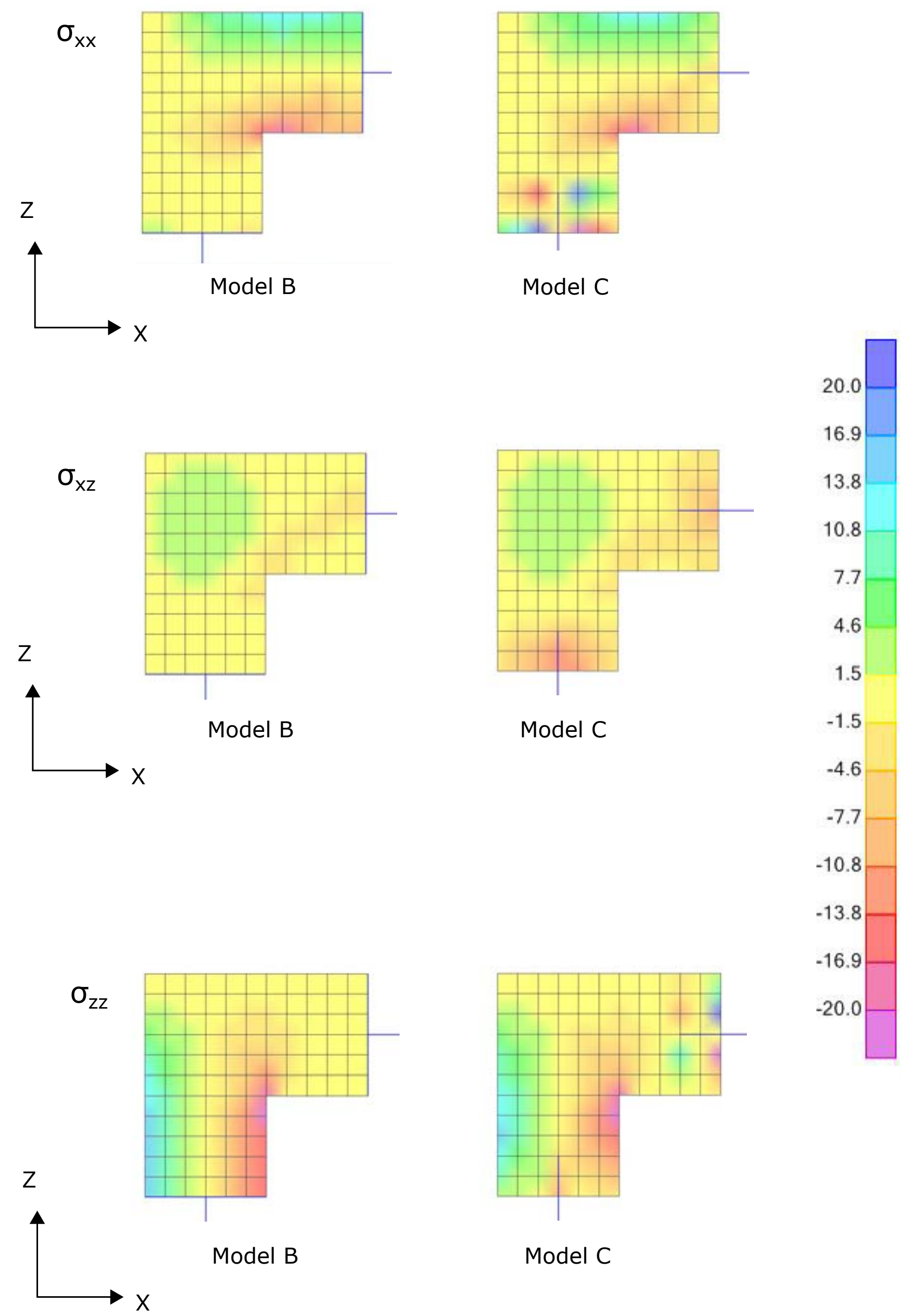


Figure 4.6: Stress field representations in the solid plane elements, for the static load case

4.1.2.2 Dynamic Case

For this test, the frame properties and measures remain the same of static analysis. Model B presents better response under the static load case than Model C, for this reason was selected for the dynamic test. The load source to perform this seismic analysis, is the earthquake of year 1985, registered in Mexico City, component East-West, "SCT" station. The considerations for analysis are concentrated mass, damping ratio of 5% and Newmark integration scheme ($\beta = 1/4$).

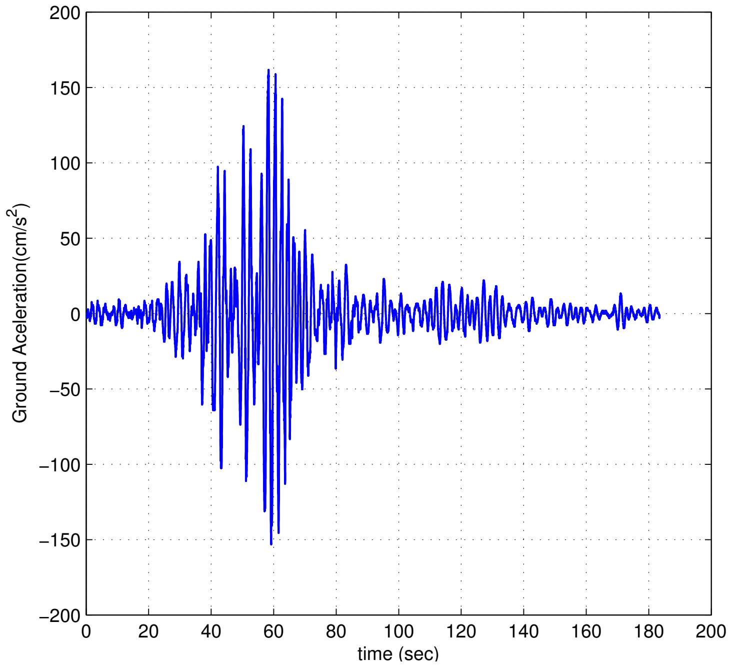
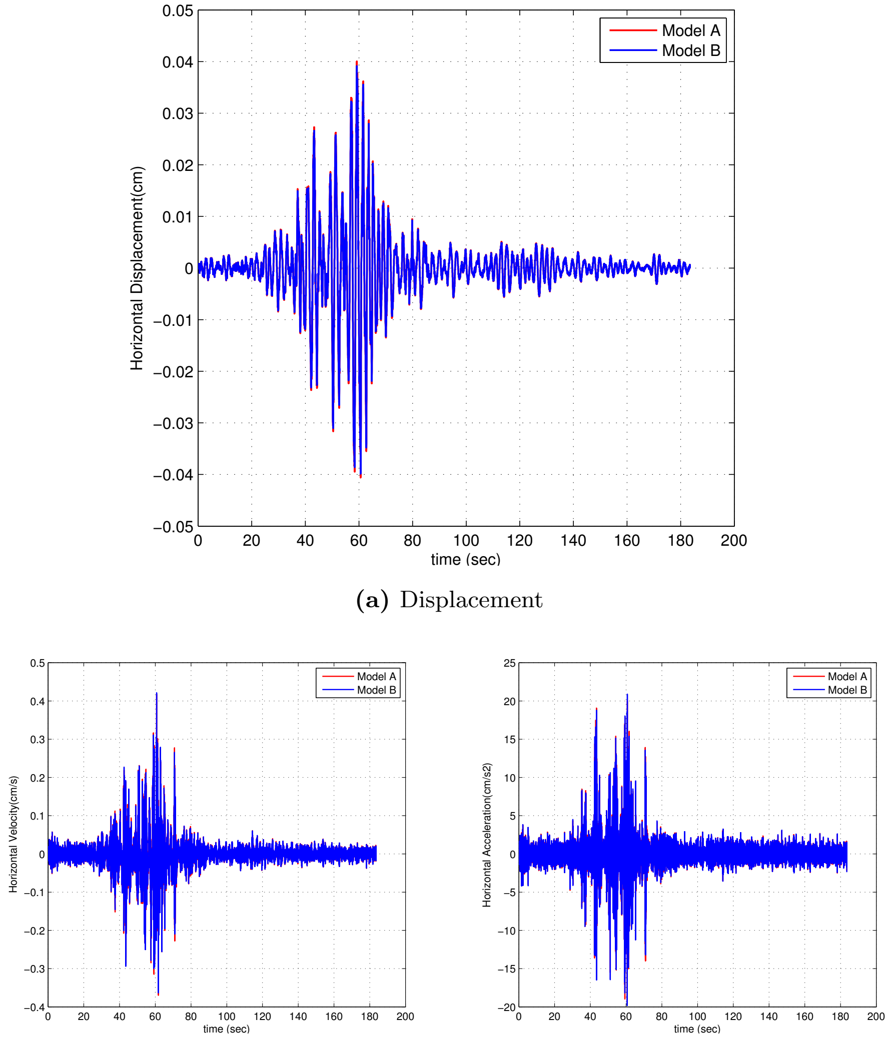


Figure 4.7: Accelerogram of Mexico City earthquake, 1985, component East-West, registered in SCT station.

Performing an Eigen-Analysis, the first 2 vibration periods are $T_1 = 0.09815 : \ sec$, $T_2 = 0.01857\ sec$ for Model A, and $T_1 = 0.09712\ sec$, $T_2 = 0.01812\ sec$ for Model B, the first modal shape is lateral displacement at top, the second corresponds to vertical displacement in the center of beam span.

The results in figures 4.8a, 4.8b and 4.8c show that the structural response under dynamic excitation does not vary importantly by the introduction of lower scale components in the model, it's important to notice that the plane stress elements do not have assigned mass, reducing drastically the quantity of eigen-values.



(a) Displacement

(b) Velocity

(c) Acceleration

Figure 4.8: Horizontal response of the coupled one bay frame under dynamic excitation, reported at Nodes 2 and 4.

4.1.2.3 Evaluation

The main task in this coupling proposal, for plane stress and frame elements, is establishing configuration(s) and properties of elements used for constraints imposition, achieving a correct transmission of effects between the related models, despite of this, it has shown multiple benefits:

- It is a functional and relatively easy to use proposal.

• Does not require the implementation of any special element.

• The force integration of the plane elements to equilibrate the frames is realized without additional operations.

• The introduction of detail zones with this coupling doesn't affect the global results, providing detailed and reliable information.

4.2 Numerical Coupling Tests

This section contains coupled analysis examples, where two structures, modeled in OpenSees, are connected using OpenFresco as middle-ware, one acting as a master and the other as slave, according to the section 3.4.2. The analysis objects specified in each of these components are:

Master:
• Corresponding part of the model (mesh, elements, loads, masses, etc).
• Since it's OpenSees, calls directly *OpenFresco processes.*
• *Experimental Control*: Control the data transmission between the master and slave process. The IP Address and Port are defined here.
• *Experimental Setup*: Specified transformations required to relate the master DOFs with the slave DOFs.
• *Experimental Site*: All the sites involved in the simulation are declared here.
• *Experimental Element*: Substitutes the portion modeled in the slave program, requires an initial stiffness matrix for the first analysis step.
• Master integration method: Dynamic or static, depends of the problem (Solution scheme, algorithm, convergence criteria, etc)

Middle-ware: Carries the communication and processes to run the simulation, but it is called from the master program, so it is not necessary to do specifications from its own interface.

Slave:
• Corresponding part of the model (mesh, elements, loads, etc).
• Adapter element that establishes communication with the middle-ware through the interface DOFs, the stiffness matrix ,that imposes the constraints between systems, k_{ij} and the IP port of the experimental control are introduced here.
• Local integration method: For the presented examples static load or displacement control (Solution scheme, algorithm, convergence criteria, etc)

4.2.1 Beam-Column

The structural model consists in a two bar elastic frame, one vertical and horizontal, fixed in both supports, according to figure 4.9, the elements assigned to the slave are in gray color, and those of the master analysis in black. The properties of the elements are unitary, i.e. Young Modulus $E = 1Pa$, Area $A = 1m^2$, Inertia $I = 1m^4$, mass $m = 1kg$.

The configuration of controls for the coupled simulation is: "SimFEAdapter" experimental control communicates with the adapter element in the slave program, "NoTransformation" experimental setup is used due the DOFs and reference systems in the slave and master processes are the same, "Experimental Site" to indicate that the slave portion of structure is modeled locally and finally a "Generic Experimental Element" with 1 node and 3 DOFs, that represents the horizontal bar, is located at the free node. On the side of the slave, the stiffness assigned to the adapter element is a diagonal 3x3 matrix with values $1x10^4$.

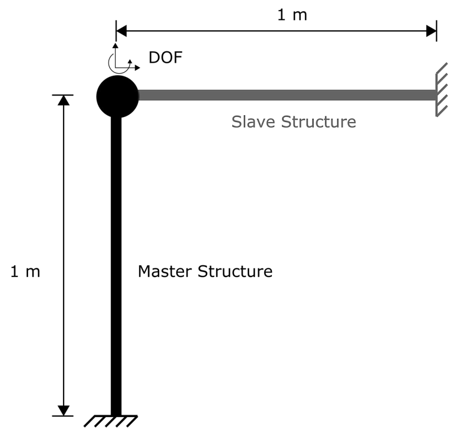


Figure 4.9: Beam-Column structural model for numerical coupled analysis.

For the experimental element, an initial stiffness matrix is required, obtained from an unit displacement imposition at one interface DOF at time while restraining the rest of interface DOFs, the determined stiffness matrix for this slave structure is:

$$K_{init} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 12 & 6 \\ 0 & 6 & 4 \end{bmatrix}$$

4.2.1.1 Static Load

The first test for the coupled analysis is a static one, a unit horizontal force is applied at the free node, since is a linear-elastic analysis the load may be applied in 1 or n load steps, obtaining the same results expressed in eqs. 4.2.c.

The contributions of the stiffness matrix by the slave and master portions are:

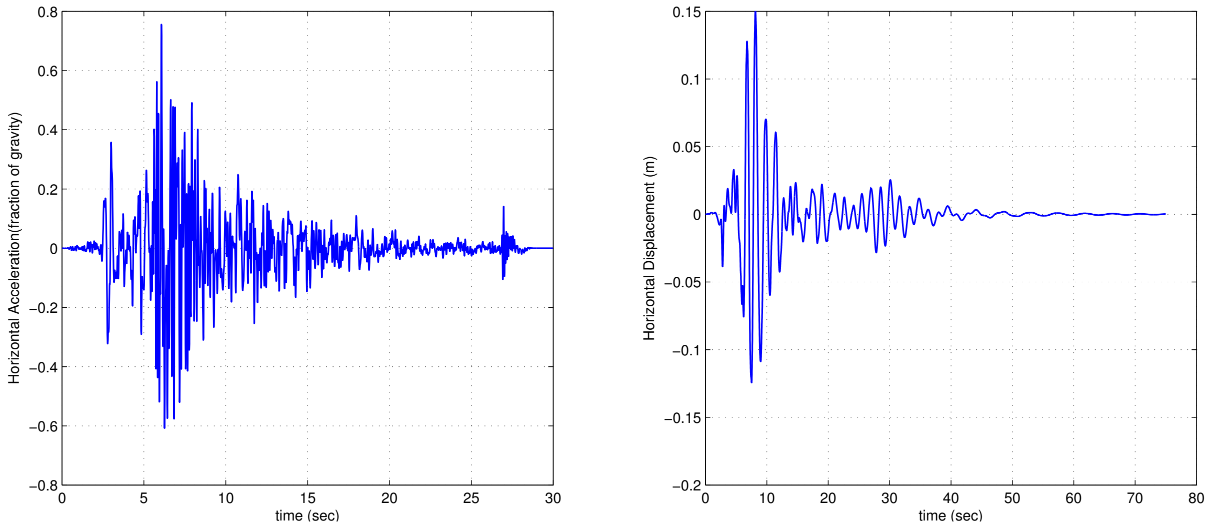
$$K_{slave} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 12 & 6 \\ 0 & 6 & 4 \end{bmatrix}, K_{master} = \begin{bmatrix} 12 & 0 & 6 \\ 0 & 1 & 0 \\ 6 & 0 & 4 \end{bmatrix}, K_{total} = \begin{bmatrix} 13 & 0 & 6 \\ 0 & 13 & 6 \\ 6 & 6 & 8 \end{bmatrix} \tag{4.2a}$$

The solution of the system under the applied load is:

$$\mathbf{U} = k_{total}^{-1} \begin{bmatrix} 1.0\,N \\ 0.0\,N \\ 0.0\,N*m \end{bmatrix}, \mathbf{U} = k_{total}^{-1}\boldsymbol{F} = \begin{bmatrix} 0.164\,cm \\ 0.0865\,cm \\ -0.1875\,RAD \end{bmatrix} \tag{4.2b}$$

4.2.1.2 Dynamic Load

The model is subjected to the SACN01 ground motion of the SAC steel project, figure 4.10a. The considerations of the analysis are concentrated mass in horizontal and rotational DOFs, Newmark integration scheme ($\beta = 1/4$) and damping ratio of 5%.



(a) Input excitation at base. **(b)** Horizontal Displacement Response at free node.

Figure 4.10: Base acceleration input for the Beam-Column numerically coupled analysis and its horizontal displacement response .

The analysis of a structure containing the whole model and the coupled, analyzed with the master-slave process, report the same results, shown in figure 4.10b.

4.2.2 Cantilever

This model consists in a simple cantilever column with concentrated mass at its top, as may be seen in figure 4.11, the gray elements are modeled in the slave and the black ones in the master, all the properties and dimensions may consulted in [29].

For the experimental element, an initial stiffness matrix is required, in this case the global DOFs do not coincide with the element local DOFs, then the local stiffness matrix is specified:

$$K_{init} = \begin{bmatrix} 1213 & 0 & 0 \\ 0 & 11.2 & -302.4 \\ 0 & -302.4 & 10886.4 \end{bmatrix}$$

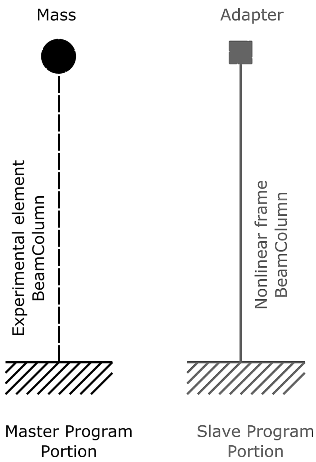
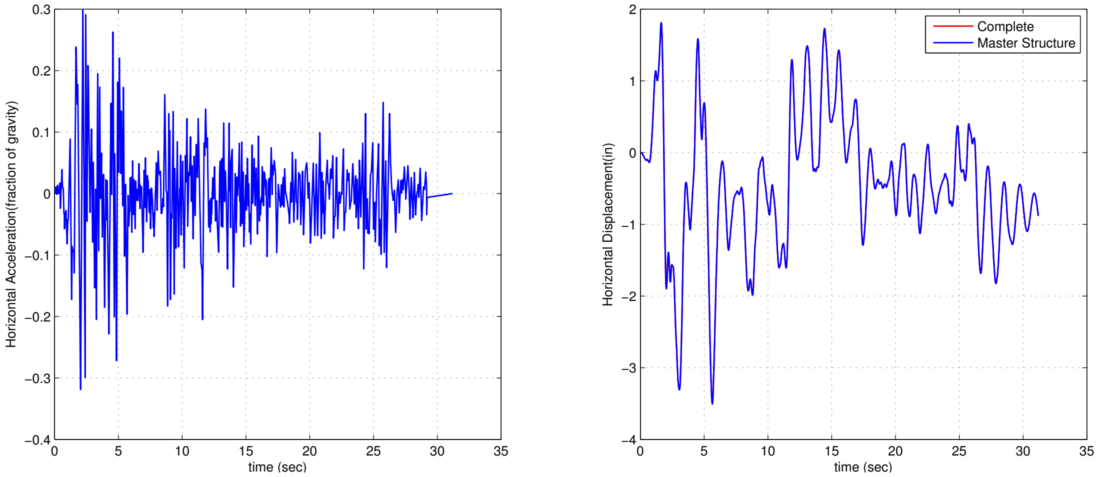


Figure 4.11: Cantilever structural model for numerical coupled analysis.

The configuration of controls for the coupled simulation is: "SimFEAdapter" experimental control communicates with the adapter element in the slave program, "OneActuator" experimental setup is used to impose conditions in only the horizontal direction of the slave, "LocalSite" Experimental Site to indicate that the slave portion of structure is modeled locally and finally a "Beamcolumn" Experimental Element, defining a two node frame with 3 DOFs at its top. On the side of the slave, the stiffness assigned to the adapter element is a diagonal 1x1 matrix with value $1x10^5$.

4.2.2.1 Dynamic Load

Based on this coupled example presented, a complete model was created for comparison. Both models were subjected to the ground motion recorded at a site in El Centro, California during the Imperial Valley earthquake of May 18, 1940, figure 4.12a. The considerations of the analysis are concentrated mass in horizontal and vertical DOFs, Newmark integration scheme ($\beta = 1/4$) and damping ratio of 8.5%.



(a) Input excitation at base.

(b) Horizontal displacement comparison.

Figure 4.12: Base acceleration input for the Cantilever numerically coupled analysis and its horizontal displacement response .

The results obtained are practically the same for both structural models, as example, the horizontal displacement comparison is shown in figure 4.12b and may be seen that there isn't an appreciable difference.

4.3 Analysis Method

Based on the satisfactory results of sections 4.1 and 4.2 a method is proposed for multi-scale plane frames by sub-structuring them with the concepts of hybrid simulation in order to involve much less DOFs in the global analysis, but with the detail level of a refined model in the selected portions. The necessary steps to apply this proposal are the following:

1.- Implementation of the generic experimental element in the FE-software that acts as master, if is OpenSees this is not necessary.

2.- Implementation of the adapter element in the slave program, responsible of imposing the boundary conditions in the sub-assemblies and recollecting their response.

3.- Selection of the detail portions of the model, this may be done with a static non-linear analysis or all the nodal zones may be chosen, otherwise.

4.- Each of the previously selected portions are modeled with a refined mesh of finite elements, in order to reduce the DOFs that are related with the global analysis model in the master program, the internal forces of a group of plane stress elements may be reduced to 3 per node using an element coupling technique, an relatively easy one is introducing orthogonal frame bars in the interface between the plane solid stress elements and the frame bar elements, imposing the plane section hypothesis in the plane stress mesh, and integrating their forces to equilibrate with the frame ones.

5.- The stiffness matrix of a sub-assembly is calculated by restraining its interface DOFs and imposing unit displacement one-at-time, these monitored reactions are the elements that integrate the stiffness matrix.

6.- Construct the global analysis model for the master program, substituting the portions represented by the sub-assemblies with the elements of step 1, experimental generic elements, that contain the initial stiffness matrices.

7.- Construct the local analysis models for the slave programs, placing the adapter elements at the boundary nodes of the model, these are responsible of data recollection, transfer and impositions, The adapter elements must have much higher values of stiffness at the principal diagonal than the rest of the sub-assembly. While the integration method and analysis options must be compatible with the master process, a static analysis may be used to simulate the condition of a pseudo-dynamic test.

Chapter 5

Application Examples and Comparative

For both test that are presented in what follows the master and slave program are OpenSees+OpenSees coupled via OpenFresco.

5.1 One Bay-One Story Frame

The One-bay frame studied in section 4.1.2 is analyzed under dynamic loads with the analysis method proposed. The properties of the model and input acceleration at base are the same.

Since both master and slave program are OpenSees, the first two steps are complete. The selected portion of the model to detail, is the left nodal zone of the first level, modeled with a plane stress mesh of finite elements, coupled by rigid bars that transmit and distribute the forces of the adjacent frames.

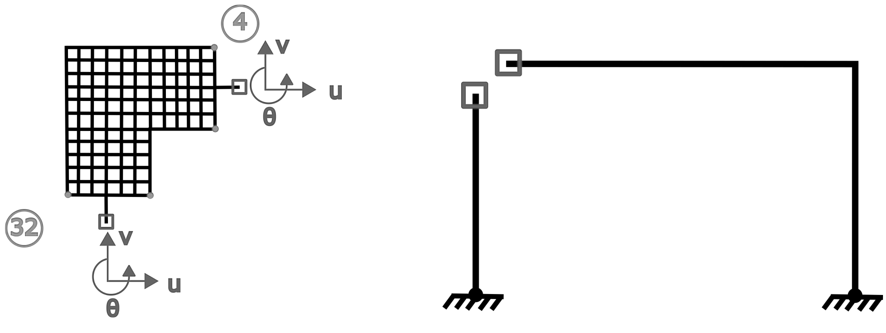


Figure 5.1: Detailed portion of the model assigned to the slave program and its interface degrees of freedom.

The master program contains the global analysis "SimFEAdapter" experimental control communicates with the adapter element in the slave program, "NoTransformation" experimental setup is used due the DOFs and reference systems in the slave and master processes are the same, "Experimental Site" to indicate that the slave portion of structure is modeled locally and finally a "Generic Experimental Element" with 2 nodes and 6 DOFs (3 per node), that represents the super-element .

$$K_{init} =$$
$$\begin{bmatrix} 420'046 & 242'157 & -2.28678x10^6 & -420'046 & -242'157 & 1.11812x10^7 \\ 242'157 & 420'046 & -1.11812x10^7 & -242'157 & -420'046 & 2.28677x10^6 \\ -2.28678x10^6 & -1.11812x10^7 & 4.59477x10^8 & 2.28678x10^6 & 1.11812x10^7 & -1.47538x10^7 \\ -420'046 & -242'157 & 2.28678x10^6 & 420'046 & 242'157 & -1.11812x10^7 \\ -242'157 & -420'046 & 1.11812x10^7 & 242'157 & 420046 & -2.28677x10^6 \\ 1.11812x10^7 & 2.28678x10^6 & -1.47538x10^7 & -1.11812x10^7 & -2.28677x10^6 & 4.59477x10^8 \end{bmatrix}$$

On the side of the slave program, an adapter element is introduced to impose the master structure displacements as boundary conditions of the local scale of analysis, a diagonal 6x6 matrix with values $1x10^{11}$ is assigned to adapter. The analysis is set as static, since the adapter element only transfer displacements.

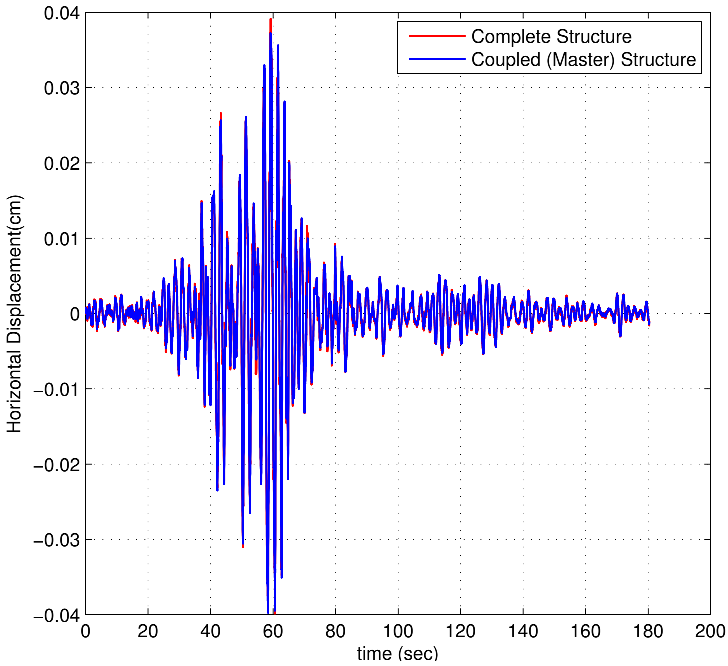


Figure 5.2: Comparison between the complete and multi-scale coupled structure horizontal displacements in the One Bay Frame under SCT E-W 1985 earthquake.

5.2 Two Bay-Three Story Frame

Finally, the multi-scale coupled simulation method is applied in a Steel structure to determinate its capacity curve to compare it with the obtained from a conventional non linear analysis, and a finite element model.

The structure consists in a plane 2 bay, 3 story steel frame, the first story height is 4.00 m, and 3.50 elsewhere; both spans of 4.00 m. The columns are considered I-cross-sections (W10x45) with flange width $b_f = 20.2cm$, flange thickness $t_f = 1.57cm$, total height $d = 26.0cm$ and web thickness $t_w = 0.89cm$, their cross section area is $A = 85.8cm^2$ and their inertia $I = 10323cm^4$. While the beams are considered I-cross-sections (W10x19) with flange width $b_f = 10.2cm$, flange thickness $t_f = 1.00cm$, total height $d = 26.0cm$ and web thickness $t_w = 0.64cm$, their cross section area is $A = 36.3cm^2$ and their inertia $I = 4008cm^4$. The material is structural steel, grade 50 with a yield stress $\sigma_y = 3525kgf/cm^2$, Young modulus $E = 2x10^6 kgf/cm^2$ and Poisson's ratio $\nu = 0.30$.

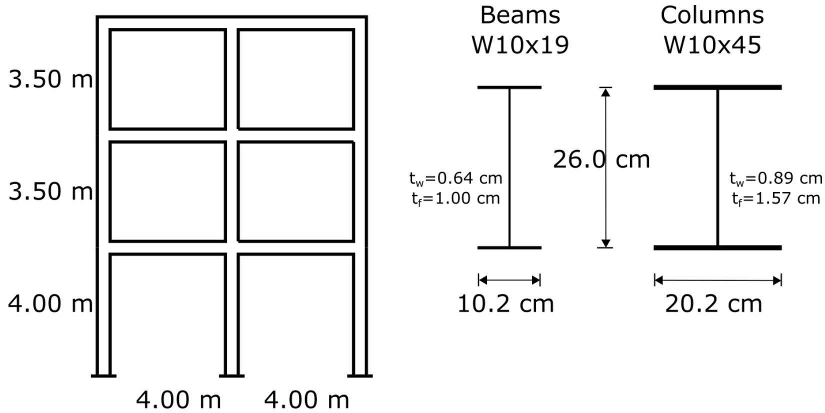


Figure 5.3: Geometry and characteristics of the two bay-three story frame example.

The structural models proposed are 3, first a fully reticular structure conformed of frame sections with a bi-linear hysteretic model to describe their non-linear behavior, the second is a multi-scale model with elastic frames and plane stress solids in the joint and base zones, and the third is a finite element model of plane stress solids.

To calculate the capacity curve a first ,vertical load, stage is realized considering distributed load of $\omega = 20kgf/cm$ in the beams of the first two levels and $\omega = 15kgf/cm$ in the third; followed by an incremental static analysis based on forces imposition, the applied lateral load pattern is proportional to the story heights, as may be seen in figure 5.5. The consideration for the non-linear behavior of the frame model is a bi-linear hysteretic relation without hardening component. While a perfect elastic-perfectly-plastic constitutive behavior (Von Misses model) is assigned to all the plain stress elements in

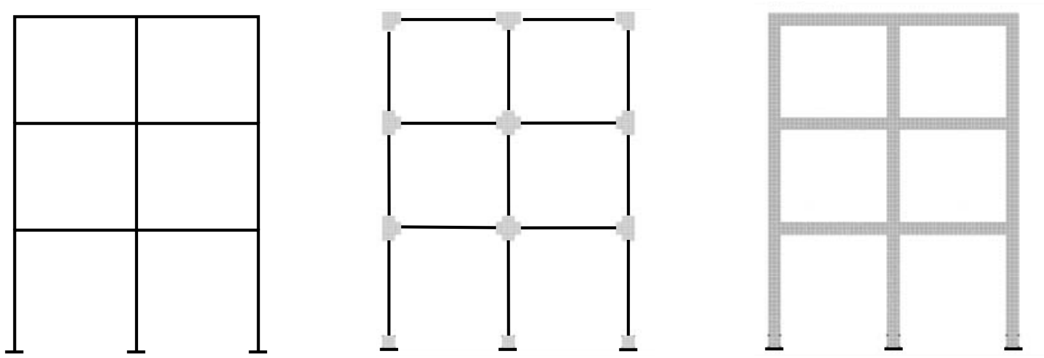


Figure 5.4: Analysis models for the two bay-three story frame example.

both, multi-scale and finite element models.

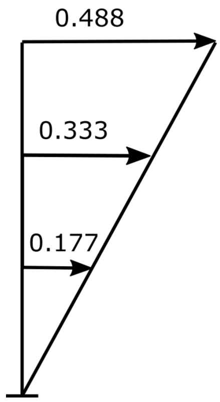


Figure 5.5: Load pattern proportions applied for the incremental static analysis applied to the models of the two bay- three story frame.

For the coupled simulation of the MS model. The master program contains the global analysis frame elements and the slaves contain the finite element models (9). This configuration requires multiple slave models, each one represents a beam-column joint or base of column. "SimFEAdapter" experimental control communicates with the adapter element, which has the same IP port direction, in the slave program. "No-Transformation" experimental setup is used due the DOFs and reference systems in the slave and master processes are the same, "Experimental Site" to indicate that the slave portion of structure is modeled locally and finally a "Generic Experimental Element" with N nodes and $3N$ DOFs (3 per node), that represents the super-element .

On the side of each slave an adapter element is included in all model boundary nodes, with a diagonal stiffness matrix of elements with 1000 times the highest value of the super-element stiffness assigned to the Generic Experimental Element in the master program.

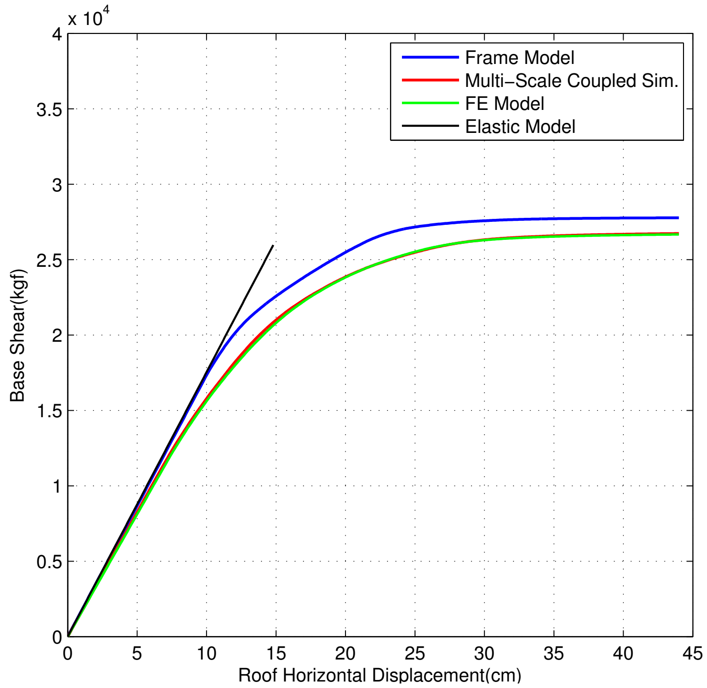


Figure 5.6: Capacity curves obtained from the incremental analysis of the three different structural models

The results of these analysis, resumed in figure 5.6, show the capacity of the proposed multi-scale method to approach the obtained results from a detailed non-linear analysis with a complex finite element model in a steel structure.

All the analyzed models start with a linear branch that express the undamaged state of the structure, and may be seen that is similar in all models (compare with a full elastic material model-black). The differences in the models become noticeable in the non-linear branch, as the multi-scale model and the full FE model lose their lateral stiffness, expressed in the slope of the tangent lines of the capacity curve (figure 5.7), smooth and gradually, this is because the non-linear effects are concentrated in the plane stress mesh in both models and each of the small finite elements incursion separately in the non-linearity. While the frame model maintains its elastic behavior during a longer interval, until the first plastic hinge appear and lost of stiffness is represented as abrupt breaks in the corresponding capacity curve.

Since the test is under load control, it may be monitored until the zero stiffness value is reached, some of the indicative values that represent the overall behavior of the structure are: the initial stiffness, the apparition of first non-linearity and ultimate base shear, values expressed in table 5.1.

Parameter	Frame Model	MS Model	FEM Model
Initial Lateral Stiffness $K(kgf/cm)$	1735	1651	1623
Damage Starting $U_{Damage}(cm)$	9.7	6.70	6.80
Ultimate Base Shear V_{Ult} (kgf)	27774	26735	26674

Table 5.1: Results from the pushover analysis of the 3 models for the two bay- three story steel frame.

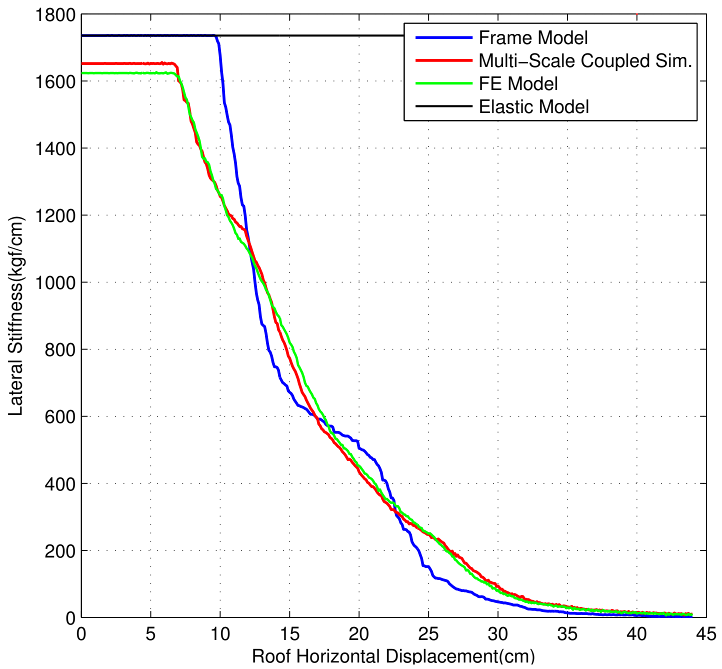


Figure 5.7: Decay of the lateral stiffness of the 3 models for the two bay- three story steel frame, during the pushover analysis.

This analysis demonstrates that a bi-linear hysteresis model in steel frames can be substituted by a coupled mesh of finite elements with a Von-Misses plasticity model, in order to simulate the damage process of a structure, allowing to introduce the required level of detail in the model and avoiding the use of an excessive quantity of finite elements, simplifying the modeling process and interpretation of results.

5.3 Evaluation of Results

The examples presented in this chapter showed the capacity of the proposed method to divide a structure in components and distribute them in multiple analysis, obtaining satisfactory results. Although, there are remarkable aspects that are exposed in what follows:

• The calculation of the initial stiffness matrix for the sub-assemblies, modeled in the slave program, might be a complex lengthy task, since all the interface DOFs must be monitored in the necessary displacement impositions (1 per interface DOF), thus, in its determination it is convenient to take advantage of symmetry and repetition, in order to ease this task.

• The selection of the penalty weights for the slave sub-assemblies is directly related to the tightness of the convergence criteria.

• The introduction of local models may act as a way to relax hypotheses in the global scale. This is shown in section 5.2, where a hysteresis model was substituted by a $J2$ plasticity model. This application is limited by the characteristics of the scale coupling and, mainly, by the available modeling features in the local scale FE code, e.g., to model a Reinforced Concrete Building for non-linear analysis, a constitutive model with different tensile and compression branches is necessary.

• The detailed portions of the global model involve different analysis considerations which may induce changes in the global behavior of the model, for this reason, the selection of the position and model properties in them is important e.g. the detail portion introduced in the example in section 5.1 induces a slight unsymmetrical behavior in the structure and does not allow the assigning of mass at internal nodes, since the slave programs conduct static analysis only.

• The geographic distribution of the calculations does not affect the results for completely numerical models, such as the presented in sections 5.1 and 5.2. However, when the dynamic experimentation of physical components is involved, time might be a critical factor as during rapid testing the load rate may inluence the dynamic properties.

Chapter 6

Conclusions

In this thesis work, a multi-scale analysis method considering the interaction between plane stress elements and frame elements has been presented. As this method is based on the principles of hybrid simulation, it has several befits as: it is capable of sub-structuring the necessary portions to obtain the grade of approximation wanted. However, it is desirable that the sub structured portions of the model have the same form and properties in order to reduce the initial stiffness calculations, necessary to initialize the master program. Multiple programs may be used in a single analysis , thanks to the common framework provided by OpenFresco software. The method is open to including physical sub-assemblies in the analysis.

The surface multi-scale coupling proposed is a feasible option, even in finite element codes with no access to source code or user elements/routines, since uses conventional frame elements to impose the kinematics of the beams into the plane stress mesh, avoiding the typically used multi-point constraints.

The proposal and validation of the method proposed was carried out in two groups of tests, one associated with the different nature element coupling, and the other with the application of the hybrid simulation to divide the structure. Obtaining that there is no excessive gaps in the studied fields (stress and displacement) near of the transition region, while the sub-structuring does not induce numerical error or instabilities. Therefore it is concluded that the goals of this investigation were fulfilled and, consequently, an initial and important step towards the development of an efficient manner to calculate the detailed behavior of structures, introducing non-linear effects from the principles of the continuum mechanics, was achieved.

6.1 Future studies

Based on the reported results in the previous chapters, the practical application potentiality of the analysis method proposed in this work suggests to continue with the following related works:

• Exploring other element coupling techniques, methods or elements feasibility, like the volumetric coupling.

• The implementation of an adapter element in other(s) finite element code(s), in order to use their unique capacities and characteristics.

• Formulation and implementation of a proper constitutive model for the representation of the non-linear behavior of reinforced concrete applicable to examples such as the presented in section 5.2.

• Implementation of an enriched finite element to model the failure process of the rebar-concrete bonding.

• Introduction of physical portions in the numerical coupled simulation process.

• Direct comparison with experimental testing results, as shaking table tests for a more explicit validation of the method.

Bibliography

[1] Bathe, K. (1996). *Finite Element Procedures*. Prentice Hall, New Jersey. 20

[2] Bazan, E. and Meli, R. (2004). *Diseño Sísmico de Estructuras*. Limusa, México. 1

[3] Borst, R. D. and Sluys, L. (1999). *Computational Methods in Non Linear Solid Mechanics*. Technische Universiteit Delft, Portugal. 17

[4] Code, M. C. B. (2004). Reglamento de construcciones del distrito federal, diario oficial. *Mexico City, Mexico*. 10

[5] Cook, R., Malkus, D., Plesha, M., and Witt, R. (2002). *Concepts and Applications of Finite Element Analysis*. John Wiley, United States. 15, 21, 24

[6] Dhia, H. B. and Rateau, G. (2005). The arlequin method as a flexible engineering design tool. *International journal for numerical methods in engineering*, 62(11):1442–1462. 19

[7] Felippa, C. A. (2013a). Introduction to finite element methods. *Course Notes, Department of Aerospace Engineeing Sciences, University of Colorado at Boulder, available at http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/Home.html*. 17, 23, 24, 25

[8] Felippa, C. A. (2013b). Nonlinear finite element methods. *Course Notes, Department of Aerospace Engineeing Sciences, University of Colorado at Boulder, available at http://www.colorado.edu/engineering/CAS/courses.d/NFEM.d/Home.html*. 16

[9] G.Frazier and Kosloff, D. (1978). Treatment of hourglass patterns in low order finite element codes. *International Journal for Numerical and Analytical Methods in Geomechanics*, 2. 17

[10] Godoy, L. (2003). Modelos y enfoques para problemas con acoplamiento de micro y macro estructuras. *Mecánica Computacional*, 22:1964–1984. 10, 18

[11] Gruttmann, F. and Wagner, W. (2013). A coupled two-scale shell model with applications to layered structures. *International Journal for Numerical Methods in Engineering*, 94(13):1233–1254. 10

[12] Houlsby, G., Liu, G., and Augard, C. (2000). A tying scheme for imposing displacement constraints in finite element analysis. *Communications in Numerical Methods in Engineering*. 23

[13] Huang, Y., Schellenberg, A., Mahin, S., and Fenves, G. (2008). Coupling fe software through adapter elements: A novel use of user defined elements. *10th International LS-DYNA Users Conference*. 13, 26

[14] Kanber, B. and Bozkurt, Y. (2006). Finite element analysis of elasto-plastic plate bending problems using transition rectangular plate elements. 21

[15] Kent, D. and Park, R. (1971). Flexural members with confined concrete. *Journal of Structural Division*, 97. 2

[16] Kim, H.-S. and Hong, S.-M. (1995). Formulation of transition elements for the analysis of coupled wall structures. *Computers & structures*, 57(2):333–344. 9

[17] Li, Z., Chan, T. H., Yu, Y., and Sun, Z. (2009). Concurrent multi-scale modeling of civil infrastructures for analyses on structural deterioration—part i: Modeling methodology and strategy. *Finite Elements in Analysis and Design*, 45(11):782–794. 7

[18] Lignos, D. G., Luna-Moreno, M., and Billington, S. L. (2011). Seismic retrofit of existing steel moment resisting frames with innovative materials: large scale hybrid simulation tests. In *Proceedings*. 13

[19] López, S. (2013). Propuesta de un método multiescalas para la transferencia de efectos mecánicos en marcos estructurales de concreto reforzado. Master's thesis, ESIA-IPN, México. 26

[20] Mander, J., Priestley, M., and Park, R. (1988). Theoretical stress-strain model for confined concrete. *Journal of Structural Engineering*, 114 No.8. 2

[21] Mata, P., Barbat, A., and Oller, S. (2008). Two-scale approach for the nonlinear dynamic analysis of rc structures with local non-prismatic parts. *Engineering Structures*, 30(12):3667–3680. 8

[22] Mazzoni, S., McKenna, F., Scott, M. H., Fenves, G. L., et al. (2006). Opensees command language manual. *Pacific Earthquake Engineering Research (PEER) Center*. 23

[23] Meli, R. (2011). *Diseño Estructural*. Limusa, México. 1

[24] Pegon, P. and Pinto, A. (2000). Pseudo-dynamic testing with substructuring at the elsa laboratory. *Earthquake engineering & structural dynamics*, 29(7):905–925. 11, 12

[25] Perelmuter, A. and Slivker, V. (2003). *Numerical Structural Analysis: Methods, Models and Pitfalls*. Springer-Verlag, New York. 22

[26] Retama, J. and Ayala, G. (2013). Procedimiento de análisis sísmico multi—escala de estructuras de concreto reforzado. *XIX Congreso de Ingeniería Sísmica*. 9, 21

[27] Ruiz, C. H. (2016). Influencia de los efectos p-delta en el desempeño sísmico de marcos planos. Master's thesis, Instituto de Ingeniería-UNAM, México.

[28] Schellenberg, A., Huang, Y., and Mahin, S. A. (2008). Structural fe-software coupling through the experimental software framework, openfresco. In *Proceedings, 14th World Conference on Earthquake Engineering*, Beijing, China. 31

[29] Schellenberg, A., Kim, H. K., Takahashi, Y., Fenves, G. L., and Mahin, S. A. (2009a). Openfresco framework for hybrid simulation: Simulation finite element adapter experimental control example. 49

[30] Schellenberg, A., Mahin, S., and Fenves, G. (2009b). *Advanced Implementation of Hybrid Simulation*. PEER, University of California, Berkeley. 28

[31] Schellenberg, A., Mahin, S. A., and Fenves, G. L. (2006). Application of an experimental software framework for international hybrid simulation. In *4Th international conference on earthquake engineering*, Taipei, Taiwan. 30

[32] Takahashi, Y. and Fenves, G. L. (2006). Software framework for distributed experimental–computational simulation of structural systems. *Earthquake engineering & structural dynamics*, 35(3):267–291. 12

[33] Takahashi, Y. and Kodera, M. (2012). Pseudo-dynamic tests in centrifugal field for structure-foundation-soil systems. 12

[34] Takanashi, K., Udagawa, K., Seki, M., Okada, T., and Tanaka, H. (1975). Nonlinear earthquake response analysis of structures by a computer-actuator on-line system. *Bulletin of Earthquake Resistant Structure Research Center*, 8:1–17. 28

[35] Taylor, R. (2013). Feap — a finite element analysis program, version 8.4 user manual. *University of California at Berkeley, Berkeley, CA*. 23

[36] Wang, F., Xu, Y., and Qu, W. (2014). Mixed-dimensional finite element coupling for structural multi-scale simulation. *Finite Elements in Analysis and Design*, 92:12–25. 7, 19

[37] Weinan, E. (2011). *Principles of multiscale modeling*. Cambridge University Press. 18

[38] Yue, J., Fafitis, A., and Qian, J. (2012). A multi-scale structural model for coupling 1d with 3d elements. 7, 18

[39] Zienkiewicz, O., Taylor, R., and Zhu, J. (2013). *The Finite Element Method: Its Basis and Fundamentals*. Elsevier, United States. 15