ON-LINE MODEL UPDATING IN EARTHQUAKE
HYBRID SIMULATION

BY

HAZEM ELANWAR

DISSERTATION

Submitted in partial fulfillment of the requirements
for the degree of Doctor of Philosophy in Civil Engineering
in the Graduate College of the
University of Illinois at Urbana-Champaign, 2014

Urbana, Illinois

Doctoral Committee:

Professor Amr S. Elnashai, Chair, Director of Research
Professor Billie F. Spencer, Jr.
Professor David A. Lange
Professor Daniel A. Kuchma
Associate Professor Larry A. Fahnestock
Assistant Professor Eun Jeong Cha
ABSTRACT

Hybrid simulation has emerged as a relatively accurate and efficient tool for the evaluation of structural response under earthquake loading. In the conventional hybrid simulation, the responses of few critical components are obtained by testing while the numerical module is assumed to follow an analytical idealization. Where there is a much larger number of analytical components compared to the experimental parts, the overall response may be dominated by the idealized parts hence the value of hybrid simulation is diminished. It is proposed to update the behavior of the material constitutive relationship of the numerical model during the test, based on the data obtained from the physically tested component.

Identifying the parameters that govern the constitutive relationship behavior from the experimental module is a challenging task. Hence, an approach based on optimization tools is developed to determine the model parameters that minimize the error between the numerical and experimental modules. Interior point methods and genetic algorithms are adopted as gradient and non-gradient optimization tools, respectively. Each of which provides different features that are suitable for various types of applications in earthquake response assessment. On the other hand, neural network is utilized as an alternative identification approach. Neural network is advantageous in case the analytical constitutive relationships are not suitable to represent the actual model behavior, as it can be trained independent from analytical guidance to find the mathematical formulas that correlate the input strain to the output stresses.

UI-SIMCOR the platform utilized to conduct the hybrid simulation analyses. It can communicate with several finite element programs. Amongst others, ZeusNL is used to analyze the numerical modules due to its efficiency in representing cases of extreme loading and non-linear problems. For model updating purposes, the source codes and the communication protocols between UI-SIMCOR and ZeusNL are modified to be able to exchange the stress-strain information during the hybrid simulation test. Several steel and concrete constitutive models included in ZeusNL library are implemented in the proposed approach. In addition, the components required for the neural network procedure are introduced to the program.
The scope of the work also includes verifying the model updating concept through analyzing several numerical problems. These problems include the assessment of regular and irregular structural systems. Moreover, it is shown that through updating the parameters of a simple constitutive model, it can capture the behavior of a more advanced one. Additionally, a number of previously conducted experiments are investigated. A procedure is presented to determine the constitutive model information from the tested component. This procedure is implemented to identify the constitutive model parameters representing a steel beam-column connection and a multi-bay concrete bridge subjected to combined loading. The identified parameters are then utilized to update the analytical model incrementally. The results of both examples show the effectiveness of model updating in minimizing the errors, compared to the pure analytical solution. The proposed approach is expected to enhance the capability of conventional hybrid simulation test to assess structures with several critical components such as high-rise buildings and multi-bay bridges.
ACKNOWLEDGMENTS

This work would not have been possible without the support of my advisor, committee members, family and friends. My deepest thanks and extreme gratitude go to my advisor Professor Amr Elnashai for his leadership, encouragement and valuable guidance throughout my research study. Above all, I will always remember his invaluable support and being a source of motivation for me.

I would like also to thank Professor Bill Spencer who always inspired me as an instructor, and for his continuous guidance during my studies. Special thanks for Professor Dan Kuchma, whose presence in the annual EKS retreats and his suggestions regarding my research always made a difference. My gratitude to my committee, Professor David Lange, Professor Larry Fahnestock, and Professor Eun Jeong Cha for providing insightful feedback and valuable comments.

I am also grateful for the support I received from many individuals at UIUC and UIUC graduates. Particular thanks for Dr. Sung Jig Kim for his effort in introducing the model-updating concept upon which I had built my research. I would like also to acknowledge Dr. Oh-Sung Kwon who always helped me in understanding and modifying UI-SIMCOR subroutines. Special thanks to Dr. DoSoo Moon who have been my friend and colleague for his continuous support in modifying ZeusNL source codes.

Thanks are due to Professor Tasnim Hassan and Machel Morrison for being generous in providing the necessary data and advices regarding the steel beam-column connection experiment. Also, thanks to Dr. Thomas Frankie whose help in understanding the CABER project experiment was crucial, and to Dr. Adel Abdelnaby for providing guidance regarding the CABER project analytical model. I am indebted to Eng. Mustafa Elzeiny for his valuable time and assistance in developing several ZeusNL subroutines.
Last but not least, my greatest thanks go to my beloved family. I cannot help but feel immense gratitude towards my parents. Special thanks to my brother Omar Elanwar, who guided me with all possible means throughout my entire career.
# TABLE OF CONTENTS

| LIST OF TABLES | .................................................................................................................. ix |
| LIST OF FIGURES | .................................................................................................................. x |
| CHAPTER 1 INTRODUCTION | ........................................................................................................... 1 |
| 1.1. Problem Statement | .................................................................................................... 1 |
| 1.2. Objective and Scope of Research | .................................................................................. 2 |
| 1.3. Organization of Dissertation | .................................................................................. 7 |
| CHAPTER 2 LITERATURE REVIEW | .......................................................................................................... 9 |
| 2.1. Conventional Hybrid Simulation | ............................................................................... 11 |
| 2.1.1. Hybrid Simulation Methodology | ........................................................................ 11 |
| 2.1.2. UI-SIMCOR | .................................................................................................. 14 |
| 2.2. Analysis Errors | ..................................................................................................... 17 |
| 2.2.1. Experimental component | ............................................................................... 18 |
| 2.2.2. Numerical analysis | .......................................................................................... 19 |
| 2.3. Current Model Updating Procedure | ....................................................................... 21 |
| 2.3.1. Rigid bar and springs | .................................................................................... 22 |
| 2.3.2. Hysteretic energy model | ................................................................................ 22 |
| 2.3.3. Alternative numerical models | ....................................................................... 23 |
| 2.3.4. Discussion | ..................................................................................................... 24 |
| CHAPTER 3 PROPOSED APPROACH | ............................................................................................... 25 |
| 3.1. Model Updating Framework | .................................................................................. 25 |
| 3.1.1. Introduction | ................................................................................................. 25 |
| 3.1.2. Model updating procedure | ........................................................................... 29 |
| 3.1.3. Discussion | ..................................................................................................... 32 |
| 3.2. Action-Deformation Characteristics | ........................................................................ 35 |
| 3.3. Identification Algorithm | ..................................................................................... 39 |
| 3.3.1. Direct application | ......................................................................................... 39 |
| 3.3.2. Optimization tools | ........................................................................................ 40 |
| 3.3.3. Neural networks | .......................................................................................... 51 |
| 3.4. Developed Software | ................................................................................................. 58 |
6.1.2. Numerical verification ......................................................................................... 170
6.1.3. Experimental verifications .................................................................................. 171
6.1.4. Remarks on model updating .............................................................................. 173

6.2. Future Work ........................................................................................................... 175

REFERENCES ................................................................................................................ 177

Appendix A ..................................................................................................................... 183
Appendix B ..................................................................................................................... 197
# LIST OF TABLES

| Table 4.1 STL1 model parameters for the different analysis cases | 76 |
| Table 4.2 STL2 model parameters for the different analysis cases | 80 |
| Table 4.3 STL3 model parameters for the different analysis cases | 83 |
| Table 4.4 Reinforcement bar model (STL1) parameters for the different analysis cases | 86 |
| Table 4.5 Concrete core model (CON2) parameters for the different analysis cases | 87 |
| Table 4.6 Irregular frame constitutive model (STL1) parameters for the different analysis cases | 91 |
| Table 5.1 The average absolute errors for the straining actions of the left pier | 156 |
| Table 5.2 The average absolute errors for the deformations of the left pier | 158 |
| Table A.1 The average errors of the left pier loads, when optimization is applied after 10 sec of the input excitation | 184 |
| Table A.2 The average errors of the left pier deformations, when optimization is applied after 10 sec of the input excitation | 184 |
| Table A.3 The average errors of the right pier loads, when optimization is applied after 10 sec of the input excitation | 184 |
| Table A.4 The average errors of the right pier deformations, when optimization is applied after 10 sec of the input excitation | 184 |
| Table B.1 The average errors of the left pier loads, while assuming soft constitutive model parameters | 198 |
| Table B.2 The average errors of the left pier deformations, while assuming soft constitutive model parameters | 198 |
| Table B.3 The average errors of the right pier loads, while assuming soft constitutive model parameters | 198 |
| Table B.4 The average errors of the right pier deformations, while assuming soft constitutive model parameters | 198 |
LIST OF FIGURES

Figure 1.1 Sub-structuring the model into numerical and physical components................................. 2
Figure 1.2 Different test components in model updating approach .................................................... 3
Figure 1.3 The main components of model updating approach ......................................................... 4
Figure 2.1 Illustration for the hybrid simulation test procedure .......................................................... 14
Figure 2.2 UI-SIMCOR framework [Kwon et al., 2007] ................................................................... 17
Figure 3.1 Layout for a multi-bay multi-story steel structure ............................................................. 27
Figure 3.2 Multi-bay multi-story frame nodal displacements for the different analysis cases ............... 29
Figure 3.3 Flowchart illustrating the procedure of model updating .................................................... 31
Figure 3.4 Constitutive models: (a) bilinear steel model, (b) non-linear concrete model, (c) Ramberg
Osgood steel model, and (d) Menegotto-Pinto steel model [Elnashai et al. 2002] ............................... 38
Figure 3.5 Constitutive model solution approaches: (a) Forward problem, and (b) inverse problem ...... 41
Figure 3.6 Flowchart describing the optimization procedure to identify the constitutive model
persons............................................................................................................................................. 43
Figure 3.7 Solution procedure of generic algorithms ....................................................................... 47
Figure 3.8 Optimization results for different constitutive models: (a) STL1, (b) CON2, (c) STL2 and (4)
STL3 .................................................................................................................................................. 49
Figure 3.9 Optimization results for different constitutive models after adding noise: (a) STL1, (b) CON2,
(c) STL2 and (4) STL3 ........................................................................................................................ 50
Figure 3.10 The main elements of a single neuron .............................................................................. 52
Figure 3.11 Multiple-layer neural network configuration ..................................................................... 53
Figure 3.12 The architect of the neural network used for the constitutive model problem ................... 56
Figure 3.13 Illustration of the neural network input requirements ....................................................... 57
Figure 3.14 The main elements of model updating approach ............................................................. 59
Figure 3.15 A schematic diagram for the model updating console ..................................................... 62
Figure 3.16 constitutive relationship: (a) steel and (b) concrete in a model updating problem.............. 67
Figure 4.1 Geometric configuration of a two bay steel frame and the cross sectional dimensions of the
columns................................................................................................................................................ 76
Figure 4.2 The nodal displacements for the STL1 frame subjected to the different analysis cases ....... 78
Figure 4.3 Error comparison in the STL1 frame for the conventional and the model updating cases ..... 79
Figure 4.4 The nodal displacements for the STL2 frame subjected to the different analysis cases ....... 81
Figure 4.5 Error comparison in the STL2 frame for the conventional and the model updating cases ..... 82
Figure 4.6 The nodal displacements for the STL3 frame subjected to the different analysis cases ....... 84
Figure 4.7 Error comparison in the STL3 frame for the conventional and the model updating cases ...... 85
Figure 4.8 Geometric configuration of a two bay reinforced concrete frame and the cross sectional
dimensions of the columns................................................................................................................. 86
Figure 4.9 The nodal displacements for the CON2 frame subjected to the different analysis cases ....... 87
Figure 4.10 Error comparison in the STL3 frame for the conventional and the model updating cases ..... 89
Figure 4.11 The geometric configuration and cross-sectional dimensions of the irregular frame model . 90
Figure 4.12 The nodal displacements for the irregular frame subjected to the different analysis cases .. 91
Figure 4.13 Error comparison in the irregular frame for the conventional and the model updating cases
........................................................................................................................................92
Figure 4.14 The geometric configuration for the frame used in updating the bilinear steel model......94
Figure 4.15 The nodal displacements for steel frame subjected to the different analysis cases.........94
Figure 4.16 Error comparison in the steel frame for the conventional and the model updating cases....95
Figure 4.17 Comparing the exact and the hybrid simulation constitutive models for; (a) the accurate (left) column, and (b) the inaccurate column .................................................................................................................96
Figure 4.18 Comparing the exact and model updating constitutive models for; (a) the accurate (left) column, and (b) the inaccurate column.................................................................................................................97
Figure 4.19 The geometric configuration for the frame analyzed using the neural network approach ....99
Figure 4.20 The response spectrum plots for the artificially generated motion and the Big Bear earthquake..............................................................................................................................................100
Figure 4.21 constitutive relationships used to train the NN; (a) the most stressed fiber and (b) a fiber in the elastic range..............................................................................................................................................101
Figure 4.22 Neural network configuration used to identify the constitutive model behavior........102
Figure 4.23 The stress-strain values identified from the neural network compared to the analytical bilinear model results ..............................................................................................................................................103
Figure 4.24 The top nodal displacements for the left column comparing the accurate to the neural network results ..............................................................................................................................................103
Figure 4.25 constitutive relationships used to train the NN; (a) the most stressed fiber distorted with noise and (b) a fiber in the elastic range..............................................................................................................................................104
Figure 4.26 The stress-strain values identified from the neural network distorted with noise compared to the analytical bilinear model results ..............................................................................................................................................105
Figure 4.27 The top nodal displacements for the left column comparing the accurate results to the neural network results distorted with noise ..............................................................................................................................................106
Figure 5.1 Constitutive model identification procedure in case the stresses can be determined directly ..............................................................................................................................................112
Figure 5.2 Constitutive model identification procedure in case of complex experimental environment 114
Figure 5.3 Test setup for the conventional beam-column steel connection [Schweizer, 2013] .........117
Figure 5.4 The instrumentation of the conventional steel beam-column connection [Schweizer, 2013]118
Figure 5.5 Actual beam-column test setup: (a) Krypton LEDs and (b) strain gauges at the beam flange [Schweizer, 2013] ..............................................................................................................................................118
Figure 5.6 The strain values at the beam flanges in the conventional connection .........................120
Figure 5.7 Identified moment values for the conventional beam-column connection....................122
Figure 5.8 Coupon test results for the steel material used in the conventional beam-column connection [Schweizer, 2013] ..............................................................................................................................................123
Figure 5.9 Instruments attached to the heat treated beam connection; (a) Krypton LEDs, and (b) strain gauges [Schweizer, 2013] ..............................................................................................................................................127
Figure 5.10 The strain records at the top and bottom flanges at the two sections of interest ..........128
Figure 5.11 The heat treated beam connection response for the different analysis cases ...............129
Figure 5.12 The percentile error normalized to the peak response for the different analysis cases in the heat treated beam connection ..............................................................................................................131
Figure B.4 Left pier longitudinal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 202
Figure B.5 Left pier transversal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 203
Figure B.6 Left pier torque-twist responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 204
Figure B.7 Right pier transversal force-deformation responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 205
Figure B.8 Right pier longitudinal force-deformation responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 206
Figure B.9 Right pier axial responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 207
Figure B.10 Right pier longitudinal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 208
Figure B.11 Right pier transversal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 209
Figure B.12 Right pier torque-twist responses and hysteretic behavior, while assuming soft constitutive model parameters ................................................................. 210
CHAPTER 1

INTRODUCTION

1.1. Problem Statement

The response of structures subjected to severe loading, such as earthquakes, is governed mainly by their inelastic behavior. However, the analytical models representing such inelastic behavior are not sufficiently accurate. Therefore, experimental investigations are the most reliable techniques to assess structural components that undergo large deformations. Executing full-scale experiments, however, is not always economic and in some cases, it is not even feasible. Thus, the idea of hybrid simulation (Pseudo-dynamic) tests was introduced [Hakuno et al., 1969; Takanashi et al., 1975]. In hybrid simulations, the structure is divided into several modules such that, the critical modules exerting behavior that cannot be accurately idealized by the numerical tools are tested in the laboratory, while the rest of the structure is simulated numerically. Figure 1.1 shows the hybrid simulation concept, which includes analyzing each module separately. Meanwhile, the global response of the different structural components is retained at the interface degrees of freedom [Kwon et al., 2008]. Hybrid simulation tests have several features that enhance the model assessment. For example, the test can be executed at slow rates to allow for extensive investigation of the physical components [Elnashai et al., 1990]. In addition, they outperform quasi-static tests since the loading history is identified during the experiment based on the interaction between the different structural components; and hence, the path dependency of the model is preserved.
In hybrid simulations, few members are tested in the laboratory, which implies that the rest of the structure should be accurately represented by analytical platforms. However, the accuracy of these analytical modules is questioned, especially when the structure is subjected to extreme loading. In addition, there are several applications where many structural components undergo non-linear deformations such as the cases of long span bridges and high-rise buildings. In these cases, if all the critical components are tested in the laboratory, hybrid simulations lose the advantage of simplicity and cost efficiency. On the other hand, testing fewer members will result in an inaccurate analytical module, which affects the reliability of the overall experiment results.

1.2. Objective and Scope of Research

In this research, it is proposed to develop, implement and verify an instantaneous model updating approach to enable the conventional hybrid simulations to handle a wider range of applications and to enhance the accuracy of the analytical modules. During hybrid simulations, a significant
amount of information can be acquired from the physically tested component. Model updating approach aims to modify the behavior of the analytical modules in a stepwise manner, based on the data learned from the physically tested component, as shown in Figure 1.2 [Yang et al., 2012; Hashemi et al., 2013; Kwon and Kammula, 2013]. Therefore, model updating allows for a more realistic representation of the numerical modules and hence, improves the overall accuracy of the experiment. Additionally, this approach provides a suitable environment for testing structures with several critical members such as multi-bay bridges and high-rise buildings.

![Figure 1.2 Different test components in model updating approach](image)

The framework of model updating includes three main components as illustrated in Figure 1.3: (1) model sub-structuring, (2) action-deformation characteristics selection, and (3) identification algorithms application. Each of these components can be subdivided into several alternatives to allow for various applications for the proposed approach.
The first step in model updating is to discretize the structure into experimental, updated and numerical modules. The critical members that represent a behavior which is not well understood are tested in the laboratory. While the analytical modules that share close characteristics to the critical members are updated incrementally based on the information learned during the test. The rest of the structure showing simple response are simulated numerically. This approach reduces the effort and cost required for conducting a hybrid simulation experiment, where it is sufficient to test the few critical components that are representatives of the others.

Second, it is important to determine the action-deformation characteristics to be identified from the physically tested component. Examples of the characteristics to be determined during the test include; (1) the constitutive relationship behavior, (2) the hysteretic model, (3) the global member response represented by a set of springs, etc. In this research, it is proposed to identify the
constitutive relationship behavior as a fundamental property on the material level of the structure. The analytical models frequently fail to represent the actual structural response, especially when the structure is subjected to repeated intense loading that impose inelastic deformations. However, through identifying the stress-strain relationship that best represents the tested specimen, it is expected to eliminate several sources of uncertainties in the model. In addition, there might be uncertainties in the parameters governing the constitutive response such as the initial stiffness, the yield strength, the crushing strain, etc. In the suggested framework, some of the constitutive relationships available in literature will be utilized, while the factors governing their behavior will be updated incrementally throughout the analysis.

The third step is applying the identification algorithms that can be used to determine the action-deformation characteristics from the tested components. These algorithms include: (1) the direct approach, where the structural components are identical in terms of geometry and applied loading. In this case, the global response of the physical specimen is duplicated in the numerical modules. (2) Gradient and non-gradient based optimization tools are proposed as an identification alternative. Optimization tools aim to determine the constitutive model factors that minimizes a certain objective function. Usually this objective function is the error between the analytical results and experimental measurements. Additionally the problem is subjected to constraint equations, which satisfies the fundamental properties of the model behavior. (3) Neural network on the other hand, recognizes the response pattern of the physical specimen and then it creates a set of mathematical functions that define the intermediate relation between the given input and output values. As a mathematical tool, neural network is advantageous in cases where the available analytical models are not suitable to represent the behavior of the tested specimen.
Model updating can be used for several numerical and experimental applications. Numerically, it can be used to modify a simple analytical model and guide its behavior to match ones that are more complicated. This procedure can improve the accuracy of simple models in addition to saving a lot of computational time. On the other hand, there are several applications where model updating can enhance conventional hybrid simulation experiments. For example, instead of testing all the members that share close characteristics, it is sufficient to test one representative of these while preserving the accuracy, which saves unnecessary cost, time, and effort. Additionally, it provides the possibility to assess massive structures that include repetitive pattern such as high-rise buildings. It is important to mention that for both numerical and experimental applications, model updating preserves path dependency, which is crucial for achieving realistic structural response. In other words, any modification in the current loading step will have an impact on the future steps.

It can be observed that all the model updating components are based on incremental modification of the analytical module. Moreover, applying the identification algorithms and imposing the new parameters to the model require calculation time. Yet, hybrid simulation can be conducted at slow rates and it can be paused at any loading step, which provides the required time for necessary computations and hence, the application of model updating procedure.

The objective of this research is to set a framework for model updating. This framework includes developing a program that integrates the different components governing the updating procedure. The program must provide a feature to exchange the stress-strain information between the numerical and experimental environments. The scope of the dissertation also includes verifying
the proposed approach through several analytical examples. These analytical examples point out
the main features of the model updating approach and show its ability to improve the conventional
hybrid simulation results. On the other hand, previous laboratory experiments are investigated to
confirm that model updating can achieve significant enhancement in the actual test environment.

1.3. Organization of Dissertation

This dissertation divided into six chapters. Chapter 1 introduces the objectives and the scope of
research. Chapter 2 gives an overview of the conventional hybrid simulation in terms of its
procedure, features and drawbacks. In addition, it presents some examples included in the literature
related to model updating framework. Chapter 3 discusses in details the framework of model
updating. Thereafter, the constitutive relationship as a representative of the action-deformation
characteristics will be investigated. Additionally, the identification techniques used to determine
the constitutive relationship characteristics will be introduced. In this research, the main focus will
be on the methodologies of optimization tools and neural networks. Finally, the features of a
software that integrates the different model updating components will be discussed.

In Chapter 4, several analytical examples are presented. These examples demonstrates the ability
of model updating to interact with different problem configurations and to improve their results.
In this chapter, the proposed model updating approach will be verified for several constitutive
relationships in addition to analyzing unconventional problems using the optimization and neural
network techniques.
Chapter 5 is intended to confirm that model updating is effectively operational in actual laboratory applications. First, a procedure for identifying the stress-strain model parameters from experimental measurements is presented. Then two previous experiments from literature are investigated to validate the proposed framework, which include (1) a simple quasi-static beam-column steel connection, and (2) a more complicated hybrid simulation experiment for multi-span concrete bridge. This chapter includes detailed information about the test setup of each experiment and the results achieved using the different techniques. Chapter 6 summarizes the findings and the conclusions of this research study followed by the potential future work.
Structural response assessment is a challenging task especially under severe and reversing loads. There are different techniques that can be applied for structural assessment such as field inspection, experimental investigation and analytical modeling. Each of these techniques has its own advantages and limitations. For example, in field inspection, the structural response is evaluated in their existing condition, which gives the most realistic behavior compared to the other approaches. However, field inspection does not satisfy a number of research needs such as applying intense loading to the structure or studying its failure mechanisms. Experimental investigations is another assessment technique; in which, either the full-scale models or a sub-structural components are tested in the laboratory. The tested components can be subjected to different states of loading; hence, the crack propagation and the failure mechanisms can be studied explicitly. Shaking table tests are considered among the most reliable experimental techniques, because the input motion is applied in real time and it flows naturally to the system through its supports [Mahmoud, 2011]. It is obvious that there are several difficulties in executing shaking table tests such as: the cost of constructing the model, the space availability, the actuators capacity and the intensive labor. For instance, it is not feasible to assess structures such as full-scale high-rise buildings in the laboratory. Moreover, it is always challenging to represent the soil structure interaction accurately in shaking table experiments [Mahmoud, 2011]. Quasi-static tests offer another alternative for experimental investigations, whereby one component of the structure is tested, which is subjected to a predefined loading history. Therefore, they require much lower cost and less effort compared to the shaking table experiments. The main drawback of quasi-static tests
is that the loading history is not realistic as it is generated prior to conducting the test. Hence, it does not take into consideration the path dependency of the model and as such, it neglects the actual behavior of the tested specimen and its effect on the overall response of the model [Mahin and Shing, 1985].

Analytical modeling is widely used in evaluating structural response due to its simplicity and cost efficiency, where the actual model is idealized mathematically based on either the theoretical understanding or the given empirical formulations [Kwon et al., 2008]. There are several platforms that can be utilized to analyze structures. Each platform is capable of representing some aspects of the model in details. Therefore, complex structures are usually subdivided into multiple-components such that each component is assessed by the software package that can best represent its characteristics. Although analytical platforms had developed significantly over the years, yet they could not achieve the level of complexity exerted by the actual structural components. The inconsistency becomes more obvious in the case of investigating inelastic response such as crack propagation and failure mechanisms. Therefore, it is recommended that the models that undergo non-linear deformation to be investigated experimentally [Elnashai et al., 1990; Kwon et al., 2008].

This chapter addresses hybrid simulation as a tool that benefits from the advantages of both analytical and experimental approaches. The chapter first discusses the framework of hybrid simulation. Then some drawbacks in this framework are presented including the errors in the experimental and the numerical modules. However, it should be noted that the focus of this research is directed towards improving the accuracy of the numerical modules only. Finally, some examples from the literature that utilized model updating approach are discussed. The main
differences between the proposed approach and the procedure provided in literature will be also highlighted.

2.1. Conventional Hybrid Simulation

Analytical platforms cannot accurately assess structural response subjected to complex loading such as earthquake records. On the other hand, it would be costly and infeasible to investigate all the structural components experimentally. Therefore, both the analytical and the experimental approaches are indispensable for research and none of them outperforms the other on all counts [Kwon et al., 2008]. Hence, the approach of hybrid simulation was introduced. Whereby, the structure is subdivided into multiple-components such that, the components that are expected to undergo large deformations are tested in the laboratory, while the rest of the structure is simulated numerically [Hakuno et al., 1969; Taknashi et al., 1975]. As such, cost efficiency is achieved by simulating most of the model numerically; meanwhile, the critical members that experience complex behavior are tested in the laboratory. Moreover, hybrid simulation allows for testing the model components in different facilities and the response can be synchronized through online protocols [Kwon et al., 2008].

2.1.1. Hybrid Simulation Methodology

The framework of hybrid simulation (pseudo-dynamic tests) was inspired by quasi-static tests. However, hybrid simulations are advantageous because the path dependency is considered in the deformations applied to the tested module [Mahin and Shing, 1985]. In other words, the actions
applied to the tested module are determine in a stepwise manner based on the interaction between the different structural components [Kwon et al., 2008]. In literature, hybrid simulations have been proved a reliable technique, even the accuracy of their results were close to full-scale shaking table tests [Shing et al., 1984; Thewalt and Mahin, 1987]. There are several integration schemes that can be used to analyze both the conventional and the real time pseudo-dynamic tests [Takanashi et al., 1975; Kwon et al., 2008; Nakashima et al., 1992; Carrion and Spencer, 2006]. In these integration schemes, the equation of motion is solved in a stepwise manner, where it receives the actions from all the structural modules and then predict the deformations applied in the next loading step. The procedure of solving hybrid simulation problem can be illustrated as shown in Figure 2.1, which is summarized in the following steps [Kwon et al., 2008]:

- The structure is divided into numerical modules and experimental components, where the critical members are tested in the laboratory, while the modules subject to modest straining actions are numerically represented.
- The equation of motion integrates the discrete structural systems to determine the deformations applied in the next loading step. The output vector is disassembled such that the deformations applied to the numerical and experimental modules are called $U_{\text{Num}}$ and $U_{\text{Exp}}$, respectively.
- For the numerical modules, the analytical software finds the restoring forces ($R_{\text{Num}}$) that corresponds to the deformations ($U_{\text{Num}}$). For the experimental modules, the actuators record the restoring forces ($R_{\text{Num}}$) required to apply the predefined deformations ($U_{\text{Exp}}$).
After finishing the first cycle, the hybrid simulation experiment can be paused in order to perform the necessary calculation for the next loading steps. It is worth noting that, there are similar approaches that can be utilized to conduct real time hybrid simulations.

Finally, the same procedure is repeated after each loading step.

Based on the discussed procedure, some important features regarding hybrid simulation can be highlighted such as: (1) the restoring forces acquired from the numerical modules, follow a simplified idealization of the actual model. However, the accuracy of this numerical solution is crucial to achieve a realistic representation of the global system. (2) The level of confidence in the response of the physical specimen is usually higher than any idealization, as it can capture complex details of the specimen behavior, even until failure. (3) Unlike other experimental techniques such as shaking table tests, hybrid simulations provides some properties to simplify executing the test. For example, it does not required to fix actual masses to account for static loads. However, these loads can be applied through the actuators before starting the dynamic analysis [Takanashi et al., 1975]. Moreover, the model can be discretized into separate components; thereby, the same experiment can be conducted at different facilities. Therefore, facilities with different specialties can interact to build a system capable of representing realistic structures as close as possible, which have been implemented successfully in literature [Spencer et al., 2006]. (4) Finally, hybrid simulations can be executed at a slow rate, which allows the user to perform detailed inspection of the tested specimen and interfere in the experiment if required [Elnashai et al., 1990]. This property is crucial for the proposed model updating framework as will be discussed in Chapter 3. It is worth noting that, the slow rate feature does not apply in case of analyzing rate dependent structure such as using viscous dampers. In these cases real time approaches are required.
In the next section, a simulation coordinator program (UI-SIMCOR) is introduced [Kwon et al., 2007]. This program is capable of interacting with the different numerical and experimental modules to find the global response of the structure.

2.1.2. UI-SIMCOR

There are few software packages that can be used as coordinators for hybrid simulation tests. Among others, the simulator coordinator program UI-SIMCOR is used in this research project [Kwon et al., 2007]. UI-SIMOR is a MATLAB® based software, which integrates the response of different structural components through the interface degrees of freedom (DOFs) [Kwon et al.,
Two implicit stepwise integration schemes are implemented in the program that can analyze pseudo-dynamic problems: (1) an unconditionally stable iterative implicit method [Shing et al., 1991] and (2) a predictor corrector α-operator splitting scheme [Combesure and Pegon, 1997; Ghaboussi et al., 2006]. UI-SIMCOR interacts with the tested components even if they exist in different laboratories. In addition it can communicate with several finite element platforms over different networks, which are Zeus-NL [Elnashai et al., 2002], OpenSees [McKenna and Fenves, 2001], ABAQUS [Hibbit et al., 2001], FedasLab [Filippou and Constantinides, 2004] and Vector2 [Vecchio and Wong, 2003]. This variety of platforms allows the users to subdivide the structure into multiple-modules and analyze each module with the program that best represent its behavior. In this research, Zeus-NL is utilized as the analytical platform as will be discussed in subsequent sections.

Figure 2.2 shows the framework of UI-SIMCOR, where the main subroutine interacts with the different structural components through the network protocols. The program procedure can be summarized in the following steps [Kwon et al., 2008]:

1. The initial stiffness of each module is determined by applying small deformations at the interface DOFs. These deformations must be small enough to ensure that the tested specimen is still behaving in the elastic phase.
2. This initial loading is not a part of the actual loading history. However, it is only used to determine the stiffness matrix of each structural component. The stiffness matrix will be combined later to achieve the global model response.
3. The static loads are applied to the physical specimen by the actuators until it reaches the level of stresses defined in the designed model. There is no need to fix the nodal masses in a hybrid simulation. These masses are implicitly accounted for in the actuators’ actions.

4. The dynamic excitations, such as the earthquake records, are applied in a stepwise manner to the investigated model.

5. The equation of motion coordinates between the discrete components of the experiment. It integrates the input loading and the restoring forces from the different modules to determine the corresponding deformations applied to each of these modules. The deformations are applied incrementally at the interface DOFs, based on the model behavior in the previous loading step.

6. Specific protocols are utilized to transfer the data through the network to interact with the numerical and the experimental modules. As shown in Figure 2.2, this is achieved through the use of multiple servers. This class in UI-SIMCOR is called MDL_RF.

7. Another class called MDL_AUX controls the auxiliary equipment such as the data acquisition system and the measurement instruments. The measured information is sent back to the network to start the new loading step.

8. At each loading increment, the same procedure is repeated where the integration scheme predicts the deformations applied in the future step.
UI-SIMCOR is an open source program, which allows for modifications that can suit different types of research. This feature will be utilized for the model updating approach.

2.2. Analysis Errors

Although it is beneficial to utilize both numerical and experimental approaches in hybrid simulations, however the errors related to both approaches are induced [Schellenberg et al., 2009]. In addition, the response of the experimental module is path dependent, which implies that any error that occurs at the initial stages will keep accumulating along the loading history. Although,
this section will briefly discuss some of the possible experimental and numerical sources of inaccuracy, the focus of this research study is to improve the accuracy of numerical model response.

2.2.1. Experimental component

In literature, the issues associated with the experimental component of the hybrid simulation are discussed thoroughly [Ahmadizadeh and Mosqueda, 2008; Mosqueda, 2003; Chang, 2001]. In this section, a brief overview of some potential errors is introduced. Pseudo-dynamic tests can be executed at slow rates. This feature allows for detailed inspection of the physical specimen and it increases the capacity of the actuator. However, in slow rate tests the load is applied in a ramp then hold pattern. In some cases, the actuators cannot sustain the load during the hold phase and relaxation might occur for displacement control tests (or softening in the force control tests) [Schellenberg et al., 2009]. A possible solution is to perform a correction step on the measured restoring forces, which takes into consideration the relaxation effect. This technique is specific for the equipment characteristics and the tested material; hence, more investigations are required to develop a more generic approach [Molina, 2002]. The issue of relaxation does not apply when running the test in real time. Usually real time tests are performed to assess structural components that are velocity dependent such as using viscous dampers [Mosqueda, 2003].

During the experiment, random noise might interfere with the measurements of the higher modes of vibration. Therefore, the overall response of the model is distorted. Since the lower modes of vibration are the ones of interest in most civil engineering applications, several numerical
integration algorithms are used to introduce artificial damping which reduces the response of the higher modes without changing the characteristics of the lower ones [Chang, 1997]. Inaccuracies can also occur in the measurement devices and data acquisition systems. Checking and calibrating the different test components is very important to avoid a number of potential errors [Mosqueda, 2003].

Another source of errors in experimental approaches is the time lag between the signal sent to the actuator and its response. This issue is more significant when performing real time tests. A possible solution is to predict the delay in the signal and to include this inconsistency while applying the load in the future load step [Mosqueda, 2003].

2.2.2. Numerical analysis

The numerical component in hybrid simulation interacts with the whole model through the interface degrees of freedom (DOFs). Consequently, the characteristics of the continuous substructure are lumped at the DOFs, which might yield inaccurate results. Therefore, special consideration is required to determine the appropriate locations to divide the model without having a significant impact on the overall model response [Schellenberg et al., 2009].

Moreover, several modifications have been proposed to both the implicit and the explicit stepwise integration algorithms to improve their performance. Each of the approaches has advantages and drawbacks. For example, the explicit algorithms are fast in calculation, but they add constraints on the time increment to avoid model stability issues (i.e. conditionally stable). On the other hand,
implicit algorithms provide excellent stability conditions, yet they require an iterative procedure. Iterative approaches are feasible for analytical solutions, but they cannot be applied for experimental components, as the loading path might be irreversible [Mosqueda and Ahmadizadeh, 2010]. Many techniques have been introduced to reduce the drawbacks of the integration schemes. Combined implicit and explicit schemes were proposed as a possible integration scheme. In the combined techniques, the stability conditions are resolved, while avoiding iterations on the experimental component. Whereby, for these components the elastic stiffness can be used instead of the tangent ones [Combescure and Pegon, 1997]. There is still ongoing research to achieve a better performance for the integration algorithms.

This section summarized some of the experimental and numerical errors that might be induced during the hybrid simulation analysis. In addition, it introduced some alternatives to reduce the effect of these errors. The objective of this discussion is to highlight the fields that can be addressed to improve the efficiency of hybrid simulations. Many of the presented drawbacks have been thoroughly studied in literature. In this research, the reliability of the numerical module to represent the actual structural behavior will be investigated. The analytical idealization is well established when the structure is subjected to simple deformation actions. However, for a more complicated environment, the ability of such models to achieve realistic behavior is still questioned. The next section will present the model updating approach, which aims to enhance the overall efficiency of the numerical modules.
2.3. Current Model Updating Procedure

Before introducing the model updating approach, it is convenient to summarize the most relevant hybrid simulation features, which are as follows; (1) the model is path dependent; hence, it is crucial to guarantee the accuracy of all the structural components governing the response. (2) Traditionally, the critical members tested in the laboratory acquire the majority of the attention, while the numerical modules are assumed to follow an analytical idealization. (3) The analytical idealization cannot represent complex systems especially, when the structure is subjected to intense loading and approaches the failure state. (4) Hybrid simulation tests can be conducted at a slow rate, which provides sufficient time to investigate the structural response and to perform model analysis if required. Model updating addresses these features, which aims to update the numerical modules dynamically during the test, depending on the data obtained from the physically tested component [Yang et al., 2012]. This approach is relatively recent and few applications have been investigated. It should be noted that the expression of model updating applies also for cases of calibrating the numerical model parameters to match the response of the tested structure [Jang et al., 2013]. However, the focus of this research is to update the model dynamically during hybrid simulations to preserve the impact of path dependency. The following section will introduce some of the examples found in literature discussing model updating.
2.3.1. Rigid bar and springs

Yang et al. introduced the idea of model updating in 2012. The research study was a simple analytical assessment of a bridge response. In this example, a one bay bridge supported on two piers was analyzed twice. First, the two piers were represented by a detailed fiber analysis model, which was subjected to an earthquake loading. The fiber analysis model can relatively capture the detailed response of the pier and hence, its results were assumed to be the exact solution. In the second model, one of the piers was replaced by a rigid bar connected to a rotational spring. As expected, the results of that model could not simulate the level of details achieved in the fiber analysis model. Consequently, a significant error in the second case was observed. The same model was assessed one more time, but after updating the parameters governing the rotational spring in a stepwise manner during the analysis, based on the global response of the accurate fiber-modeled pier. The results showed significant improvement when the proposed model updating technique was applied. The objective of this problem was to demonstrate that a simple system can be adapted to represent a more sophisticated one.

2.3.2. Hysteretic energy model

Later on, the model updating concept was applied by Hashemi et al., 2013. In this study, the problem was verified both numerically and experimentally. An experiment was conducted on a one-story one-bay frame, where one column was tested in the laboratory and the other was simulated numerically. It was required to determine the parameters that govern the hysteretic behavior using the modified Bouc-Wen model [Baber and Noori, 1985]. These parameters were
identified using the downhill simplex optimization tool, then the results were improved using unscented Kalman filter, which reduced the effect of the unwanted noise in the experimental measurements. For both numerical and experimental verification examples, the parameters of interest were identified incrementally during the analysis. The overall structural response showed that the model updating technique successfully calibrated the numerical module to match the experimentally recorded data.

2.3.3. Alternative numerical models

In this technique, a different philosophy was adopted. Instead of updating the numerical component, several shadow numerical components were created and the combination that generates the most realistic behavior was preserved [Kwon and Kammula, 2013]. The objective of this research was to determine the Bouc-Wen hysteretic model from the tested specimen. Multiple-numerical modules were designed such that each one of them represents a different hysteretic behavior. The choice of the hysteretic parameters was defined such that all the modules cover a wide range of responses. Then optimization tools were applied to determine the weight factors of each numerical module that result in the closest response to the physically tested specimen. Again, model updating technique showed better results than using the conventional analytical solution [Kwon and Kammula, 2013]. The distinctive characteristic of this approach, compared to the other two examples, is that the inherent parameters of each numerical module did not change. Instead, it selects the combination of models that best represents the accurate experimental component.
2.3.4. Discussion

In literature, it was shown that model updating has the potential to enhance the capabilities of the conventional hybrid simulation. The main objective of this technique is to benefit from the rich information learned during the test to modify the characteristics of the corresponding numerical modules. However, all the analytical and the experimental examples presented herein focused on determining the global response of the physical specimen. In other words, smearing all the uncertainties in the model behavior, then finds a way to achieve a similar response as the accurate one on the global level, while overlooking the inter element behavior. The proposed approaches operated on characteristics such as spring elements or the hysteretic response of the member. In this research, the concept of model updating will be extended to identify the uncertainties in the constitutive relationship to update the structural components on a more fundamental level. The framework and the characteristics of the new approach will be discussed in details in Chapter 3.
CHAPTER 3
PROPOSED APPROACH

In Chapter 2, the main components of the hybrid simulation were discussed. It was shown that this approach is an efficient tool to assess the structures in an accurate and economical way. However, a number of limitations were highlighted, which hinder hybrid simulations from reaching their full potential. Therefore, model updating approach is proposed to address one of the main limitations; which is the reliability of the numerical module response. This chapter introduces the framework of model updating and discusses its procedure in details. Then, an approach based on learning the constitutive relationships as the action-deformation characteristics to be modified will be presented. Next, an overview of the identification algorithms that can be used to determine the required parameters during the analysis are explored. Finally, the features of a software developed to support model updating purposes and some of its applications are presented.

3.1. Model Updating Framework

The objective of model updating is to enhance the conventional hybrid simulation procedure to assess a wide range of applications. Therefore, a number of modifications are required to the current hybrid simulation procedure, which are highlighted in this section.

3.1.1. Introduction

Analytical models aims to idealize the actual structural system response. However, they usually cannot represent the complex real systems, especially for the conditions of intense loading and
inelastic response [Takanashi et al., 1975]. Even contests had been held among researchers and field engineers with the intention to evaluate the ability of analytical platforms to mimic the experimental results. Nevertheless, the analytical models experienced considerable inconsistencies with the experimental results [Collins et al., 1985; Ohsaki et al., 2008; PEER and NEES, 2010]. Therefore, experimental investigations provide a reliable technique for structural assessment. Hybrid simulation approach was developed to combine the advantages of both numerical and experimental tools. To guarantee an efficient experiment, all the critical members must be tested in the laboratory, while the simple and well-established structural elements are simulated numerically. The number of critical members that can be tested is governed by the number of available actuators in the facility. In literature, it is reported that for the largest size hybrid simulation experiment, only three components were tested in the laboratory [Kwon and Kammula, 2013].

There are several other applications, where the structure includes several critical members. For example, multi-span bridges, high-rise building and braced structures, amongst others show that the ability of hybrid simulation to evaluate these structures diminishes. Whereby, it is infeasible to accommodate all the critical members in the testing facility. The traditional solution is to test few components in the laboratory and to represent the rest of the structure numerically. To quantitatively evaluate the efficiency of this approach, a multi-bay multi-story steel structure subjected to an earthquake motion was analyzed. Figure 3.1 shows the layout of the model, where all the beams were defined to be rigid and all the inelastic deformations were represented at the columns. The purpose of this illustrative example is to investigate the effectiveness of applying
hybrid simulation on one member, while overlooking the behavior of the numerically analyzed ones.

![Figure 3.1 Layout for a multi-bay multi-story steel structure](image)

In order to find the impact of the analytical component on the overall structural response, this model was analyzed for each of the following three cases:

1 - Accurate case: the columns were defined with the bilinear steel model and the values used in this model are assumed to be the exact ones. This case represents an imaginary full-scale experiment, whereby all the components are behaving accurately.
2- Analytical case: the structural response was distorted by increasing the yield strength values of the steel columns by 10%. This case represents a numerical solution, whereby the behavior of the structural columns deviates from the accurate ones due to the lack of complete theoretical understanding of the member response.

3- Hybrid simulation case: one column is defined by the accurate material parameters (i.e. yield strength). Meanwhile, the rest of the columns were defined with the distorted yield strength as in the analytical case. This case is analogous to a typical hybrid simulation experiment, where only few components are tested in the laboratory to give the accurate behavior, whereas the rest of the structure is simulated using analytical models.

The results of the three analyses are shown in Figure 3.2. The plot represents the top nodal displacement of the structure against the duration of the ground excitation. It is obvious that there is an error in the case of the inaccurate analytical solution when the yield strength of the columns was distorted by 10%. However, the hybrid simulation case did not provide valuable improvement to the structural response. The difference between the analytical and the hybrid simulation case is nearly negligible.

This example illustrated that for multi-critical member structures it is not sufficient to test one component only in the laboratory. Hence, the ability of hybrid simulation tests is hindered for such applications, unless an approach is developed to enhance the accuracy of the analytical module.
Figure 3.2 Multi-bay multi-story frame nodal displacements for the different analysis cases

3.1.2. Model updating procedure

Model updating is an approach that aims to modify the numerical model response during the analysis based on the response of the more reliable components. The idea is intuitive and does not entail any further complications of the conventional test. The flowchart shown in Figure 3.3 summarizes the framework of the model updating approach, as follows:

- The structure is divided into three main categories based on the importance and complexity of each component:
  - The first category represents the critical members that are expected to undergo large non-linear deformations. A representative sample of these members is tested in the laboratory. Moreover, a shadow finite element model (FEM) is created for each representative critical member. The FEM model does not participate in the
formulation of the equation of motion, but it is only used to calculate the error in the numerical model compared to the actual response.

- The second category represents the members that share close characteristics to the tested member. As such, these members are simulated numerically and they are updated incrementally based on the response of the corresponding tested members in the first category.

- The third category represents the members that exhibit low level of response mainly in the elastic range and can be accurately represented through the analytical platforms. Therefore, they are simulated numerically without updating their properties during the test.

- Next, each component is analyzed separately to determine the restoring forces, which are either identified from the laboratory measurements in case of the experimental modules, or analytically solved in case of the numerical modules. The equation of motion (EOM) integrates the restoring forces of the different components and determines the deformations applied for the next loading step.

- The deformations are applied to each module based on the categories previously defined including the displacements applied to the tested module \( (U_{\text{Exp}}) \), the displacements applied to the modules to be updated \( (U_{\text{Upd}}) \), and the displacements applied to the numerical modules \( (U_{\text{Num}}) \).

- At each loading step, the error between the responses measured from the physical specimen and the shadow FEM model is evaluated. If the error exceeds a predefined tolerance, model updating will be performed to modify the numerical module parameters; otherwise, the numerical module will proceed using the latest defined parameters. The figure shows that
the restoring forces are used as an example for the evaluation criterion. However, in this research the constitutive model response will be utilized to evaluate the errors, as will be discussed in the following section.

- This procedure is then repeated incrementally through the entire loading history.

![Flowchart illustrating the procedure of model updating](image)

Figure 3.3 Flowchart illustrating the procedure of model updating

The framework of model updating is similar to the conventional hybrid simulation. The main difference is adding a shadow finite element model to compare the numerical and experimental solutions. Then, the identification algorithm determines some action-deformation characteristics to update the numerical modules. To that extent, the model updating procedure does not introduce much complexity to the conventional experiment. It is obvious that the successful implementation of this approach can reduce the errors in the numerical modules, which in turn will enhances the
accuracy of the overall structural response. This can be achieved while saving a lot of unnecessary cost and effort for testing several structural members.

3.1.3. Discussion

Model updating is an approach that aims to modify the behavior of the analytical model, based on the response of the experimental parts that share similar characteristics. In this approach, the analytical modules are usually represented by finite elements. Before discussing the features of model updating, it is important to summarize the main components of finite element analysis. The process of analyzing the structure using the finite element method may be divided into three main components as follows [Cook et al., 2001]:

- Classification: which addresses the physical understanding of the problem to be analyzed. For instance, determining the nature of the problem whether it is subjected to static or dynamic loading has a significant impact on the analysis procedure. In addition, classification of the problem includes addressing the non-linearity, the required output, the level of precision, and selecting the solution algorithm for the investigated model.

- Modelling: where the physical system is represented through analytical idealizations aiming to capture the essential features of the actual component. It is important to achieve sufficient accuracy, while preserving the computational simplicity of the analytical model. Modelling is the core component of FEM, which defines some important features of the structural system such as the geometric configuration, the material properties, the loading representation, the boundary conditions, the constitutive relationships, etc.
Discretization: where the continuous physical model is discretized into smaller meshes of finite elements that are connected at the interface nodes. This approach includes approximation of the actual structural representation and these approximations are more considerable for models with complex non-linear configuration. However, the more fine meshes used in the model, the more reliable the outcome will be and the more computationally involved the problem becomes.

As presented above, the classification procedure lays out the foundation for the assessment of the structural problems, yet it is noted that a number of assumptions are required for the modelling and the discretization components. These assumptions lead to uncertainties in the analytical idealization and hence, experimental tools are considered as more reliable techniques. The concept of model updating takes advantage of the observations that the physically tested component in hybrid simulations bares resemblance to the numerical modules. As a result, the uncertainties in the numerical modules can be reduced through investigating the behavior of the tested components. The framework of model updating can be applied to several characteristics of the assessed structural component such as boundary conditions, hysteretic behavior, and constitutive relationship.

The focus of this thesis is on updating the constitutive relationship behavior as a fundamental property of the numerical module. However, there are several sources of uncertainties in the analytical idealization. These uncertainties are inter-dependent. Therefore, in model updating, the analytical constitutive relationship behavior is modified to account for various sources of uncertainty, not just those associated with the constitutive relationship. In future work, it would be
convenient to segregate the model uncertainties, and to update the analytical model by addressing each source of uncertainty explicitly. There are some limitations in the proposed approach, which can be summarized as follows:

- For problems with complex configuration, loading, and boundary conditions, the modifications in the constitutive relationship will have to represent the uncertainties from several sources rather than just the actual material behavior.
- For structures subjected to combined loading or responding in 3-dimensions fields, the analysis on the stress-strain level would be complicated and it will required advanced analysis tools.
- Uniaxial constitutive models may not be able to represent some failure mechanisms such as buckling of reinforcement bars or shear failures of the tested components.
- The test setup must be suitable to support model updating applications, such as providing the necessary records required to perform the identification procedure. In addition, real-time hybrid simulations will require advance techniques to perform the identification and the updating procedure instantaneously.

In this section, the framework of model updating was introduced. In the following sections, more details about the constitutive model, identification algorithms, and the communication between the different model components will be presented. Additionally, in Chapters 4 and 5 analytical and experimental verification examples will be presented to explore the capabilities of the proposed approach and highlights some of its limitations.
3.2. **Action-Deformation Characteristics**

The numerical modules might not accurately represent the actual constitutive model for a number of reasons. For instance, the actual structural behavior is very complex especially in the non-linear stage or when subjected to combined loading patterns [Collins et al., 1985; Mahin and Shing 1985]. Furthermore, there is uncertainty in the values of the parameters that govern the material stress-strain model, such as the initial stiffness, yielding strength, ductility, etc. For example, it is reported that the manufacturer error in the strength values of steel and concrete might reach 10% [Elnashai and Chryssanthopoulos, 1991; ACI Committee, 2002]. To address these limitations, the model updating approach aims to acquire reliable information from the physically tested component.

There are different action-deformation characteristics that can be learned from the tested component. In literature, it was proposed to identify the hysteretic relationship using Bouc-Wen model [Hashemi et al., 2013; Kwon and Kammula, 2013]. It was also suggested to represent the physical specimen by a rigid bar connected to springs at the end nodes. Then, the parameters governing the spring response can be updated to yield a more realistic response [Yang et al., 2012]. All these approaches modify the member’s global response, which simplifies the identification procedure. However, updating the model on the global level has some limitations, including (1) its inability to analyze the inter element response. (2) The tested and the updated modules must share very close characteristics in terms of boundary conditions and geometric dimensions. (3) The inability of using hysteretic behavior to represent members with different modal parameters. (4) The fact that Bouc-Wen relationship in its basic formula sometimes violates Drucker’s postulate.
of plasticity when it experiences short loading and unloading paths. This fourth limitation is because Bouc-Wen formula might induce displacement drift and negative energy to the model [Charalampakis and Koumousis, 2009].

To address the aforementioned limitation, it is proposed in this research to update the model on a more fundamental level through modifying the material constitutive relationship. This approach aims to identify the parameters that govern the constitutive relationship such as the initial stiffness, the yield strength, the confinement factor, etc. based on the experimental component behavior. Updating the constitutive relationship allows the user to investigate the inter element response. Moreover, the source and the modified modules do not need to fulfill the stringent conditions such as having the same geometric, modal and loading conditions as in the case of updating the model on its global level. Most importantly, updating the constitutive relationship behavior allows flexibility to discretize the structural member into smaller segments. The response of each segment can be modified based on the behavior of the corresponding component in the physically-tested specimen.

In Chapter 2, the simulation coordinator program (UI_SIMCOR) was presented, which will be utilized in this research to communicate between the numerical and the experimental modules of the hybrid simulation test. Furthermore, ZeusNL will be used to analyze the numerical modules. ZeusNL provides several constitutive relationships that for steel and concrete materials [Elnashai et al. 2002]. Amongst others, four models will be considered which are (1) a bilinear elasto-plastic steel model with kinematic hardening (STL1). (2) A modified version of the nonlinear concrete model developed by Mader et al. (CON2) [Mander et al., 1988; Martinez-Rueda and Elnashai,
(3) Ramberg-Osgood steel model with Masing type hysteretic curve (STL2) [Ramberg and Osgood, 1943]. (4) Menegotto-Pinto model with isotropic strain-hardening (STL3).

During hybrid simulation tests, it is required to identify some parameters that govern the behavior of each constitutive model. Figure 3.4(a) shows the bilinear steel model with strain hardening pattern (STL1) and its behavior can be described in terms of three parameters: (1) the initial stiffness (E), (2) the yield strength (Fy) and (3) the strain hardening factor (F). The strain hardening factor defines the material stiffness after it reaches the yielding point. Figure 3.4(b) shows the nonlinear concrete model (CON2), which can be characterized by four parameters: (1) the ultimate compressive strength (Fcu); (2) the ultimate tensile strength (Ftu); (3) the crushing strain ($\varepsilon_{cu}$); and (4) the confinement factor (CF). This factor specifies the confinement level of the concrete core by the steel stirrups. Figure 3.4(c) shows the Ramberg-Osgood steel model, which is defined by Young’s modulus (E), and the three other parameters (a,b and n) that specify the behavior of the back bone curve. These parameters can be obtained by curve fitting of the experimental data [Lowes, 1999]. Finally, Menegotto-Pinto steel model can be characterized by eight parameters as shown in Figure 3.4(d). These parameters define the yield strength, modulus, curvature and isotropic strain hardening. This model can represent some important features, such as Bauschinger effect, buckling and fracture of the reinforcing bars. There is a detailed discussion of the Menegotto-Pinto model in literature [Abdelnaby, 2012; Elnashai et al. 2002].
Each of the stress-strain models included represents a different structural behavior. Hence, they can cover a variety of applications as will be demonstrated in Chapter 4. For future work, it is expected to expand the model updating library with additional alternatives of the material models. Moreover, finite element programs other than ZeusNL can be implemented to give the user the freedom to analyze the structure using the most appropriate platform. After introducing the model updating framework and the action-deformation characteristics that can be updated, the next
section will focus on the different tools that can be used to extract the required information from the tested specimen.

3.3. Identification Algorithm

Determining the physical specimen characteristics is a challenging task. Moreover, this issue becomes more complicated in case of identifying the structural member response on a fundamental level such as the constitutive model behavior. This section, will present a number of alternatives of the identification algorithms. First, the direct approach will be introduced as the simplest application for such algorithms. However, this approach is applied to a limited number of structural problems that satisfy stringent conditions. Therefore, later in the section, more advanced techniques, which are optimization tools and neural networks will be proposed. These techniques are suitable for the constitutive model identification procedure as will be presented.

3.3.1. Direct application

Direct application is the simplest and most intuitive tool for model updating, whereby the global response of some numerical modules is duplicated by the response of the physical component. In order to directly apply the characteristics of one module onto the other, the source and the updated modules must be identical in terms of geometric configuration, applied loading and boundary conditions. For such cases, the global displacements and restoring forces will be the same for these structural components, and hence no need to test all the critical members.
There are few applications that fulfill the stringent requirements for this approach. For example, a building constructed from several duplicates of the same frame, or truss members subjected to the same loading. Although it is not expected to implement the direct approach for many cases, yet it provides a basic understating the model updating procedure.

3.3.2. Optimization tools

Optimization is considered the main approach used to identify the constitutive model parameters in this research and hence, it will be addressed in details. First, the adequacy of optimization as an identification tool are demonstrated. Next, the main components that govern the optimization procedure are presented. Followed by, the difference between the gradient and non-gradient based optimization tools. Then, genetic algorithms and interior point methods are described as the identification techniques used in this research. Finally, some analytical examples will be presented to verify the ability of optimization to handle the constitutive relationship problem.

The constitutive model identification problem is not straightforward; hence, it requires powerful identification tools. The key elements of the identification problem is summarized in terms of four properties as follows: (1) the constitutive equations that defines the relationship between the actions and the deformations. (2) The parameters that define the pattern of the constitutive equations (i.e. initial stiffness, yield strength, etc.). (3) The strain values. (4) The stress values. In the conventional forward problem, the strain and the model parameters are given as input and it is required to determine the stress values as shown in Figure 3.5(a). However for model updating applications, the inverse identification problem must be analyzed, as shown in Figure 3.5(b). In
the inverse problem, it is assumed that the values of the strain and stress values acquired from the experimental component are the input, and then it is required to find the parameters that best fit numerical and experimental responses. Although the forward and inverse approaches share the same elements, yet the inverse problem is much more complicated and requires several iterations to achieve the optimum solution. Optimization algorithms are one of the tools that can be used to solve such problems. Optimization is a technique which aims to determine the values of the decision variables (unknown parameters) that maximize or minimize a certain objective (fitness) function subjected to specific constraints [Haftka and Gurdal, 1992].

Figure 3.5 Constitutive model solution approaches: (a) Forward problem, and (b) inverse problem
A better description of the main elements of the optimization technique can be achieved through discussing the constitutive model application, which is the main focus of this research. These elements can be demonstrated as follows:

- The decision variables (unknowns): are the model parameters such as: the initial stiffness, the yield strength, the crushing strain, etc.
- The objective function: is the difference between the actual stresses and the stresses determined by a theoretical model. This difference is considered as the error in the identified values, and hence optimization algorithm aims to search for the decision variables that minimize the objective function.
- The constraints: are the set of equation that govern the boundaries of the analyzed problem. For example, the upper and lower bound values specify the domain where the solution is located within.

The flowchart shown in Figure 3.6 illustrates the optimization procedure. Where the stresses from the accurate component are compared with the stresses from the mathematical idealization (i.e. the theoretical constitutive model). If the error is at the minimum possible value, the initial parameters defined in the mathematical model are preserved. Otherwise, a search algorithm is processed to find a new set of parameters. It is important to check if the identified vector of parameters satisfies all the constraints (i.e. the upper and lower bounds). Finally, the same steps are repeated iteratively, until the minimum possible objective function is achieved.
The most critical component in optimization is the searching algorithm, which determines the direction that will be followed to find new values for the decision variables in the next increment. In that regards, optimization tools can be classified into gradient and non-gradient based approaches. For gradient-based approaches the derivatives of the objective and the constraint functions are evaluated. These derivatives direct the solution toward the local maxima and minima. In some techniques, the Hessian matrix is also required to determine the minimum solution. Since the searching algorithm is guided mathematically, it is expected to reduce the required calculations to reach the optimum values of the decision variables. However, following a certain search direction might cause the solution to be trapped in a local minimum region. It is worth noting that the accuracy of the optimization output depends significantly on the value of the initial guess. The initial guess needs to help the search tool to determine the global minimum of
the function. Gradient-based approaches include Interior-Point Methods, Steepest Descent, Newton and Quasi-Newton methods [Haftka and Gurdal, 1992].

On the other hand, non-gradient (random) based approaches such as simulated annealing and genetic algorithms are better than gradient-based ones in reaching a global minimum solution. In addition, most of these approaches can be applied to discrete and discontinuous problems because they do not rely on the derivatives of the function. The fact that non-gradient based methods depend on some randomness in searching for the optimum solution helps them to avoid being trapped in a certain zone of the function. Instead, they use random-based approaches to guess the best candidates amongst the feasible solutions in the domain. Although the random search optimizations can solve the local minimum issue, they usually require long computational times. Additionally, in cases of complex systems such as having a large number of decision variables, they might not be feasible [Haftka and Gurdal, 1992].

In this research study, both gradient and non-gradient approaches are included. Specifically, Interior-Point Methods (IPM) is used as a gradient-based tool, and Genetic Algorithms (GA) is used for the non-gradient method. The features of each approach will be discussed in more details as will be shown in later parts of this section. The application of identifying the constitutive model parameters is a small size problem and can be solved using both techniques. As will be shown later in the examples, both IPM and GA were able to determine the optimum solution accurately. However, IPM has a relative advantage in the required calculation time. Hence, it will be the default algorithm used for the following examples. On the other hand, GA is also provided in the model updating program as an alternative for future applications, these applications might include:
(1) non-traditional problems, where the equations include many fluctuations and the IPM cannot avoid local minimum solutions. (2) Problems where the objective function is not continuous or not differentiable. (3) In some problems, the constitutive relationship formulations are defined in an external program and cannot be accessed by the optimization algorithm; then, random based techniques such as GA are suitable to find the optimum solution. Therefore, the frameworks of both IPM and GA will be discussed in the following sections.

In MATLAB® there is a built-in toolbox for IPM and GA; hence, they can be directly utilized for identifying the constitutive model parameters. Each of the two methods has its distinct features. The IPM is a gradient based optimization techniques that can be applied to convex problems. The objective function must be continuous and differentiable. IPM reconstructs the objective function to include self-concordant barriers [Nemirovskii and Todd, 2008]. The IPM can handle sparse problem and always satisfies the constraints. Unlike other methods, it can recover from infeasible solution such as complex numbers. In addition, IPM can deal with large-scale problems because it uses linear algebra, which does not need to operate on the full system matrix. Therefore, it requires small amount of computer memory and storage when dealing with large-scale problems [Byrd et al., 1999]. In IPM, the Hessian matrix must be provided in the solution scheme, and in MATLAB it can be user-defined or can be numerically calculated during the analysis. The default option in the program is the Broyden-Fletcher-Goldfarb-Shanno algorithm (BFGS), which has the advantage of utilizing relatively small storage. The optimization problem can be analyzed using either the direct step or the conjugate gradient step approaches. The verification examples that will be shown later confirms that IPM is very efficient in solving the constitutive model problem within a reasonable computational time.
On the other hand, GA is a non-gradient based approach motivated by nature. This approach provide some randomness in searching for the optimum parameters. The algorithm selects some possible solutions from the entire domain. Then, the ones that show promising potential to include the optimum results have a higher probability to be preserved for the next iteration. Therefore, GA cannot be easily trapped in one searching path, as it relies on the fitness of the candidate solutions rather than direct mathematical formulations. As earlier mentioned, non-gradient based algorithms require long computational time, because they need to explore a wide range of the solution domain. For large-scale problems, non-gradient based methods might fail in reaching the optimum decision variables. GA can handle the proposed constitutive relationships problem in a small number of iterations. As such, this approach will be offered as an alternative for the IPM when solving problems with special features. The procedure that GA follows can be described with the aid of Figure 3.7 as follows [Sastry et al., 2005]:

- For each decision variable (DV) a set of values are assumed between the upper and lower bound domains. The DVs can also be called genes. Using a random approach a gene is selected for each unknown parameter and these values are combined to generate a candidate solution called chromosome (i.e. DVs1).
- The previous step is repeated several times to create a population of chromosomes (i.e. DVs1, DVs2 … DVsN).
- For each chromosome, the objective function is calculated (F1, F2 … FN) and the ones that show potential to include the optimum genes receive a higher probability to be preserved for the next iteration, while the rest of chromosomes are neglected.
• Crossover and mutation are applied to improve the results of the current iteration.
  
  o Crossover: the genes in the parent chromosomes are overlapped together to generate children population with new properties.
  
  o Mutation: some genes are susceptible to have negative impact on the results. Therefore, they are replaced with others from the original solution domain, which can be referred to as mutation in the current chromosome.

• After crossover and mutation a new population chromosomes having the same size of the original one is created.

• Then the new objective functions (i.e. $F_i'$) are evaluated to select the best candidates and same procedure is repeated iteratively until convergence.

![Diagram of solution procedure of generic algorithm](image)

Figure 3.7 Solution procedure of generic algorithms
After discussing the theoretical procedure and the features of both IPM and GA, it is important to verify their efficiency of identifying the parameters in an actual constitutive model problem. Therefore, several numerical examples were analyzed using ZeusNL. Where a cantilever column with a mass concentrated at the top was subjected to an earthquake record. This structural system represents a single degree of freedom structure, and it was analyzed several times with a different column material.

Four material models were used for the analysis, which include (1) a bilinear steel model with kinematic hardening (STL1), (2) a modified Mander model for non-linear concrete (CON2), (3) a Ramberg-Osgood model with Masing type hysteretic curve (STL2), and (4) a Menegotto-Pinto model with isotropic strain-hardening (STL3). This four models were discussed thoroughly in section 3.2. Furthermore, ZeusNL manuals can be referred to for more information about parameters definition [Elnashai et al. 2002]. For each analysis case, the stress-strain values were recorded from the Gauss point subjected to the higher straining actions in ZeusNL. Then, the inverse identification problem was solved, whereby the equation governing the constitutive relationship, the stress values and the strain values were provided to the optimization algorithm. The optimization algorithm aims to find the optimum constitutive model parameters and compares the output using these parameters to the exact solution provided by ZeusNL, as shown in Figure 3.8. The figure illustrates that for all the analyzed stress-strain models, the identified results nearly coincide with the accurate ones. The results were determined using the IPM.
The results illustrated in Figure 3.8 show that optimization is capable of handling the different types of the constitutive relationship problem. The only concern is that the optimization algorithm operated on stress and strain values that are based on theoretical representation. However, in actual specimens, the model parameters and the constitutive relationship behavior do not strictly follow the analytical ones. To have a more realistic verification example, the same optimization problem was solved once more but after adding random noise to the stresses determined by ZeusNL. The random noise followed Gaussian distribution, with a factor of 5% of the maximum stress values. Figure 3.9 shows that optimization was still able to capture the behavior of the stress-strain models, even if the values had some distortions. The error in the identified parameters compared to the accurate ones was lower than 3%. In Chapter 5, it will be shown that the same procedure was applied to actual experimental data and it achieved reasonable results.
Figure 3.9 Optimization results for different constitutive models after adding noise: (a) STL1, (b) CON2, (c) STL2 and (4) STL3

It is worth noting that the results shown in Figures 3.8 and 3.9 were determined using the gradient based interior point methods (IPM). However, the same set or problems was analyzed using genetic algorithms and achieved a similar output. The only difference to be observed is the calculation time. For instance, in the bilinear model example the IPM requires few seconds, while it took about one minute using GA. Hence, IPM method will be used in the following examples as it excels in the computational efficiency. Although the errors were nearly negligible, yet the optimization outcome can be improved through increasing the number of iterations or by reducing
the stoppage tolerance criteria. On the other hand, such approaches will increase the required computational time. In this section, it was shown how the constitutive relationship parameters were identified from the accurate component. In Chapters 4 and 5, these characteristics will be implemented for some applications within the model updating framework.

3.3.3. Neural networks

Identifying the action-deformation properties from an accurate component is a challenging task. Optimization tools were introduced to determine the constitutive model parameters that minimize the error with a reference solution. Neural network offers another alternative that can be used as an identification technique. Unlike optimization, neural networks (NN) do not depend on the theoretical relationships. Instead, it generates a set of mathematical formulas that connects the input data to the output results. However, NN can be considered as an empirical approach; where, the generated mathematical relationships do not give the user a deep understanding about the properties of the studied problem. Thus, it is important to discuss the main features of NN and to explore its possible applications.

The neural networks concept was first inspired by the human brain architecture, which is composed of neurons that interconnect to provide the required knowledge and skills for humans [Ghaboussi et al., 1991]. The fundamental component of NN is the neuron, which can be discretized into three main elements: (1) weight factor \(W\), (2) bias \(B\) and (3) transfer function \(f\), as shown in Figure 3.10. The input is multiplied with a certain weight factor, and then a bias is added to the outcome before it is activated by a transfer function [Yun et al., 2007]. There are several alternatives for the
transfer function. The ones used in this research are the linear, hyperbolic tangent and log-sigmoid functions. NN can be perceived as an inverse identification problem. Whereby, the input and the output values are provided and the network aims to determine the three main components of the neurons that satisfies the given relationship.

![Diagram of a single neuron](image)

**Figure 3.10 The main elements of a single neuron**

Usually, the basic formulation of the neuron cannot handle the actual engineering problems. Instead, several neurons are generated in the same layer, in addition to creating multiple hidden layers as shown in Figure 3.11. The multi-layer NN architect can identify the behavior of models with complicated input requirements and output responses. It can be observed that the simple scalar weights and biases are transferred into matrices; also, the connection scheme between the neurons adds another level of capability to the network [Kim, 2010]. The capacity of the NN is determined by the number of hidden layers, the number of neurons in each layer, and the connection between them. On the other hand, the number of input and output parameters is governed by the physical understanding of the analyzed model [Ghaboussi et al., 1991].
The weight factors are determined iteratively by measuring the error through the forward and the backward propagations. The learning process of NN can be summarized in the following steps [Yun et al., 2007]

1. The feed forward network is activated through supplying the input parameters to the first layer of the NN.
2. Afterwards, the input parameters propagate through the hidden and the output layers, where their values are modified based on the assigned weight factors, biases and transfer functions.
3. The output measured from the forward network is then compared to accurate solution and the error is evaluated.
4. The next step is the back propagation of the evaluated error. Gradient descent method is used to find the share of each connection in the errors. Thereafter, the connections that
are responsible for larger portion of error are given smaller weight factors in the next iteration.

5. The forward and backward propagation procedure is repeated iteratively until the stoppage criterion is satisfied.

In literature, NN was used in many disciplines such as system identification, financial application, and pattern recognition [Kim, 2010]. However, it was not until 1991 when Ghaboussi et al., introduced NN to determine the uniaxial constitutive model behavior for plain concrete. Then, several researchers extended the same methodology to support different action-deformation applications such as determining the hysteretic behavior of materials [Yun et al., 2008a], identifying the moment-rotation behavior for connections [Yun et al., 2008b; Yun et al., 2008c; Kim, 2010] and for geotechnical engineering purposes [Ghaboussi and Sidarta, 1998; Hashash et al., 2006; Johari et al., 2011]. These are some of the examples, amongst others, where NN showed its ability to identify the hysteretic response of structural components with high precision. The framework of the neural network used to identify the action-deformation behavior has evolved over the years. Each application had its distinct NN features based on the physical characteristics of the problem. The focus of this research is directed towards identifying the constitutive relationship parameters, where the NN components will be briefly discussed. For more details about the NN hysteretic model applications, it is recommended to revise the previously listed references.

The main challenge in NN approach is to determine the configuration of the input parameters, the number of neurons, the number of hidden layers and the connection between them. In literature,
some advanced techniques have been proposed such as the nested adaptive neural network (NANN) [Ghaboussi and Sidarta, 1998; Kim, 2010]. For the NANN technique, the nested feature describes the connection between a certain module and the sub-layers related to it, while the adaptive approach is concerned with introducing additional neurons whenever the network requires. For model updating purposes, a simpler procedure is adopted that utilizes the built-in toolbox in MATLAB. The built-in feed-forward network was sufficient to develop the required constitutive relationship responses. In future work, a more advanced approach such as NANN might be included depending on the application of concern.

Figure 3.12 illustrates an example of a NN architect, which will be used in Chapter 4 to represent the bilinear steel model. As shown in the figure, there is an input layer, a hidden layer, and an output layer composed of 5, 3, and 8 neurons, respectively. As suggested by Ghaboussi et al. (1991), each node has weighted connections with all the nodes in the previous layer, which allows the network to handle the complex stress-strain relationship problems. However, one of the main concerns regarding the hysteretic model representation is the one-to-many mapping [Yun et al., 2008b]. The one-to-many mapping means that for each value of strain there are several corresponding values of stresses, and hence the network requires the tools to evaluate the correct path that includes the appropriate stress. Therefore, it is crucial to select the input parameters that can describe the physical characteristics of the hysteretic model problem.
The neural network architecture shown in Figure 3.12 is based on five input requirements, which are: (1) $\varepsilon_{i-1}$: strain in the previous step, (2) $\sigma_{i-1}$: stress in the previous step, (3) $\varepsilon_{i}$: strain in the current step, (4) $\xi = \varepsilon_{i-1} \times \sigma_{i-1}$: hysteretic energy in the previous step, and (5) $\eta = (\varepsilon_{i} - \varepsilon_{i-1}) \times \sigma_{i-1}$: additional energy for the current step. A better illustration for the NN input parameters is shown in Figure 3.13. There are six different possibilities for the loading and unloading paths in the material hysteretic model. However, the network can identify the appropriate loading/unloading path depending on the sign conventions of the input parameters. Using this approach the one-to-many mapping issue can be solved and hence, the NN can efficiently handle the hysteretic model problems. The logic behind this network configuration and its ability to analyze several engineering applications is discussed thoroughly in literature [Kim, 2010; Ghaboussi et al., 1991; Yun et al., 2008b].
In Chapter 4, some analytical examples will be presented utilizing the proposed NN network procedure. The main difference between the optimization and NN verification examples is that NN could not be applied in a stepwise manner. In another words, the whole constitutive relationship history is provided to the network in order to be efficiently trained. Therefore, it might be reasonable to conduct a simple quasi-static experiment before the main hybrid simulation test. Then, the NN developed from this experiment can be used to represent the analytical modules in the hybrid simulation test to achieve an accurate response.

Up to this point, the main engine for the model updating approach is developed, where different alternatives for the action deformation identification techniques were introduced. Optimization tools showed a very good potential to modify a pre-existing constitutive model to match the accurate structural behavior. On the other hand, NN is useful in the absence of any analytical
representation of the accurate module, as it can mathematically generate the intermediate relationships between the input and output parameters. Nevertheless, NN can be considered as a black box, which does not help in understanding the theoretical behavior of the investigated component. The remaining component of the model updating is to integrate all the presented schemes in one global framework.

3.4. Developed Software

UI-SIMCOR is a MATLAB® based program used to coordinate between the numerical and the experimental components in hybrid simulations [Kwon et al., 2007]. For model updating purposes, UI-SIMCOR was modified to be able to exchange information about the stress-strain information and to update the numerical module parameters during the analysis. Additionally, the identification approaches such as optimizations and neural networks were included to determine the parameters required for updating the model. In order to achieve these results, UI-SIMCOR and ZeusNL source codes were modified to allow for the required communication between the two platforms. In this section, the capabilities, input requirements, as well as the output and feedback of the software developed to support model updating problems will be presented. Moreover, some of the challenges encountered while modifying ZeusNL source codes will be discussed.

3.4.1. Capabilities

In conventional hybrid simulation tests, UI-SIMCOR and Zeus-NL are directly connected through a TCP/IP protocol [Kwon et al, 2008]. Figure 3.14, shows a schematic diagram for UI-SIMCOR,
which can exchange the deformations and restoring forces between the numerical (i.e. ZeusNL) and experimental modules. As can be observed from the figure, the purpose of model updating is to learn the constitutive relationship behavior from the tested component and to send it to the analytical platform. This additional step requires some modifications in the source codes of both programs.

Figure 3.14 The main elements of model updating approach

Opening a new port to exchange the stress-strain data was a challenging task. The protocol that connects UI_SIMCOR to ZeusNL is referred to as Network Interface for Console Applications (NICA). However, the source codes of NICA protocol is not accessible and hence, it cannot modified. This issue was addressed through developing an external loop to support model updating applications such as receiving the stress-strain values from ZeusNL and sending the identified
parameters to it. Figure 3.15 shows a schematic diagram for the framework adopted to communicate between the different programs. The subroutines of interest in UI-SIMCOR and ZeusNL are the Transient $\alpha$-OS integration scheme and the dynamic analysis algorithm, respectively. The procedure followed in this problem can be summarized as follows:

- At the beginning of the analysis, UI-SIMCOR and ZeusNL run in parallel. In UI-SIMCOR, the transient integration scheme predicts the deformations for the current loading step based on the response of the previous increments. Meanwhile, the dynamic analysis algorithm in ZeusNL initializes the program and then it is paused by NICA waiting for any response from UI-SIMCOR.
- To take advantage that ZeusNL is in a pending condition, a new MATLAB file is processed to run the identification algorithms, which determines the new parameters that will be used in the next loading increment. The new parameters are saved in an external text file.
- In the transient $\alpha$-OS method, the predicted deformations are sent to the analytical program through NICA. At this stage, UI-SIMCOR will be paused waiting for the feedback from ZeusNL.
- When ZeusNL receives the nodal deformations, it triggers the subroutines that will enquire about the text file and read the necessary parameters to modify the constitutive relationship.
- Next, ZeusNL run the main solution algorithms to determine the restoring forces on each module that corresponds to the applied deformations.
• The stresses and strains used to calculate the restoring forces are sent to another text file, which will be analyzed in later steps.

• NICA then receives the restoring forces of the analytical module and sends it back to the integration scheme. In addition, this file also triggers UI-SIMCOR to resume the remaining calculations.

• After receiving the restoring forces from the different hybrid simulation components, the equivalent forces are determined, which will be used in the corrector step by the transient $\alpha$-OS algorithm.

• Finally, all the acquired values are used to initialize the next integration increment. Meanwhile, ZeusNL saves the output response and holds in a waiting mode for the deformation from the NICA console.

This framework was developed because NICA source codes could not be accessed. Instead, the feature that NICA can pause both platforms was advantageous to open an external loop that is capable of exchanging the stress-strain information. To achieve this objective, the data was saved in an explicit text file, which can be enquired whenever UI-SIMCOR or ZeusNL are ready to conduct the solution algorithm. It is worth noting that this procedure slightly increases the communication time between the two programs. However, for future applications it would be more convenient to utilize NICA directly to exchange the required data. This approach can be crucial in case of conducting a real time hybrid simulation experiment, where the time efficiency is a challenging task.
Figure 3.15 A schematic diagram for the model updating console

The purpose of the modified communication protocol is to send the new constitutive relationship information to ZeusNL, which in turn will use it to determine the analytical solution of the structural module. The constitutive relationship information can be either the modified model parameters (i.e. Young’s modulus, yield strength, etc.) identified from the optimization algorithm,
or the network configuration in case of utilizing the neural network approach. Initially, several convergence issues were experienced, where ZeusNL could not find the state of strains and stresses that satisfies equilibrium with the applied deformations and forces. In order to solve this problem, the solution procedure adopted by ZeusNL was briefly investigated.

In ZeusNL, the elasto-plastic cubic formulation are used to represent the material non-linearity and the spread of plastic hinges is analyzed in the Eulerian system [Izzuddin and Elnashai, 1993]. In this approach, the local axes are not fixed; they track the motion of the structural component, which is efficient in handling large deformation problems. The solution steps can be summarized as follows. First, the nodal displacements are determined from the integration scheme. Then the strains, rate of twist and curvatures are calculated. Next, the strains are generalized for two Gauss points, where two points are sufficient because this procedure is developed for short length members. Fiber analysis method is used to identify the stresses based on the material tangent modulus and hence, the local forces on the structural members are calculated. Finally, the global characteristics of the structure are determined [Izzuddin and Elnashai, 1993]. This procedure is repeated until convergence is achieved, where the out-of-balance forces at each iteration are used to correct the formulas in the next step.

By revising the solution algorithm adopted by ZeusNL it was observed that the tangent modulus does not appear only to determine the stress based on the given strains. The tangent modulus is also required in the iterative algorithm to get the out-of-balance forces. This finding justified the convergence issue experienced after modifying the constitutive model parameters, where the tangent modulus was modified for stress calculations only, while a different value was utilized for
the iterations and correction steps. Consequently, equilibrium was not achieved and the program could not converge. After adjusting the tangent modulus in the different subroutines, the program processed robustly.

This section highlighted the main routine of the model updating program. Additionally, the challenges occurred during the analysis and the possible solutions were introduced. Next, some features such as the input requirements and the output responses will be presented.

3.4.2. Input requirements

The implementation of the model updating approach required significant modifications to the conventional hybrid simulation procedure. Additionally, several action-deformation characteristics can be determined during the test and there are different tools that can be used to identify them, such as optimization algorithms or neural networks. In order to integrate the different model updating components and to make them easily accessible, a MATLAB based graphical user interface program (GUI) was developed. The GUI simplifies the definition procedure for the input requirements. Moreover, it gives the user control over the procedure as it allows for observing the output responses. A summarized description for the main components of the input requirements are listed below:

- At the end of the original hybrid simulation configuration file, additional code line must be added to trigger the model updating program (i.e. Sys.Enable_updating=1). Then the new GUI will be launched rather than conventional UI-SIMCOR interface.
The structure is divided into several modules. The program will ask the user to specify the source and updated modules. The source modules are assigned such that they represent the more reliable behavior, which will be identified.

Next, each updated module will be matched with the corresponding source component. In many cases, the same source module will be used to modify more than one analytical component.

In actual physical specimens, the whole member is not expected to behave in the same manner, because each segment is subjected to a different level of loading or even represent a different geometric properties. For example, the zone of the plastic hinge might not exert the same constitutive behavior as another section subjected to small loads. Therefore, the program allows the user to divide the numerical module into several segments and to relate them to the corresponding ones in the physical specimen.

For each source module or segment, the identification tool must be specified; either optimization (genetic algorithm or interior point methods) or neural networks, and for each case some features need to be provided as follows:

- For optimization tools, the constitutive relationship must be selected from the available alternatives (i.e. STL1, STL2, STL3 and CON2). Additionally, the upper and lower bounds for the decision variables must be specified. These values determine the search domain for optimization. In addition, these bounds prevent the solution from reaching illogical values (i.e. negative yield strength or Young’s modulus).

- For neural networks, it is required to define the configuration of the network, such as the transfer functions, the number of hidden layers and the number of neurons in
each layer. Moreover, some features such as the number of epochs and the stoppage criteria must be provided.

- A threshold value is specified to determine whether the identification algorithm need to modify the constitutive model parameters or not. Whereby, the error between the source and the updated modules is calculated. If the error exceeds the predefined threshold value, this implies that the identification process is required, otherwise it continues with the last defined parameters. This approach saves unnecessary computations, and identification is only applied when an abrupt change occurs in the behavior of the accurate component.

- Finally, the user selects either to run all the analyses directly in one step or to perform the analysis in several steps (i.e. determine modules stiffness, static loading and dynamic loading), which was also provided in the original UI-SIMCOR version.

These are the main input requirements for the model updating program. It must be noted that some improvements can be developed in future work. For example, the program was customized for each application separately. Whereby, the version of the program used in the optimization problems is different than the one used for the neural networks. A framework can be developed to generalize the program to support different applications. In addition, it is important to address the communication protocols to handle actual experimental modules, as for now it was applied to analytical examples. Therefore, there was no need in this research to exchange data with physically tested components. The following section presents the GUI output, which helps in analyzing the hybrid simulation problem.
3.4.3. Output and feedback

The output of the program shows the nodal displacements of the structure. Moreover, it illustrates the stress-strain relationship for the source module compared to the updated one, as shown in Figure 3.16. The identified constitutive model parameters are also included within the output information. It is crucial to give the user the ability to observe the feedback from the tested specimen, because experimental investigations are path dependent, which implies that any illogical results at any loading increment will be preserved and might escalate in future steps.

![Figure 3.16 constitutive relationship: (a) steel and (b) concrete in a model updating problem](image)

As demonstrated in this discussion, the developed model updating program combined the different components of the proposed framework. It can communicate with the numerical and experimental modules and it can handle the transfer of the stress-strain information between them. The optimization and neural network identification tools are also integrated. All these elements are incorporated in a user-friendly graphical user interface program to facilitate the analysis of model updating applications.
3.5. Range of Applications

The general idea of model updating is to promote the performance of a simple structural component to match the behavior of a more detailed one. This concept was utilized in many numerical applications. However, this concept was not popular in the experimental environment as the tests were usually conducted in real time and model updating requires additional time to identify the system behavior. The development of hybrid simulation, which can be executed at slow rates, motivated the implementation of the model updating.

Several applications can be designed to benefit from model updating approach. For example, structures that include more than one critical member can be investigated by testing one component only and updating the others that share close characteristics. This technique can help in reducing the time, cost and effort required for the conventional hybrid simulation tests. Additionally, it can help in overcoming the limited capacity of testing facilities. Another analytical application is to represent a detailed system with a much simpler model, provided that this model will be updated incrementally to acquire the behavior of the accurate component. In Chapter 4, a numerical example will be analyzed to demonstrate that model updating can be applied to modify the response of bilinear steel model to achieve similar performance of the more complicated Ramberg-Osgood model.

On the other hand, neural networks can be used to learn the response of the material hysteretic behavior of the structural elements. This method is advantageous in case the available idealizations
are not suitable to represent the actual model. Therefore, the neural network configuration and parameters can be trained to match the accurate component response. Then, the identified network parameters are used to update the numerical model in a stepwise manner. Model updating can be also used to investigating a local region in a structural element, and then to use it in representing the rest of the model. In one of the verification examples, it will be shown that behavior of heat treated beam section was identified form experimental data, which was used in updating the numerical model afterwards. Therefore, there is no need to test the entire component; instead, a small segment can be tested in a hybrid simulation test and its behavior can be duplicated in the numerical module.

Other applications, such as identifying the constitutive model parameters from a hybrid simulation test, will be discussed in subsequent sections. However, it is important to note that model updating concept is not restricted to hybrid simulation applications only. There are various applications that can be solved using similar method in the finite element field such as mesh resizing to reduce the computational time. In addition, there are soil mechanics examples that utilize tools analogous to model updating procedure [Andrade and Tu, 2009].

3.6. Summary

In this chapter, different aspects of model updating approach were discussed including; (1) the purpose and the framework of model updating. (2) The action-deformation characteristics, which describe the behavior of the tested member. (3) A number of identification algorithms that can be utilized to determine the characteristics of the tested member. (4) The features and the capabilities
of a newly developed program that supports model updating procedure. (5) The range of applications that can benefit from the proposed approach. Some conclusions can be highlighted as follows:

- Model updating is an approach that aims to improve the performance of the numerical modules based on the information acquired from the physically tested component. The effectiveness of approach requires that the updated and the source module share close characteristics.

- Four constitutive models are included in the scope of the current research, namely: (1) the bilinear steel model with kinematic hardening (STL1), (2) the non-linear concrete model with active confinement (CON2), (3) Ramberg-Osgood model (STL2) and (4) Menegotto-Pinto steel model (STL3). For future applications, more stress-strain relationships can be added to serve a wide range of problems.

- Two identification techniques were proposed to determine the behavior of the accurate module, namely (1) optimization tools, which are utilized when a constitutive relationship exists and this relationship can be modified to best match the response of the accurate component. (2) Neural network, which has the advantage of developing a mathematical formulations that find the relationship between the input and the output parameters.

- For optimization tools, two approaches were employed that represent gradient and non-gradient based methods. The gradient-based tools (interior point method) provide a fast algorithm to reach the optimum solution. Meanwhile, the non-gradient based methods (genetic algorithms) require long computational time, yet they impose much less restrictions on the objective function. Since, both methods were able to determine the
model parameters effectively; the interior point methods will be used to benefit from its fast computation abilities.

- The proposed neural network configuration was designed to address the one-to-many mapping issue, where for the hysteretic model problems; each value of strain corresponds to several values of stresses. However, the input parameters suggested in literature were considered such that their sign conventions can guide the solution to the correct loading/unloading path.

- UI-SIMCOR is utilized as the simulator coordinator program for hybrid simulation tests, which can communicate with several analytical platforms. For this research, ZeusNL is the analytical platform used to represent the numerical module.

- UI-SIMCOR was modified to be able to exchange the stress-strain information between the experimental and numerical modules. Nevertheless, some challenges were discussed related to the communication protocol between UI-SIMCOR and ZeusNL.

- A graphical user interface program (GUI) was developed to integrate all the different components of model updating. The input requirements, as well as the output and feedback features of the program were presented. These features were designed to be user friendly and to allow for the observation of the output characteristics. The GUI was customized to serve each application separately. In future work, it is recommended to combine the different versions of the program in one toolbox that can handle the different model updating problems.

- Finally, a number of applications were introduced that draw potential benefit from the proposed model updating approach. The main concept behind all the suggested applications was to modify the performance of a simple analytical component to behave
in more complicated manner. The focus of this study is to update the constitutive relationship parameters, which addresses the model response on a fundamental level. Such approach gives much flexibility and allows for wide variety of applications.

As shown in this chapter, model updating is an approach that aims to enhance the ability of conventional hybrid simulations to assess structures in a more reliable way. It can be used to analyze structures with several critical members such as long span bridges and high-rise buildings. In the following chapters, some analytical examples will be presented. In addition, the data of previously conducted experiments will be investigated in an effort to verify the model updating concept.
CHAPTER 4
NUMERICAL VERIFICATION

The framework of model updating was introduced, which showed its potential to improve the conventional hybrid simulation results. Nevertheless, it is crucial to assess the efficiency of the proposed approach through analyzing several applications. In this chapter, a number of numerical examples are presented to demonstrate various features of model updating. These examples are designed to cover a wide variety of applications for regular and irregular structural configurations. The general concept of all the problems is to determine the characteristics of one component and impose them on the rest of the members.

4.1. Constitutive Model Examples

In this section, model updating approach is applied to modify the behavior of four different constitutive relationships. The analytical platform used for these examples is ZeusNL and the investigated constitutive relationships are the bilinear steel model (STL1), Ramberg-Osgood model (STL2), Menegotto-Pinto model (STL3), and the non-linear concrete model (CON2). The purpose of these examples is to verify the model updating framework and to ensure the robustness of the communication protocol between the different model components. For verification purposes, the analyzed system is designed to represent a simple geometric configuration, which is a symmetric two-bay one-story frame. The results of these examples will be discussed to explore the advantages and the challenges encountered while adopting the proposed approach.
4.1.1. Bilinear steel model

Zeus-NL was used to analyze a two-bay one-story frame, as shown in Figure 4.1. The beam was defined to be rigid, while the columns were defined using a bilinear steel model. The middle column supports a mass of 20 N/(mm/sec^2), while each of the end columns supports half this mass. The structure was subjected to the North-South component of the Imperial Valley earthquake, which occurred in May 18, 1940 [Chopra, 1995]. The geometric configuration of the frame and the cross-sectional properties of the columns are shown in Figure 4.1. The top nodes of the frame will experience the same displacements due to the rigidity of the beam. The objective of this example is to identify the parameters governing the constitutive relationship of the left column and to use this information to update the other two columns. As previously mentioned, the three parameters governing the bilinear steel model are the initial stiffness (E), the yield strength (F_y) and the strain hardening factor (F). In the conventional hybrid simulations, the physically tested component represents the accurate behavior of the member. However, for analytical examples, the accuracy of the structural module depends on the parameters assigned by the user to define the model response. Therefore, this problem was solved three times to represent the different analysis cases as follows:

- Exact case: The bilinear model parameters (i.e. E, F_y and F) for the three columns were given certain values, which were assumed to represent the exact solution. The structure was subjected to the input excitation and the output of this system was recorded. For all the following cases, the recorded results are considered the reference solution to evaluate the accuracy of the other cases.
- Hybrid simulation case: the left column stress-strain model parameters were defined with the same accurate values assumed in the exact case, but for the other two columns a 10% error was imposed to account for the uncertainties in the material properties and the non-linear behavior of the structural component. This case is analogous to experimental hybrid simulation applications, where one element represents the accurate response (i.e. the tested component), while the rest of the structure is modeled based on the theoretical understanding, which usually deviates from the realistic behavior.

- Model updating case: The configuration and the parameter definition of this case are the same as in the hybrid simulation case. The main difference is that the middle and left columns (i.e. the components represented by the inaccurate parameters) will be instantaneously updated during the analysis. Whereby, at each loading step, the optimization algorithms will identify the constitutive model parameters from the left column (source module) and use them to update the constitutive model in the other two columns.

It should be noted that for model updating, the identified parameters are only applied to the structural elements that share close characteristics to the source module. For example, in this problem the three columns share similar geometric properties, boundary conditions, and subjected to similar loading combinations. Through updating the constitutive model, these conditions do not need to be strictly satisfied. It is sufficient if the source and modified module share overall close behavior as will be shown in later examples. The values utilized to define the constitutive relationship parameters for the three analysis cases are listed in Table 4.1.
After running the three analyses, several features can be considered to evaluate the response of each analysis cases such as the nodal displacements, restoring forces, modal properties, hysteretic response etc. However, there is no need to show the results of all these features, because the accuracy of one of them will imply the accuracy of the defined constitutive relationship. Hence, the displacement of the top node will be used to assess the response of the model. Since, the beam is rigid, all the columns produce the same top nodal displacements. Figure 4.2 shows the results of the three analysis cases, and several observations were noted as follows.

Table 4.1 STL1 model parameters for the different analysis cases

<table>
<thead>
<tr>
<th>Module</th>
<th>Left Column</th>
<th>Middle Column</th>
<th>Right Column</th>
</tr>
</thead>
<tbody>
<tr>
<td>Factors</td>
<td>E (MPa)</td>
<td>F (MPa)</td>
<td>F</td>
</tr>
<tr>
<td>Exact</td>
<td>200000</td>
<td>140</td>
<td>0.10</td>
</tr>
<tr>
<td>Updated</td>
<td>200000</td>
<td>140</td>
<td>0.10</td>
</tr>
<tr>
<td>Hybrid simulation</td>
<td>200000</td>
<td>140</td>
<td>0.10</td>
</tr>
<tr>
<td></td>
<td>200000</td>
<td>140</td>
<td>0.10</td>
</tr>
<tr>
<td></td>
<td>210000</td>
<td>160</td>
<td>0.12</td>
</tr>
<tr>
<td></td>
<td>210000</td>
<td>160</td>
<td>0.12</td>
</tr>
<tr>
<td></td>
<td>210000</td>
<td>160</td>
<td>0.12</td>
</tr>
<tr>
<td></td>
<td>210000</td>
<td>160</td>
<td>0.12</td>
</tr>
</tbody>
</table>

Figure 4.1 Geometric configuration of a two bay steel frame and the cross sectional dimensions of the columns
There is a clear gap between the results of the hybrid simulation case compared to the exact response. On the other hand, through applying the model updating approach, the structural response improved significantly, and the results nearly coincide with the exact solution.

For the hybrid simulation case, the nodal displacements are on average smaller in value than the exact case. This pattern can be justified because $E$ and $F_y$ defined for the hybrid simulation case are stiffer than the accurate values.

Due to the stiffer parameters in the hybrid simulation case, a phase lag can be observed in the structural response. It is clear that the inaccurate constitutive relationship affected the dynamic modal properties of the system.

At the beginning of the analysis, the differences in responses were negligible. However, as the analysis proceeded, the error kept accumulating and the worst behavior occurred near the end of the analysis. In addition to the error accumulation, abrupt changes occurred in the displacement values for the hybrid simulation case. Probably these abrupt errors were induced to the model at critical loading stages, such as the yielding point and the non-linear loading/unloading paths.

These observations highlight some important aspects of the model updating approach. Usually for hybrid simulations, the attention is placed on the physically tested component. In this example, it was shown that the behavior of the analytical module affected the overall structural response, including the accurate module itself. On the other hand, in the model updating analysis case, the constitutive relationship parameters were acquired from the accurate column to modify the behavior of the other components, which had a positive impact on the global structural behavior. Another observation is the accumulation of errors in the system, because the errors that occur at
any loading step cannot be recovered in future steps. Therefore, it is preferred to select the source element to be ahead of the modified sources in terms of loading. Hence, it provides enough time for the identification algorithm to determine the accurate model parameters, before the other components experience the same level of loading. Moreover, it is recommended to use small time increments to allow for more analysis steps in the model updating process. The final observation is specific to this example, where the value of the yield strength had the most critical effect on the accuracy of the nodal displacements. This is because in the exact case, some loading cycles caused yielding in the material and inelastic response was observed. However in the hybrid simulation case, the yield strength was defined with higher values and hence for the same loading stages the member responded in the elastic range.

![Graph](image)

**Figure 4.2** The nodal displacements for the STL1 frame subjected to the different analysis cases

**Figure 4.3** shows the error percentage normalized by the peak response for the different analysis cases. The average absolute displacement errors in the hybrid simulation and the model updating
cases are 12.8 mm and 1.21 mm, respectively. It is clear that the pattern of error propagation keeps accumulating in the hybrid simulation case. The maximum normalized error in the hybrid simulation analysis is 36.1% compared to 3.8% in the model updating case. These values demonstrate the impact of the numerical module response in path dependent problems. Therefore, model updating is an excellent approach that can be utilized to enhance the results of the conventional hybrid simulation tests.

Figure 4.3 Error comparison in the STL1 frame for the conventional and the model updating cases

As discussed in Chapter 3, both IPM and GA can be used as optimization tools to analyze this problem. Both techniques gave nearly the same results; however, as earlier mentioned IPM is preferred due to its relative advantage in the required computational time. It is important to mention that; a constraint was imposed on the optimization tool to assign the upper bound value for the
yield strength at the elastic stages, until the source column experience yielding. This approach is adopted to avoid early yielding in the updated columns.

4.1.2. Ramberg-Osgood model

The problem configuration presented in the previous example is preserved for the other two steel constitutive relationships problems (i.e. Ramberg-Osgood and Menegotto-Pinto models). The geometric properties of the analyzed frame is same as the one illustrated in Figure 4.1, except for replacing the column material from the bilinear steel model (STL1) by the Ramberg-Osgood (STL2) model. STL2 represents the model non-linearity with a higher level of details than STL1. STL2 response is governed by four parameters, which are Young’s modulus (E), in addition to three parameters a, b, and c, represents the hysteretic behavior of the material model. As described in the earlier example, the frame was subjected to an input earthquake motion, and it was analyzed for the exact, hybrid simulation and model updating cases. The values used for constitutive relationship parameters are summarized in Table 4.2.

<table>
<thead>
<tr>
<th>Module</th>
<th>Left Column</th>
<th>Middle Column</th>
<th>Right Column</th>
</tr>
</thead>
<tbody>
<tr>
<td>Factors</td>
<td>E</td>
<td>a</td>
<td>b</td>
</tr>
<tr>
<td>Exact</td>
<td>200000</td>
<td>0.001</td>
<td>120</td>
</tr>
<tr>
<td>Updated</td>
<td>200000</td>
<td>0.001</td>
<td>120</td>
</tr>
<tr>
<td>Hybrid sim,</td>
<td>220000</td>
<td>0.001</td>
<td>120</td>
</tr>
</tbody>
</table>

Figure 4.4 shows the nodal displacement at the top of the left column plotted against the time series. It is observed that the model updating case provided output results that nearly coincide with the exact solution. Few remarks are noted by comparing the results of this example with the
previous one, which are (1) the nodal displacement values and the time history pattern changed dramatically through replacing the stress-strain model from STL1 to STL2. This behavior is expected because STL1 cannot represent the same level of complexity as STL2, either in the linear or non-linear stages. Therefore, it is extremely important, even for the conventional hybrid simulation test, to assign the most suitable constitutive model in the analytical component. (2) The error in the hybrid simulation case in this example was not as obvious as in the previous example. This observation can be justified as in the STL1 model, yielding occurred in the hybrid simulation case several cycles later than the accurate case. Therefore, this error was preserved and it kept accumulating throughout the loading history. On the other hand, due to the stochastic nature of the input loading, this issue was not significant while using STL2 model.

![Figure 4.4 The nodal displacements for the STL2 frame subjected to the different analysis cases](image-url)
Figure 4.5 shows the percentile error normalized to the peck response. The average absolute error was reduced from 8.10 mm in the hybrid simulation case to 0.33 mm when model updating was utilized. These values emphasize the ability of updating the numerical module to achieve a much better structural response. Meanwhile, the maximum peak error reduced from 10.2% in hybrid simulation case to 0.8% in model updating case. The maximum peak error values show that in the model updating case the errors were nearly negligible. By comparing the results of this example to the STL1 model, it is noted that the error in the hybrid simulation case was accumulating. However, in this problem the error keeps fluctuating around the reference solution. This fluctuating pattern confirms the previous conclusion that the residual stresses due to early yielding had the major impact in the STL1 example.

Figure 4.5 Error comparison in the STL2 frame for the conventional and the model updating cases
4.1.3. Menegotto-Pinto model

Following the procedure of the previous two example, the same frame is utilized herein, but the columns are defined using Menegotto-Pinto model (STL3). STL3 has the ability to represent Bauschinger effect and the fracture in reinforcement bars. Its behavior is governed by eight parameters, which characterize the elastic stiffness, yield strength, curvature and isotropic strain hardening. The parameters assigned for the different analysis cases are listed in Table 4.3. The nodal displacement of the left column for the three analysis cases are shown in Figure 4.6.

The figure shows that at the early stages of loading, the errors in the nodal displacements for the different cases were minimal, but as the analysis proceeded, especially at the non-linear stages, the errors in the hybrid simulation case became significant. This behavior was expected because most of STL3 parameters characterize the non-linear model behavior rather than the elastic response such as the curvatures, the isotropic strain hardening, and the hysteretic response. In the hybrid simulation case, these parameters were defined inaccurately and hence, the non-linear behavior was affected considerably.

<table>
<thead>
<tr>
<th>Analysis</th>
<th>Exact case</th>
<th>Model updating case</th>
<th>Hybrid simulation case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Column</td>
<td>Left</td>
<td>Middle</td>
<td>Right</td>
</tr>
<tr>
<td>$E_o$</td>
<td>200000</td>
<td>200000</td>
<td>200000</td>
</tr>
<tr>
<td>$F_y$</td>
<td>120</td>
<td>120</td>
<td>120</td>
</tr>
<tr>
<td>$R_o$</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
<tr>
<td>Param1</td>
<td>18.5</td>
<td>18.5</td>
<td>18.5</td>
</tr>
<tr>
<td>Param2</td>
<td>0.15</td>
<td>0.15</td>
<td>0.15</td>
</tr>
<tr>
<td>Param3</td>
<td>0.01</td>
<td>0.01</td>
<td>0.01</td>
</tr>
<tr>
<td>Param4</td>
<td>7</td>
<td>7</td>
<td>7</td>
</tr>
</tbody>
</table>
Figure 4.6 The nodal displacements for the STL3 frame subjected to the different analysis cases.

Figure 4.7 shows the errors in the hybrid simulation and model updating cases. The average absolute error in the nodal displacements dropped from 18.33 mm in the hybrid simulation case to 1.48 mm, when model updating approach was applied. Meanwhile the peak error was reduced from 40.3% to 5.84% for the same cases. These results show model updating approach has a significant impact in reducing the errors in the numerical module compared to the conventional hybrid simulation results.

It must be noted that, the behavior of STL3 model defined in ZeusNL was initially unrealistic, as a sudden drop was observed at the hysteretic response. This issue was due to an error in the source codes of the program, where the curvature parameter was defined in two different subroutines in ZeusNL, which caused some conflict in the solution algorithm. Once this problem was addressed, the results STL3 model became realistic.
4.1.4. Nonlinear concrete model

In this example, the model configuration is different from the previous ones. The structure is a two-bay one-story frame analyzed by Zeus-NL but with modified geometric properties as shown in Figure 4.8. A concentrated mass of 1500 N/(mm/sec^2) is added at the top of each of the end columns and a mass of 3000 N/(mm/sec^2) is added on the middle column. The structure is subjected to Elcentro earthquake, as mentioned in the previous examples. The beam is rigid and the columns are modeled using the modified Mander concrete relationship (CON2). CON2 is a more complicated model than the other steel models as it requires the assignment of three different elements (1) the reinforcement bars, (2) the confined concrete core, and (3) the unconfined concrete cover. The reinforcement bars are defined using the bilinear steel model. Meanwhile, the concrete core and cover are defined using CON2. The parameters that govern CON2 are the
ultimate compressive strength ($F_{cu}$), the ultimate tensile strength ($F_{tu}$), the crushing strain ($\varepsilon_{cu}$) and the confinement factor (CF).

The constitutive model parameters used to define the reinforcement bars and the concrete core are shown in Tables 4.4 and 4.5, respectively. As in the previous example, an error of about 10% is induced in the model parameters for the hybrid simulation and model updating cases to account for the uncertainties in the analytical response. However, in this example, model updating will be applied to the confined concrete core and steel bars only, since the concrete cover has a small contribution on the column response.

![Figure 4.8 Geometric configuration of a two bay reinforced concrete frame and the cross sectional dimensions of the columns](image)

Table 4.4 Reinforcement bar model (STL1) parameters for the different analysis cases

<table>
<thead>
<tr>
<th>Module</th>
<th>Left Column</th>
<th></th>
<th>Middle and Right Columns</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Factors</td>
<td>E (MPa)</td>
<td>$F_y$ (MPa)</td>
<td>F</td>
<td>E (MPa)</td>
</tr>
<tr>
<td>Exact</td>
<td>200000</td>
<td>360</td>
<td>0.1</td>
<td>200000</td>
</tr>
<tr>
<td>Updated</td>
<td>200000</td>
<td>360</td>
<td>0.1</td>
<td>210000</td>
</tr>
<tr>
<td>Hybrid sim.</td>
<td>200000</td>
<td>360</td>
<td>0.1</td>
<td>210000</td>
</tr>
</tbody>
</table>
Table 4.5 Concrete core model (CON2) parameters for the different analysis cases

<table>
<thead>
<tr>
<th>Module</th>
<th>Left Column</th>
<th>Middle and Right Columns</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Factors</td>
<td>F$_{cu}$ (MPa)</td>
</tr>
<tr>
<td>Exact</td>
<td></td>
<td>40</td>
</tr>
<tr>
<td>Updated</td>
<td></td>
<td>40</td>
</tr>
<tr>
<td>Hybrid sim.</td>
<td></td>
<td>40</td>
</tr>
</tbody>
</table>

The results of the nodal displacements representing the three analysis cases are shown in Figure 4.9. It is obvious that the error in the hybrid simulation case was much larger than the model updating cases. The errors in the hybrid simulation case were negligible at the beginning of the analysis, but they kept accumulating throughout the analysis. However, when model updating was applied, the results improved considerably. It is important to note that, optimization required longer computational time compared to the steel examples. This issue is due to the following reasons; (1) for concrete models; optimization is applied for both CON2 and STL1, simultaneously. (2) The formulation of CON2 is much more complicated than STL2 model. Therefore, optimization required longer time to identify the model parameters.

Figure 4.9 The nodal displacements for the CON2 frame subjected to the different analysis cases
The average absolute displacement errors in the hybrid simulation and the model updating cases were 15.44 mm and 2.56 mm, respectively. Figure 4.10 shows the percentile error normalized to the peak response for the different analysis cases. It can be observed that the maximum error in the hybrid simulation case was 17.62%, while it was reduced to 2.70% in the model updating case. The improvement in this example after applying model updating was not as obvious as in the other steel problems. Since, CON2 is a highly non-linear model, any error that occurs even at early stages has a permanent effect on the structural response. The impact of this issue can be reduced through using smaller loading increments, which provides more loading steps for the optimization tools to identify the error and to update the model parameters.

Updating the constitutive parameters of CON2 in ZeusNL was a challenging task. The subroutines used to analyze the non-linear concrete model were nested and they depended mainly on the previous history points. Therefore, while applying the new constitutive parameters, critical issues were observed. For instance, the determined stresses occasionally yielded illogical values such as imaginary number or unrealistic high values. Moreover, some convergence issues were experienced with ZeusNL. In CON2 subroutines, there are seven loading and unloading branches for tension or compression actions. These branches are guided by the constitutive model parameters and the response of the former cycles. Consequently, when the model parameters were modified, sometimes the stresses were guided to wrong loading/unloading path. This problem was addressed through imposing additional constraints in the program subroutines to confirm that the stresses follow the suitable path.
This series of verification examples showed the ability of model updating approach to identify the constitutive relationship parameters from one structural component to modify the others. Moreover, it was important to evaluate the efficiency of the communication protocols between UI-SIMCOR and ZeusNL to exchange the information required for modifying the different constitutive relationships. During this procedure, some challenges were encountered with STL3 and CON2 models, which were addressed by modifying some of ZeusNL subroutines. These four examples were designed to be relatively simple in configuration to introduce the model updating procedure and to get a better understanding of its features. The following examples will explore other advanced applications for the proposed approach.
4.2. Irregular configuration

The objective of this example is to show that updating the constitutive model does not require the source and the updated structural components to share the exact characteristics. Figure 4.11 illustrates the analyzed model, which is a two-bay one-story frame with unsymmetrical configuration. The assigned masses and the lengths of the structural members are shown in the figure. The beam is rigid and the columns are represented by STL1 model. The column lengths and the cross-sectional dimensions are designed such that the three columns share approximately the same stiffness. In this problem, the short column (i.e. left column) will be used as the source components to update the other ones. STL1 parameters used for this problem are listed in Table 4.6. As in the previous examples, the frame was subjected to Elcentro earthquake and then it was analyzed for the three cases, namely; exact, hybrid simulation and model updating case.

![Diagram of the frame model](image)

Figure 4.11 The geometric configuration and cross-sectional dimensions of the irregular frame model
Table 4.6 Irregular frame constitutive model (STL1) parameters for the different analysis cases

<table>
<thead>
<tr>
<th>Module</th>
<th>Left Column</th>
<th>Middle Column</th>
<th>Right Column</th>
</tr>
</thead>
<tbody>
<tr>
<td>Factors</td>
<td>E (MPa)</td>
<td>Fy (MPa)</td>
<td>Fl (MPa)</td>
</tr>
<tr>
<td>Exact</td>
<td>200000</td>
<td>140</td>
<td>0.10</td>
</tr>
<tr>
<td>Updated</td>
<td>210000</td>
<td>160</td>
<td>0.12</td>
</tr>
<tr>
<td>Hybrid simulation</td>
<td>200000</td>
<td>140</td>
<td>0.10</td>
</tr>
</tbody>
</table>

The structural response for the three analysis cases are shown in Figure 4.12. There is an obvious error in the hybrid simulation case. The errors in the model updating case were nearly negligible. Surprisingly, the improvement in of this example response is more considerable than the regular frame problems. This result confirm that updating the model on a fundamental level, such as the constitutive model, can be efficient even if the source and the modified modules are not strictly symmetric in terms of geometric and loading conditions.

Figure 4.12 The nodal displacements for the irregular frame subjected to the different analysis cases
Figure 4.13 illustrates the percentile error normalized to the peak response for the different analysis cases. It can be observed that for the model updating case the error is nearly negligible. The average absolute displacement errors in the hybrid simulation and the model updating cases are 8.54 mm and 0.17 mm, respectively. The maximum error in the hybrid simulation analysis is 24%, compared to 0.44% in the model updating case. In this example, the stiffness of the source column was slightly higher than the other columns. Therefore, it experienced the applied loads in earlier stages. For example, the source module yielded few steps before the rest of the columns. Hence, the optimization algorithm had enough time to identify the yield values before it was experienced by the modified components. The results of this problem emphasized on two points (1) the source and the modified components do not need to be identical, where updating the constitutive relationship provides flexibility to the model updating procedure. (2) Selecting the source module requires close attention, because it allows for better results if the source module is ahead of the modified components.

![Figure 4.13 Error comparison in the irregular frame for the conventional and the model updating cases](image-url)
4.3. Upgrading the bilinear steel model

The purpose of model updating is to determine the behavior of the experimental module and use it to modify the numerical components that share close characteristics. This procedure implicitly implies that the theoretical idealization of the numerical component is not sufficient to represent the sophisticated response of the actual experimental module. This example represents a simplified analogy of the model updating concept. Whereby, the modified members are defined using STL1, while the source component is defined using the STL2. STL2 exerts a complicated behavior compared to STL1, where its backbone curve and non-linear response are governed by four parameters. On other hand, STL1 is represented by three parameters, and the non-linear response is guided by the strain hardening factor only. The geometric configuration of the analyzed frame is shown in Figure 4.14, where a mass of 150 N/(mm/sec²) is applied on the middle columns and each of the outer columns supports 75 N/(mm/sec²). The structure is subjected to Elcentro earthquake and its analysis procedure follows the one adopted in previous example. In this example, the exact case is represented such that all the columns are defined using STL2 model. Meanwhile, in the hybrid simulation and model updating cases, only the left column (source component) is defined using STL2 and the rest of the columns are defined using STL1 model.

As in the previous examples, the structure was analyzed and the nodal displacements were plotted to evaluate the performance of the three analysis cases as shown in Figure 4.15. There was an apparent inconsistency between the exact and the hybrid simulation cases. However, when model updating approach was applied the error gap was reduced to a great extent. These results verifies that model updating was able to modify the behavior of STL1 to match the response of STL2 as close as possible.
Figure 4.14 The geometric configuration for the frame used in updating the bilinear steel model

Figure 4.15 The nodal displacements for steel frame subjected to the different analysis cases
Figure 4.16 shows percentile error of the displacements normalized to the peak value. The average absolute displacement errors in the hybrid simulation and the model updating cases are 12 mm and 5.1 mm, respectively. The maximum error in the hybrid simulation analysis is 41.42%, which is reduced to 9.46% in the model updating case. Figures 4.15 and 4.16 highlights the significant improvement in displacement especially at the peak value. Usually, the peak value is of crucial importance for the structural assessment. For instance, if the structural member exceeded a certain displacement value, this might violate a serviceability limit or even the failure state of the member.

![Graph](image.png)

Figure 4.16 Error comparison in the steel frame for the conventional and the model updating cases

Model updating provided significant improvement to the overall structural response. However, it can be observed that at the peak values the improvement was more obvious than at the low stress values. This behavior was expected because at low stresses STL2 model shows a curved backbone behavior. On the other hand, STL1 model is represented by Young’s modulus only (straight line relationship) at low stresses. Therefore, any modification in the model parameters will not be able
to reduce the errors at low stresses considerably. Nevertheless, the non-linear response provides more flexibility to update the yield strength and the strain hardening factors to match the accurate response. It is worth noting that, for the first attempts to solve this example, the bilinear model was subjected to unjustified early yielding, because it aimed to fit the STL2 backbone curve. To avoid this problem, an additional constraint was imposed to guarantee that STL1 model would not yield, unless the gradient of Ramberg-Osgood relationship becomes smaller than a certain tolerance.

Due to its importance, the results of this example were investigated in more details. Figure 4.17 shows the constitutive responses of the source column and a representative of the modified members. The figure compares the exact and the hybrid simulation cases. As expected, STL1 was not able to represent the behavior of STL2, which had a negative impact on the structural response as shown in Figure 4.17b. On the other hand, Figure 4.17a illustrates the constitutive model response of the source column. The results of the left column were affected by the inaccurate STL1 response of the other columns. This observation confirms that the accuracy of the numerical modules in hybrid simulation experiments cannot be overlooked.

![Figure 4.17 Comparing the exact and the hybrid simulation constitutive models for; (a) the accurate (left) column, and (b) the inaccurate column](image)
Figure 4.18 shows the same features, but this time it compares the exact case to the model updating case. Figure 4.18b is very illustrative in describing the effect of modifying STL1 parameters to approach the response of the source component. By contrasting Figures 4.17b and 4.18b, it can be observed that in the model updating case, optimization imposed smaller yield strength values in STL1. In addition, it modified the values of the Young’s modulus and the strain hardening factors to best represent STL2 curvatures. Additionally, Figure 4.18b helps in understating the significant improvement at the peak values (maximum strain values) in the nodal displacement plots shown earlier. However, at small strains STL1 could not represent the accurate model curvatures. The improvements in the modified columns performance had a positive impact on the source column as shown in Figure 4.18a. This conclusions, emphasizes the importance of model updating in the assessment of path dependent problems.

![Figure 4.18 Comparing the exact and model updating constitutive models for; (a) the accurate (left) column, and (b) the inaccurate column](image)

This example is unique in its configuration and objectives relative to the previous ones. As its main objective was to verify the fundamental concept of model updating, which is modifying the behavior of a simple system to represent the response of a more sophisticated one. The analysis
results showed that through applying model updating, the simple bilinear relationship was able to approach the behavior of Ramberg-Osgood model. Moreover, it was observed that the errors reduced significantly at the peak displacements. Finally, it was shown that for path dependent problems the numerical components have a considerable impact even on the accurate module. In another words, the response of the physically tested specimen in a hybrid simulation test is affected by the accuracy of the numerical components.

4.4. **Neural Network**

In this chapter, optimization tools were used to identify parameters that best represents the accurate constitutive model. A different philosophy is adopted in the neural networks (NN) applications. First, there is no need to assign a numerical constitutive model before analyzing the problem, where the NN generates the mathematical relationships representing the model behavior. Moreover, the input requirements are related to the stress and the strain values of the accurate module. Finally, the proposed NN configuration does not identify the constitutive model in a stepwise manner. Instead, the network is trained for a certain loading history and then, the trained network is used for a different problem.

The configuration of the frame analyzed in this example is similar to the previous ones. Where a two-bay one-story frame was subjected to an earthquake recorded as shown in Figure 4.19. The beams were represented by a rigid member, while the columns were defined using a bilinear steel model. The geometric properties, cross-sectional dimensions and the assigned masses are shown in the figure. The analysis procedure adopted in this example can be summarized as follows:
1. The frame is subjected to an input excitation (i.e. earthquake record) and the output results are recorded.

2. The stress-strain data of the source member are used as input for the NN. At this stage, the stress and strain values are determined from a predefined constitutive relationship (i.e. bilinear steel model).

3. The NN is trained based on the input and output values provided from the previous step. The weight factors and biases generated by the network are saved in a matrix form to be used by the analytical platform.

4. The frame is analyzed once more, but it is subjected to a different earthquake record. However, in this analysis the columns constitutive relationships are represented by the NN parameters identified in the previous step.

5. The output results of the frame are compared to the exact solution to evaluate the effectiveness of the NN representation. The exact solution is the one obtained if the bilinear steel model would have been used to define the columns response.

Figure 4.19 The geometric configuration for the frame analyzed using the neural network approach
After several analyses, it was observed that the performance of the NN is highly dependent on the characteristics of the input ground motion used in the network training procedure. These characteristics need to be as general as possible to excite different modes of the structure. For example, if the NN is trained for input that is rich in high frequency content, it will not be suitable to represent a model subjected to low frequency excitation. Therefore, an artificially generated ground motion based on 475 years return period was used to train the network. This record was chosen because it is rich in its frequency content. Meanwhile, the earthquake record used for verification is the horizontal component of the Civic Center station in the Big Bear earthquake (6/28/1992). The time history acceleration of this records was obtained from the Next Generation Attenuation (NGA) strong motion data bases provided by the Pacific Earthquake Engineering Research (PEER) center [PEER, 2010]. Figure 4.20 shows the response spectrum plots for the artificial record used to train the network (ACC475) and the Big Bear earthquake used in the problem verification. It is important to mention that, the Big Bear time history values were scaled down to make sure that its amplitude is smaller than ACC475 records.

![Figure 4.20 The response spectrum plots for the artificially generated motion and the Big Bear earthquake](image-url)
ZeusNL was utilized to analyze the steel frame subjected to the artificially generated record. The stress-strain values generated from one of the Gauss point in the left column were used as input for the NN. Initially, the most stressed fiber was used for the identification procedure. However, this approach sometimes resulted in overtraining the network, which means that the NN tend to curve fit the stresses and strain values rather than capturing the model behavior. To avoid this issue the response of two fibers were used as shown in Figure 4.21; (1) the most stressed fiber and (2) a fiber that is still in the elastic stage.

Figure 4.21 constitutive relationships used to train the NN; (a) the most stressed fiber and (b) a fiber in the elastic range

The network configuration used to represent the constitutive model problem is constructed of three layers, namely; input, hidden, and output layers. These layers include five, three, and eight neurons respectively [5-3-8]. All the neurons are interconnected between the layers as shown in Figure 4.22. The two hidden layers apply the log-sigmoid transfer function, while the output layer utilizes a linear transfer function. A detailed discussion about the input and output requirements can be found in Chapter 3.
After training the network, the weight factors and biases were saved and sent to ZeusNL for analysis. In ZeusNL, the bilinear model was replaced by the identified NN configuration, which required developing a new subroutine in the program. Figure 4.23 shows the stress-strain relationship of a representative fiber in ZeusNL. This figure shows the results of the conventional STL1 model compared to ones generated by the NN. It is obvious that the NN successfully generated the constitutive relationship required for that problem.

There are several characteristics that can be used to evaluate the efficiency of structural response using the proposed NN approach. As discussed in the previous examples, the nodal displacement of the left columns will be used as a representative criterion. Figure 4.24, show the top nodal displacement of the left column compared to the accurate solution. Big Bear earthquake was applied as input motion for the analyzed frame, while the network was trained based on ACC-475. The results of the NN model almost coincided with the accurate STL1 model response.
Figure 4.23 The stress-strain values identified from the neural network compared to the analytical bilinear model results.

Figure 4.24 The top nodal displacements for the left column comparing the accurate to the neural network results.
Although these results confirmed the ability of NN to represent the constitutive model accurately, yet it can be argued that the network was trained based on a theoretically perfect analytical data. However, in actual problems, the model will not follow such behavior. Therefore, the example was solved again, but after adding random noise to the input values as shown in Figure 4.25. To have a more realistic representation, the noise was induced only to the non-linear component of the source data, as the elastic stage generally follow the analytical models.

Figure 4.25 constitutive relationships used to train the NN; (a) the most stressed fiber distorted with noise and (b) a fiber in the elastic range

The same NN configuration used in the previous examples was used herein and the identified parameters were saved and used in ZeusNL to evaluate the accuracy of the results. Figure 4.26 illustrates the response of the most stressed fiber in ZeusNL model. The figure compares the constitutive relationship identified by the NN to the exact STL1 results. As expected, due to the distortions induced in the non-linear stage, the model behavior in this example is less accurate than model identified in the previous one. Although random noise was added to the input records, yet NN was able to represent the overall features of the actual model behavior.
The structural performance can be assessed through evaluating the top nodal displacements of the left column as in the previous example, Figure 4.27 shows the accurate results compared to the NN ones. Once more, the displacements determined based on the NN approach were close to the accurate solution and followed the same pattern. Minor errors can be observed, yet they do not affect the overall response of the structural response. At the latest loading stages, it can be noted that there is a small phase shift between the exact and the identified results. This pattern can be explained through the stress-strain model shown in Figure 4.26, where the stresses and hence the stiffness values did not match the accurate results perfectly. Therefore, the slight changes in the modal behavior affected the modal properties of the structure. Finally, it is important to mention that in neural networks procedure, the output at any loading step depends entirely on the previous history. Consequently, any error in the identification procedure will propagate through the loading stages.
This example demonstrated the ability of neural networks to replace the analytical constitutive models. The results showed that the errors in the identified network results were almost negligible. However, when random noise was added to the source data used to train the network, some errors were observed. Additionally, the analyzed example verified the ability of the model updating program to exchange the neural network information between UI-SIMCOR and ZeusNL. For future applications, it would be useful to use more advanced neural network architect, which can be used to update the analytical component instantaneously during the analysis.

4.5. **Summary and Discussion**

In this chapter, a number of numerical examples were analyzed. The purpose of these examples was to evaluate the ability of model updating to learn some information from an accurate structural
component to modify the behavior of the corresponding counterparts. Additionally, the performance of the communication protocol to exchange the stress-strain data between UI-SIMCOR and ZeusNL was verified. Each of the analyzed example was designed to serve a different objective. First, a series of problems were solved to evaluate the efficiency of model updating procedure in general. For that purpose, STL1, STL2, STL3 and CON2 models were used to represent the behavior of the structural components. The results showed a significant enhancement in the overall structural performance when model updating was applied. It is worth noting that STL3 and CON2 subroutines have been slightly modified in ZeusNL to support the updating procedure.

Second, an example was solved to demonstrate that updating the constitutive relationship provides flexibility in defining the source and the modified components in the structure. An irregular frame was analyzed, where the source and modified modules had different geometric properties. It was also recommended to choose the source component to be ahead of the modified ones in terms of the applied straining actions, which results in a better numerical model performance. Third, an example was solved to show that the simple bilinear steel model could be modified to approach the behavior of the more complicated Ramberg-Osgood model. This concept is analogous to the actual hybrid simulations, where some properties of the simple analytical module can be updated based on the data acquired from the detailed physically-tested component.

Finally, in this chapter, neural network was used to replace the traditional constitutive relationship. NN is advantageous because it does not require the presence of a predefined analytical model. On the other hand, it does not provide much physical understanding about the behavior of the
investigated model. Two example were solved, (1) the network was trained based on the theoretical bilinear stress-strain data, and (2) the stress-strain data were distorted with noise to represent a more realistic problem. In both examples, the results showed that NN was able to capture the capture the essential features of the constitutive relationship. It is important to mention that the neural network was not updated incrementally. Instead, it was trained based on the full history of a certain time record. Then, the trained network was used to analyze the model subjected to a different record. For future work, more advanced network configurations can be applied to instantaneously update the numerical module response.

This chapter showed that model updating can be used for various hybrid simulation applications. The next chapter will develop a more advanced step in verifying the proposed approach, where the records of previously conducted experiments will be investigated within the model updating framework.
CHAPTER 5
EXPERIMENTAL DATA ANALYSIS

The numerical examples discussed in Chapter 4 showed that model updating can substantially improve the results of conventional hybrid simulations. However, some difficulties can be expected while analyzing the actual experimental data. For instance, measuring the stress values from a physically tested specimen is a challenging task. Additionally, the ability to identify the constitutive relationship parameters from an actual specimen, then using them to update the numerical module behavior requires more investigation. Therefore, it is important to verify the model updating concept on the experimental level.

Instead of conducting actual experiments, it is decided to analyze previous test results from the available database. This procedure is advantageous because these experiments provide a wide variety of applications and hence, different aspect of model updating can be explored. Moreover, updating the constitutive relationship in a hybrid simulation test is a novel approach. Therefore, it is more convenient to analyze the data offline, which allows for operating on the same specimen several times. Analyzing previous experimental data is sufficient to verify the model updating framework in terms of identifying the stresses from an actual specimens and to confirm that updating the constitutive model can improve the overall results of the numerical module. This chapter starts by discussing the constitutive relationship identification procedure, while considering the related experimental complications. Then, a quasi-static test for beam-column steel connection will be investigated to evaluate the proposed identification procedure. Next, model updating approach will be applied to a similar steel connection, but the beam section is heat treated, which adds complexity to the model behavior. Finally, a full-scale hybrid simulation experiment
will be analyzed. This experiment represents a multi-bay concrete bridge subjected to a combined loading, which provides a suitable environment to verify model updating concept.

5.1. Stress Identification Procedure

The constitutive model provides the relationship between the strains and the corresponding stress values for a certain material. Measuring strains from the tested specimen is well established and there are several instruments that can be used for that purpose, which include strain gauges, linear variable transformers, and fiber optic strain gauges [Ayranci et al, 2008]. On the other hand, measuring stresses in the laboratory is not straightforward. Conventionally in the coupon tests, the specimen is subjected to the load in one direction and by assuming uniform distribution, the stresses can be determined. However, this approach cannot be applied in case of having a set of combined loading and hence, inverse identification techniques are required [Cooreman et al., 2008]. Several research studies discussed the idea of inverse identification problems to determine non-homogenous material parameters [Endelt and Nielsen, 2005; Cooreman et al., 2008]. These studies motivated the procedure that can be applied in hybrid simulation environment and hence, supporting model updating applications.

In this section, a procedure based on optimization tools to update the constitutive relationship parameters during the analysis is presented. First, the approach followed in the analytical examples, illustrated in Chapter 4, will be revised. This approach is intuitive and helps in defining the main components of the identification procedure. Afterwards, the modifications required to this procedure to be able to handle the data provided in the physically-tested problems will be highlighted. It is important to mention that the main differences between the two approaches is
that in the analytical problems the stresses can be evaluated explicitly. On the other hand, for actual problems, the stresses cannot be measured directly from the laboratory. The identification procedure applied for the analytical problems is shown in Figure 5.1 and it can be summarized as follows:

- The structure is subdivided into several components. The critical component that shows complex behavior is investigated in more details to provide a reliable response (i.e. physically tested in the hybrid simulations).
- An auxiliary numerical model is developed, which mimics the characteristics of the critical component. The auxiliary model is used to evaluate the differences between the analytical idealization and the accurate member response.
- The auxiliary model is divided into sections and fibers, where the stress at the fiber of interest ($\sigma_{Num}$) is compared to the stress determined from the accurate module ($\sigma_{Acc}$).
- If the error exceed a predefined tolerance, it implies that the numerical model is not a suitable representative for the accurate module. Hence, the optimization program determines the constitutive relationship parameters that minimizes that error.
- The new parameters are assigned to the numerical module and the errors are evaluated once more. This procedure is repeated iteratively, until the best constitutive model representation is achieved.
- On the other hand, if the error between the stresses is within the tolerance value, then the initial parameters are saved for the next loading increment.
After presenting the approach followed to update the constitutive model parameters for the analytical problems, this section discusses the corresponding procedure used to handle actual experimental data. As earlier discussed, it is not possible to determine the stresses explicitly in the laboratory. In 2008, Cooreman et al. presented an inverse identification method used to determine the constitutive relationship parameters in case the model is subjected to combined loading. This method is modified to be adequate for model updating applications. The identification procedure is shown schematically in Figure 5.2, which can be summarized in the following step;
- The stepwise integration scheme determine the deformations assigned to the different structural modules. For the physically tested component, the restoring forces required by the actuators to apply the predefined deformations are recorded.

- Along the length of the tested component, a cross section is selected based on the purpose of the application. Strain measurement instruments, such as strain gauges, are attached to the section of interest, which are used to evaluate the strain distribution.

- Through utilizing the equilibrium conditions, the straining actions applied to the same section (S.A-Exp.) can be calculated based on the restoring forces measured by the actuators.

- An auxiliary finite element model is developed that resembles the characteristics of the tested component. The same deformations are applied to the finite element model and the straining actions (S.A-Num) are also determined at the section of interest. It is important to note that, the straining actions values are evaluated based on an analytical constitutive relationship and the model parameters initially defined.

- The values of the straining actions identified for the experimental and auxiliary models are compared. If the difference between these values exceeds the allowed tolerance, then optimization algorithms will update the constitutive model parameters in the numerical model. This step is repeated iteratively until the difference reaches its minimum possible value.

As it can be observed, the two identification approaches share similar philosophies. The former approach was much simpler because the accurate stresses were explicitly provided, which was compared to the analytical solution. On the other hand, in case of experimental model subjected
to combined loading, the stresses cannot be directly evaluated. Therefore, the identification procedure was modified, where the equilibrium conditions on a certain section were utilized to compare the response at a certain section rather than a certain fiber. It is important to mentions that there might be additional sources of uncertainty in the model such as boundary conditions, and reinforcement bars buckling. Therefore, the constitutive relationship modifies the model by proxy to include the other source of uncertainty as discussed in Chapter 3.

![Diagram](image)

**Figure 5.2 Constitutive model identification procedure in case of complex experimental environment**

The inverse identification approach provides flexibility regarding the error evaluation criterion. For instance, the discussed procedure utilized the straining actions such as forces and moment to...
assess the adequacy of the numerical module. However, for other applications it might be convenient to adopt different characteristics to evaluate the system response such as deformations, model energy, modal parameters, etc. Moreover, a combination of these characteristics can be utilized, which offers redundancy in the identification process. Additionally, the analyzed section can be discretized into several fibers and each fiber can be assigned with a different constitutive model. For example, the inner fibers are expected to behave in the elastic range, while the outermost might be subjected to non-linear response. Therefore, different constitutive models can be used for these fibers.

The inverse identification procedure is very useful in updating the constitutive relationship parameters as earlier discussed. Its main advantage that it does not require direct measurement of the stresses in the laboratory. However, there are some limitations related to the proposed approach that need to be highlighted, such as; (1) this method update the parameters of a pre-existing stress-strain model, where optimization can be utilized for that purpose. On the other hand, neural networks require the strain and stress values to be provided explicitly; therefore, they cannot be utilized for this method. (2) Some assumption are implicitly imposed within the identification procedure; such as plane section remains plane during the analysis and the strain distribution along the section of interest must be also assumed. (3) In order to calculate the straining actions at a certain section of the tested member, it is preferable that this member show simple geometric configuration (i.e. statically determinant), otherwise the evaluation procedure will be more complicated. In this section, the modifications imposed to the constitutive model identification procedure were presented. The following section will verify its applicability on the experimental data. Later on, some model updating applications will be explored using the proposed procedure.
5.2. Beam-Column Steel Connection

Several experiments were investigated to verify the proposed model updating concept. Each of these experiments were selected to serve a different objective. The recorded data of the first experiment will be used to validate the inverse identification approach described in the previous section. In 2012, a beam-column steel connection quasi-static test was conducted as a part of the Network for Earthquake Engineering Simulation (NEES) program [Morrison et al., 2012]. In this experiment, a series of beam-column steel connections were tested to evaluate the seismic performance of the heat-treated beam section (HBS) connections compared to the conventional one. The experiments were conducted in the Multi-Axial Subassemblage Testing (MAST) facility at the University of Minnesota [Morrison et al., 2012]. The simple test setup of the conventional connection experiment provide an excellent environment to verify the constitutive model identification procedure.

5.2.1. Problem description

Figure 5.3 shows the experimental set up for the tested connection, where the column and beam cross-sections were designed to be W14X257 and W30X148, respectively. Two actuators applied lateral load on the tip of the beam at distance 134 inches from the column face. The beam was supported by lateral bracing to avoid out of plane deformations. The actuators applied cyclic loading as suggested by the ASIC 2005 seismic provisions, with lateral drift reaching 6% at the end of the test [Morrison et al., 2012]. The height to the beam cross-sectional aspect ratio confirms that the beam is subjected to flexural action, which provide the suitable environment for some of the assumptions that will be presented later on.
The tested specimen was heavily instrumented, which provided the records necessary for the model assessment. The attached instruments included load cells, displacement transducers, strain gauges, string potentiometer, Krypton LEDs and linear variable displacement transducers (LVDTs). Figure 5.4 illustrates the orientation and the different types of the utilized instruments. On the other hand, Figure 5.5 shows the actual representation of the experiment, where Figure 5.5a illustrates the overall setup, while Figure 5.5b focuses on the strain gauges attached to either flanges of the beam. As earlier mentioned, the identification procedure requires the strain distribution and the straining actions at the section of interest. The strain distribution can be acquired using either the strain gauges or the Krypton LED records. Meanwhile, the straining actions applied at any section of the beam are the shear forces and the moments. The moments can be determined through equilibrium conditions, based on the forces applied by the actuators and the arm of moment at the section of interest.
Figure 5.4 The instrumentation of the conventional steel beam-column connection [Schweizer, 2013]

Figure 5.5 Actual beam-column test setup: (a) Krypton LEDs and (b) strain gauges at the beam flange [Schweizer, 2013]
Through analyzing the strain gauge records, it was observed that the pattern of the measurements was not logic at some phases of loading. Upon a personal communication with the experiment team members, it was noted that, the performance of the strain gauges for the conventional beam-column connection test was not satisfactory. However, this issue was not significant in the other experiments. Consequently, to analyze the conventional connection it was suggested to utilize the Krypton LEDs to find the relative deformations and hence, the strains along the beam cross-section [Morrison and Elanwar, 2013].

In order to determine the strains from the LEDs records, the deformation between any two LEDs is calculated in the Cartesian coordinates. Then, this deformation is normalized by the initial gauge distance to find the strains. In this example, the strain was assumed to follow a linear distribution. Therefore, it is sufficient to find the strains at any two fiber of the beam’s section. However, two issues need to be highlighted: (1) the initial gauge distance was about 2.5 inches, this distance is relatively long, which might have a negative impact on the accuracy of the strain measurements, and (2) the pattern of some LED records included illogic distortions, which is not expected in the conventional strain gauge results. Nevertheless, the beam’s web and flanges were heavily covered with measuring instruments as shown in Figure 5.5a.

To eliminate any noise and distortions in the records, the strain distribution between any two LED candidates was calculated. Among those, the average of 18 combinations was determined. After following this procedure, the strain distribution along the beam section was reasonable. Figure 5.6 shows the identified values at the top and bottom beam flanges. It can be observed that strain pattern is symmetric at the beam edges. It is important to note that, this long procedure was adopted
because the strain gauge results were not trustworthy. For the next example, the strain gauges records will be used directly.

![Graph showing strain values at the beam flanges in the conventional connection](image)

**Figure 5.6** The strain values at the beam flanges in the conventional connection

Before describing the identification procedure, some of the main features of the experiment can be summarized as follows; (1) this experiment was a quasi-static test, and hence there was no communication protocols with a numerical module. (2) Lateral loads were applied by the actuators at the beam tip, which generated moments and shear forces only on the beam cross-sections. (3) The experiment was rich in records and measurement instruments such as strain gauges and krypton LEDs, which simplified the analysis. (4) The beam cross section was fabricated from conventional steel and hence, the available constitutive models are expected to efficiently represent the actual beam response. These features are important to consider while discussing the identification approach, where the solution steps are listed below;
1. A section in the beam was selected for analysis and the moments at this section were calculated based on the applied forces and the arm of moment. The arm of the moment at the chosen section was located between the first and second rows of the LEDs at distance of 131 inches from the actuator (i.e. Moment (kip.in) = Force (kip) x 131 (inches)).

2. The strain distribution along the section of interest were determined, using the Krypton LED data as earlier discussed. In order to fulfil the requirements of the fiber analysis approach, a constitutive relationship must be assumed for the model. For simplicity, the bilinear steel model was assigned to represent the behavior of the beam.

3. Interior point methods (IPM) optimization tool was then utilized to identify the parameters that govern the bilinear constitutive model (i.e. Young’s modulus, yield strength and strain hardening factor). The objective of the IPM is to minimize the difference between the moment resulting from the fiber analysis and the experimentally evaluated ones.

4. To verify this procedure, the identified parameters were compared to the coupon test results provided in literature [Schweizer, 2013]. If the proposed procedure is suitable for this problem, the analytical moments and the identified parameters should be as close as possible to the experimental moments and to parameters defined in the coupon test report, respectively.

The next section will present the results of the analysis, which can be compared to the reference solution. Then, in the discussion section, the outcome of the identification procedure will be evaluated.
5.2.2. Results

Due to the simple configuration of the test setup, there was no need to use a complicated finite element program such as ZeusNL. It was sufficient to develop a MATLAB program to perform the fiber analysis calculations and to communicate with the optimization toolbox. The procedure presented in the previous section was utilized and the results of the identification procedure are illustrated in Figure 5.7. The figure shows the loading steps against the moment values evaluated from the experimental data. These moments are compared to the values identified by the fiber analysis program. The pattern and the values of the analytical solution were consistent with the experimental results. The errors were insignificant at the early loading steps (elastic range), while they became more obvious in later stages. This observation was expected as the bilinear model is governed by the strain hardening factor only in the inelastic range, which is not sufficient to represent the actual model behavior.

Figure 5.7 Identified moment values for the conventional beam-column connection
Another approach to verify the ability of the constitutive model identification procedure was adopted, which is comparing the analytically evaluated bilinear model parameters to the coupon test results provided in the literature. Figure 5.8, shows the stress-strain relationship for the steel material used in the connection. The plots and the model parameters were included in the experiment report [Schweizer, 2013]. The identified parameters matched very well with the coupon test ones, where the average value of Young’s modulus from the identification algorithm was 28,512 ksi compared to 29,312 ksi from the coupon test results. Meanwhile the identified average yield strength was 56.4 ksi compared to 56.5 ksi for the coupon test values. Moreover, the pattern of the stress-strain curve confirms the conclusions from the moment plots, that the bilinear steel model can efficiently capture the elastic stage of loading. However, it cannot strictly follow the inelastic behavior.

Figure 5.8 Coupon test results for the steel material used in the conventional beam-column connection [Schweizer, 2013]
The identified model parameters and the analytical moments, show that the proposed identification procedure can be applied to actual experimental data. Next, a brief summary about the features of this examples and its output will be discussed.

5.2.3. Discussion

The experiment of the conventional beam-column steel connection was utilized in this example due to its simple test setup. The experiment was a quasi-static problem and the beam was subjected to cyclic lateral loading. Moreover, the beam aspect ratio was sufficient to assume flexural beam behavior. Additionally, the stress-strain model of the construction material was provided. These features reduced the impact of external factors that might bias the analysis of the results. Hence, it was suitable for verifying the identification procedure. A MATLAB program was developed to perform the required fiber analysis calculation, which utilized the optimization algorithms to determine the model parameters. The program required less than 10 minutes to execute the identification procedure for 600 loading increments. This duration satisfactory considering the long time required for an actual hybrid simulation or quasi-static experiments.

There were some challenges regarding the preparation and the organization of the recorded data. However, the most significant challenge was the inadequacy of the strain gauges records used in this experiment. Therefore, Krypton LEDs were used instead to find the strain distribution along the beam cross-section. The analysis of the LEDs was not as straightforward as the conventional strain gauges. Moreover, the gauge distance was 2.5 inches for the LEDs, which had an impact on the accuracy of the data. In addition, the pattern of the results was unrealistic for some LED records. To avoid random biases, the strain values were determined based on the average of 18
different combinations of LED records. The final strain distribution at both flanges was consistent with the input loading pattern and the physical understanding of the problem.

The results of this problem were very promising, as the error in the identified constitutive relationship parameters was less than 2% as compared to the coupon test results. In addition, the analytically determined moments were consistent with the experimental ones. It is worth noting that, the bilinear model used for this problems was able to represent the actual model behavior to a great extent. However, for other applications, more complicated constitutive model can be used. This example did not address model updating applications. It only focused on the identification procedure that can be used for that approach. The following example will utilize the identification procedure for model updating purposes.

5.3. Heat Treated Beam Section

In the same project presented in the previous section, several other experiments were conducted to evaluate the seismic performance of beam-column connections. The second experiment evaluated the performance of a heat treated beam section (HBS) compared to the conventional one. Heat treatment changes the properties of the beam, where the yield strength is expected to drop down without losing much of the initial stiffness. Hence, the inelastic deformations are enforced in the beam rather than the column [Schweizer, 2013]. Therefore, the conventional constitutive models are not a suitable representatives for the behavior of steel material used in the connection. In this example, the constitutive relationship parameters will be determined from a certain section of the beam. Afterwards, these parameters will be used to represent the behavior of the beam at another section subjected to different loading conditions. The identification procedure introduced in the
previous example is applied directly herein. This procedure will be applied incrementally, such that it can be utilized for future model updating applications.

5.3.1. Problem description

The same test setup demonstrated in the previous example is preserved. Figures 5.3, 5.4 and 5.5 show the main components and the geometric configuration of the experiment. Additionally, the figures illustrate the layout of the measuring instruments attached to the specimen. This section will highlight only the main differences between the procedure adopted in this example and the previous one. The steps followed in the analysis are listed below:

1. Two sections were determined along the beam, which were selected such that they overlap with the location of the attached strain gauges as shown in Figure 5.9. The two sections were located at distances of 1 and 3.5 inches from the column face.

2. The moment values at the two sections were calculated based on the forces at the actuator and the arm of the moment. It is worth noting that, the total distance from the point of action of the actuator to the face of the column is 134 inches. Therefore, the arms of the moment for sections 1-1 and 2-2 are 133 and 130.5 inches, respectively.

3. The strain gauge values at the predetermined sections were organized and plotted in Figure 5.10. The figure shows the records at the top and bottom flanges. The behavior of the stain values is consistent with the physical understanding as they follow the pattern of the applied loads. Additionally, the section that was subjected to higher moments, show higher strain values.
4. In the fiber analysis program, the bilinear steel model was used to represent the beam response. Then, the identification procedure was applied to the first section (at a distance of 133 inches from the actuator height) to determine the model parameters that minimizes the error between the analytical and the experimentally evaluated moments.

5. The bilinear model parameters determined from the previous step were used in the second section. Then, the moment values were calculated analytically, which will be compared to the accurate experimental response.

It is important to mention that during the experiment the strain gauges could not record the data until the end of the experiment. However, the available data were sufficient as they cover all the elastic phase and few cycles from the inelastic response.

Figure 5.9 Instruments attached to the heat treated beam connection; (a) Krypton LEDs, and (b) strain gauges [Schweizer, 2013]
The fiber analysis program developed in the previous example is used in the problem as well. This program was sufficient to apply the optimization algorithms and to calculate the moment values at the section of interest. Slight modifications were required to the program subroutines to allow for identifying the parameters from one section and applying it to the data of another section.

5.3.2. **Results**

The fiber analysis program was used to determine the moment values at section 2-2 for two different cases; (1) the analytical case, where the bilinear model parameters were defined according to the coupon test values provided in project report for the heat treated section [Schweizer, 2013]. (2) The updated case, where the model parameters were modified incrementally based on the information acquired from section 1-1. Figure 5.11 shows the loading steps against the moments for both cases compared to the experimentally determined values.
Some observations can be noted from Figure 5.11. First, the moment values after updating the constitutive model parameters were much closer to the experimentally evaluated results, when compared to the pure analytical solution. Moreover, the updated case was more efficient in the early elastic stages than the later inelastic parts. This behavior is expected as described in the previous example, where the bilinear model is governed by Young’s modulus, which can fairly represent the elastic response of the actual member. However, for the non-linear characteristics of the bilinear model is not sufficient to represent the actual complex behavior. Generally, updating the bilinear model parameters had a positive impact on the overall performance of the analytical solution.

Figure 5.11 The heat treated beam connection response for the different analysis cases
Figure 5.12 shows the percentile error of the response normalized to the peak moment value. This figure clarifies the discrepancies between the different approaches. It is obvious that the model updating approach provided more accurate responses than the analytical case through the entire loading history. These errors at the early stages were nearly negligible, but they became more significant near the last stages as earlier discussed. Quantitatively, the average absolute error in the moment values dropped from 1940 kip.in in the pure analytical case to 920 kip.in when the updating procedure was applied. In addition, the peak error reduced from 47.5% to 26.6% in the model updating case. These values show model updating approach can have a significant impact in improving the response of the numerical module in the actual hybrid simulations.

There is an interesting pattern that can be observed in the updated approach at the steps were the actuators were not applying any forces (i.e. between steps 6200 and 9800). Whereby, the identified moments were fluctuating rapidly around the reference line. This phenomenon occurred because the correlation between the strain and moment values did not follow exactly the theoretical understanding. In addition to the measurement noise and distortions that might have been induced in the records during the experiment. At the stages were no deformations were applied, the optimization algorithms aimed to minimize these inaccuracies by modifying the bilinear model parameters. If the identified moments overshoot the reference solution, even by a small value, optimization will try to recover this error in the next loading step. Therefore, the model kept fluctuating around the equilibrium position as long as the analytical and the experimental records did not match perfectly. For future applications, this issue can be addressed by postponing the identification algorithm, whenever the test is in a static state.
This example demonstrated the importance of analyzing the actual constitutive relationship response. It was shown that the coupon test results might not be sufficient for some applications. On the other hand, applying the model updating approach to identify the constitutive relationship parameters improved the response of the numerical model. The next section summarized the main finding of this example, in addition to discussing its potential applications in the hybrid simulation experiments.

5.3.3. Discussion

The test setup of the HBS connection was similar to the conventional connection experiment. However, the main difference was the significant change in the beam behavior due to heat treating the beam flanges. This technique reduced the yielding strength of the steel material. In addition, it
changed the constitutive model behavior. In this problem, the results of the analytical module could not represent the actual member response. Yet, when model updating approach was applied the errors in the analytical solution were reduced considerably. At the initial loading stages, it was observed that the identified stiffness parameter was softer than the coupon test values. This finding agrees with one of the conclusions provided in the experiment report regarding the HBS connections as shown in Figure 5.13 [Schweizer, 2013]. The figure compares the overall moment-rotation behavior of the different types of connections. The legend in the figure follow the expressions in the report, where the conventional connection was named Ctrl#1 and the HBS connection discussed in this section was called HBS#2 [Schweizer, 2013]. From the figure it is observed that the HBS connections show soft behavior, even in the elastic range, which complies with the pattern of the identified parameters.

On the other hand, the improvement in the non-linear stage using the model updating approach was not as efficient as it was in the elastic range. This was expected because the analytical constitutive models in general cannot capture the actual member behavior accurately. Therefore, the engineering judgment is crucial in order to choose the appropriate constitutive relationship before running the test, where utilizing a more capable constitutive models is expected to improve the results. Finally, it is important to mention that several assumption were implicitly included in the identification procedure. For instance, the strain distribution followed a linear relationship also, plane section was assumed to remain plain through out the experiment. However, these assumptions might be violated when the model is subjected to cases of intense deformations or combined loading actions.
Figure 5.13 Moment-rotation curves for the different types of connections [Schweizer, 2013]

Some challenges were encountered while analyzing the response of the connection. First, the available strain gauge records were terminate before the end of the test. Nevertheless, they covered all the elastic response and some cycles after the yielding phase. For verification purposes, these records were sufficient to evaluate the performance of the constitutive model. The second challenge was the unexpected results of the analytical model at the stages where no deformations were applied by the actuators. At these stages, the optimization algorithm was still searching for new model parameters, which caused the identified moment values to fluctuate around the accurate experimental results (zero reference line). This phenomenon occurred because the optimization always aims to minimize the error, even if this error is not significant. Since the theoretical values never match exactly with the experimental records, the identified parameters will keep fluctuating around the accurate values. This issue can be avoided, by preventing the identification algorithm from operating at the cases of very small loading. Another alternative is to increase the tolerance
of the optimization, which will discard minor errors. It is important to mention that the identification procedure required few minutes, which is expected due to simple test setup.

In this chapter, two steel beam-column connections were investigated. The first example verified that the proposed identification procedure could determine the material constitutive relationship parameters based on experimental record. Meanwhile, the second example applied the identification procedure to update the behavior of a certain beam cross-section based on the records of a different section. This concept has several application for the model updating approach. For example, the zones that show complex behavior such as plastic hinge zones or the members subjected to severe loading can be tested in hybrid simulation experiments. Then, their constitutive model can be identified to update numerical modules that share close characteristic. The following example will address an experiment with more complicated test setup.

5.4. Multi-bay Concrete Bridge

In this example, a hybrid simulation experiment for a multi-bay concrete bridge is investigated. The test setup of this experiment was more complicated than the previously discussed experiments, which provides a suitable case of study to verify the model updating concept. The Combined Actions on Bridges Earthquake Research project (CABER project) was conducted in the Multi-axial Sub-Structuring Testing and Simulation (MUST-SIM) facility at the University of Illinois at Urbana-Champaign [Frankie, 2013; Abdelnaby et al., 2014]. The objective of the project was to evaluate the seismic behavior of a concrete bridge subjected to a combined set of loading. The test setup and the quality of the experiment allows the exploration of different model updating features. The following sections will discuss the experiment configuration. Then, the procedure applied and
the results of the model updating approach will be introduced. Finally, some comments and discussions will be presented.

5.4.1. Problem description

In this section, the main components of the CABER project are presented. However, for more details about project, some sources from literature can be revised [Frankie, 2013; Abdelnaby et al., 2014]. The experiment layout and the geometric configuration of the bridge are demonstrated in Figure 5.14. The superstructure was designed to be curved and it was supported on three piers. The structural dimensions and cross-sectional properties for each pier were different. The piers were subjected to multi-directional earthquake loading and hence, torsional moments were induced at the pier cap. In this experiment, each of the three piers was subjected to all the six degrees of freedom actions. To ensure realistic response, all the piers were tested in the MUST-SIM facility. Meanwhile, the bridge deck was simulated numerically using ZeusNL. The communication between the experimental and numerical components of the hybrid simulation was conducted through UI-SIMCOR platform [Frankie, 2013].

The main objective of this experiment was to evaluate the seismic response of a concrete with complex configuration. Therefore, the characteristics of the input earthquake record were selected carefully. For a realistic representation, the bridge was subjected to two horizontal components of earthquake records. In the transversal direction, the full-scale record was assigned. Yet in the longitudinal direction, one-fourth of these values were applied [Abdelnaby et al., 2014]. As earlier mentioned, applying the earthquake in two directions induce torsional and multi-directional moments, which complicated the analysis procedure.
Figure 5.14 The structural configuration of the CABER project [Frankie, 2013]

The preliminary design of the tested bridge follows one of the projects used by the National Cooperative Highway Research Program (NCHRP). This project was designed to evaluate the seismic behavior of structures in Seattle, Washington region. Therefore, artificial ground records were generated to represent the response spectrum of this zone [Frankie, 2013]. For the model updating purposes, only the main characteristics of the input motion will be highlighted. However, full details can be found in literature for the generation procedure of the records [Frankie, 2013]. The records were designed to represent the maximum considered earthquake (MCE) for a return period of 2500 years. A 10 sec time history segments were combined four successive times to generate an input of total 40 sec, where each segment induced a certain level of loading as follows:

- The first segment (i.e. 0-10 sec) represents the cracking performance level, where the crack in the concrete initiates. The design value was assigned to be 0.08 of MCE.
- The second segment (i.e. 10-20 sec) implies the start of yielding in the reinforcement bars. The design value was 0.3 of MCE.
- The third segment (i.e. 20-30 sec) simulates the design criteria with peak value of 1 MCE.
- The fourth segment (i.e. 30-40 sec) is designed to result in complete failure of the tested piers. For that purpose, the peak value reached 2 of MCE.

Figure 5.15 shows the pre-described characteristics of the input earthquake. Figure 5.15a illustrates the time history records, while Figure 5.15b demonstrates the acceleration spectrum of each segment separately compared to the MCE values. This pattern is important in understanding the bridge response as will be presented later.

Figure 5.15 Artificial earthquake used in caber project analysis; (a) time history records, (b) response spectrum plot [Frankie, 2013]
The hybrid simulation experiment conducted for the CABER project was divided into three main numerical and experimental components as follows [Abdelnaby et al., 2014];

- The superstructure and abutments were expected to respond in the elastic range. Therefore, it was sufficient to simulate these components numerically using ZeusNL.

- The inner pier was physically tested at a scale of one-twentieth the size of the original design. The size of this pier was adequate to be tested at a small scale facility located at the same site of the MUST-SIM laboratory, Figure 5.16a.

- The two outer piers were tested in the laboratory with the geometric dimensions (i.e. height, cross-sectional dimensions and reinforcement) scaled down to one-third of the original sizes. Figure 5.16b, shows the piers tested in MUST-SIM facility using the large scale actuators. It should be noted that, these piers were heavily instrumented with Krypton LEDs, strain gauges and displacement transducers.

As described, each component of the CABER project was selected cautiously to achieve reliable structural response. Therefore, the experiment team conducted several tests to confirm the accuracy of the piers scaling down procedure. The original cross-sectional dimensions of the outer piers are shown in Figure 5.17a, while Figure 5.17b illustrates the 1:3 dimensions used in the laboratory experiments. Moreover, the efficiency of discretizing the bridge into several components using UI-SIMCOR was compared to the behavior of the whole bridge have been verified analytically [Frankie, 2013].
Figure 5.16 The experimental components of the CABER project; (a) 1:20 scale pier, and (b) 1:3 scale pier [Frankie, 2013]

Figure 5.17 The cross-sectional dimensions of the outer pier; (a) full scale, and (b) 1:3 scale [Frankie, 2013]

A final remark regarding the problem description is the utilized coordinate system. Figure 5.18 shows the coordinate system of the experimental module and the corresponding one used in ZeusNL. It can be observed that the Y and Z directions are reversed between the two modules.
This observation is important to be able to compare the model updating results with the accurate experimental values.

Figure 5.18 The coordinate system used in UI-SIMCOR for the different modules [Frankie, 2013]

The CABER project setup provides an excellent environment to investigate the model updating approach. The proposed idea is to learn the constitutive relationship parameters from one of the tested piers. Then, these parameters are used to update the analytical modules. It is obvious that the test setup of the CABER project is much more complicated than the steel connections discussed earlier, some of its features are summarized as follows;

- All the critical components (piers) of the bridge were tested in the laboratory. Therefore, it can be fairly assumed that the output results of this experiments represents the accurate bridge response.
- As shown in Figure 5.14 the three piers were not symmetric, where the height of piers 1, 2 and 3, before scaling down, were 28.5, 37.5 and 22.5 ft, respectively. The success of this
approach, proves that updating the model on the constitutive relationship level can be applied even if the source and the modified components are not perfectly matching.

- The two outer piers were heavily instrumented, and their records were well organized. Hence, there are sufficient experimental data for the model updating procedure.
- The hybrid simulation test utilized ZeusNL as an analytical platform. ZeusNL is already included in the developed model updating program. In addition, all the analytical models were already provided by the experiment team, which saved a lot of unnecessary effort.
- Some challenges are expected while conducting the model updating approach. For instance, the piers were constructed of concrete, which is more difficult to analyze than the steel materials. Moreover, the piers were subjected to six degrees of freedom forces. The interaction between several forces might add complexities to the concrete model behavior. However, these challenges are useful to explore the ability of model updating in evaluating complex models.

In this section, some features of the CABER project were briefly presented. More details about the test setup and the bridge response can be found in literature [Frankie, 2013; Abdelnaby et al., 2014]. The next section, discusses the application of the model updating approach and its output results.

5.4.2. **Model updating procedure and results**

The objective of this example is to determine the constitutive relationship parameters from one of the tested piers, to update the characteristics of the corresponding numerical modules.
incrementally. To satisfy this objective, three analysis cases are considered, which can be described as follows:

1- The exact case, which utilizes the experiment records as a reference solution. In the hybrid simulation experiment, all the three piers were physically tested. On the other hand, the bridge deck and abutments were expected to behave in the elastic range and hence, these components were simulated numerically using ZeusNL.

2- The analytical solution, where all the bridge components were represented numerically using ZeusNL. The material parameters used to define the piers response were defined based on the coupon test results for the reinforcement bars, and based on the concrete cylinders tests for the concrete core and the pier cover. [Frankie, 2013].

3- The model updating case, where the identification procedure discussed in the previous examples, will be used to determine the constitutive relationship parameters from one of the tested piers. Then, the identified parameters will be used to update the numerical piers incrementally. The superstructure will be simulated using ZeusNL without any updating procedure.

In the CABER project, the outer left pier was subjected to the higher level of loading. Therefore, its response is expected to be ahead of the other piers. In addition, its geometric configuration complies with the assumption of the flexural beam theory. Thus, the left pier is selected to be the source module used to update the corresponding numerical piers. As in the previous example, a fiber analysis program is required to analyze the pier cross-section. The program was developed with the purpose of finding the constitutive model parameters used to update the numerical model.
First, it is important to select the section along the pier that includes sufficient instruments to determine the strain distribution.

Figure 5.19 shows the locations of the strain gauges that were attached to the outer piers during the experiment. For the left pier (name column 1 in the figure), strain gauges were instrumented at 10 different levels. Four strain gauges were attached at the longitudinal reinforcement bars for each level. However, several strain gauge records were either lost or the instrument failed before the end of the test. The four strain gauge records were available only for sections at levels 2, 3, and 4. The records at level 4 (section at distance 93 inches from the pier cap) showed the most consistent pattern and hence, it will be used for further analysis.

Figure 5.19 The strain gauges attached to the outer piers in the CABER project [Frankie, 2013]
Another challenge in the CABER project is the circular cross-section of the piers. Whereby assuming linear relationship is not sufficient to determine the strain on each fiber. Instead, two-dimensional interpolation was utilized. The following points summarize the approach followed and the assumptions adopted to calculate the strains on each fiber, and then the procedure used to determine the constitutive model parameters will be described:

- The chosen cross-section is divided into several fibers in the radial coordinates. Figure 5.20a shows an illustration for the fiber discretization. It is worth noting that, the actual problem that will be solved for this example is more complicated than the one shown in the figures, as the pier cross-section is divided into three materials, which are: (1) confined concrete core, (2) concrete cover and (3) longitudinal reinforcement steel bars. Each concrete material includes a number of fibers (i.e. 100 fibers), while the area of steel bars will be analyzed without any further discretization.

- From the 17 longitudinal reinforcement bars only 4 of them had strain gauges (SGs) attached. This is expected as the original experiment was designed to be a conventional hybrid simulation test, and attaching additional SGs to determine the constitutive relationship behavior was not of crucial importance.

- Due to the limited number of strain gauges, the procedure used to determine the strain on each fiber is divided into several step. First, the circle quadrant that includes the fiber of interest is selected. Then, the radial distance (R) and its inclination angle (Φ) of that fiber are calculated as shown in Figure 5.20. Next, the coordinates of Points (A) and (B) can be determined based on the angel (Φ) and the geometric relationships of the pier cross-section. Hence, linear interpolation is used to evaluate the strain values at these two points. Finally,
to get the strain values at any fiber, another interpolation is applied between the strains of Points (A) and (B) depending on the radial distance (R). These steps are then repeated for all the fibers included in the section.

- In order to identify the constitutive model parameters, the straining actions at the section of interest (section at a distance of 93 inches from the pier cap) are required. However, the restoring forces were only recorded at the actuator level. Therefore, equilibrium conditions can be applied to determine the straining actions at the pier section as follows:
  - $M_x^{\text{level4}} = M_x^{\text{top}} \text{ (kips.in)} + 93 \text{ (inches)} \times F_y^{\text{top}} \text{ (kips)}$
  - $M_y^{\text{level4}} = M_y^{\text{top}} \text{ (kips.in)} - 93 \text{ (inches)} \times F_x^{\text{top}} \text{ (kips)}$
  - $N_{\text{level4}} = N^{\text{top}} \text{ (kips)}$

Where, $M_x$ and $M_y$ are the bi-directional moments applied at the pier cap. $F_x$ and $F_y$ are the lateral forces in the global directions, while $N$ is the axial load applied to the pier by the actuators.

- At this stage, the straining actions determined from the fiber analysis program, based on an assumed constitutive model, can be compared to the straining actions evaluated from the experimental records. Optimization tools can be applied to find the constitutive model parameters that minimizes the difference between the straining actions determined from the analytical and experimental components.

- At each loading increment, the identified constitutive model parameters are sent to ZeusNL to update the response of its piers. The communication protocols used to exchange the stress-strain information between UI-SIMCOR and ZeusNL were described in Chapter 3.
The procedure introduced for identifying the constitutive relationship parameters shows that it is computationally involved. Whereby, optimization algorithms need to operate on three levels of the section, which are dependent on each other. First, it identifies the stresses on the fiber level. Next, the stresses are combined to find the straining actions on the section level. This step requires analyzing three construction materials, which are concrete core, concrete cover and reinforcement bars. Finally, optimization compares the analytical straining actions to the experimental records. There are some assumptions that are implicitly included in the identification procedure such as plane section remains plane and considering linear strain distribution along the pier cross-section. These assumptions are acceptable regarding the purpose of model updating, as it tries to improve the response of the analytical modules guided by the physically tested component. As mentioned earlier in Chapter 3, there are several reasons of uncertainties in the analytical representation of the actual engineering problems. Therefore, updating the constitutive relationship will tend to smear those uncertainties to achieve an overall more reliable response.
The strain values recorded at level 4 of the left pier are plotted in Figure 5.21. The results are reasonable relative to the moments generated in the longitudinal and transverse directions. Whereby, the higher strain values corresponds to the larger moments. Moreover, it can be observed that the strains have much larger values in the tension direction as concrete usually cracks at early loading stages and the reinforcement bars are mainly responsible for the tensile stresses.

![Strain Gauges](image)

**Figure 5.21** The strain gauge records at level 4 of the left pier for the longitudinal and transversal strains

For the numerical modules, ZeusNL was utilized and the pier cross section required three types of stress-strain models to be defined. The concrete core and cover were both defined using modified Mander non-linear concrete model (CON2). Meanwhile, the longitudinal reinforcement bars were represented by the bilinear steel model (STL1). The initial values of the concrete compressive
strength, the steel Young’s modulus and the steel yield strength were defined to be 6.5 ksi, 29000 ksi and 66 ksi, respectively as suggested by the CABER project report [Frankie, 2013]. These values were obtained from some concrete cylinders and coupon tests for the materials used in this experiment. Figure 5.22 shows the stress-strain plots for concrete and steel materials.

![Strain Calculated from 300 kip Crosshead Readings](image)

(a) (b)

Figure 5.22 Stress-strain results for the pier material; (a) concrete cylinders, and (b) steel bars coupon tests [Frankie, 2013]

The coupon test constitutive model parameters were used to calculate the straining actions on the pier cross-section. These straining actions were compared to the ones evaluated from the experimental records using the equilibrium equations. Figure 5.23 shows the results of the analytical and the experimental straining actions. The three plots represent the flexural moments (Mx/My) and the axial forces (N) acting on the left pier cross-section. It is important to note that, until this point no optimization or identification algorithms are applied. These values represent the analytical solution based on the suggested coupon test results. In the following section, the improvement due to the identification procedure will be discussed.
Several observations can be noted from the results shown in Figure 5.23. For example, the pattern of the fiber analysis solution followed the experimental records to some extent. As expected, the values are not matching because the analytical constitutive relationship cannot perfectly represent the actual model response. Moreover, through the entire loading history the analytical solution showed higher values for Mx, My and N than the experimental ones, which implies that the
assumed constitutive model was stiffer than it should be. This observation agrees with one of the conclusions suggested in literature, which states that springs need to be added at the pier boundaries to reduce the stiffness of the numerical model response [Abdelnaby et al., 2014].

These results helped in getting more understanding of the bridge model, yet it was important to do more investigations about the experiment. Therefore, some concerns regarding the experiment were discussed with the CABER project team leader [Frankie and Elanwar, 2014]. This discussion was very constructive and some the important notes are summarized as follow:

1. The coupon test results used to evaluate the constitutive model parameters included many uncertainties, because some of these tests were conducted at different facilities and the identification procedure was not very accurate. For example Figure 5.22a shows that there are three different values for the compressive strength of concrete (i.e. 6.5 kis, 5.5 ksi and 4 kis approximately). In addition, for the reinforcement bars coupon tests, some specimen gave reasonable results for Young’s modulus (29000 ksi) but in other cases the values were much lower than expected. Therefore, it is acceptable to identify softer parameters during the model updating procedure.

2. The results during the first segment of the input loading (i.e. 0-10 seconds) were not as accurate as the rest of the experiment. The reason of this phenomenon is that the load was extremely small at this stage, and the actuator tolerances were on a close order of magnitude to the input command. Hence, the hybrid simulation equipment was applying a correction step after each increment, which resulted in records contaminated with many distortions.
It was suggested to ignore the first 10 seconds and start the optimization procedure afterwards.

3. It was noticed that the fiber analysis response are much stiffer in Mx compared to My values. This observation was consistent with one of the findings of the research team, which demonstrates that the pier-cap could not transfer all the loads from the actuator to the pier in the transversal direction. To overcome this issue, they added springs at the pier boundaries to induce some flexibility to its response.

Based on this discussion, the main features of the optimization procedure can be characterized, where the identification procedure will start after the first 10 seconds to avoid the distortions in the initial loading phase. In addition, the moment in the longitudinal direction (Mx) will be the only criteria used to determine the error between the analytical and the experimental models. This approach is adopted because there were uncertainties related to the boundary conditions in the transversal direction (My) as earlier mentioned. On the other hand, the response in the axial direction is usually sensitive to errors due to its high stiffness. These assumptions are implemented to the optimization approach, which in turn will be applied in the model updating procedure. Figure 5.24 shows the moments and normal forces after modifying the concrete and steel bars constitutive models identified through optimization. It is important to notice that in all three plots, the fiber analysis and the updated results coincide in the first 10 seconds of loading, because optimization was not yet applied.
Figure 5.24 Comparing the fiber analysis and optimization results for moments and normal force against the experimental ones

The figure shows a significant improvement in the moment and normal force values, which implies that the optimization algorithms successfully reduced the difference between the analytical and the experimental model. Additionally, it can be observed that the moment values were more consistent.
with the experimental data than the normal forces as earlier discussed. It is can also be noted that the identification procedure started after the first 10 seconds of the time history. The constitutive model is path dependent and hence, and any error that occurs at the early stages could not be recovered in later steps. Therefore, it is expected that the results would have improved if the initial segment was suitable for the optimization application.

Although the gradient-based Interior point method was used for optimization, the identification program required a long computation time. Therefore, two techniques were adopted to reduce the identification duration: (1) the original loading increments were 4000 steps, for updating purposes this number was resampled to 1000 steps only. (2) Originally, all the loading increments were used to evaluate the error between the analytical and experimental values. Consequently, the calculation time increases as the time history proceeds. Instead, another approach was implemented, by evaluating the behavior of the last 4 seconds only starting from the current step. The 4 seconds were assigned as they cover two loading cycles on average. This approach is convenient, as the identification algorithm is mainly affected by the recent events in the pier response, otherwise, the model will be always dominated by the behavior of the early stages. After successfully implementing these techniques, the identification time reduced significantly (i.e. around 2 hours), which is convenient for applications such as hybrid simulations. For future applications, the analysis code can be further optimized to reduce the calculation time; in addition, faster platforms other than MATLAB can be utilized.

The previous analyses showed that updating the constitutive model was able to achieve more reliable response on the section level. Whereby, the different sources of uncertainty in the
analytical model were modified within the constitutive relationship response. Next, the identified parameters will be induced incrementally to update the behavior of the three piers in ZeusNL model to evaluate the improvement in the response on the global level of the bridge. Figures 5.25 and 5.26 show the moment and force values developed at the top node of the left pier. The plots illustrates the response of the analytical response, where the constitutive model parameters were defined based on the coupon test values compared to the numerical response after applying the model updating approach. These values are plotted against the exact experimental results. The right side of the each figure shows the plots for the percentile errors in the straining actions for the updated and the pure analytical solutions normalized to the peak response. Meanwhile, the quantitative errors for both models are listed in Table 5.1.

Several observations can be noted from the two figures and the values listed in the table. The model updating approach reduced the errors in most of the straining actions characteristics significantly. For the lateral forces, the average absolute error was reduced by about 45%, while the moment error reduced by 30%, approximately. On the other hand, negligible improvement occurred in the axial forces (Fz) and the torsional moments (Mt). This result is acceptable, because the error in the axial forces before updating the model was not significant, and hence it was not expected that updating the parameters will cause much improvement. Meanwhile, the toque-twist model in ZeusNL follows a linear relationship. This model is independent on the constitutive relationship. Therefore, changing its parameters does not have a direct effect on the torsional response.

Due to some experimental issues discussed earlier, the model updating procedure did not start until the end of the first loading segment. This decision had a negative impact on the accuracy of the
results. Additionally, in this experiment the static loading applied on the pier at the beginning of the experiment had a considerable contribution to its response. These loads created permanent deformations that could not be recovered after updating the model. This phenomenon is obvious in Figure 5.25 for the moment records (Mx and My). Whereby, permanent errors occurred at the initial stages. If the model updating was applied from the beginning of the test, the response of the numerical model would have improved considerably. The figures also show that there is a shift in the pattern between the numerical model response before and after updating the constitutive relationship parameters. The shift in the response pattern is due to changing the modal parameters in ZeusNL. As earlier mentioned, the optimization approach identified a new set of parameters, which were softer than the initially defined ones. Consequently, the pattern of the updated numerical model is closer to the experimental values.

On the other hand, the six deformations for the analytical and model updating cases are shown in Figures 5.27 and 5.28, while the summary for the evaluated errors are listed in Table 5.2. The improvement in the deformation results after using model updating was not as obvious as in the straining actions. For example, the average absolute error for most of the deformations reduced within the range of 10-20%, because the deformations were more sensitive than the forces to the effect of rigid boundary conditions and to the permanent errors that occurred initially by the static loading. The only exception was the twist angle (rz), which did not improved and even became slightly worse. As mentioned earlier, ZeusNL represents the torque-twist model with a linear relationship, which cannot capture the actual model response. Although the improvement in the deformations was not as clear as in the straining actions, yet they still provide more convenient structural response than the pure analytical case.
Table 5.1 The average absolute errors for the straining actions of the left pier

<table>
<thead>
<tr>
<th>Load response</th>
<th>Fx (kips)</th>
<th>Fy (kips)</th>
<th>Fz (kips)</th>
<th>Mx (kips-in)</th>
<th>My (kips-in)</th>
<th>Mz (kips-in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>7.35</td>
<td>10.99</td>
<td>10.73</td>
<td>872.62</td>
<td>248.27</td>
<td>256.1</td>
</tr>
<tr>
<td>Updated</td>
<td>3.57</td>
<td>6.78</td>
<td>9.96</td>
<td>612.47</td>
<td>187.47</td>
<td>245.1</td>
</tr>
</tbody>
</table>

Figure 5.25 Comparing the lateral and axial forces of the different analysis cases for the left pier of the bridge

156
Figure 5.26 Comparing the normal and torsional moments of the different analysis cases for the left pier of the bridge
Table 5.2 The average absolute errors for the deformations of the left pier

<table>
<thead>
<tr>
<th>Deformations response</th>
<th>Dx (in)</th>
<th>Dy (in)</th>
<th>Dz (in)</th>
<th>rx (rad)</th>
<th>ry (rad)</th>
<th>rz (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>0.6789</td>
<td>0.2783</td>
<td>0.0364</td>
<td>0.0046</td>
<td>0.0088</td>
<td>0.0039</td>
</tr>
<tr>
<td>Updated</td>
<td>0.5067</td>
<td>0.2377</td>
<td>0.0436</td>
<td>0.0043</td>
<td>0.0071</td>
<td>0.0031</td>
</tr>
</tbody>
</table>

Figure 5.27 Comparing the deformations of the different analysis cases for the left pier of the bridge
The analysis of the CABER project showed that the pure analytical model could not represent the response of the actual structural component. On the other hand, through applying the proposed model updating approach, results improved considerably. This section focused on the analysis of
the outer left pier as a representative of the bridge. Meanwhile, using the model updating technique reduced the errors for the right pier with factors between 25-40% for the lateral forces and moments. In Appendix A, the plots showing the response of the two outer piers are illustrated. In addition, the hysteretic response plots are included. The following section will summarize the main features discussed about the experiment setup and it will recommend some approaches to improve the numerical model response.

5.4.3. Discussion

This section presents a brief summary about the CABER project test setup. Next, it introduces some of the conclusions discussed by the research team that conducted the experiment, which will be reflected on the model updating results. Finally, the main challenges encountered in this example and the recommendations for future experiment will be presented.

The CABER project provided an excellent environment to explore the feature of the model updating approach. The output resulting from the hybrid simulation experiment was considered a reference solution for the curved bridge response, where in this experiment all the critical components (three piers) were physically tested in the laboratory. Meanwhile, the superstructure and abutments, which were designed to behave in the elastic range, were simulating numerically using ZeusNL. Moreover, the procedure used to scale down the piers was verified experimentally, meanwhile the discretization approach used in UI-SIMCOR was verified analytically [Frankie, 2013]. The tested piers were subjected to multi-directional earthquake records, which developed six degrees of freedom actions on the bridge piers. The experiment was rich in the records, because
all the tested piers were covered with a large number of measurement instruments. The test setup and the bridge configuration was found suitable to verify the model updating concept.

Some of the conclusions discussed in the CABER project report were found useful for the model updating analysis, where a part of the original project objectives, was to investigate the accuracy of the numerical ZeusNL models, and to calibrate its components if needed. However, the results of the un-calibrated ZeusNL model were extremely inconsistent with the experimental response. Therefore, several features of numerical model were modified to achieve more realistic bridge response [Frankie, 2013]. Two main features were calibrated, which are; (1) the linear torque-twist model available in ZeusNL was not sufficient to represent the actual structural response. This issue was solved through inducing a torsional spring at the top and bottom of the pier. These springs provided the necessary non-linear behavior, which required some modifications in ZeusNL source codes. (2) The boundary conditions of the piers did not follow the theoretical expectations. For example, the top node in the transversal direction and the bottom node in both directions could not develop the assumed infinite rotation capacity. This issue was not encountered in the longitudinal direction, because the deck of the bridge supported the top node of the pier. This problem was solved by including non-linear rotational springs at the two nodes. The spring parameters were determined based on the Krypton LED data, which showed the actual inflections in the piers during the test [Abdelnaby et al., 2014]. After applying these calibration techniques, the results of the numerical model approached the experimental results.

This discussion can be reflected on the model updating results. For example, the torsional-twist model in ZeusNL does not depend on the constitutive relationship parameters. Therefore, model
updating could not improve the torsional response, which supports the approach of using non-linear torsional springs to solve this issue. On the other hand, using the springs to adjust the moment-rotation behavior might not be sufficient, because the fiber analysis results on one of the pier sections did not match with the experimental records. The main factor that govern the fiber analysis results is the constitutive model behavior. In addition, there were many uncertainties related to the values provided by the coupon test. Moreover, the calibration procedure proposed in literature was not applicable for all the input segments and hence, it was divided into slight, moderate and severe calibrations based on the level of loading [Frankie, 2013]. This approach confirms that the model calibration using spring is not sufficient and that these issues can be related to the complex constitutive model behavior when the member is subjected to combined loading.

The approach proposed in this research helped in improving the numerical model response. Whereby, the constitutive relationship parameters were identified based on the experimental records of the left pear, and then they were used to update the response of the corresponding numerical pier incrementally. Several conclusions can be summarized to highlight the main feature of model updating approach such as;

- The model updating results showed a significant reduction in the errors of the pure analytical model. For some straining actions, the error was reduced by 30-50%. However, the deformations did not improve in the same manner. The overall hysteretic behavior of the pier was more convenient, when model updating was applied.
- The identified constitutive relationship parameters such as the compressive strength and Young’s modulus were on average much softer than the initial values. This behavior is
acceptable, because applying combined loading on the structural component might have a negative impact on the strength of the model. In addition, the parameters defined initially were determined from the coupon test results, where the quality control on them was not satisfactory.

- The torsional moment did not achieve an observable change after updating the model, which confirms that the linear torque-twist relationship implemented in ZeusNL was not sufficient to generate a realistic behavior.

- Initially, the optimization algorithm required very long time to determine the model parameters. Nevertheless, after resampling the loading steps and after evaluating the response of few loading cycles, the computational time was reduced. The overall time required to complete one analysis was in the order of 2 hours, which is reasonable for slow rate experiments such as hybrid simulations.

- The moments in the longitudinal direction (Mx) of the left pier were used to evaluate the difference between the fiber analysis and the experimental records. The axial forces and the moments in the transversal direction were excluded, because the axial forces showed high sensitivity to errors especially, near the failure zones. Meanwhile, some uncertainties were related to the pier behavior in the transversal direction during the test.

- Optimization was not applied until the end of the first loading segment (i.e. after 10 seconds of loading). This was one of the shortcomings in the experimental data, as the input deformations were of close order of magnitude to the actuator tolerance. Therefore, the static loading and the first earthquake segment did not participate the identification procedure, which had a negative impact on the result.
In the CABER project, the static loading had a considerable contribution to the permanent deformations that occurred in the model at the early stages. Consequently, model updating could not be utilized except after the static loading and the first 10 seconds of the input records were applied. However, the trend of the model updating afterwards showed that the identified parameters were reduced approaching the lower bound. In order to investigate more about this issue, the average of the identified parameters were manually assigned to ZeusNL. In this approach, model updating was not applied. Instead, the new constitutive relationship parameters were used as the initial values for the analytical solutions. The results and figures are presented in Appendix B. The error decreased to more than 50% for the lateral forces and moments.

The main conclusion of the CABER project experiment would be that the calibrating the numerical model requires two complementary approaches. First, adding nodal spring to achieve more realistic boundary conditions for the analyzed component. Especially for the torsion-twist relationship, as suggested in literature [Abdelnaby et al., 2014]. However, this approach is not sufficient because it was not successful for all the loading stages. In addition, equilibrium was not achieved on the section level in the fiber analysis results, which implies that the constitutive relationship required some modifications. The second approach is to apply model updating to achieve a more reliable constitutive behavior. Model updating reduced the errors in the structural response significantly. Moreover, it addressed the uncertainties in the coupon test results as it identified the required parameters directly from the hybrid simulation measurements. It should be noted that, adding a non-linear springs for the torque-twist model, would have a positive impact on the reaming characteristic of the model. Finally, the constitutive relationship behavior becomes more complicated near failure stages or when subjected to combined loading. Therefore, the
The proposed model updating procedure is beneficial as it acquire the behavior from the physically tested component.

The analysis of the CABER project was very useful to evaluate the capabilities of updating the constitutive model during the analysis. In this example, and due to the complicated model configuration, it was obvious that the identified model parameters aimed to smear the different uncertainties in the analytical model such as the boundary conditions. Although in this experiment, the ability of model updating approach was explored, yet a number of challenges were encountered during the analysis. These challenges are listed along with some recommendations to guarantee a more convenient test setup for future model updating applications;

1. Model updating approach is still in the development stages and all its features are not yet explored. On the other hand, the CABER project has several complex properties, which added difficulties on the evaluation process. For example, each pier was subjected to six degrees of freedom loading, and hence, several uncertainties in the constitutive relationship characteristics were induced. In addition, the circular cross-sections required a long fiber analysis procedure and many assumptions to evaluate the strain distribution.

2. The concrete piers were represented by the modified Mander model (CON2). During the analysis procedure, it was observed that in some steps the model determine unrealistic stress values such as imaginary numbers or reaching exceptionally high values. However, after detailed investigations, it was observed that the stress was occasionally guided to a different loading or unloading branch, because of changing the model parameters. This issue was discussed in details in Chapter 4, where the issue was solved by modifying
some features in CON2 subroutines to make sure that the stress are evaluated through the correct path.

3. For the two outer piers, most of the strain gauge data were lost and could not be recovered. Only three sections had all the strain data available. Therefore, the identification procedure focused on one local zone of the pier. Increasing the number of strain gauges would add redundancy to the evaluation process, which helps in eliminating any outliers. Consequently, for future experiments, it is suggested to design the configuration of the measuring instruments to support model updating applications.

4. The experimental records for the first input earthquake segment were not reliable. Hence, the identification procedure was applied while overlooking the static loading and the first segment of the dynamic loading. This approach had a negative impact on the efficiency of model updating as shown in the results of Appendices A and B.

5. The low quality control on the coupon test caused many uncertainties in the analysis of the experiment. Whereby, the initial constitutive model parameters were not good representative for the actual model response.

These comments show that for future experiments it is important to design the test setup to serve model updating applications. In general, the analysis of the CABER project confirmed the ability of model updating to improve the response of the numerical modules without adding much experimental effort. Moreover, it was verified that updating the constitutive model parameters could be applied even if the source and the modified components have slightly different geometric and loading conditions. However, this approach was not sufficient, because the torsional-twist
response in ZeusNL does not depend on the constitutive model parameters. Therefore, adding nodal springs can also further help in reducing the numerical model errors.

For the structural assessment, there is always a compromise between the cost, effort and time required on one side and the accuracy of the model on the other side. For the bridge analyzed in the CABER project, three piers were physically tested. However, the fabrication and instrumentations of each pier requires a lot of cost and several days of preparation. In addition, as the number of tested modules increase, convergence issues might occur during the experiment. On the other hand, through adopting the proposed model updating approach, testing 1 pier might be sufficient, yet the accuracy will not be the same as the testing all the piers. Therefore, engineering judgment is crucial to determine the modules that can be simulated numerically, without losing much accuracy by updating its constitutive model behavior. Moreover, the value of model updating is emphasized in case of having structures with multiple critical components such as high-rise buildings.
CHAPTER 6
CONCLUSIONS

Model updating is an effective approach, which can be integrated with conventional hybrid (experimental-analytical) simulation framework to improve the overall quality of the structural assessment. In this thesis, it was shown that updating the constitutive relationship of the numerical modules during the test, based on the instantaneously-measured response of the tested components, can substantially reduce the errors in the hybrid simulation results. Optimization tools and neural networks were proposed as two possible alternatives to identify the stress-strain information from the physical specimen. The simulation coordinator platform UI-SIMCOR was modified to integrate the different components of model updating. Moreover, its communication protocols were adapted to exchange the stress-strain data with the finite element program ZeusNL, during the analysis. The proposed approach was verified through investigating several numerical example, and through evaluating the output results of previously conducted experiments.

6.1. Main Findings

In this section, a brief description of the model updating approach components is presented. In addition, the main findings related to the numerical and experimental verification examples are discussed. Finally, general remarks regarding the proposed concept are highlighted.

6.1.1. Model updating approach

In order to apply the model updating approach, a number of modifications were necessary to the conventional hybrid simulation procedure. In the following points, the motivation of this research study and the main features of model updating are summarized;
Hybrid simulation is an intuitive approach that integrates the advantages of both numerical and experimental techniques. Whereby, the critical members are tested in the laboratory, which achieves accuracy, while cost efficiency is preserved as the rest of the structure is simulated numerically.

There are few platforms that can communicate between the different hybrid simulation modules. The simulation coordinator program UI-SIMCOR was utilized in this research, which can integrate with several finite element programs.

Model updating aims to enhance the efficiency of the numerical modules response in the conventional hybrid simulations, where the behavior of the numerical modules is updated incrementally based on the data learned from their physically-tested counterparts.

Several action-deformation characteristics can be acquired from the physical specimen. In this research, the constitutive relationship was utilized as a fundamental property that update the model on the material level.

In order to identify the constitutive model parameters from the accurate component, genetic algorithms and interior point methods were used as optimization tools. Optimization is suitable in case of having a theoretical constitutive model, which requires modification to match the behavior of the accurate component. On the other hand, neural network was proposed as an alternative, which handles the structural response mathematically, without the need of a pre-existing analytical model.

A graphical user interface program was developed to support model updating purposes. The program can exchange the constitutive relationship data between UI-SIMCOR and
ZeusNL during the analysis. Moreover, several models were integrated in the program library, which are STL1, STL2, STL3 and CON2.

- The model updating framework was developed for slow rate hybrid simulation tests. However, for real time applications the communication protocols and the identification algorithms need to be modified.

6.1.2. Numerical verification

In order to verify the efficiency of model updating and to explore its potential applications, several numerical examples were analyzed. The main findings are listed as follows:

- Four analytical problems were solved using ZeusNL to assess the efficiency of the model updating procedure. These problems were designed to evaluate the performance of four different constitutive relationships. The results showed that model updating can substantially reduce the errors compared to the conventional approaches.
- It was demonstrated that model updating can be applied even if the source and modified modules do not share the same geometry and cross-sectional dimensions. This conclusion was expected because updating the model on the constitutive level allows for flexibility in the analysis procedure.
- Model updating was applied to modify a simple stress-strain model to match the behavior of a more complicated one. It was shown that through using optimization, the bilinear relationship captured the main features of the more advanced Ramberg-Osgood model, which had a significant effect on improving the overall structural response.
Neural network was used to represent the model behavior in the absence of any analytical constitutive relationship. The results showed very good agreement with the accurate numerical results. For a more realistic representation, the input data used to train the network was contaminated with noise, which had a minor effect on the accuracy of the response. Yet, the current neural network was not applied incrementally. Instead, the network was trained based on the full history of a previous analysis results.

6.1.3. Experimental verifications

The objective of model updating is to enhance the ability of hybrid simulations to assess the structural response. Therefore, it was important to confirm that the proposed approach can handle actual experimental data. Instead, of conducting validation tests, experimental models available in literature that cover a wide variety of applications were investigated. A summary of the major outcomes is shown as follows:

- An inverse identification procedure was utilized to determine the stress-strain data during the test. In this procedure, optimization tools searches for the constitutive model parameters that achieves equilibrium on the section level through comparing the analytically determined and experimentally evaluated forces applied on the section.
- In order to verify the identification procedure, a steel beam-column connection test was analyzed. The results showed that the determined constitutive model parameters matched accurately with the coupon test values provided in the literature. Additionally, the moment values from the experimental measurements were consistent with the fiber analysis results.
The same test setup was analyzed again, but in this example the beam flanges were heat treated, which changed its constitutive behavior significantly. The inverse identification procedure was applied to determine the constitutive model parameters from a certain section in the beam. Then, the new parameters were implemented to calculate the moment values at a different section. The results showed that the calculated moments matched closely to the experimental ones. This example demonstrated that, model updating can be used to determine the constitutive behavior from a local zone in the experimental component, to represent the behavior of the corresponding numerical modules.

The most comprehensive example used to verify the model updating concept was the CABER project, which was a hybrid simulation experiment conducted at the MUST-SIM facility to evaluate the seismic response of a curved bridge subjected to combined loading. The measurements and records available for this experiments were utilized to determine the constitutive parameters of one of the concrete piers tested in the laboratory. Then, these parameters were supplied to the analytical model incrementally to update its response. The main conclusion from this analysis are summarized as follows:

- Applying model updating reduced the error for the lateral forces and the moments (i.e. 30-50%) compared to the pure analytical model. This results show the value of updating the constitutive model behavior.

- Negligible effect was observed regarding the torque-twist response, because the torsional model adopted in ZeusNL is independent from the constitutive relationship parameters. Hence, it is convenient to add non-linear spring to represent the actual pier behavior. Moreover, adjusting the torque-twist model would have a positive indirect effect on the other straining actions.
Due to some inaccuracies related to the experimental measurements, the identification procedure started after the first segment of the input loading was already processed (i.e. after 10 seconds). This approach had a negative impact on the ability of model updating to improve the results.

Based on the detailed investigation of this example, the procedure suggested to achieve an accurate numerical module is to adopt two complementary techniques; (1) updating the constitutive model parameters using the proposed identification procedure, which was applied in this thesis and (2) adding non-linear springs to regulate the behavior of the pier boundary conditions as suggested in literature. It is believed that either of the two approaches cannot outperform the other, and that both are required to achieve realistic analytical response.

6.1.4. Remarks on model updating

Several examples were analyzed adopting the model updating approach. A number of conclusions were drawn. The most significant conclusions are listed below:

- In traditional hybrid simulation tests, focus is placed on the global behavior of the tested member. However, the model updating examples showed that it is important to investigate the constitutive relationship behavior especially, if the member is subjected to combined loading, where the coupon tests are not sufficient to characterize its behavior.
- For path-dependent problems, the accuracy of the numerical modules has a direct effect on the experimental component, as shown in one of the analytical examples. Therefore, model updating is valuable in providing a more reliable test environment.
– Applying model updating at the early loading stages has a positive impact on the overall accuracy of the analytical model, because permanent errors that occur during the analysis cannot be recovered at later stages. Using smaller time increments helps in reducing the errors because the identification algorithm has more points to evaluate the model response.

– The selection of the source and the modified modules requires astute engineering judgment to confirm that they share close characteristics. Additionally, it is important that the source module be ahead of the modified ones in terms of loading.

– Depending on the model configuration, optimization may require long computational time. However, some techniques can be adopted to reduce the processing time such as using limited number of points to evaluate the error in the analytical model.

– It is preferable to design the test setup of hybrid simulation taking into account model updating requirements. For instance, the zones of interest in the physical specimens need to be heavily instrumented, as they will be used to identify the model behavior. Moreover, the redundancy in the attached instruments reduces the impact of outliers and increase the accuracy of the identified features.

Model updating is a powerful approach used to achieve a more realistic numerical model response. Hence, it limits the number of the critical members that need to be physically-tested in hybrid simulations. Such approach reduces the cost of conducting the experiment without losing much of the model accuracy. In addition, it allows for assessing structures with several critical components such as multi-bay bridges and high-rise buildings.
6.2. Future Work

A number of issues had not been addressed in this research study and others that are worthy of explorations. A selection of topics for future research is listed below;

- The current model updating framework can be modified depending on the requirements of the future applications. For example, additional constitutive models might be added to the developed program library, if it better represents the behavior of the tested specimen. Moreover, alternative finite element programs other than ZeusNL can be integrated in the model updating framework.

- Real time hybrid simulation is used to analyze a number of structural engineering problems. However, the time required for running the identification algorithms hinders the ability of applying real time techniques. Therefore, it is suggested to developed more advanced tools that can determine the required model parameters in a shorter time.

- There are several complexities in representing the actual constitutive model behavior using analytical idealizations. However, model updating can be used to explore the behavior of some model characteristics such as rate dependency, thermal effect, isotropic hardening, etc. In addition, it is essential to develop new constitutive relationships that support model updating applications to avoid the challenges encountered in the previously discussed examples.

- Conducting a number of experiments that utilize model updating would be very beneficial to explore the different properties of this approach. The design of the test setup must be customized to allow model identification from the experimental member. For verification
purposes, it is recommended to adopt a simple test setup in order to focus on the model updating components, without considering intrusions due to the structural complexities.

In this research study, the concept and the main components of model updating approach were presented. It was shown that, model updating has the ability to reduce the gap between the numerical and the actual structural response. In addition, it can be used for a wide range of applications without entailing further effort on the conventional hybrid simulations. Nevertheless, some issues from different engineering fields, that are not yet explored, were discussed in this section. Therefore, it is important to extend the current model updating efforts to serve more comprehensive applications.
REFERENCES


Molina, F. G., Verzeletti, G., Magonette, G., Buchet, Ph., Renda, V., Geradin, M., Parducci, A.,


Department of Civil and Environmental Engineering, North Carolina State University, Raleigh, NC.


Appendix A

This appendix shows the response of the left and right piers utilized in the CABER project. The results compare the pure analytical response to the model updating results. The figures show the 6 degrees of freedom deformations and restoring forces as well as the hysteretic behavior of each pier. In this research, the focus was on improving the response of the lateral forces and moments. However, the torsional-twist results and the axial force plots are included herein for completeness.

The tables included in this appendix list the average absolute error for the analytical solution and the model updating cases. They cover the average errors for deformations and restoring forces for both the outer piers. It worth noting that, the included illustrations are based on updating the numerical model after analyzing the first segment of the input loading (i.e. after 10 sec).
Table A.1 The average errors of the left pier loads, when optimization is applied after 10 sec of the input excitation

<table>
<thead>
<tr>
<th>Load response</th>
<th>Fx (kips)</th>
<th>Fy (kips)</th>
<th>Fz (kips)</th>
<th>Mx (kips-in)</th>
<th>My (kips-in)</th>
<th>Mz (kips-in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>7.35</td>
<td>10.99</td>
<td>10.74</td>
<td>872.62</td>
<td>248.27</td>
<td>256.1</td>
</tr>
<tr>
<td>Updated</td>
<td>3.57</td>
<td>6.78</td>
<td>9.96</td>
<td>612.47</td>
<td>187.47</td>
<td>245.1</td>
</tr>
</tbody>
</table>

Table A.2 The average errors of the left pier deformations, when optimization is applied after 10 sec of the input excitation

<table>
<thead>
<tr>
<th>Deformations response</th>
<th>Dx (in)</th>
<th>Dy (in)</th>
<th>Dz (in)</th>
<th>rx (rad)</th>
<th>ry (rad)</th>
<th>rz (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>0.6789</td>
<td>0.2783</td>
<td>0.0345</td>
<td>0.0046</td>
<td>0.0088</td>
<td>0.0039</td>
</tr>
<tr>
<td>Updated</td>
<td>0.5067</td>
<td>0.2377</td>
<td>0.0436</td>
<td>0.0043</td>
<td>0.0071</td>
<td>0.0031</td>
</tr>
</tbody>
</table>

Table A.3 The average errors of the right pier loads, when optimization is applied after 10 sec of the input excitation

<table>
<thead>
<tr>
<th>Load response</th>
<th>Fx (kips)</th>
<th>Fy (kips)</th>
<th>Fz (kips)</th>
<th>Mx (kips-in)</th>
<th>My (kips-in)</th>
<th>Mz (kips-in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>3.22</td>
<td>9.56</td>
<td>116.7</td>
<td>451.04</td>
<td>209.12</td>
<td>232.44</td>
</tr>
<tr>
<td>Updated</td>
<td>2.14</td>
<td>5.60</td>
<td>116.1</td>
<td>274.60</td>
<td>158.32</td>
<td>197.36</td>
</tr>
</tbody>
</table>

Table A.4 The average errors of the right pier deformations, when optimization is applied after 10 sec of the input excitation

<table>
<thead>
<tr>
<th>Deformations response</th>
<th>Dx (in)</th>
<th>Dy (in)</th>
<th>Dz (in)</th>
<th>rx (rad)</th>
<th>ry (rad)</th>
<th>rz (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>0.417</td>
<td>0.500</td>
<td>0.407</td>
<td>0.0013</td>
<td>0.0045</td>
<td>0.0066</td>
</tr>
<tr>
<td>Updated</td>
<td>0.374</td>
<td>0.500</td>
<td>0.407</td>
<td>0.0012</td>
<td>0.0037</td>
<td>0.0068</td>
</tr>
</tbody>
</table>
Figure A.1 Left pier transversal force-deformation responses and hysteretic behavior
Figure A.2 Left pier longitudinal force-deformation responses and hysteretic behavior
Figure A.3 Left pier axial responses and hysteretic behavior
Figure A.4 Left pier longitudinal moment-rotation responses and hysteretic behavior
Figure A.5 Left pier transversal moment-rotation responses and hysteretic behavior
Figure A.6 Left pier torques-twist responses and hysteretic behavior
Figure A.7 Right pier transversal force-deformation responses and hysteretic behavior
Figure A.8 Right pier longitudinal force-deformation responses and hysteretic behavior
Figure A.9 Right pier axial responses and hysteretic behavior
Figure A.10 Right pier longitudinal moment-rotation responses and hysteretic behavior
Figure A.11 Right pier transversal moment-rotation responses and hysteretic behavior
Figure A.12 Right pier torque-twist responses and hysteretic behavior
Appendix B

This appendix shows the response of the left and right piers utilized in the CABER project. These responses do not represent the proposed model updating approach. Instead, the constitutive model parameters were defined manually to the numerical program. The purpose of this appendix is to overcome some experimental challenges in the bridge analysis. The figures compare the pure analytical response to the model updating results. The results show the 6 degrees of freedom deformations and restoring forces as well as the hysteretic behavior of each pier. In this research, the focus was on improving the response of the lateral forces and moments. However, the torsional-twist results and the axial force plots are included herein for completeness. The tables included in this appendix list the average absolute error for the analytical solution and the model updating cases. They cover the average errors for deformations and restoring forces for both the outer piers.
Table B.1 The average errors of the left pier loads, while assuming soft constitutive model parameters

<table>
<thead>
<tr>
<th>Load response</th>
<th>Fx (kips)</th>
<th>Fy (kips)</th>
<th>Fz (kips)</th>
<th>Mx (kips-in)</th>
<th>My (kips-in)</th>
<th>Mz (kips-in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>7.35</td>
<td>10.99</td>
<td>249.20</td>
<td>872.62</td>
<td>248.27</td>
<td>366.54</td>
</tr>
<tr>
<td>Updated</td>
<td>3.66</td>
<td>5.00</td>
<td>251.0</td>
<td>362.77</td>
<td>126.54</td>
<td>354.47</td>
</tr>
</tbody>
</table>

Table B.2 The average errors of the left pier deformations, while assuming soft constitutive model parameters

<table>
<thead>
<tr>
<th>Deformations response</th>
<th>Dx (in)</th>
<th>Dy (in)</th>
<th>Dz (in)</th>
<th>rx (rad)</th>
<th>ry (rad)</th>
<th>rz (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>0.679</td>
<td>0.278</td>
<td>0.0364</td>
<td>0.0046</td>
<td>0.0088</td>
<td>0.0047</td>
</tr>
<tr>
<td>Updated</td>
<td>0.480</td>
<td>0.242</td>
<td>0.0248</td>
<td>0.0039</td>
<td>0.0065</td>
<td>0.0055</td>
</tr>
</tbody>
</table>

Table B.3 The average errors of the right pier loads, while assuming soft constitutive model parameters

<table>
<thead>
<tr>
<th>Load response</th>
<th>Fx (kips)</th>
<th>Fy (kips)</th>
<th>Fz (kips)</th>
<th>Mx (kips-in)</th>
<th>My (kips-in)</th>
<th>Mz (kips-in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>3.22</td>
<td>9.56</td>
<td>116.7</td>
<td>451.04</td>
<td>209.12</td>
<td>232.44</td>
</tr>
<tr>
<td>Updated</td>
<td>2.15</td>
<td>5.33</td>
<td>114.9</td>
<td>267.2</td>
<td>144.6</td>
<td>197.36</td>
</tr>
</tbody>
</table>

Table B.4 The average errors of the right pier deformations, while assuming soft constitutive model parameters

<table>
<thead>
<tr>
<th>Deformations response</th>
<th>Dx (in)</th>
<th>Dy (in)</th>
<th>Dz (in)</th>
<th>rx (rad)</th>
<th>ry (rad)</th>
<th>rz (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>0.417</td>
<td>0.500</td>
<td>0.407</td>
<td>0.0013</td>
<td>0.0045</td>
<td>0.0066</td>
</tr>
<tr>
<td>Updated</td>
<td>0.402</td>
<td>0.407</td>
<td>0.407</td>
<td>0.0014</td>
<td>0.0041</td>
<td>0.0068</td>
</tr>
</tbody>
</table>
Figure B.1 Left pier transversal force deformation responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.2 Left pier longitudinal force-deformation responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.3 Left pier axial responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.4 Left pier longitudinal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters
Figure B.5 Left pier transversal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.6 Left pier torque-twist responses and hysteretic behavior, while assuming soft constitutive model parameters
Figure B.7 Right pier transversal force-deformation responses and hysteretic behavior, while assuming soft constitutive model parameters
Figure B.8 Right pier longitudinal force-deformation responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.9 Right pier axial responses and hysteretic behavior, while assuming soft constitutive model parameters
Figure B.10 Right pier longitudinal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.11 Right pier transversal moment-rotation responses and hysteretic behavior, while assuming soft constitutive model parameters.
Figure B.12 Right pier torque-twist responses and hysteretic behavior, while assuming soft constitutive model parameters.