Modeling and Analysis of Bridges Subjected to Vessel Impact

by

Wyatt Robert Henderson, B.S. Civil Engineering

Thesis

Presented to the Faculty of the Graduate School of

The University of Texas at Austin

in Partial Fulfillment

of the Requirements

for the Degree of

M.S. Civil Engineering

The University of Texas at Austin

May 2005

Modeling and Analysis of Bridges Subjected to Vessel Impact

APPROVED BY
SUPERVISING COMMITTEE:

____
Eric Williamson

____
Lance Manuel

Dedication

To my family and friends.

Acknowledgements

I would like to thank the Texas Department of Transportation for sponsoring TxDOT Project 0-4650, Vessel Impact on Bridges, which this report is part of. I would also like to thank my fellow students on the Vessel Impact on Bridges project, Kenny Berlin and Adam Cryer. I also need to thank Dr. Lance Manuel and Dr. Loukas Kallivokas for their contributions to this research. I am especially grateful to my advisor and supervisor on this project, Dr. Eric Williamson. This report would not have been possible without his effort and guidance. Finally, I would like to thank all of the people who have made the last two years at the University of Texas at Austin a great experience for me.

May, 2005

Modeling and Analysis of Bridges for Vessel Impact Design

Wyatt Robert Henderson, M.S.E
The University of Texas at Austin, 2005
SUPERVISOR: Eric Williamson

Vessel collision is an important consideration in the design of bridges crossing navigable waterways. The American Association of State Highway and Transportation Officials (AASHTO) *LRFD Bridge Design Specification* governs vessel collision design of bridges in the United States. The AASHTO recommended design procedure for vessel collision is a probability-based calculation that returns an annual frequency of collapse for a given bridge. One of the important calculations in determining the annual frequency of collapse is the ultimate lateral strength of a bridge element, which AASHTO defines as a bridge pier or bridge span. The current AASHTO Design Specification provides

little guidance in the calculation of this value. The primary objective of this report is to provide engineers with the necessary tools to calculate the ultimate lateral strength of bridge elements. This report outlines procedures for modeling and analyzing bridge piers and bridge systems subject to vessel impact loads using a typical structural analysis software package. The methods presented in this report focus on modeling reinforced concrete bridge piers, both with and without shear walls. In addition, the effect of considering system-wide response on the ultimate lateral strength of a bridge is investigated by including the bridge superstructure and adjacent bridge piers in the models.

Table of Contents

List of Tables

List of Figures

CHAPTER 1
Introduction

1.1 Background on Vessel Collision

Vessel collision is an important consideration in the design of bridges crossing navigable waterways. This section clearly illustrates this importance by showing the consequences of vessel collision accidents. General information on the current state of vessel collision design is outlined along with an analysis of where the current design procedures could be improved.

1.1.1 The Significance of Vessel Collision with Bridges

Recent bridge failures in Texas and Oklahoma resulting from barge collisions indicate that engineers need better methods of design and analysis to counter these catastrophic events. On September 15th, 2001 a fully-loaded four-barge tow struck a pier on the Queen Isabella Causeway (QIC) in Texas, destroying a 240-foot section of the bridge and killing 8 people. Figure 1-1 shows the damage caused by the collision. The accident closed the QIC for over two months, the only road link between South Padre Island and the Texas mainland. Repair costs for the bridge were approximately $4.3 million according to the Texas Department of Transportation (TXDOT press release, 2001)

On May 26th, 2002 a tow boat pushing two empty barges struck a pier of the I-40 Bridge outside of Webbers Falls, Oklahoma, collapsing a 503-foot section. Figure 1-2 shows the aftermath of the collision. The incident resulted in 14 deaths and an estimated $30 million in damage, including the cost of re-routing traffic while repairs were made, according to the National Transportation and Safety Board Accident Report (NTSB, 2002). These two events clearly show the damage that vessel collision can cause and the importance of carefully considering this load case in the design of bridges crossing navigable waterways.

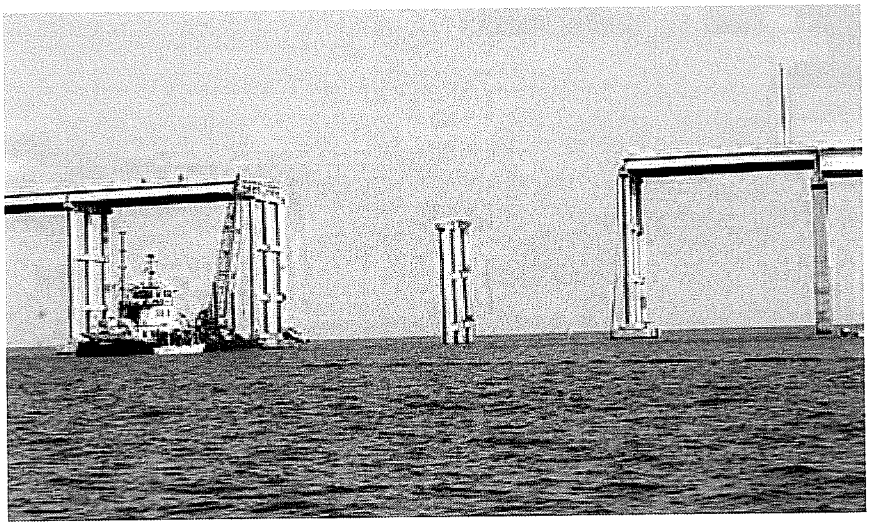


Figure 1-1. Queen Isabella Causeway Damage

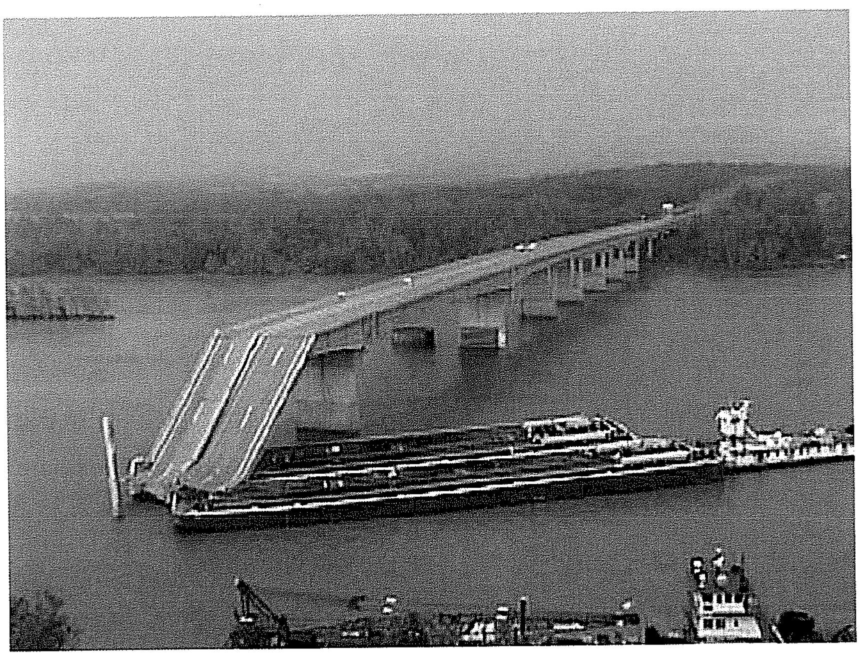


Figure 1-2. Webbers Falls, OK I-40 Bridge Damage

1.1.2 Vessel Collision Design in the United States

Bridge design in the United States is governed by the American Association of State Highway and Transportation Officials (AASHTO) *LRFD Bridge Design Specification* (AASHTO, 2003). Section 3.14 of this document covers vessel collision and is based on the AASHTO *Guide Specification and Commentary for Vessel Collision Design of Highway Bridges* (AASHTO, 1991). The AASHTO Guide Specification provides three methods for the evaluation of bridges spanning navigable waterways. Method I provides the most simple procedure for selection of a design vessel and calculation of an equivalent static impact force to apply to a bridge. A structural analysis is then performed to check if the bridge can resist the applied load. Method II is a probability-based procedure that calculates an Annual Frequency of Collapse (*AF*) for a bridge based on waterway characteristics, vessel traffic data, and bridge geometry. AASHTO provides minimum acceptable *AF* values for various bridge types. Method III is a cost effectiveness analysis procedure where the cost of protecting a bridge is compared against the benefits of reducing the risk to a bridge (AASHTO, 1991). Method III is intended to be used only for unique cases where the risk acceptance criteria using Method I or II result in designs that are unreasonably expensive (AASHTO, 1991)

The AASHTO Guide Specification recommends the use of Method II. Therefore, the AASHTO Design Specification includes only the Method II procedure. Methods I and III are only found in the AASHTO Guide Specification. A brief review of Method II is given below to provide some essential background on the procedure. All three methods are explained in greater detail in Chapter 3 of this thesis.

1.1.3 AASHTO Method II Vessel Collision Design Basics

Design Method II is a detailed, probability-based analysis procedure. It requires a wide range of data on the waterway characteristics, the vessels traversing the waterway and the geometry of the bridge being analyzed. This information is used to compute an annual frequency of collapse for a bridge. A minimum acceptable annual frequency of collapse is given depending on bridge classification. Bridges are classified as 'regular' or 'critical', and the AASHTO Design Specification provides guidance on the factors and parameters that should be considered when determining bridge classification. This topic will be discussed in more detail in Chapter 3.

The annual frequency of collapse calculation is based on the number and type of vessels traversing the waterway, the probability of a given vessel being aberrant, the geometric probability of a collision between an aberrant vessel and a bridge element, and the probability of a bridge collapsing due to a collision with an aberrant vessel. The AF is given by the following equation (4.8.3-1 in the AASHTO Guide Specification):

$$AF = (N)(PA)(PG)(PC) \tag{1-1}$$

Where:

- $AF =$ annual frequency of bridge element collapse due to vessel collision
- $N =$ annual number of vessels classified by type, size, and loading condition which can strike a bridge element
- $PA =$ probability of vessel aberrancy
- $PG =$ geometric probability of a collision between an aberrant vessel and a bridge pier or span
- $PC =$ probability of a bridge collapse due to a collision with an aberrant vessel

Vessel aberrancy is usually the result of human error, mechanical failure or adverse environmental conditions (AASHTO, 1991). The probability of aberrancy (*PA*) calculation is based on several factors including current speed and direction, location of a bridge within a waterway, and vessel traffic density. The geometric probability (*PG*) that an aberrant vessel will strike a bridge element is based primarily on bridge geometry and vessel traffic data. The probability of collapse (*PC*) from vessel collision is a function of two primary variables, the load imparted to a bridge from the colliding vessel and the lateral capacity of the bridge.

The input data and calculations required to calculate the probability of vessel aberrancy (*PA*) and the geometric probability of a collision between an aberrant vessel and a bridge (*PG*) are clearly defined. For example, the probability of vessel aberrancy is increased if a bridge is located in bend/turn regions of a waterway, or if there is a high density of vessel traffic. The geometric probability of collision increases if there are a greater number of bridge piers exposed in the waterway, or if a barge tow has greater overall length. Calculating the probability of collapse term, however, is less well defined than the other terms.

The probability of collapse term is defined as the probability that a bridge will collapse when an individual bridge element (pier or span) is struck by an aberrant vessel. AASHTO defines the probability of collapse as a function of two variables: the impact force of a vessel and the ultimate strength of a bridge element. Determining the impact forces from a vessel collision requires consideration of many factors, including vessel type, size, mass, speed, location of impact on a bridge, and the direction of the impact against a bridge. AASHTO does provide guidance for the calculation of impact forces, but offers little information on the calculation of bridge element ultimate lateral strength. The probability of collapse is given by a curve defined by the following equations:

For $0.0 \leq H/P < 0.1$, PC shall be computed as:

$$PC = 0.1 + 9\left[0.1 - \frac{H}{P}\right] \tag{1-2}$$

For $0.1 \leq H/P < 1.0$, PC shall be computed as:

$$PC = \frac{\left[1 - \frac{H}{P}\right]}{9} \tag{1-3}$$

For $H/P > 1.0$:

$$PC = 0 \tag{1-4}$$

where

$H =$ ultimate bridge element strength (kips)

$P =$ vessel impact force (kips)

These equations are shown as a graph in Figure 1-3.

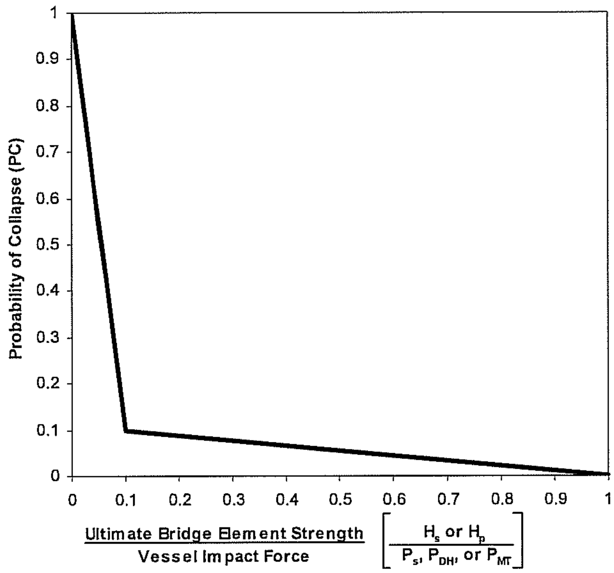


Figure 1-3. Probability of Collapse Curve (Adapted From AASHTO, 1991)

1.1.4 Improving the Probability of Collapse Term

While the basics of the *PC* equations seem reasonable, that is, if the force with which a bridge element is struck increases, the probability of collapse increases, or if the strength of a bridge element being struck increases, the probability of collapse term decreases, looking deeper into the development and background of the equations raises some questions. The AASHTO equations for the *PC* term above are based on historical *ship to ship* collision data collected by Fujii in Japan (AASHTO, 1991). How well the damage from ship to ship collisions correlates to ship to bridge or barge to bridge collision damage is questionable.

In addition, current AASHTO guidelines only require the calculation of the ultimate strength of an individual bridge element (defined as either the pier or span). In reality, consideration should be given to system-wide bridge response and strength rather than individual pier or span strengths. Furthermore, AASHTO provides little guidance in the calculation of bridge element, or system ultimate lateral strength. These factors raise further questions about the validity of the probability of collapse term in the recommended AASHTO design procedure for vessel impact.

1.1.5 Summary of the Problem

Vessel collision is a complex problem involving many factors, including the physical characteristics of the waterway, the type and number of vessels traversing the waterway, and the geometric properties of the bridge under consideration. Environmental, human and mechanical factors can all lead to serious accidents. Characterizing bridge response to vessel collision is an equally complex problem and requires the understanding of both local and system-wide behavior of a bridge pier, nonlinear material behavior and dynamic response of structures. While current design codes attempt to capture all of the variables involved in vessel collision design of bridges, there exists an opportunity to make improvements to the AASHTO design specification. Specifically, the probability of collapse term in the AASHTO Method II annual frequency of collapse equation deserves critical examination. With a better understanding of the ultimate lateral strength of bridge elements and systems, and the loads imparted to a bridge during collision, a more accurate equation for the probability of collapse can be developed that better reflects the actual phenomena of barge to bridge, or ship to bridge impact.

1.2 OBJECTIVES

The primary objective of this thesis is to outline a method for accurately calculating and characterizing the ultimate strength and response of bridge elements or systems subjected to vessel collision forces. AASHTO currently offers no guidance on how to calculate the ultimate strength of a bridge element or system.

In achieving the main objective of this report, emphasis will be placed on improving the probability of collapse (*PC*) term in the annual frequency of collapse calculation. The calculation of this term is currently based on outdated ship to ship collision tests that perhaps do not correlate well to the problem of ship or barge collision with bridges. Additional work underway at the University of Texas at Austin, as part of TxDOT Project 0-4650, is seeking to better understand the loads imparted from a ship or barge to a bridge during vessel collision. This research, along with the methods presented in this report for calculating the ultimate lateral strength of a bridge can be used to improve the *PC* term.

1.3 SCOPE

The bridges being investigated for this research are all from inland waterways in the state of Texas and are subject primarily to tug and barge traffic. Two types of bridge piers will be investigated, those with and those without shear walls. Bridge modeling and analysis guidelines will be specifically tailored for use in SAP 2000, but they should be applicable to other structural analysis software packages with similar features. The analysis results will focus on one representative bridge pier of each type and will compare the results from individual element response and system-wide response.

1.4 APPROACH

The objectives of this research will be accomplished using computational analysis methods. Computer modeling and analysis guidelines will be presented for two primary bridge pier configurations, those with and those without shear walls. Nonlinear material behavior will be captured through the use of plastic hinges. Further guidance will be given if consideration of system-wide response and redistribution of forces throughout a bridge system, including the effect of the superstructure (deck and girders) and adjacent piers is desired. The outlined procedure will allow a user to calculate a load versus displacement curve and ultimate strength in a straightforward manner using a typical structural analysis software package such as SAP 2000. The simplified modeling and analysis procedures developed will be verified using more detailed, nonlinear finite element analyses

1.5 ORGANIZATION OF THESIS

A brief summary of previous work and additional background information is provided in Chapter 2. This summary includes work leading up to and influencing the development of the AASHTO Guide Specification and Commentary for Vessel Collision Design in 1991, as well as more recent work that has occurred since the guide specification was completed. Chapter 3 reviews in greater detail the design procedures outlined in the AASHTO Guide Specification for bridges subject to vessel collision, with a heavy emphasis on Method II as it is the AASHTO recommended procedure. In Chapter 4, the modeling procedures used to compute bridge ultimate lateral strengths will be outlined. The modeling of two representative bridges from Texas, one with piers containing shear walls, the other with piers comprised of just beams and columns, will be presented as examples. SAP 2000 (SAP 2000, 2002) will be used to model these bridges. Chapter 5 will present the analysis cases and the ultimate strength analysis results for the two bridges constructed in Chapter 4 and will draw conclusions on the validity of the modeling guidelines. In addition, the affect of considering system-wide response will be examined. Chapter 6 will summarize the work contained in this report and explain how the modeling guidelines from Chapter 4 and the results from Chapter 5 could be used to improve the current AASHTO design procedures for bridges subject to vessel impact. Lastly, future research areas to continue to improve vessel collision design in the United States will be suggested.

CHAPTER 2

Historical Background on Vessel Collision Design of Bridges

2.1 Introduction

This chapter reviews the history and development of vessel collision design in the United States. Important events and research that led to the introduction of the *AASHTO Guide Specification and Commentary for Vessel Collision Design of Highway Bridges* in 1991 are presented. A review of research conducted since the development of the AASHTO Guide Specification is also included. An assessment on the direction that research in the area of vessel collision design is going and what areas need further examination is provided. In addition, work currently underway at the University of Texas at Austin is reviewed, along with a discussion of how this work (of which this document is part of) fits into the current spectrum of vessel collision design research, and how this work can be used to further improve vessel collision design in Texas and the rest of the United States.

2.2 Sunshine Skyway Bridge Accident

On May 9th, 1980 the freighter Summit Venture, under poor weather conditions, collided with one of the piers of the Sunshine Skyway Bridge crossing Tampa Bay in Florida. The struck pier was destroyed, and a 1300-foot section of the bridge superstructure collapsed into the water. Thirty-five people lost their lives in vehicles that drove off the bridge and into the bay. The extensive damage caused by this event can be seen in Figures 2-1 and 2-2.

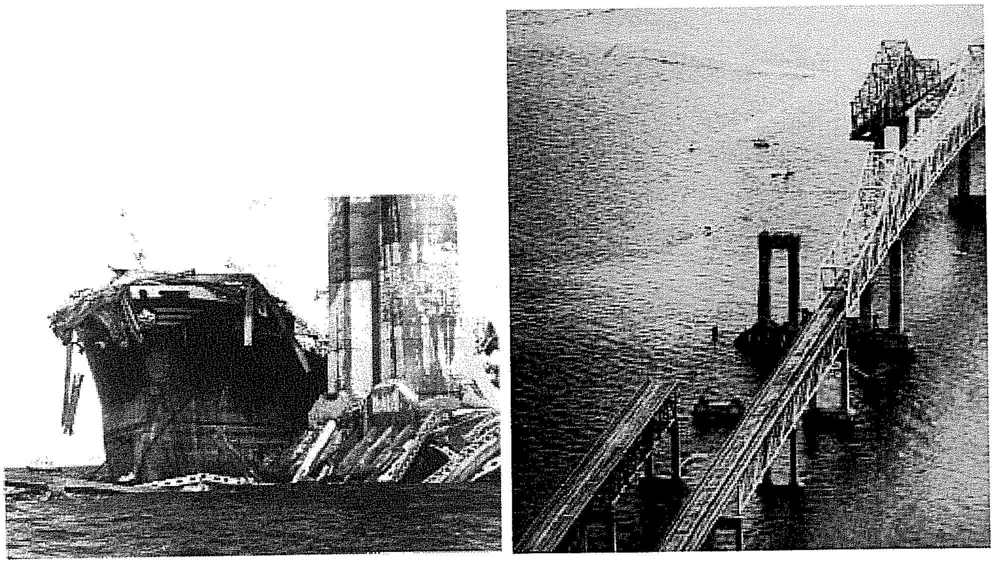


Figure 2-1. Sunshine Skyway Damage

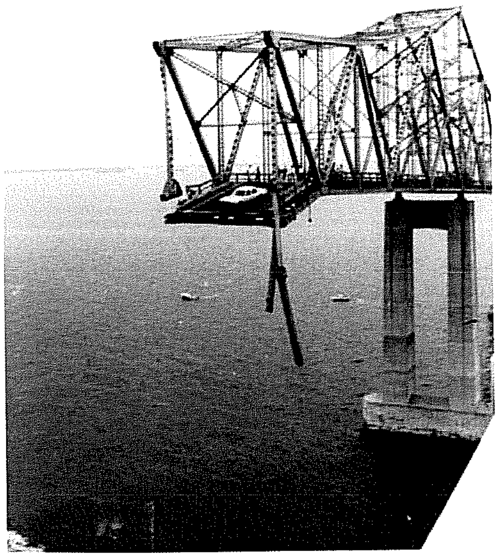


Figure 2-2. Sunshine Skyway Damage

The severe nature of the Sunshine Skyway accident and the large loss of life served to bring significant attention to the problem of vessel collision in the

United States and around the world. It is recognized as a major turning point in the development of vessel collision design criteria for bridges in the United States'(AASHTO, 1991).

2.3 DEVELOPMENT OF THE AASHTO GUIDE SPECIFICATION

2.3.1 Introduction

Historically, vessel collision forces have been ignored in the design of bridges (AASHTO, 1991). For many years it was believed that vessel collision with bridges was a highly unlikely event and it was not possible or economical to protect bridges from serious collision (AASHTO, 1991). However, as accident data grew over the years, it became clear that vessel collision loading needed to be considered in bridge design. Between 1965 and 1989 there occurred, on average, one catastrophic vessel-bridge collision accident per year (AASHTO, 1991). Through the 1980s, attention on vessel collision design grew and significant work was done to develop some basic criteria for vessel collision design. This section seeks to highlight the research that led to the development of the *AASHTO Guide Specification and Commentary for Vessel Collision Design of Highway Bridges* (AASHTO, 1991) which is still the basis for vessel collision design in the United States today.

2.3.2 1983: National Research Council Marine Board

In 1983, the Marine Board of the National Research Council in Washington D.C. appointed a committee to investigate the issue of vessel collision with bridges in the United States. The group was specifically charged with looking into the risk posed by vessel collision and analyzing the consequences of vessel impact with bridges (AASHTO, 1991). Some of the important conclusions reached by the group include the following:

- No one agency is responsible for the protection of bridges subject to vessel collision (AASHTO, 1991).
- Greater coordination between agencies or groups with a vested interest in protecting bridges from vessel collision is needed (Modjeski and Masters, 1984).
- Criteria and standards for the design, protection, and placement of bridges over navigable waterways have not been developed in the United States (Modjeski and Masters, 1984; AASHTO, 1991).
- There exists a large amount of research data in the area of risk assessment, calculation of vessel collision forces, and the design of collision-resistant structures that has yet to be applied in the United States (Modjeski and Masters, 1984).
- Criteria and standards for vessel collision design in the United States needs to be developed by AASHTO (Modjeski and Masters, 1984).

2.3.3 1983: IABSE Colloquium on Ship Collisions with Bridges and Offshore Structures

In 1983, consulting engineers and researchers from around the world gathered in Copenhagen, Denmark to present results from a wide range of vessel collision studies (AASHTO, 1991). Some of the areas covered include historical accident studies, risk assessment studies, determination of collision forces, and vessel behavior during collision, design of pier protection systems and design of motorist warning systems (IABSE, 1983). The work published as part of this colloquium served as an important source of information during the development of the AASHTO Guide Specification (AASHTO, 1991).

2.3.4 1984: Modjeski and Masters Vessel Collision Guidelines

In November of 1984, the consulting engineering firm of Modjeski and Masters completed a document titled, *Criteria for the Design of Bridge Piers with Respect to Vessel Collision in Louisiana Waterways* for the Louisiana Department of Transportation and Development, and the Federal Highway Administration (FHWA). The Louisiana DOT and the FHWA were motivated to sponsor the work based on recognition of the increased occurrence and severity of vessel-bridge collision accidents (Modjeski and Masters, 1984). The document and recommendations contained within were prepared specifically for bridges crossing navigable waterways in Louisiana, but the basic principles and methods developed are applicable for any waterway (Modjeski and Masters, 1984).

The Modejeski and Masters report illustrated the serious nature of the problem posed by vessel collision with bridges and notes the lack of consideration the issue had been given up to that point, especially in the United States. It emphasized the need for the development of a consistent approach to vessel collision design and greater oversight from appropriate governing bodies, such as AASHTO and the United States Department of Transportation (Modjeski and Masters, 1984). Furthermore, they suggested increased research to both better understand the problem of vessel-bridge collision and improve and speed up the development of technology and knowledge to mitigate the problem.

Modjeski and Masters also presented specific methods for bridge design for vessel collision. The report provided guidance for collection of the necessary waterway and vessel traffic data information, determination of the risk of vessel collision and calculation of collision forces. Finally, a design procedure for both deep and shallow waterways was outlined using those inputs. Many of the basics of the Modjeski and Masters approach to vessel collision design were eventually incorporated into the AASHTO Guide Specification.

2.3.5 1988: FHWA Establishment of a Design Specification

In 1988, eleven states helped to fund a Federal Highway Administration (FHWA) research project to develop a design specification to address vessel collision design (AASHTO, 1991). This work led to the development, in 1991, of the *AASHTO Guide Specification and Commentary for Vessel Collision Design of Highway Bridges*. Method II presented in the Guide Specification was later adopted by the *AASTHTO LRFD Bridge Design Specification* in Section 3.14. Both the Guide Specification and Bridge Design Specification have not seen significant changes related to vessel collision design since their introductions.

2.4 CURRENT RESEARCH

2.4.1 Introduction

Research conducted since the introduction of the AASHTO Guide Specification in 1991 has focused on two areas. The first and largest area of ongoing work is in understanding and more accurately characterizing the mechanics of vessel-bridge collision. The second primary area of research is in understanding, implementing and better utilizing the AASHTO Method II design procedure.

2.4.2 Understanding Vessel-Bridge Collision Mechanics

Understanding the mechanics of vessel collision design presents a unique challenge in that conducting actual tests of vessel collisions with existing bridges is not easily accomplished. The current equations in the AASHTO Design Specification related to the mechanics of vessel-bridge collision are based primarily on historic accident data and limited physical testing. In many cases, the AASHTO equations are based on related areas of study. For example, the calculation of the probability of collapse term (*PC*) in AASHTO is based on data from ship to ship collisions. Another example is the determination of impact loads for barges. Current AASHTO equations for barge impact loads are based on laboratory tests on reduced-scale barges conducted by Meir-Dornberg in Germany in the 1980s (AASHTO, 1991).

Recent work, through the use of finite element analyses and expanded physical testing, has sought to better understand the behavior of barges, ships and bridges under the condition of vessel impact. Researchers have focused on improving the ability to calculate the damage to both vessel and bridge from a collision as well as accurately determining the load imparted to a bridge from a colliding vessel. An effort has been made to better understand the influence of both sub- and superstructure elements by considering soil-structure interaction during vessel collision and increased bridge strength from the redistribution of loads through the deck to adjacent piers.

Researchers at the University of Florida and the Florida Department of Transportation have been leaders in vessel collision research. Dr. Gary Consolazio and Dr. Ronald A. Cook have published results from both finite element analyses of barge impacts with bridges (Consolazio, 2003) as well as the first results from actual barge to bridge collision tests (Consolazio, 2005). Both studies have sought to capture barge bow damage and barge impact loads due to collision with a bridge pier, an inherently dynamic problem, and compare those results to the equivalent static load equation suggested by the AASHTO Design Specification (Consolazio, 2003, 2005). Of special interest are the full-scale experiments completed on the St. George Island Causeway Bridge. The bridge was replaced in 2004, allowing for the opportunity to safely conduct barge collision tests on the old bridge. Test results showed good correlation for barge bow damage equations used in AASHTO, but found that the equations for calculating an equivalent static load were overly conservative. The study found that the load imparted to the bridge by the barge was limited by the plastic capacity of the barge bow.

Other barge impact tests have been carried out by the United States Army Corps of Engineers (Patev, 2003). The Army Corps of Engineers work has focused on understanding the mechanics of barge collision with navigation structures such as lock walls. Two types of full-scale impact tests on barges have been conducted. The first examined barge collision with different types of lock walls and rail systems, and considered various barge speeds and impact angles. The second set of tests involved crushing of the bow of a jumbo hopper barge using a Statnamic load device (typically used for foundation testing) to impart a lateral load (Patev, 2003).

2.4.3 Improving Implementation of the AASHTO Method II Design Procedure

As of 1996, five years after the release of the AASHTO Guide Specification, no inland waterway bridges had been designed using the recommended Method II procedure due to the large amounts of data required to complete that analysis (Whitney, 1996). For the most part, designers used the simple Method I design procedure. Research work in Kentucky and Florida has focused on improving the collection and processing of the necessary waterway and vessel traffic data needed to apply Method II of the AASHTO Guide Specification. M.W. Whitney, I.E. Harik, J J Griffin, and D.L. Allen, a team of researchers and engineers from Kentucky and Tennessee, conducted a study of vessel traffic on inland waterways in Kentucky and proposed a method to organize barge and flotilla data for use in the AASHTO Method II design procedure (Whitney, 1996). In 2001, Chunhua Liu and Ton-Lo Wang, from Florida International University, proposed a strategy for collection and analysis of vessel traffic data in Florida so the AASHTO recommended Method II design procedure could be utilized and implemented throughout the state (Liu, 2001).

2.4.4 Current Work at the University of Texas at Austin

Research work at the University of Texas at Austin, funded by the Texas Department of Transportation (TxDOT Project 0-4650), is seeking to integrate research in several areas in order to improve vessel collision design in Texas and rest of the United States. This report is part of that project. The work being done will be used within the current framework of the AASHTO Method II procedure and can be divided into four main areas. An effort has been made to develop a comprehensive database of waterway, vessel traffic and bridge information for the state of Texas. This information is critical for an AASHTO Method II analysis. In addition, a user-friendly, windows-based analysis program has been developed to guide an engineer through the Method II design calculations. With access to the necessary data and a program to run the required calculations, the Texas Department of Transportation will be able to easily analyze and assess the threat of vessel impact for both existing bridges and new bridge designs.

Additional work is focused on accurately characterizing the loads imparted to a bridge during vessel impact. The focus for the impact load study has been on the loads imparted to a bridge pier by a typical barge. Computer simulations have been run to capture the full dynamic effect of a vessel striking a bridge and the loads determined from these analyses will be compared against the current AASHTO provisions for calculating impact forces.

The last area of research, which this document covers, is focused on the calculation of ultimate strength for bridge elements and bridge systems that are subject to vessel impact. The primary goals of this research are to provide guidelines for modeling the ultimate lateral strength of a bridge subject to vessel impact in typical structural analysis programs and to investigate the effect of the surrounding bridge system on the strength of an individual element that has been struck by a vessel.

By investigating and calculating impact loads and ultimate lateral strengths for a bridge, a critical examination of the Probability of Collapse term can be made. The calculation of the *PC* term was identified in Chapter 1 as a potential limitations in the Annual Frequency of Collapse equation used in the Method II procedure. This research project will propose an alternate or adjusted method for calculating the Probability of Collapse, which can then be integrated into the vessel impact analysis program.

2.5 SUMMARY

Vessel collision design is a relatively new and still evolving field. It was not until 1991 that a wide ranging design code was introduced for use in the United States. This chapter has introduced events and research that led to the development of the AASHTO Guide Specification for Vessel Collision Design, which provides a probability and risk-analysis based approach to vessel impact design. Additional works that have been completed since the introduction of the Guide Specification were also reviewed. This research has focused on improving vessel impact design of bridges by staying within the framework of the AASHTO Guide Specification. Research has focused on two primary areas, understanding and characterizing vessel impact mechanics and improving implementation of the AASHTO Method II design procedure. Work currently underway at the University of Texas at Austin is seeking to integrate research in both of these areas to improve vessel collision design of bridges.

CHAPTER 3

AASHTO Vessel Collision Design

3.1 Bridge Design in the United States

The American Association of State Highway and Transportation Officials (AASHTO) is the leading authority on bridge design in the United States. AASHTO is made up of state department of transportation officials for all fifty states. They are responsible for producing and maintaining a wide range of documents related to bridge design. Primary among these documents is the *AASHTO LRFD Bridge Design Specification* (AASHTO, 2003). AASHTO provides additional documents which offer more detailed information regarding specific design issues. An example is the *AASHTO Guide Specification and Commentary for Vessel Collision Design of Highway Bridges* (AASHTO, 1991). This document will be referred to as the 'AASHTO Bridge Design Specification' throughout the remainder of this thesis.

3.2 Vessel Collision Design

Section 3.14 of the AASHTO Bridge Design Specification covers vessel collision and is based of the *AASHTO Guide Specification and Commentary for Vessel Collision Design of Highway Bridges* (AASHTO, 1991). This document will be referred to as the 'AASHTO Guide Specification' from this point forward. The AASHTO Guide Specification provides three methods for the evaluation of bridges for vessel collision design. A comprehensive flow chart in Section 1.5 of the AASHTO Guide Specification outlines the analysis steps needed for each of the three evaluation procedures.

A brief review of the three design methods in the AASHTO Guide Specification was presented in Chapter 1. A more detailed review of each is included in this chapter with a focus on Method II as it is the AASHTO recommended design procedure.

3.3 AASHTO Vessel Collision Design Method I

Method I uses a semi-deterministic procedure to select the design vessel for a given waterway. The Method I procedure for design vessel selection is based on bridge design criteria in the *Common Nordic Regulations* used in Scandinavian countries with slight modifications (AASHTO, 1991). With this approach, the design vessel is selected such that a maximum number or percentage of vessels that are larger than the design vessel is not exceeded (AASHTO, 1991). AASHTO states the no more than 50 vessels per year, or 5% of the vessel traffic, can be larger than the design vessel (AASHTO, 1991).

The selected design vessel is used to calculate a design impact force which can be expressed as an equivalent static load at the mean water level. Equations for calculation of design loads based on the design vessel are contained in Chapter 3 of the AASHTO Guide Specification. The procedure and equations used for this calculation are the same for all three design methods. Calculation of impact forces will be discussed in more detail in Section 3.4 of this thesis, which covers design Method II. Once the design loads have been determined, a linear elastic structural analysis can be completed to check the adequacy of the bridge members.

Method I is intended to be a simple, conservative approach to vessel collision design. Limited vessel traffic and waterway data are required for Method I, and the analysis equations and calculations are less complicated than in Method II. Method I, however, is only applicable in limited situations. The

Method I design procedure is not appropriate for bridges classified as 'critical', or for bridges crossing waterways that see a wide distribution of vessel types and sizes, or for waterways that see significant numbers of large ships (AASHTO, 1991). Method I is most appropriate for shallow, inland waterways that are subjected primarily to barge traffic (AASHTO, 1991).

3.4 AASHTO VESSEL COLLISION DESIGN METHOD II

Method II is the recommended design procedure presented in the AASHTO Guide Specification and is the only method presented in the AASHTO LRFD Bridge Design Specification. Method II is a detailed, probability-based risk analysis procedure. It requires a significant amount of data on the waterway characteristics, the vessels traversing the waterway and the geometry of the bridge being analyzed. The essential data needed for application of Method II are vessel description, speed and loading conditions, waterway geometry, navigable channel geometry, water depths, environmental conditions and bridge geometry (AASHTO, 1991). The specific data requirements can be found in Sections 3 & 4 of the AASHTO Guide Specification and in Sections 3.14.5-3.14.11 of the AASHTO LRFD Bridge Design Specification. The required data are used to compute an annual frequency of collapse for a bridge element. A minimum acceptable annual frequency of collapse for bridges is given based on bridge classification (i.e., regular or critical).

3.4.1 Importance Classification and Acceptance Criteria

Under AASHTO Method II, bridges must be assigned an importance classification as a: 1) Regular or 2) Critical bridge. Bridges are classified based on society/survival demand and security/defense requirements (AASHTO, 1991). Bridges that provide important links for police and fire departments, emergency personnel, hospitals and schools are classified as critical, as well as bridges in areas where few alternate waterway crossings are available.

Heavily traveled bridges can be also be classified as critical, both because of large disruption costs if the bridge is struck by a vessel and because of the possibility of greater loss of motorist life in the event of an accident. The designation of a critical bridge is somewhat subjective, but the AASHTO Guide Specification provides some guidance in the classification process. Bridges not given a critical classification are marked as regular bridges. For critical bridges, the acceptable annual frequency of collapse is less than or equal to 0.0001, or once every ten-thousand years. For regular bridges, the acceptable annual frequency of collapse is less than or equal to 0.001, or once every thousand years.

3.4.2 Annual Frequency of Collapse Calculation

The result of using the AASHTO Method II design procedure is the calculation of an annual frequency of collapse for a given bridge. The equation appears quite simple, but the calculation of each individual term in the equation can be quite complex and may require several levels of calculations. The equation for determining annual frequency of collapse (*AF*) was shown previously in Chapter 1 and is presented below in Equation 3-1. Also shown are the equations for calculating the individual terms in the *AF* calculation as well as some additional information regarding each term and the data required to complete the calculations.

$$AF = (N)(PA)(PG)(PC) \tag{3-1}$$

Where:

$AF =$ annual frequency of bridge element collapse due to vessel collision

$N =$ annual number of vessels classified by type, size, and loading condition which can strike the bridge element

$PA =$ probability of vessel aberrancy

$PG =$ geometric probability of a collision between an aberrant vessel and a bridge pier or span

$PC =$ probability of a bridge collapse due to a collision with an aberrant vessel

3.4.3 Probability of Aberrancy Calculation

There are three primary causes of vessel aberrancy — pilot error, adverse environmental conditions, or mechanical failure (AASHTO, 1991). The probability of aberrancy (PA) calculation attempts to capture the likelihood that if one of these events occur, a vessel will become out of control. AASHTO recommends using a statistical analysis based on historical data on vessel collisions, rammings, and groundings along a waterway to calculate the probability of aberrancy. Given that this information can be difficult to compile, or that there may not be enough information available, AASHTO also provides an equation requiring information on waterway characteristics, bridge location and geometry, and vessel traffic data to compute PA (Equation 3-2).

$$PA = BR(R_B)(R_C)(R_{XC})(R_D) \tag{3-2}$$

where

$PA =$ probability of aberrancy

$BR =$ aberrancy base rate

$R_B =$ correction for bridge location

$R_C =$ correction factor for current acting parallel to vessel transit path

$R_{XC} =$ correction factor for crosscurrents acting perpendicular to vessel transit path

$R_D =$ correction factor for vessel traffic density

Based on historical accident data on U.S. waterways, AASHTO suggests the following values for aberrancy base rates:

for ships: $BR = 0.6x10^{-4}$

for barges: $BR = 1.2x10^{-4}$

AASHTO provides equations for the calculation of the other variables used in the calculation of the probability of aberrancy. These equations can be found in Section 4.8.3.2 of the AASHTO Guide Specification (AASHTO, 1991), or Section 3.14.5.2 of the AASHTO Bridge Design Specification (AASHTO, 2003).

3.4.4 Geometric Probability of Collision Calculation

The geometric probability is the probability that a vessel will collide with a bridge given that the vessel has already lost control. The geometric probability (*PG*) is computed based on a normal distribution of vessel accidents about the centerline of the vessel transit path (AASHTO, 1991). The graphic in Figure 3-1 illustrates the *PG* calculation. The AASHTO guide specification recommends using a standard deviation value of $\sigma = LOA$. *LOA* is length overall of the design vessel. For ships, length overall is simply the length of the ship. For barges, length overall is the length of the entire barge tow including the tow boat. The use of 1 *LOA* as the standard deviation in the geometric probability calculation is based primarily on ship collision data (AASHTO, 1991). Despite the fact that barge collisions are more common in the United States, these accidents have not been as well documented as ship collisions, and less accident data is available.

Therefore, AASHTO recommends using the same value for the standard deviation for both barge and ship calculations (AASHTO, 1991).

Because of the assumed normal distribution of accidents about the water navigation channel centerline, by definition, 99.7 percent of accidents will occur within a distance of 3 *LOA* from the centerline. AASHTO states that bridge elements located outside of 3 *LOA* from the centerline need not be considered in the analysis. As Figure 3-1 shows, the *PG* is the area under the normal distribution in the ship/barge impact zone. The impact zone is defined by the pier location and width, plus ½ of the ship/barge width on each side of the pier.

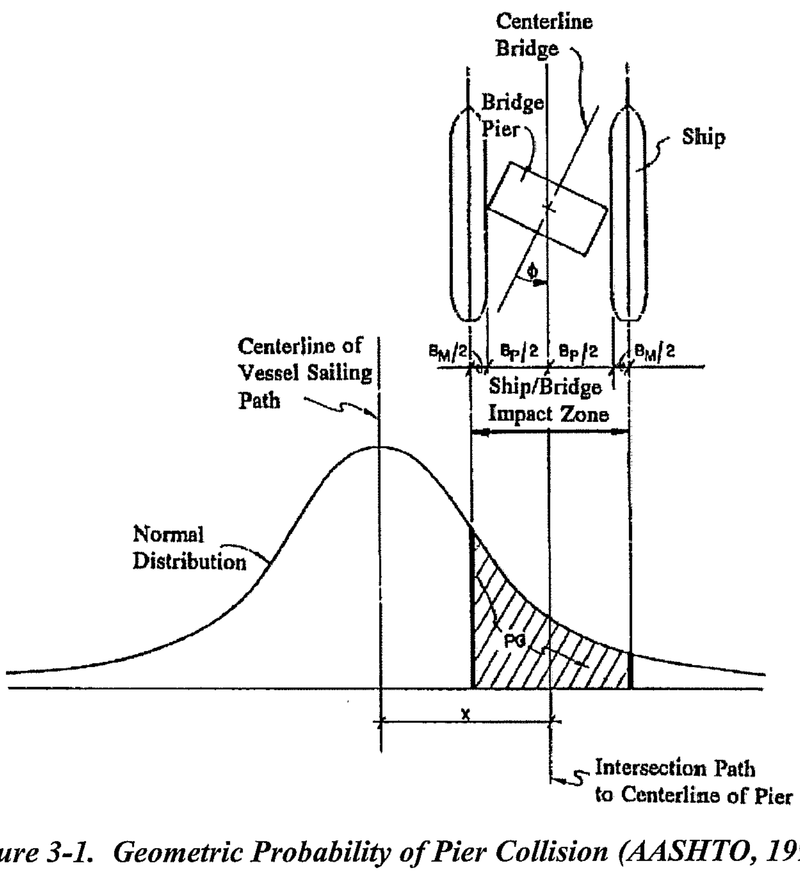


Figure 3-1. Geometric Probability of Pier Collision (AASHTO, 1991)

3.4.5 Probability of Collapse Calculation

The probability of collapse calculation is covered in Section 4.8.3.4 of the AASHTO Guide Specification and in Section 3.14.5.4 in the AASHTO Bridge Design Specification. The probability of collapse is a function of just two variables — the ultimate strength of the bridge element (pier or span) being struck and the load imparted by the vessel. Bridge element ultimate strength and impact force are used in simple equations (shown below as Equations 3-3, 3-4, 3-5) to calculate the *PC* factor. These equations were developed based on historical data from ship to ship collisions collected by Fujii in Japan (AASHTO, 1991). Fujii used data on the damage caused during ship to ship collision events to develop a damage relationship based on the angle at which the two ships collided and the gross tonnage of the two colliding vessels. This damage relationship was used by Conwiconsult to develop the *PC* term for ship to bridge and barge to bridge collisions that was later adopted by AASHTO (AASHTO, 1991).

The calculation of impact loads is covered in Chapter 3 of the AASHTO Guide Specification. AASHTO currently recommends the calculation of an equivalent static load that is applied as a point load, or as a distributed load over the bow length of a barge or ship, at the mean high water level. The actual impact load can be calculated using empirical equations based on vessel velocity and dead weight tonnage (*DWT*). There are separate equations for calculating impact forces for ships or barges. There are also different equations for impact on a bridge pier or against a bridge span. The AASHTO empirical equation for ship impact forces is based on tests performed in the 1970s in Germany by Woisin. The AASHTO equations for barge impact forces are based on work done by Meir-Dornberg in Germany in the 1980s (AASHTO, 1991). It is significant to note that the AASHTO Design Specification does not provide guidance in the calculation of bridge element ultimate strengths.

For $0.0 \le H/P < 0.1$, PC shall be computed as:

$$PC = 0.1 + 9\left[0.1 - \frac{H}{P}\right] \tag{3-3}$$

For $0.1 \le H/P < 1.0$, PC shall be computed as:

$$PC = \frac{\left[1 - \frac{H}{P}\right]}{9} \tag{3-4}$$

For $H/P > 1.0$:

$$PC = 0 \tag{3-5}$$

where

$H =$ ultimate bridge element strength (kips)

$P =$ vessel impact force (kips)

These equations are shown as a graph in Figure 3-2.

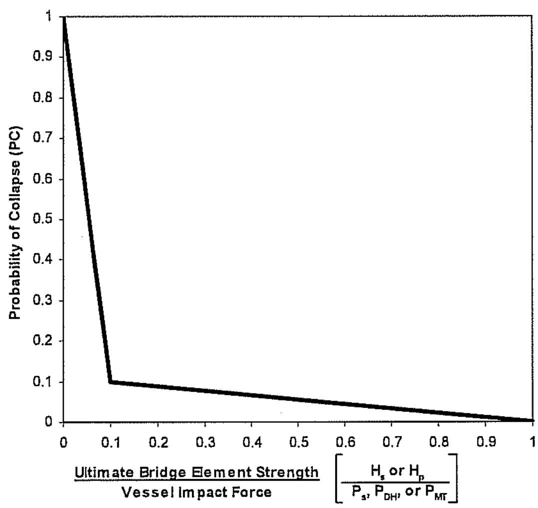


Figure 3-2. Probability of Collapse Curve (Adapted From AASHTO, 1991)

3.5 AASHTO VESSEL COLLISION METHOD III

Method III is a cost-effectiveness analysis procedure that uses standard engineering economic principles. It was developed for situations where design Method II does not accurately capture the acceptable risk levels for a bridge and results in designs that are cost-prohibitive or not technically feasible (AASHTO, 1991). A possible scenario for this case is a bridge with a large number of piers in the water that are exposed to vessel collision (AASHTO, 1991).

Method III allows a designer to make decisions based on a typical cost/benefit analysis. In this case, the 'costs' represent the present worth of the costs of making a bridge stronger or providing some additional protection for a bridge, while the 'benefits' side of the equation is represented by the present value worth of the avoidable disruption cost. The avoidable disruption costs are equal to whatever losses are expected should a bridge collapse due to vessel collision. Section 4.9.3 of The AASHTO Guide Specification provides an equation for calculation of the disruption costs should a vessel collision accident occur. The equation is as follows:

$$DC = PRC + SRC + MIC + PIC \tag{3-6}$$

Where:

$DC =$ disruption cost

$PRC =$ pier replacement cost

$SRC =$ span replacement cost

$MIC =$ motorist inconvenience cost

$PIC =$ port interruption cost

The cost/benefit analysis should be carried out over the lifetime of a bridge. A bridge design or improvement to an existing bridge is effective when benefits outweigh costs over the expected life span of the bridge.

3.6 SUMMARY

This chapter has outlined the various design methods presented in the *AASHTO Guide Specification and Commentary for Vessel Collision Design of Highway Bridges*. The remainder of this document will focus on the AASHTO recommended Method II design procedure for calculation of an Annual Frequency of Collapse (*AF*) of a bridge. Procedures for computer modeling and analysis of bridges for the calculation of ultimate lateral strengths will be presented. This procedure will be applied to several bridge pier configurations and analysis results will be presented.

CHAPTER 4
Bridge Ultimate Strength Modeling

4.1 INTRODUCTION

For application of Design Method II, AASHTO requires the calculation of the ultimate lateral strength of a bridge element being impacted by a vessel. This capacity is used in the probability of collapse term (*PC*) in the Annual Frequency of Collapse (*AFC*) calculation. Chapter 3 of this document outlines in detail the AASHTO Method II design procedure and calculation of both the *PC* and *AFC* terms.

While AASHTO requires the calculation of the ultimate lateral strength, it provides no guidance in the determination of this value. The judgment of what represents the ultimate strength of a bridge element is left to the engineer. There are several possible ways to interpret this requirement, and each interpretation could lead to significantly different values of bridge element capacity for the same structure. In addition, for certain bridge geometries, determination of the ultimate lateral strength can require a complex analysis. Furthermore, AASHTO only considers the lateral capacity of an element, which it defines as a single pier or single span, as opposed to the lateral strength of the bridge system as a whole. AASHTO does not consider the interaction between a bridge pier and deck, and the redistribution of forces from one bridge element to the next.

The goal of this chapter is to provide a modeling procedure that is simple, consistent, and conservative that can be used for a design to capture the inelastic behavior and the ultimate or limit strength of a bridge pier or bridge system subject to vessel collision. Special emphasis is placed on developing modeling techniques and procedures that can be used within commonly used structural analysis software packages and will not require the use of more complex finite element analysis programs.

4.2 SCOPE

The method for determination of ultimate lateral strength outlined in this chapter is intended to be applied to reinforced concrete bridge piers that may or may not contain shear or web walls. Figure 4-1 shows a half-elevation and section drawing of a bridge pier with a web or shear wall extending upwards from the pile cap. Notice that the wall is flush with one edge of the column. This configuration is not typical. Normally, the wall will be centered on the face of the column. Figure 4-2 shows a simple bridge pier, consisting of beams and columns without a wall.

This chapter presents modeling guidelines for use with SAP 2000 (version 8) [SAP 2000, 2002], a commonly used software package for structural analysis. Primary emphasis is placed on modeling bridge piers subject to vessel impact, but additional guidelines to capture system-wide response and the effect of redistribution of forces through the deck to adjacent piers is also considered. As they are presented, the models are not intended to be used as part of a dynamic analysis, although dynamic effects could be considered by applying a dynamic response factor to the static analysis results that will be presented in Chapter 5.

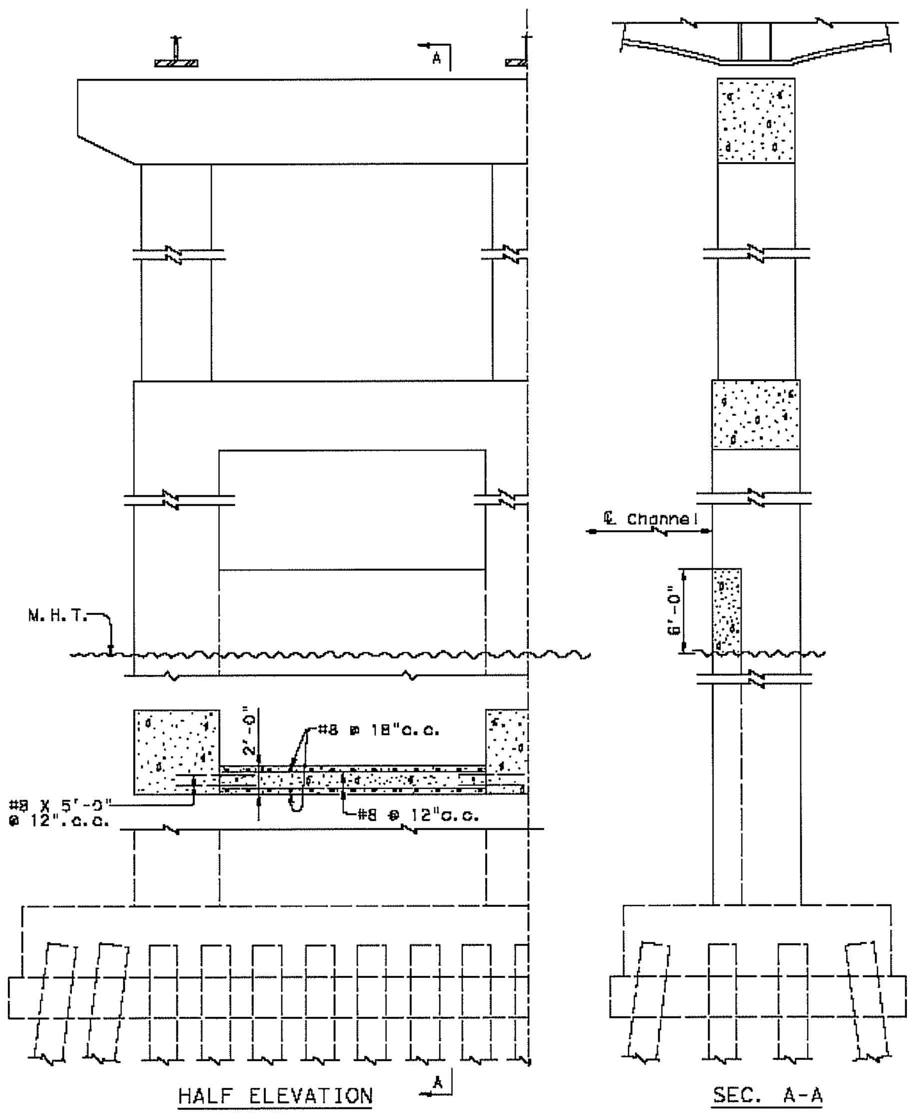


Figure 4-1. Bridge Pier with Shear Wall (TXDOT, 2001)

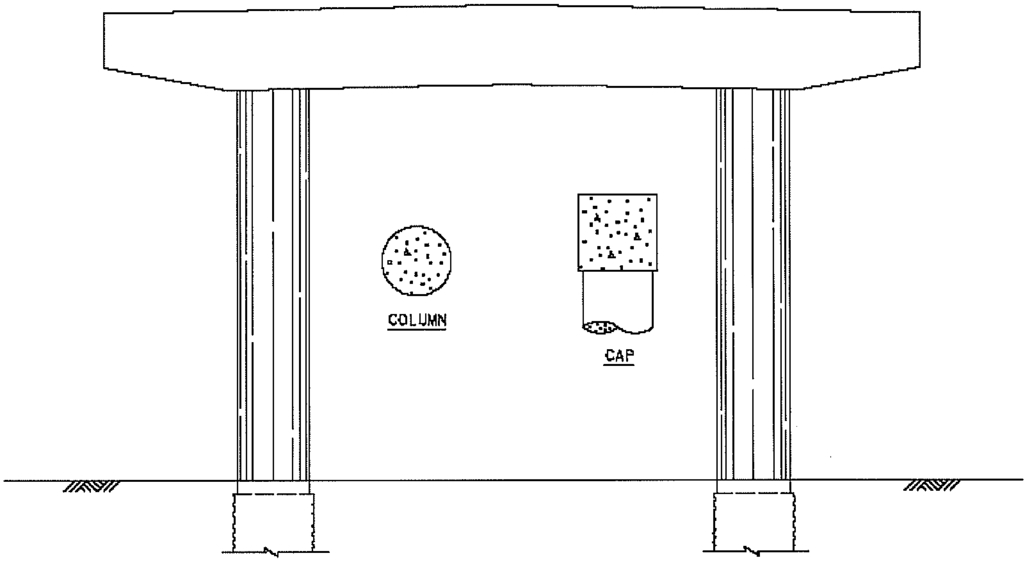


Figure 4-2. Bridge Pier without Shear Wall (TXDOT, 2001)

4.3 APPROACH

Because SAP 2000 and other typical structural analysis software packages do not have the capability to capture inelastic behavior for wall elements, an approximate method to capture response is presented. The approximate method is verified using ANSYS, a general finite element analysis software package. ANSYS has the capability to capture inelastic response of all types of elements, including shell/wall elements. Other aspects of the SAP 2000 and ANSYS models are defined in a similar manner, so the only variable in the two sets of models is how the wall is being modeled. Once the SAP 2000 approximate wall model has been verified, ultimate lateral strength analyses are conducted for two bridges.

4.4 Modeling Basics In SAP 2000

This section outlines basic background information on the creation of models within SAP 2000 to capture the inelastic behavior and ultimate strength of bridge piers subject to vessel collision. The following sub-sections are intended to provide introductory details on the major areas that need to be addressed within SAP 2000 to build an accurate bridge model. Later sections provide a step-by-step procedure, along with screen shots from SAP 2000, for the construction of specific bridge models.

4.4.1 Defining Bridge Geometry

A wide range of bridge geometries can be easily defined within SAP 2000 by establishing gridlines along the centroid of beam and column members and around the boundaries of wall areas. Walls are defined by shell elements if the wall behaves in a linear elastic manner. To capture inelastic behavior of wall elements, shells cannot be used. Instead, an approximation of the wall needs to be developed. The method proposed in this chapter utilizes a grid of truss elements to replace the wall. Both rigid and axially deformable members are used in the grid. The specifics of the approximate wall model are discussed in detail in later sections.

4.4.2 Element Types

Two basic element types are used within SAP 2000 (frame elements and shell elements), to construct bridge pier and bridge system models. Other types of elements can be defined in SAP 2000 by modifying frame elements. Rigid members can be used by assigning large stiffness modification factors to the desired elements. Truss elements can be defined by releasing moments at member ends.

4.4.3 Material Model

A simplified approach is used to model reinforced concrete. Smeared material properties, considering the concrete and reinforcing steel as a single material with similar properties in tension and compression are specified for the analyses. Taking this approach, a reasonable determination of strength and stiffness characteristics of the elements in a bridge pier can be made without having to address the difficulties of modeling reinforced concrete material properties directly. Modeling reinforced concrete requires not only accurately capturing the material behavior of steel, which is not especially difficult, and concrete, which is more difficult because it behaves differently in tension and compression, but also the interaction between the two materials. Taking the two materials as a single smeared material, while not as accurate, is considerably easier and more appropriate for design calculations. Using this approach, the key material properties a user needs to input are E, the modulus of elasticity, f_y, the yield stress, and f_u, the ultimate stress.

4.4.3.1 Defining Modulus of Elasticity

The Modulus of Elasticity is calculated in accordance with American Concrete Institute (ACI) guidelines (ACI 318-02, 2002). Section 8.5.1 of ACI 318-02 (Building Code Requirements for Structural Concrete and Commentary) recommends the following expression to define E:

$$E = 57000\sqrt{f_c^{'}} \qquad (4\text{-}1)$$

where

$E =$ Modulus of Elasticity in psi

$f'_c =$ Concrete Strength in psi

4.4.3.2 Defining f_y and f_u

Values for the yield stress and ultimate stress can be determined by a reinforced concrete section analysis. Separate values of f_y and f_u must be defined for each element (column, beam, or wall) in a bridge pier. Thus, a separate section analysis must be completed for each element. There are several readily available computer programs that will perform a reinforced concrete section analysis. An example of such a software package, and the program used for calculations contained in this report, is Response-2000, developed at the University of Toronto (Bentz, 2001). This program allows a user to input the geometric and material properties of a reinforced concrete section, including longitudinal and transverse steel reinforcement, and returns the strength and ductility characteristics of that section in the form of a moment-curvature plot. Figure 4-3 shows the basic section information produced by Response-2000 for a typical circular column section. Note that geometric properties of the section are shown along with the user specified material properties for concrete and reinforcing steel, as well as the layout of longitudinal and transverse steel.

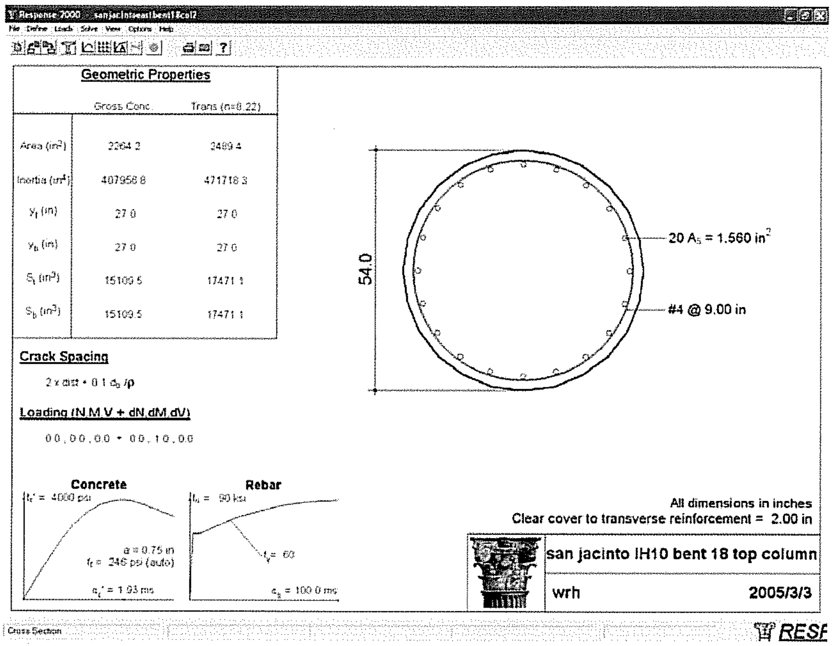


Figure 4-3. Response Section Input

Figure 4-4 shows a moment-curvature plot from the analysis of the 54-inch-diameter column section shown in Figure 4-3. The section analysis results shown are for a section with no axial load. It will be shown later that neglecting axial load will result in material property values that are slightly conservative. On Figure 4-4, values for the yield moment, M_y, and the plastic moment, M_p, have been estimated. Response-2000 returns a value for M_p, and the user will need to estimate a value for M_y, although doing so is straightforward as there is generally a clear point at which the stiffness begins to change. These values will be used to determine the yield stress and ultimate stress using the following equations:

$$f_u = \frac{M_p}{Z} \tag{4-2}$$

$$f_y = f_u \frac{M_y}{M_p} \tag{4-3}$$

where

$f_u =$	Ultimate Stress
$f_y =$	Yield Stress
$M_y =$	Yield Moment
$M_p =$	Plastic Moment
$Z =$	Plastic Section Modulus

The ultimate stress, f_u is defined based on the plastic section modulus and the ultimate or plastic moment. The yield stress is then defined based on the ratio between the yield and plastic moment. It is important to note that the definition for the yield stress value is not consistent with the typical definition which is as follows:

$$f_y = \frac{M_y}{S} \tag{4-4}$$

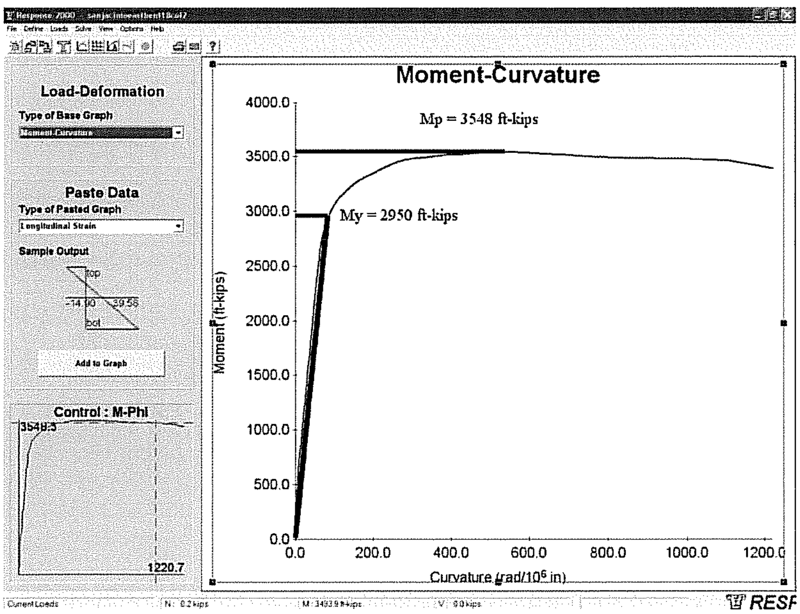


Figure 4-4. Response Moment-Curvature Analysis Results

The expressions shown are being used because they are consistent with how SAP 2000 defines the values of moment at which hinges first form and then reach their ultimate strength capacity. Table 4-1 summarizes the calculation of the smeared material properties for the 54-inch-diameter column section shown previously. The section properties and actual material properties are entered into Response-2000, a sectional analysis is performed, and the results are used to calculate the new, smeared material properties for use within SAP 2000.

Table 4-1. Smeared Material Properties for 54" Diameter Column Section

Section Properties	
Section Diameter, d (in)	54
Section Modulus, Z (in^3)	26244
Real Material Properties	
Concrete Strength, $f_c{}'$ (ksi)	4
Reinforcing Steel Strength, f_y (ksi)	60000
Response Section Analysis Results	
Yield Moment, M_y (ft-kips)	2950
Plastic Moment, M_p (ft-kips)	3548
Smeared Material Properties	
Ultimate Stess, f_u (ksi)	1.62
Yield Stress, f_y (ksi)	1.35
Modulus of Elasticity, E (ksi)	3605

It was stated earlier that performing a section analysis with zero axial load will result in slightly conservative values for the smeared material properties. The moment-axial interaction diagram shown in Figure 4-5 helps to explain why neglecting axial load on the section is conservative. This plot is for the same 54-inch-diameter column that has been discussed throughout this section. The vertical line represents the value of M_p that was used to calculate the smeared material properties. The plot shows that for axial loads between 0 k and 7750 k, the actual moment capacity for this section is actually greater than the value of M_p used. In addition, an axial load of 7750 k represents approximately 75% of the crush load, P_u. An axial load of this magnitude is not only extremely unlikely, but would also fail to meet code requirements. Realistically, bridge piers will see axial loads much less than 50% of the crush load, and in many cases the axial loads will be closer to 10-15% of the crush load.

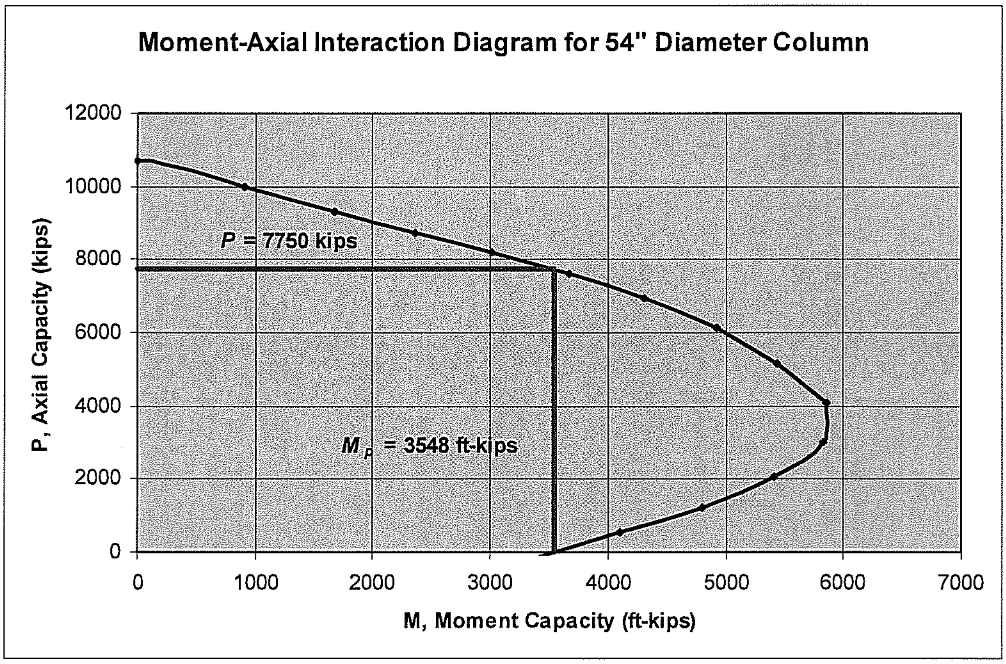


Figure 4-5. Response Moment-Axial Interaction Results

4.4.4 Section Properties

A wide range of user-defined sections can be entered in SAP 2000. Modeling of most reinforced concrete bridge piers will require the use of regular geometries, usually rectangular or circular sections. As the section is being defined, the user needs to assign a material model to that particular section. Figure 4-6 shows the SAP 2000 input for a rectangular section called

'COLUMN', which is made of a material called 'MAT1'. The windows shown in Figure 4-6 were reached through the 'Define-Frame/Cable Sections' menu in SAP 2000. Once elements have been created, the user-defined sections are then assigned to the appropriate members. The shapes required to define the bridge pier geometries are assigned using sections that exactly match the geometry of the actual bridge. SAP 2000 also has the option of applying section modification factors for a specific property. For example, a user could enter a large value for the cross-sectional area modification property, essentially making the element axially rigid. Figure 4-6 also shows that the section 'COLUMN' has been given a large axial modification factor.

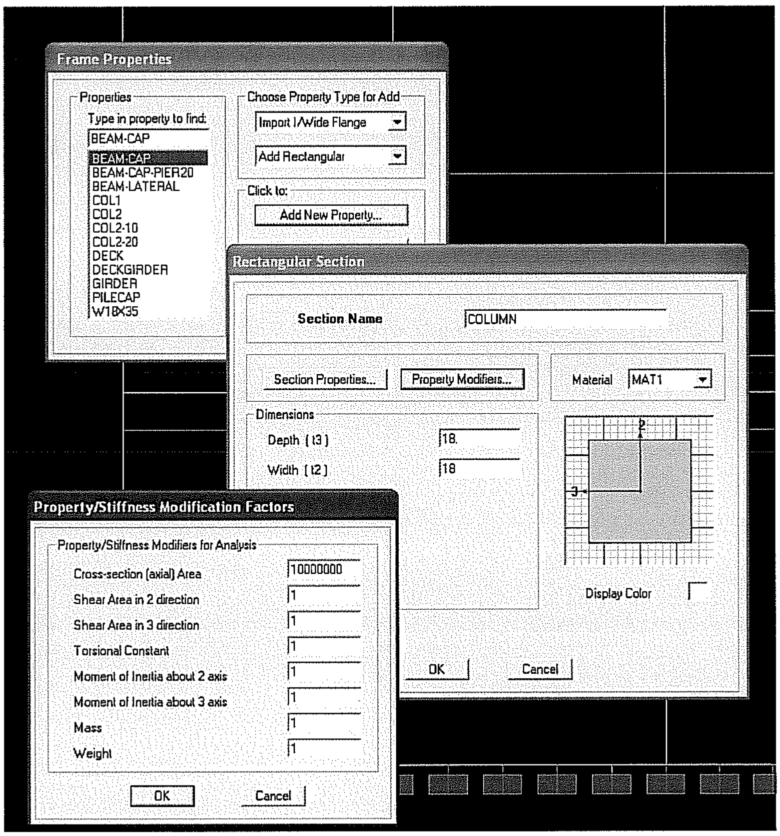


Figure 4-6. Entering Section Properties in SAP 2000

In addition, SAP 2000 has the option of using a 'general' section, which can be defined based only on the section properties such as area and the moment of inertia, without specifically defining the geometry. Figure 4-7 shows the input box for a general section, which SAP 2000 names 'FSEC1' by default. Notice that the section properties are entered directly and no geometrical parameters need to be defined. The general section option is particularly useful for defining section properties for elements to represent the bridge deck and girders. The strength and stiffness of these members can be determined and then applied to a 'general' section rather than explicitly drawing sections to represent the girders and the deck. In the case of the girders it may be necessary to model members with unique geometries such as AASHTO prestressed girder types or steel trapezoidal girders, or in the case of the deck, require the use of shell elements. In either case, the geometry for these elements may not be easily defined using the default shapes in SAP 2000, and use of general sections is favorable.

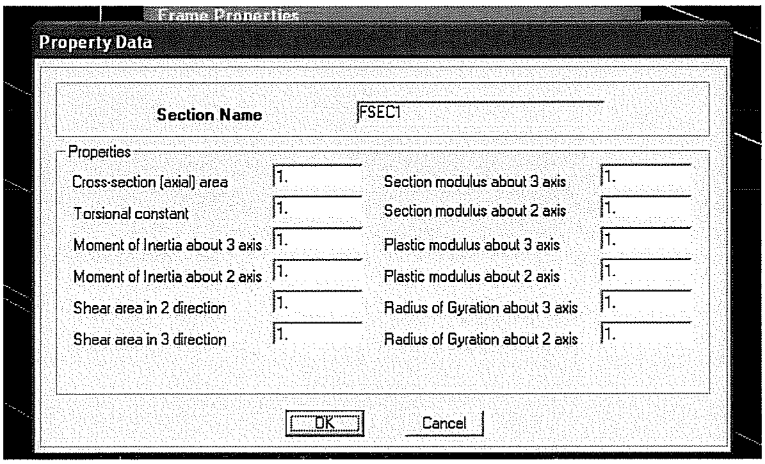


Figure 4-7. General Section Input in SAP 2000

4.4.5 Plastic Hinges

Inelastic behavior and nonlinear material properties are captured through the use of plastic hinges acting at the member ends. Hinges can be defined as axial hinges, shear hinges, moment hinges, or moment-axial interaction hinges within SAP 2000. For the bridges modeled within this report, axial hinges and moment-axial interaction hinges are be used.

Plastic hinge properties are defined in SAP 2000 based on the strength and deformation capacities of the member to which they are assigned. The yield and ultimate strength are based on the material properties from a reinforced concrete section analysis. Therefore, because of the smeared material model approach, different hinge properties must be defined for each column, beam and grid section. If there are two different column sections in a bridge pier, a different set of hinge properties is needed for each section and must be applied at the ends of members with that section.

For the bridge models in this report, hinges are defined as infinitely plastic, and system ductility will be assessed in the post-analysis phase. This approach was taken to simplify the analyses and to ensure that the bridge models have adequate ductility to form a failure mechanism. Another option would be to define the deformation or rotational capacity of the hinges, either as a multiple of the yield deformation or rotation, or by their actual deformation or rotational limits, in inches or radians. This approach is slightly more difficult than the approach described above given the variation that will be seen in rotational and deformational capacities based on the specific concrete section that is being considered. Figure 4-8 shows the typical hinge profile that is used. Note that the values assigned for the ultimate strength and the rotation when ultimate strength is reached are merely representative of a typical hinge. Exact values for these properties will be determined by the material properties and will vary for each hinge used. Also, a more detailed discussion of the actual plastic hinge inputs, with SAP 2000 screen captures, is provided in Sections 4.6.1 and 4.6.2, which outline the modeling of two specific bridges.

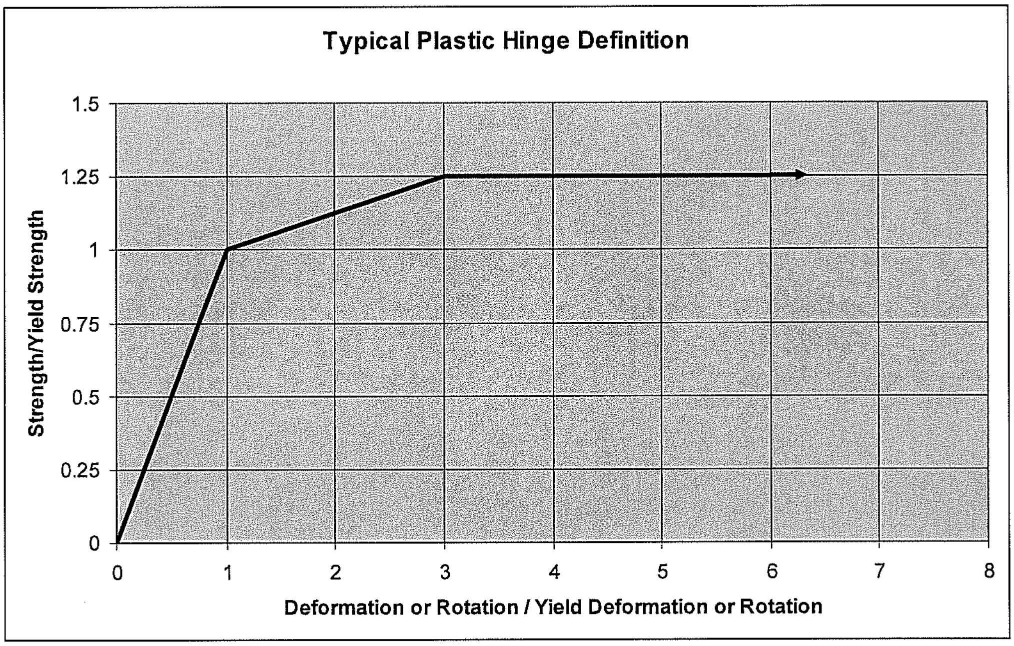


Figure 4-8. Typical Plastic Hinge Definition

To properly capture post-yield behavior in a model, plastic hinges must be assigned to the regions of the model that are subject to inelastic deformation. Therefore, the location of plastic hinges must be carefully selected. Hinges can be placed at any relative distance along the length of a member, but it is often easiest to apply hinges only at the ends of members in SAP. Using this approach it may be necessary to subdivide elements, such as columns, to assign plastic hinges at locations along the length.

4.4.6 Boundary Conditions

Boundary conditions at the base and at the top of a bridge model need to be considered carefully. The boundary conditions can have a significant effect on the stiffness and strength characteristics of a bridge system. A single base condition is considered at the base of the structure, while several boundary conditions are considered at the top of a pier.

4.4.6.1 Bridge Pier Base Boundary Condition

The base conditions for all of the bridge piers being modeled for this research are assumed to be fixed. This assumption is reasonable for the connection between the wall and columns of a bridge pier and the foundation cap beam, but it ignores the interaction between the foundation piles or piers and the surrounding soil or water. Figure 4-9 shows a graphical representation of the assumption being made. The resulting system will be less flexible than what actually exists, and there are several important implications to this statement.

First, a stiffer system will attract more loads to the elements in the bridge pier. This observation can be viewed as both conservative and unconservative, depending on how the problem is being considered. If vessel impact loads are known *a priori* and a bridge model with a fixed base is being analyzed for those *known* loads, the forces in the members will be greater than they are in reality. In comparing these loads to member capacities, a conservative approach is being taken. If instead a bridge model with a fixed base is subject to a static nonlinear analysis, in which the load is increased incrementally until failure, the stiffer structure will again attract more load, and a artificially high, or unconservative value of ultimate lateral strength will result. Second, with a stiffer system, the displacements are expected to be underestimated. For systems that are controlled by ductility, the modeling approach used for this research could also result in unconservative ultimate lateral strength results.

While the fixed base assumption may not be the most accurate representation of actual bridge base conditions, it is made both for the sake of simplicity and because accurately modeling the base condition with springs or other elements would require data on the soil conditions at a given site, which might not be easily obtained. It is important to know both the strength and ductility limits of the real structure and to understand the effect that the assumed fixed base condition has on the analysis results

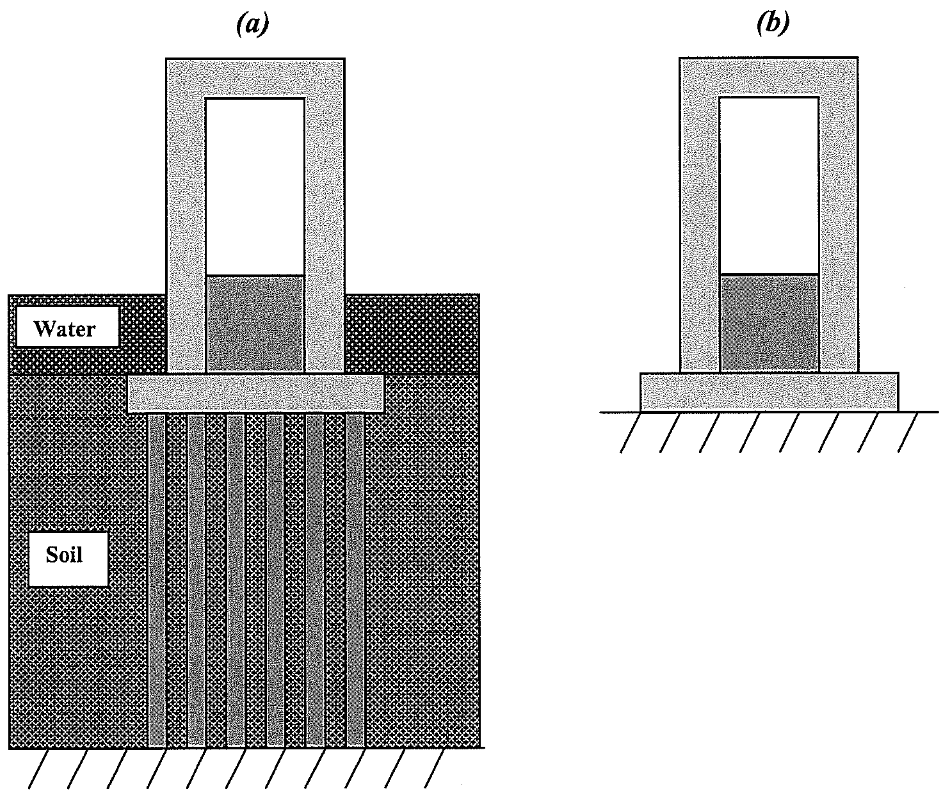


Figure 4-9. Bridge Pier Base Condition: (a) Actual Conditions; (b) Assumed Conditions

4.4.6.2 Bridge Pier Top Boundary Condition

Several boundary conditions are considered at the top of the bridge piers being modeled and analyzed. Currently, AASHTO requires the calculation of ultimate lateral strength of a stand-alone pier. Therefore, an analysis for this case will be considered. At the opposite extreme, the assumption will be made that the bridge deck and girders provide a rigid support at the top of the pier. Analyses of these two cases will provide a range of possible strengths for the bridge system under consideration. Figure 4-9 shows free and fixed top conditions for a SAP 2000 bridge pier model. To best represent an actual bridge system, a third analysis case is considered with elements at the top of the bridge pier that match the stiffness contributed by the bridge girders and deck. This case is illustrated in Figure 4-10.

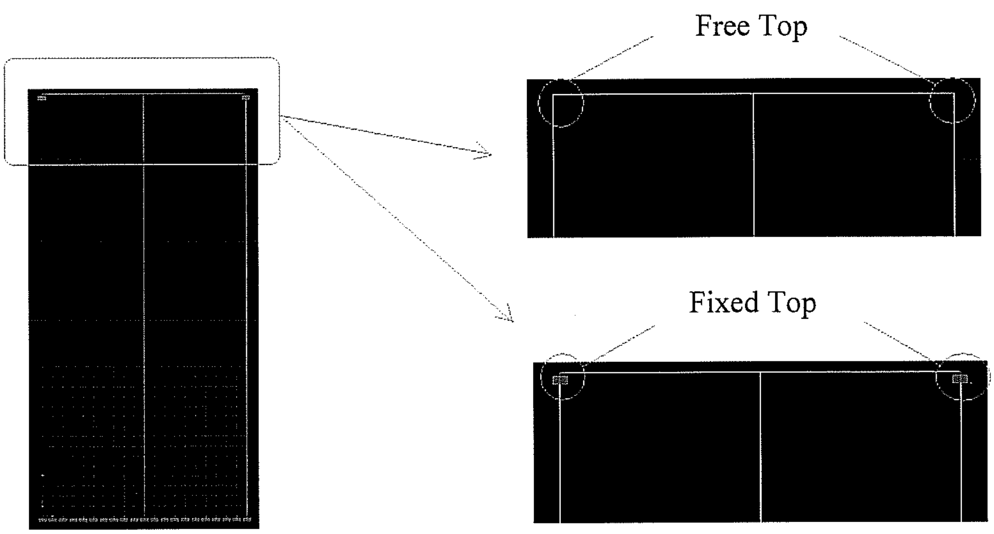


Figure 4-10. Bridge Pier Top Boundary Condition: (a) Free Top, (b) Fixed Top

Perpendicular Elements Representing

Bridge Superstructure

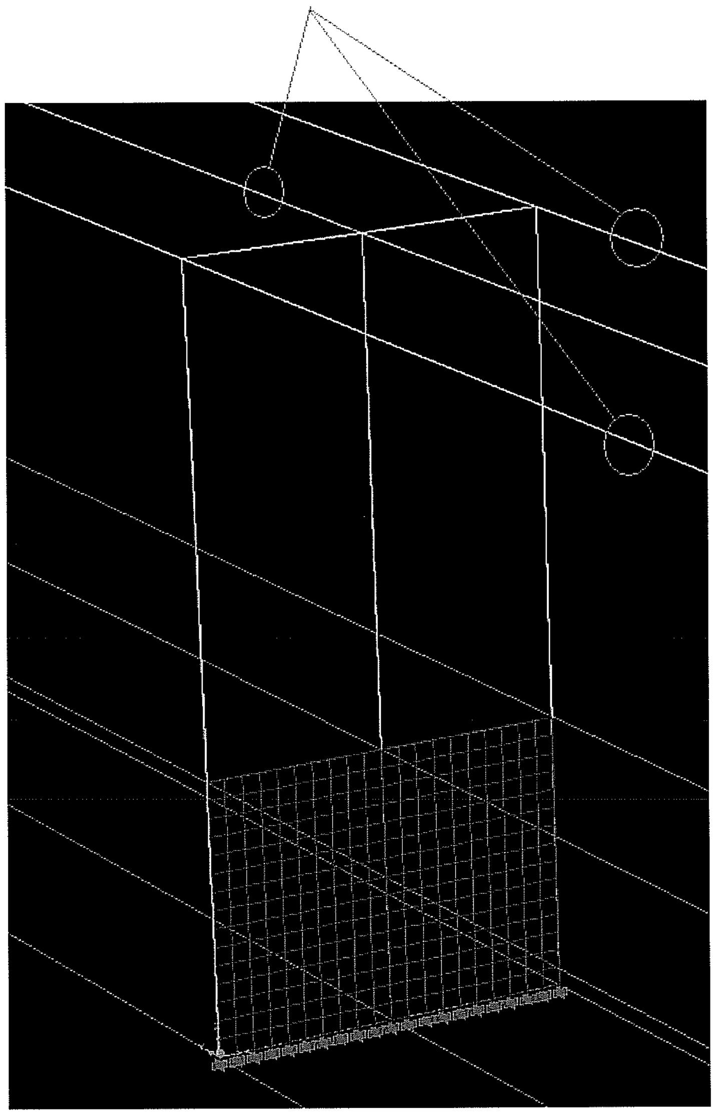


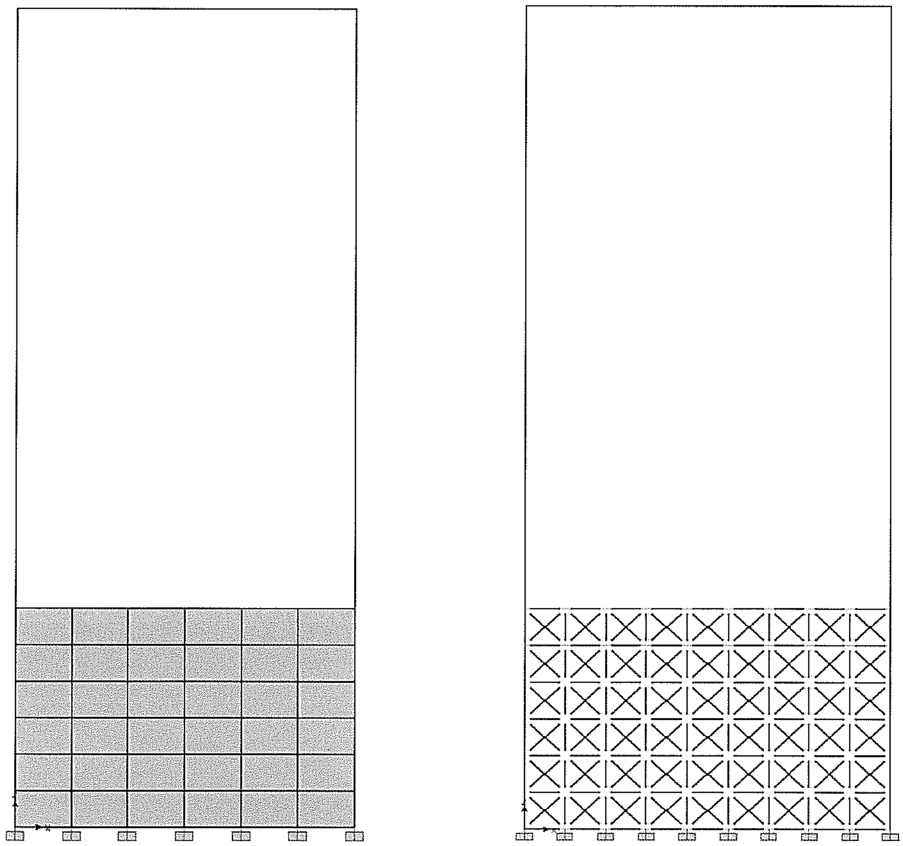
Figure 4-11. Bridge Pier Top Boundary Condition — Superstructure

4.5 MODELING SHEAR OR WEB WALLS

For a linear elastic analysis within SAP 2000 or other typical structural analysis programs, walls can be modeled using shell elements. These programs, however, usually lack the capability to capture inelastic behavior of these element types. As an alternative, one could build a model using a general finite element software package such as ANSYS. Finite element analysis programs can be rather expensive considering both dollars and computational time. Furthermore, these programs are more difficult to use and increase the chance of user error in the course of an analysis. Because of these reasons, there is a need for a simple, approximate method to capture the inelastic response of a wall within a structural analysis program like SAP. This section outlines such a method for the purposes of ultimate lateral strength prediction.

A possible solution to the problem of modeling bridge piers with shear walls to is to replace the shear wall with a truss-grid system. Figure 4-12 illustrates two models. The model on the left is comprised of frame elements for the columns and beams and shaded shell elements for the shear wall. In SAP 2000, this model is only capable of capturing linear elastic response. The model shown on the right is made up of frame elements to represent the beams and columns and a grid of truss elements to model the wall.

With the truss-grid model, the wall is replaced by rigid truss members in the vertical and horizontal direction. Non-rigid truss elements are placed on the diagonal between the rigid members. The diagonal truss members are sized such that the response for a linear elastic analysis matches the response of the shell-wall model. Once the linear elastic analysis case has been verified, plastic hinges are applied to the ends of the truss members and the analysis is rerun. Because the horizontal and vertical members are rigid, all of the inelastic deformation in the wall is captured in the diagonal truss elements. Figure 4-13 shows a close-up view of the truss-grid system for the pier shown in Figure 4-12.



Shell Wall Truss-grid

Figure 4-12. SAP 2000 Bridge Pier Models

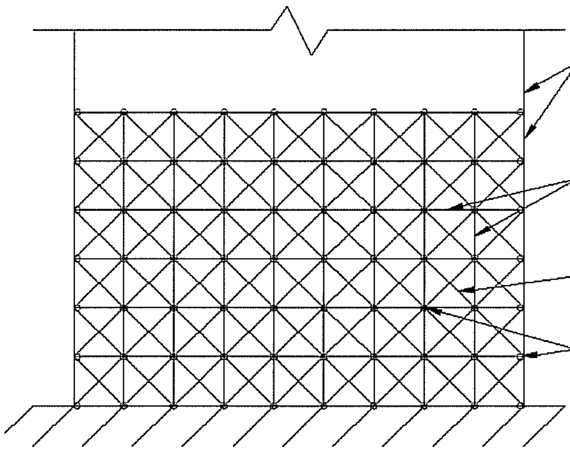


Figure 4-13. Close-up of Truss-Grid Wall Model

4.6 Procedure for Bridge Ultimate Strength Modeling within SAP

This section provides a step-by-step procedure to model bridge piers, both with and without shear walls, within SAP 2000. Along with a written description, images from SAP 2000 are included to show the necessary steps to accurately model the bridge piers and bridge systems under consideration. The bridges being modeled represent actual bridges crossing navigable waterways in the state of Texas that are subject to potential vessel collision.

4.6.1 Bridge Pier without Shear Wall

The representative example selected for this type of bridge pier is the east-bound Interstate Highway 10 (IH-10) bridge over the San Jacinto River outside of Houston, Texas. Bent 18 is one of two identical piers on each side of the main navigation channel that is subject to vessel collision. The bridge superstructure is comprised of 622-foot, 3-span continuous plate girders topped with a 10-inch reinforced concrete deck. Figure 4-14 shows a simple line sketch of the pier and includes the basic dimensions of the structure. Note the member names associated with each beam and column as those same names will be used throughout this section as labels for the material model and section definitions in SAP 2000 for those elements.

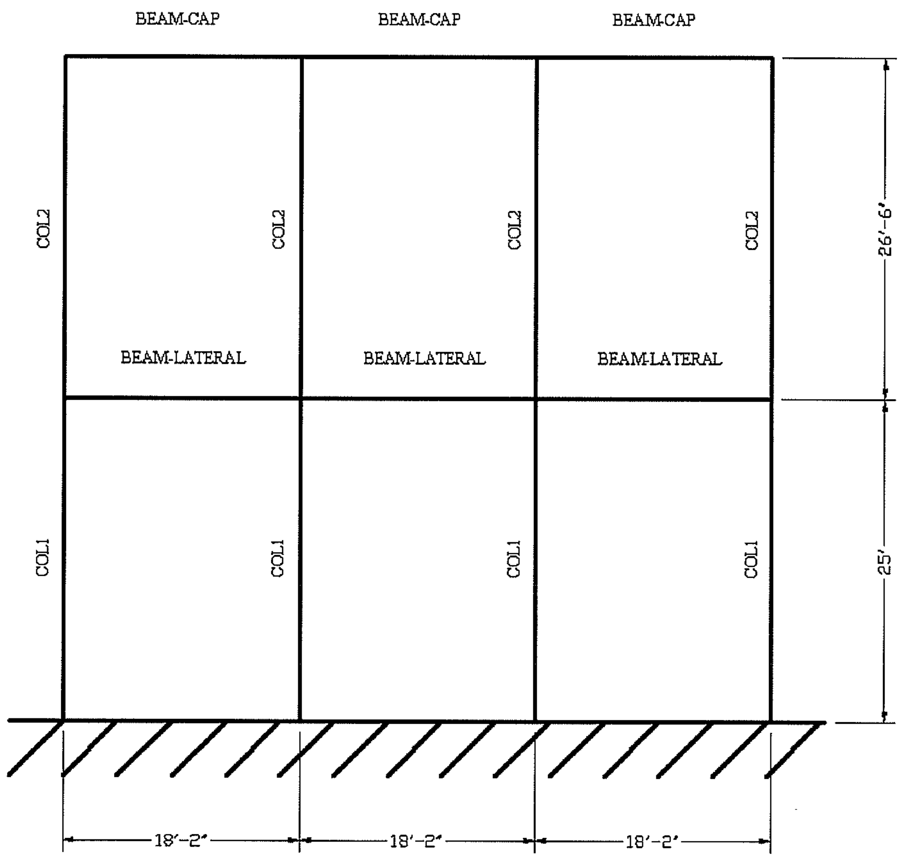


Figure 4-14. IH-10 Bridge Bent 18

Step 1: Define Bridge Pier Geometry

Bent 18 is 51.5-feet tall by 54.33-feet wide and is comprised of four equally spaced columns, connected together by a pile cap at the bottom, a beam at 25 feet above the pile cap and a cap beam at the top of the columns. To define the geometry of this pier in SAP 2000, it is convenient to start with a blank model and define gridlines along the centroids of the columns and beams. Figure 4-15 shows the creation of a grid within SAP 2000.

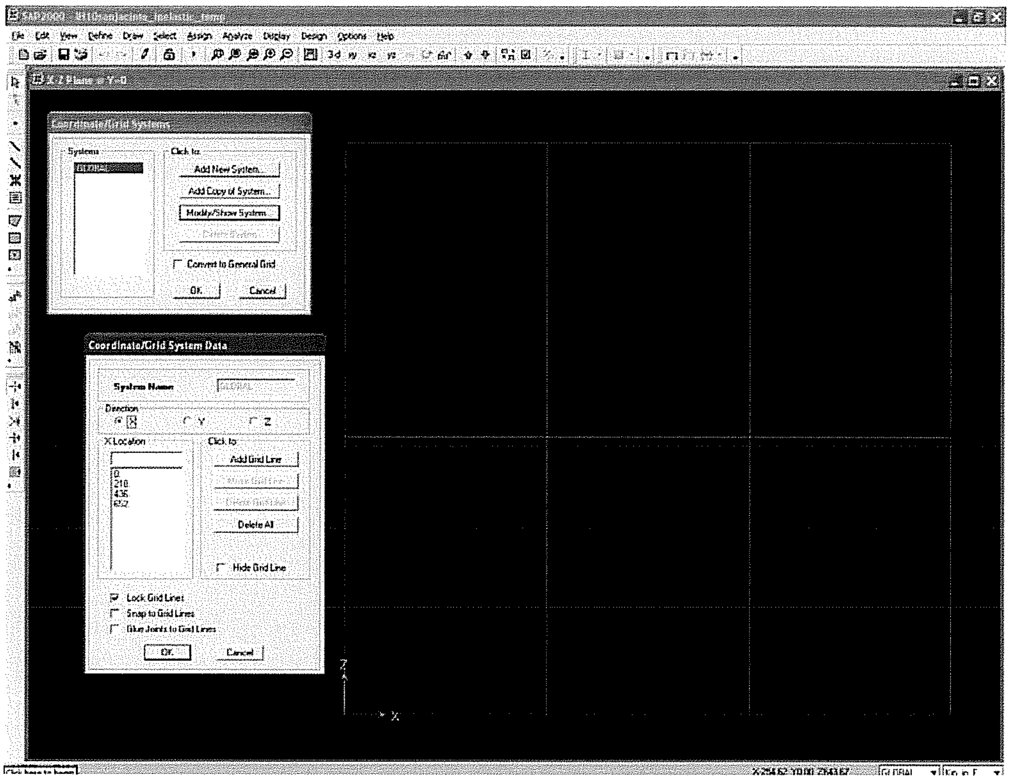


Figure 4-15. IH-10 San Jacinto Bridge Bent 18

Step 2: Define Material Models

Once the basic geometry of the system being analyzed has been established, the next step is to define a set of material properties for each of the bridge elements. As previously described, smeared material properties are being used to represent the modeling of the reinforced concrete elements. To establish the material property sets, a reinforced concrete section analysis was run for each of the bridge pier elements. These results are summarized in Table 4-2.

Table 4-2. IH-10 San Jacinto Pier 18 Material Model Summary

IH10 San Jacinto Eastbound Pier 18 Bottom Column				*IH10 San Jacinto Eastbound Pier 18 Lateral Beam*			
Basic Section Properties				*Basic Section Properties*			
width (in)	78.00	*depth (in)*	78.00	*width (in)*	84.00	*depth (in)*	48.00
Plastic Modulus (in^3)	118638.00	*Section Modulus (in^3)*	79092.00	*Plastic Modulus (in^3)*	48384.00	*Section Modulus (in^3)*	32256.00
bars	40-#11 bars	*stirrups*	#4 @ 9"	*bars*	22-#11	*stirrups*	#4 @ 9"
Response 2000 Section Analysis				*Response 2000 Section Analysis*			
My (in-kips)	126060.00	*fy (ksi)*	1.06	*My (in-kips)*	42732.00	*fy (ksi)*	0.88
Mp (in-kips)	159900.00	*fu (ksi)*	1.35	*Mp (in-kips)*	55392.00	*fu (ksi)*	1.14
		fu/fy	1.27			*fu/fy*	1.30
IH10 San Jacinto Eastbound Pier 18 Top Column				*IH10 San Jacinto Eastbound Pier Cap Beam*			
Basic Section Properties				*Basic Section Properties*			
diameter (in)	54.00			*width (in)*	84.00	*depth (in)*	48.00
Plastic Modulus (in^3)	26244.00	*Section Modulus (in^3)*	15458.99	*Plastic Modulus (in^3)*	48384.00	*Section Modulus (in^3)*	32256.00
bars	20-#11	*stirrups*	#4-9" pitch	*bars*	22-#11	*stirrups*	#6 @ 9"
Response 2000 Section Analysis				*Response 2000 Section Analysis*			
My (in-kips)	34572.00	*fy (ksi)*	1.32	*My (in-kips)*	61668.00	*fy (ksi)*	1.27
Mp (in-kips)	42576.00	*fu (ksi)*	1.62	*Mp (in-kips)*	86904.00	*fu (ksi)*	1.80
		fu/fy	1.23			*fu/fy*	1.41

Figure 4-16 shows the material property input boxes in SAP 2000. SAP 2000 contains default properties for several materials. Figure 4-16 specifically shows the material property input for 'col 1' or the bottom columns in the pier frame. Note that the values under 'Analysis Property Data', the mass, weight, modulus of elasticity, Poisson's ratio, and the coefficient of thermal expansion, are all consistent with reinforced concrete material properties. Also, note that the 'Type of Design' is set to steel. This selection is intentional, despite the fact that a reinforced concrete pier is being modeled. This box must be set to steel in order to define smeared material properties based on a yield and ultimate stress.

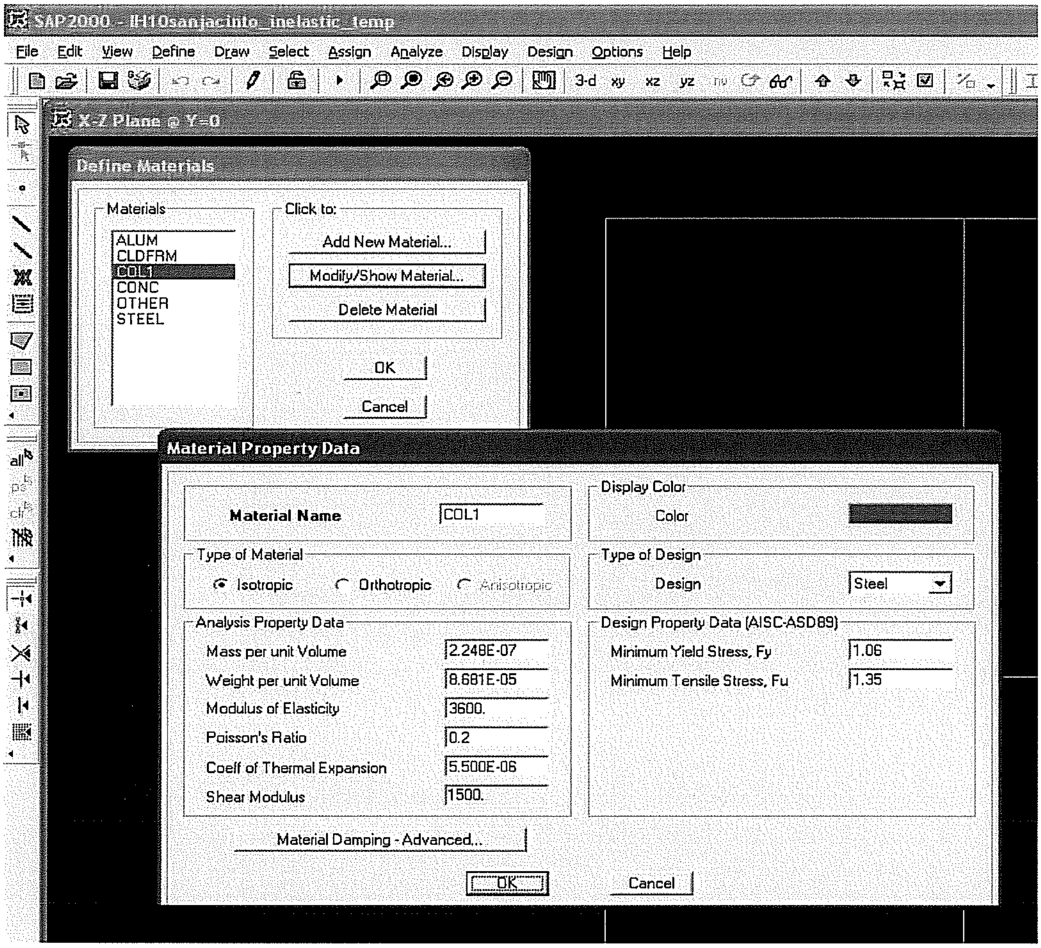


Figure 4-16. Material Property Definition in SAP 2000

Step 3: Define Element Section Properties

Figure 4-17 shows the SAP 2000 screen for entering a new section. Again, input for the bottom column or 'col 1' in the pier frame is shown. A user needs to enter dimensions and assign a material for the section. Also shown in the screen capture are the section properties, which SAP calculates based on the geometry entered by the user.

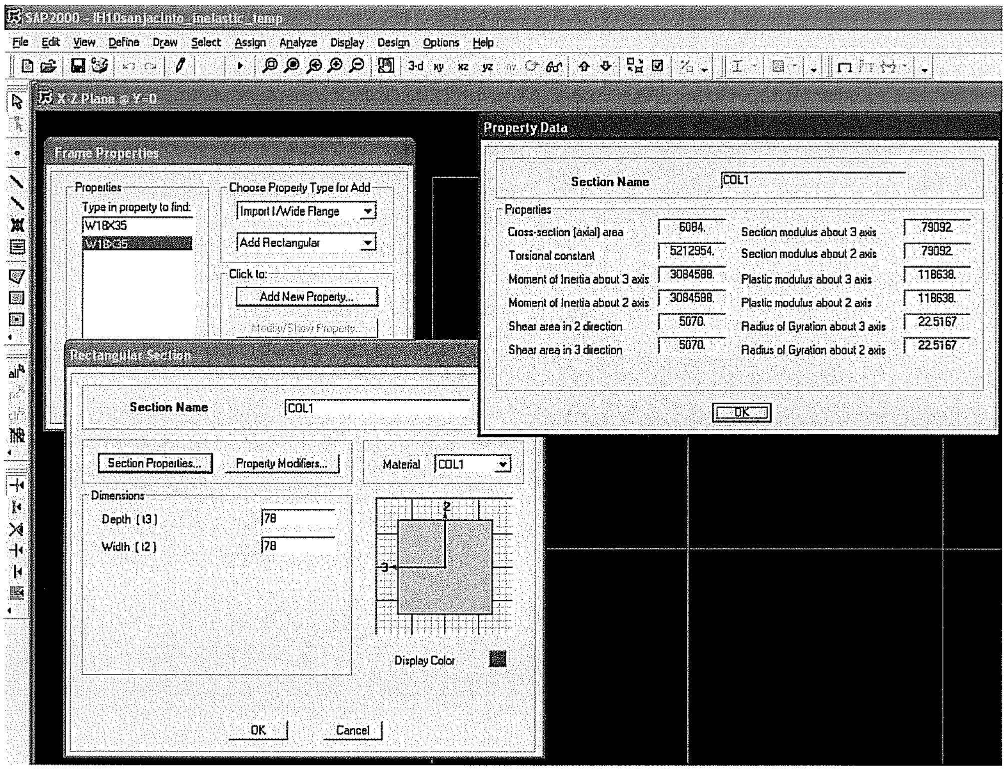


Figure 4-17. Section Input in SAP 2000

Step 4: Draw Bridge Pier Elements

After the material and section properties are entered for each element, the bridge pier can be drawn. Figure 4-18 shows 2 images of the drawn bridge in SAP 2000, one as line elements and a second comprised of 3-D solids. It is helpful to view the solid model to ensure that sections have been defined properly.

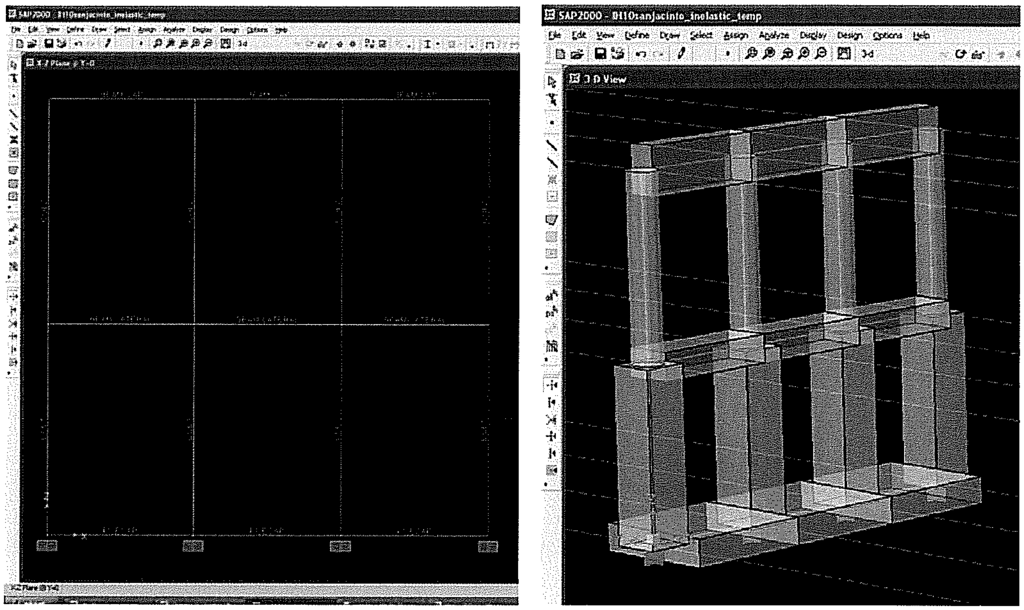


Figure 4-18. SAP 2000 Pier Models

Step 5: Define Plastic Hinge Properties and Assign to Element Ends.

The next step is to define plastic hinges, which will be applied to member ends and will be used to capture the inelastic behavior of the structure. SAP 2000 has several default hinge properties built in including moment, axial, shear and moment-axial interaction hinges. User-defined hinges can also be defined and are used for this model. Plastic hinge properties are based on the section analyses performed to determine the material properties. The hinges used in these models are defined as moment-axial hinges and are based on the yield stress and the ratio of the yield stress to the ultimate stress. This ratio defines how much additional capacity is available in the hinges after the onset of yield. Figure 4-19 shows the SAP 2000 input boxes for defining hinge properties.

Recall that, in defining the material properties it was assumed that there was zero axial load on the sections. It seems counterintuitive then, that the hinges are defined as moment-axial interaction hinges. Assuming zero axial load in defining the material was shown to be a conservative assumption, given the level of axial load on bridge members. There is, however, axial load in the real structure, and by defining moment-axial hinges, the effect of the axial load on yielding in the structure is taken into consideration. In addition, by defining moment-axial hinges, as opposed to hinges based only on moment, any change in how the material is defined could be easily integrated into the models. For example a more accurate or detailed reinforced concrete material model could be used or developed.

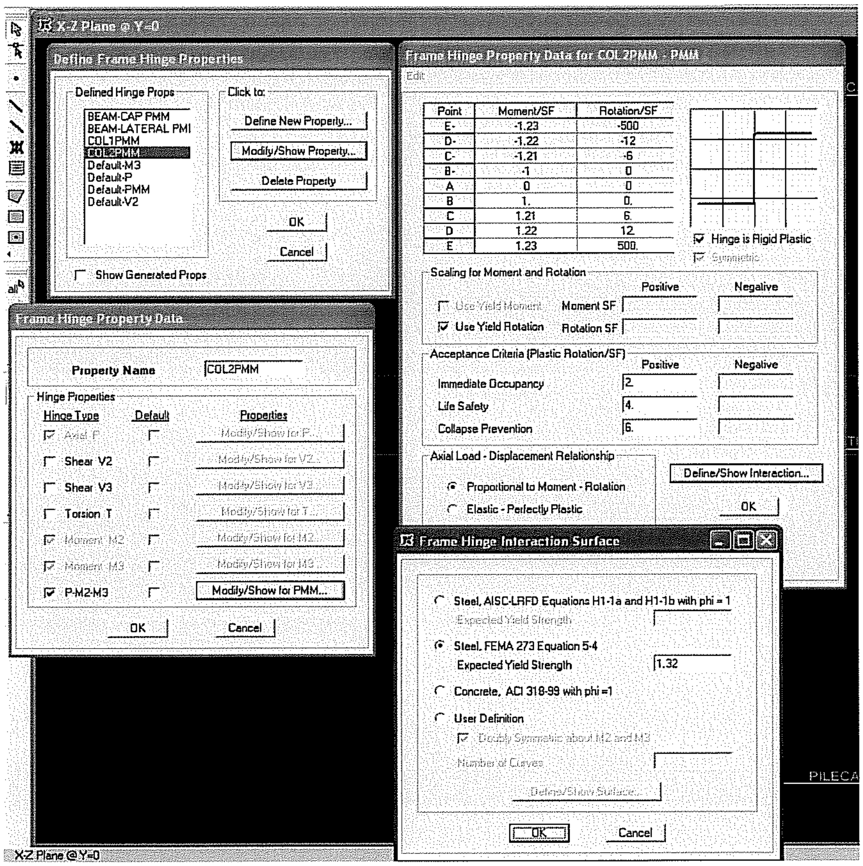


Figure 4-19. SAP 2000 Plastic Hinge Property Input

The 'Define Frame Hinge Properties' window (upper left of Figure 4-19) lists all of the default and user-defined hinges and provides the option to modify, delete or define a new hinge property. The moment-axial hinge for the top column, 'COL2-PMM', has been selected. The 'Frame Hinge Property Data' window (lower left of Figure 4-19) shows that 'COL2-PMM' is a user-defined moment-axial interaction hinge. The 'Frame Hinge Property Data for COL2-PMM' box in the upper right of Figure 4-19 shows the actual strength and deformation properties of the hinge. This hinge is defined based on strength and deformation relative to the yield strength and rotation. The strength characteristics of the hinge are defined by the left column. The values entered are based on the ratio of the plastic moment to the yield moment for this section, $M_p/M_y = 1.23$. The deformation capacity of the hinge is defined by the right column. In this case, a large value has been assigned for the ultimate rotation capacity, essentially making the hinge capable of infinite plastic deformation or rotation once the plastic strength has been reached. A real plastic hinge would not be capable of infinite deformation or rotation, but this definition is acceptable if the deformation capacity of the member or structure as a whole is assessed in the post-analysis phase.

For user defined moment-axial interaction hinges, an interaction surface must be defined. The lower right window in Figure 4-19 shows that 'Steel FEMA 273 Equation 5-4' has been selected as the yield surface. A steel interaction surface has been defined even though the bridge sections being defined are reinforced concrete because of the smeared material approach that has been taken. When using the 'Steel FEMA 273 Equation 5-4', the yield strength of the section which the hinge is being used for must be re-entered. Figure 4-19 shows that for the top column the yield stress was entered as 1.32, which is consistent with how this value was previously defined.

Once plastic hinges have been defined for each member, they need to be assigned to the proper elements. Hinges can be defined at any relative point along the length of a beam, but it is often easiest to assign hinges just to the ends of members. Therefore, it may be necessary to subdivide an element to place hinges at the desired locations. It is important to note that static nonlinear analysis results in SAP 2000 are very sensitive to the number and location of hinges used. In order to capture the inelastic response at given point in a system, a hinge must be assigned to that location. If a static nonlinear analysis case is setup in SAP 2000, but no hinges are assigned to the model, the analysis results will show the system acting in a linear elastic fashion. While possible hinge locations could vary widely depending on the specifics of a given structure, it is generally sufficient to place hinges at member ends and at points where loads are applied to a structure. Other portions of a structure that are subject to high moment or axial forces should also have hinges assigned. If necessary, several configurations of hinges may need to be tried to be certain that SAP 2000 is accurately capturing the inelastic response of the system being analyzed. Figure 4-20 shows the hinge assignment process for a specific case of the IH-10 Bridge in SAP 2000. For the results presented in Chapter 5, different hinge patterns are considered for each particular load case. Figure 4-20 is presented only as an example of assigning hinges to the model.

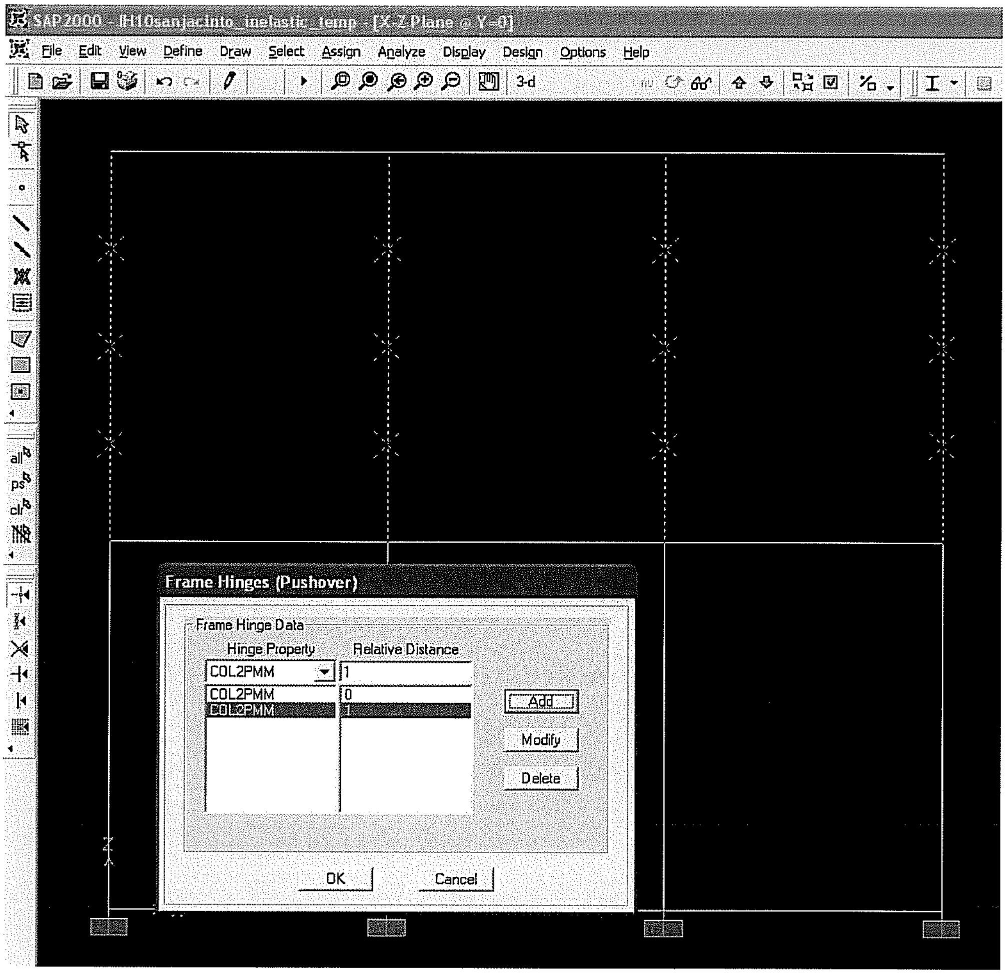


Figure 4-20. Assigning Hinges in SAP 2000

Step 6: Modeling Bridge Superstructure and Adjacent Piers

The current AASHTO specifications consider only the strength of a stand-alone pier and do not account for the redistribution of forces through a cap beam and deck to adjacent piers. By modeling the adjacent piers of a bridge, as well as the bridge deck and girders, these factors can be considered. Adding adjacent piers can be done by adding grid planes in the direction perpendicular to the pier that has already been drawn. Rather than redrawing an identical pier, a user can also simply copy and paste elements onto a new grid plane. Figure 4-21 shows the IH-10 Bridge with all of the piers connected to the 3-span continuous plate girder that spans the main navigational channel. It is assumed that the adjacent piers will behave linear elastically during a vessel collision event, so there is no need for the material model and plastic hinge property definitions described above. The pier and section geometry, along with a simple material model that accurately reflects the stiffness of reinforced concrete, needs to be defined.

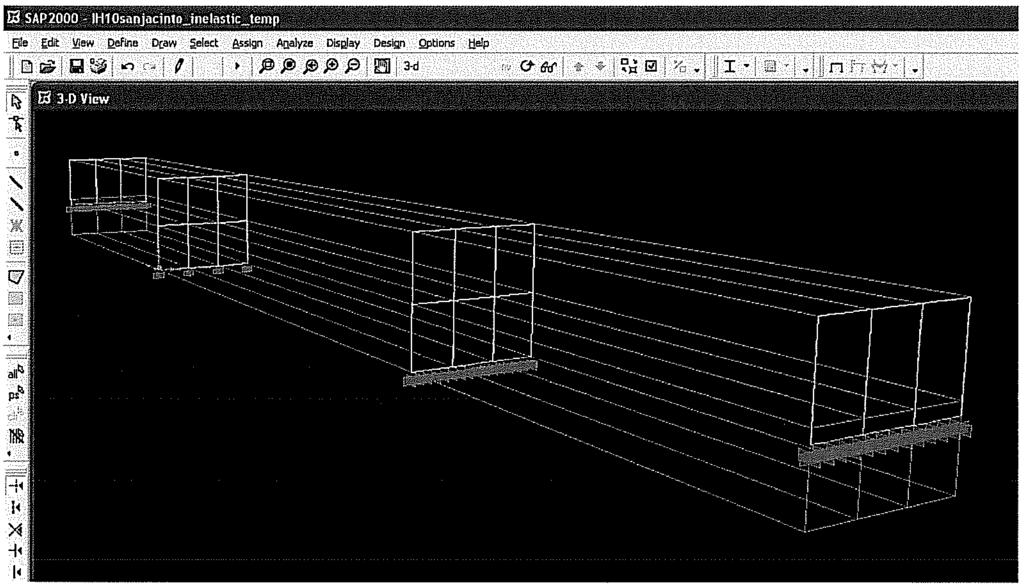


Figure 4-21. Bridge Model with Adjacent Piers

Rather than model the exact geometry of the bridge superstructure, general elements are defined that match the stiffness characteristics of the bridge deck and girders. In order to model the superstructure this way, geometric and stiffness properties of the superstructure must be determined. Table 4-3 shows the section properties for the deck and girders of the IH-10 Bridge. It is assumed that the deck and girders do not act as a composite system, so the section properties of each are merely added together to get the final section properties shown. Note that Table 4-3 also shows the dead load of the deck and girders. This information will be used in chapter 5 when loads are assigned to the model.

Table 4-3. Deck and Girder Properties

Member	Description	Modulus of Elasticity (ksi)	Cross Section Area (in^2)	Moment of Inertia (in^4)		Dead Load Contribution to Pier (kips)
				xx	yy	
Deck	7.25" Thick Deck 60' Roadway Width	3600.00	5220.00	22864.69	225504000.00	1212.60
Girder	3-span Cont Plate Girder 6 Individual Girders 84" Depth, 30" Flange Width 1" Plate Thickness	29000.00	852.00	895723.98	27041.00	586.47
Transformed Girder	Girder Properties Transformed to Account for Difference in Modulus of Elasticity	3600.00	6816.00	7165791.84	216327.98	586.47
Total	Entire Superstructure Properties--Deck and Transformed Girder Together	3600.00	12036.00	7188656.53	225720327.98	1799.07
Total/4	4 Elements will be used to represent the deck in SAP 2000	3600.00	3009.00	1797164.13	56430082.00	449.77

Once the section properties have been determined, the bridge and deck can be modeled together as a series of elements with a general section. SAP 2000 allows a user to input section properties such as area and moment of inertia without entering the exact section geometry. Figure 4-22 shows the input for a general section that is used to represent the superstructure of the IH-10 Bridge model. For this model, four elements are used to represent the deck and girders. Therefore the section properties entered reflect ¼ of the moment of inertia, I, and cross-sectional area, A, for the superstructure in each direction. Lastly, property modification factors are assigned. This step is shown in Figure 4-23. Large modification factors have been applied to the area properties, making the section axially rigid, to the torsion properties to prevent twist, and to the shear properties, so that shear deflections will be negligible. Also note that a modification factor of *0.0* has been applied to the weight and mass for these elements. The dead load of the girders and deck were determined earlier, but will not be applied directly to the general elements being defined here. Instead, the dead load as well as the live load from the bridge and deck will be applied directly to the top of the piers. Application of in-place loads is addressed in more depth in Chapter 5. These factors are used to ensure that the line elements representing the superstructure behave in a similar fashion to the actual deck.

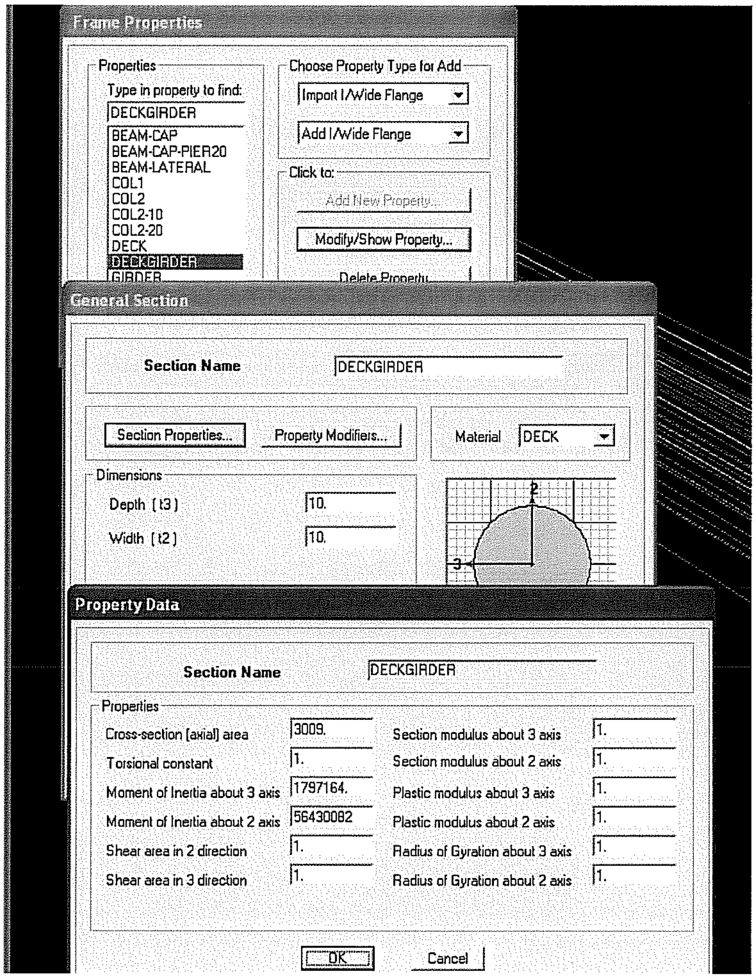


Figure 4-22. General Section Properties

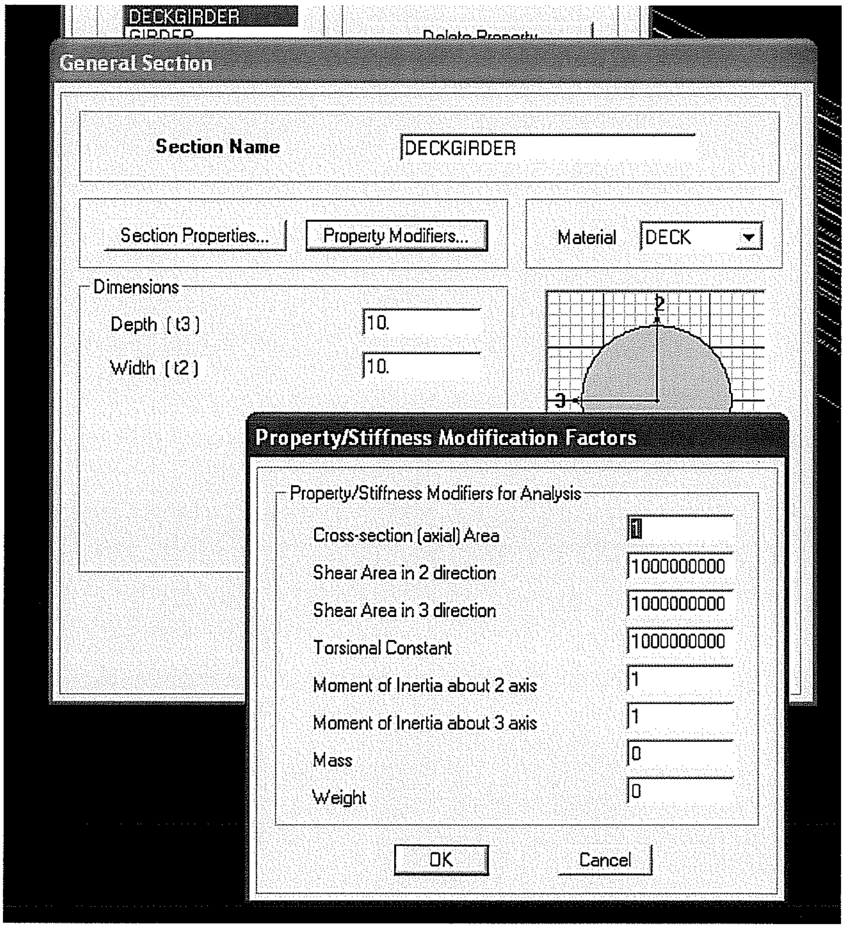


Figure 4-23. Section Modification Factors

Once a general section has been defined for the superstructure, line elements, representing the superstructure can be drawn into the model. This step is shown in Figure 4-24. At this point, it is crucial to assess and understand the connection detail between the bridge superstructure and bridge pier. Take for example a bridge with simply-supported precast concrete 'I' girders between the bridge piers with a continuous deck poured over the top. One needs to be careful in assuming how much redistribution is possible. The girders and deck will certainly provide some type of support condition at the top, but the effect may be limited if the girders are simply sitting on bearing pads on the cap beam of the pier. The IH-10 Bridge superstructure is continuous over the main channel piers. For the IH-10 Bridge being modeled for this research, the 3-span superstructure is assumed to be fully connected over the two interior piers and simply supported at the two exterior piers. After the superstructure elements have been drawn, moments can be released at the far ends of the elements.

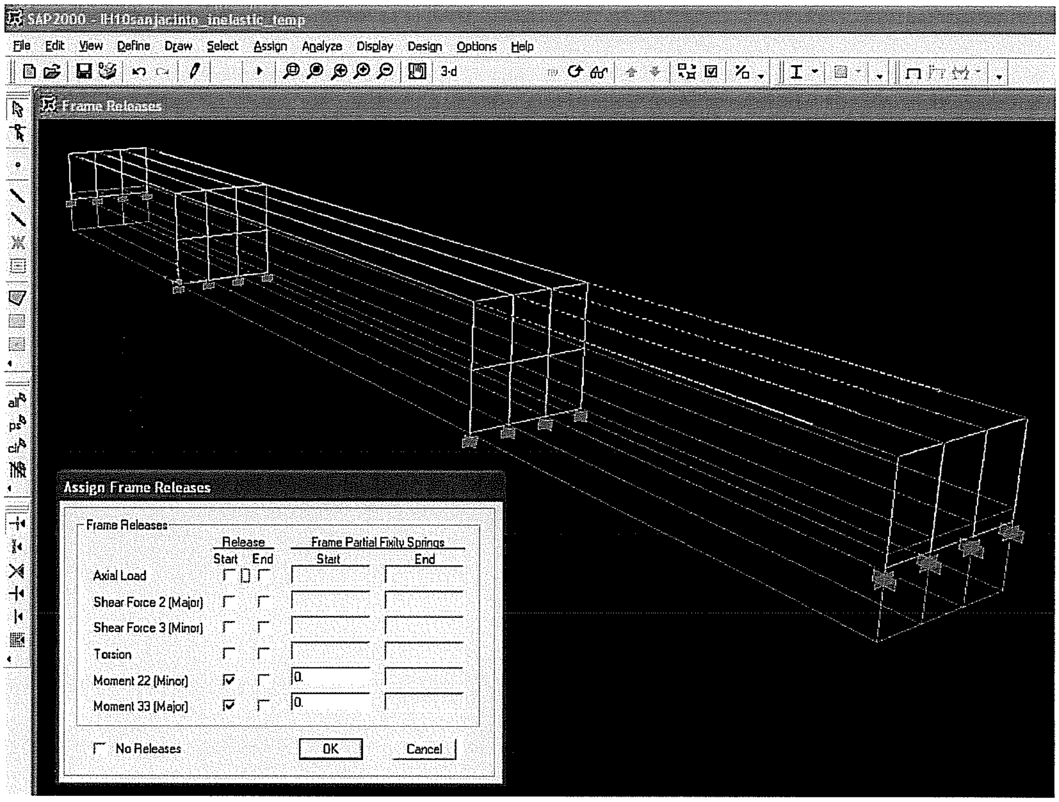


Figure 4-24. Releasing End Moments in Superstructure Elements

At this stage, the structure is ready to be analyzed. Chapter 5 of this report outlines the loads applied and the analysis cases performed for this model to assess the structural performance when subjected to vessel impact.

4.6.2 Bridge Pier with Shear Wall

The representative examples selected for this type of bridge pier are bents 21 and 22 of the State Highway 87 (SH-87) bridge over the Gulf Intracoastal Waterway (GIWW), constructed in 1969. Bents 21 and 22 are two identical piers on each side of the main navigation channel that are subject to potential vessel collision. The bridge superstructure over the waterway is a 680-foot, 3-span continuous steel plate girder unit. An elevation of the SH87 Bridge is shown in Figure 4-25. The drawing is shown to give an idea of what the SH-87 bridge profile looks like, the specific notes on the bridge are not important.

Bents 21 and 22 of the SH-87 Bridge are 3-column piers, and measure 88-feet high by 42-feet wide. A 2-foot wide shear or web wall extends 31 feet up from the pile cap. Figure 4-26 shows the construction drawing for these piers. Notice that the column sections change 27 feet above the shear wall, from a 66-inch square column to a 48-inch circular section. Again, the specific notes in the figure are not relevant to the current discussion.

The following section outlined the procedure for the modeling of bridge piers with shear walls. Many of the required steps have already been described in detail in Section 4.6.1 and will only be touched on briefly. The emphasis is on modeling of the shear wall using the truss-grid model described earlier.

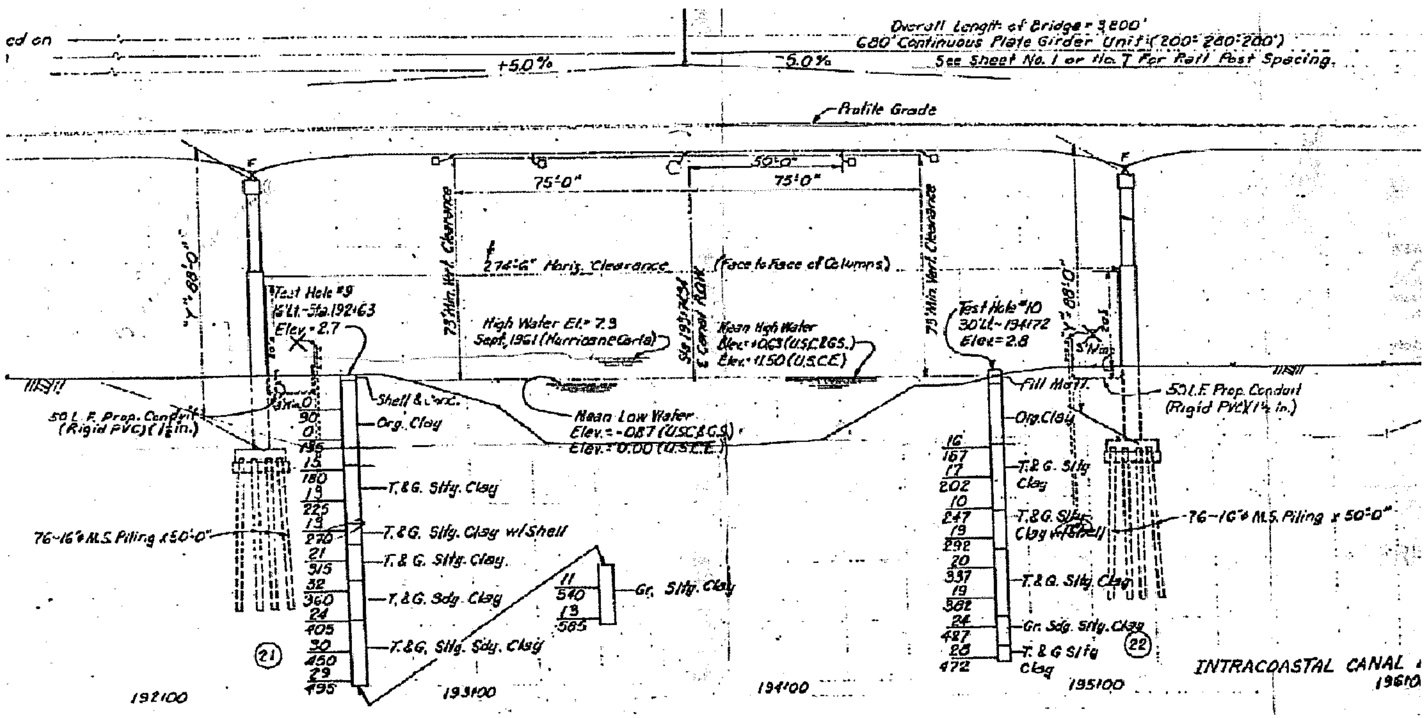


Figure 4-25. SH87 Bridge over the GIWW Elevation (TXDOT Construction Documents, 1969)

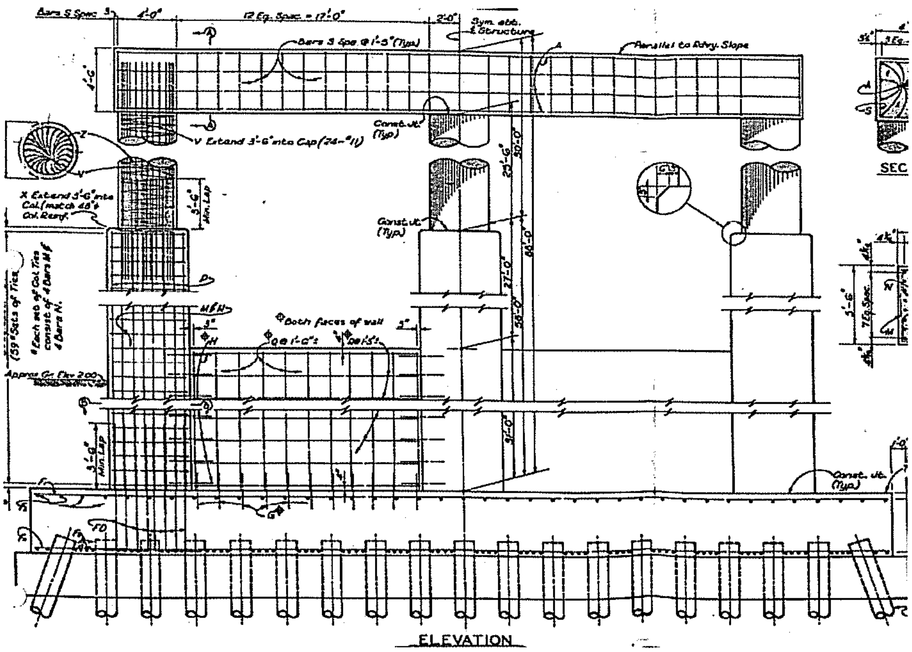


Figure 4-26. SH87 Bridge Pier (TXDOT Construction Documents, 1969)

4.6.2.1 Part A: Shell Wall Model

Step 1: Define Bridge Pier Geometry

The bridge geometry for this model can be established in the same fashion as described earlier for the IH-10 Bridge. Gridlines should be spaced and drawn to correspond with the centroids of the beams and columns. In addition, gridlines should be defined at the boundaries of the wall.

Step 2: Define Material Models

The shell-wall model being constructed in this section will only be used for a linear elastic analysis. Therefore, the lengthy process for defining material properties for each individual member in the frame described in Section 4.6.1

does not need to be completed. Instead, only the proper Modulus of Elasticity, E, needs to be defined. The Modulus of Elasticity used here is based on the ACI 318-02 equation given earlier (Equation 4-1). This value can be entered into either the SAP 2000 default concrete material property (CONC), or a new user-defined material property, as was done previously. From the construction drawings, it is known that the specified concrete compressive strength for this bridge, f_c', is 1200 psi. Therefore, the corresponding modulus of elasticity, E, is 1975 ksi.

Step 3: Define Element Section Properties

Section properties can be assigned in the same manner for the beams and columns as described earlier. Additional properties need to be defined for the wall/area section. Figure 4-27 shows the definition of an area section in SAP2000. Note that the section has been titled 'WALL' and is defined as a 'Shell'. SAP 2000 defines a shell as an area element that has both translational and rotational degrees of freedom and is capable of supporting both forces and moments (SAP2000 user manual, 2004). SAP 2000 also has sub-types for a shell element. In this case, the sub-type used is also called 'Shell' with the 'Thick Plate' option checked. The 'Shell' subtype again means that the element is capable of supporting both forces and moments. The 'Thick Plate' option is used to include shear deformations in the elements. The defined thickness values used should correspond to the dimensions of the actual shear wall being modeled.

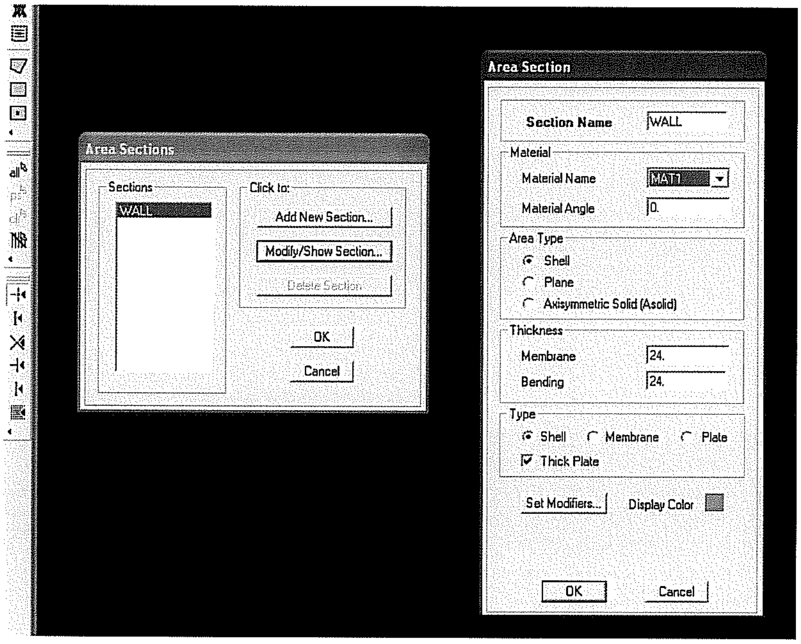


Figure 4-27. Defining Area Sections

Step 4: Draw Bridge Pier Elements

Much of the process for building the SAP 2000 model for bents 21 and 22 is the same as described for the IH-10 Bridge. The only difference is the addition of a shear wall. The wall is added by drawing two area elements between the three columns up to a height of 31 feet. Next, the user needs to mesh the large areas into a series of smaller elements. It is best to break the areas down into elements that are approximately square. In this case, each area representing the walls between the three columns are divided into a 10 by 16 mesh of elements, each 25.2 inches × 25.2 inches. After the area has been meshed, the joints at the bottom of the pier need to be fixed. Figure 4-28 shows the two wall areas drawn and ready to be meshed. Figure 4-29 shows the final meshed model.

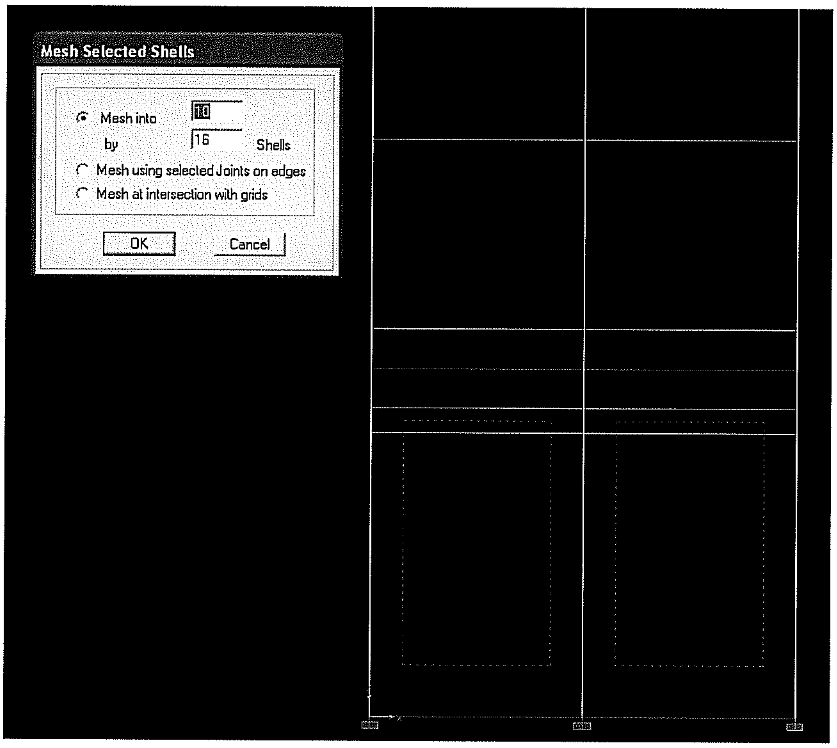


Figure 4-28. Meshing Shell Wall in SAP 2000

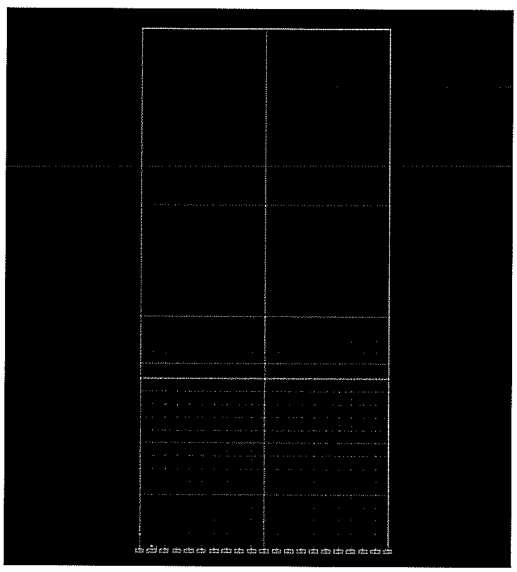


Figure 4-29. Meshed Wall

To check if the wall mesh is adequate, at least two different meshes need to be investigated. If they both models give the same solution (displacement) under the same loads, then the coarser mesh is acceptable. If the solutions are not close, a finer mesh must be used. For this model, the 10 by 16 element mesh has been verified to be adequate.

Step 5: Modeling Bridge Superstructure and Adjacent Piers

This step can be completed as previously described. However, due to a lack of information, the two exterior piers at the ends of the 3-span continuous plate girder unit are not modeled. Instead, simple pin supports are placed at the ends of the superstructure. Figure 4-30 shows the SH-87 Bridge Model with identical piers 21 and 22 defining the main navigation channel and both subject to potential vessel collision. Table 4-4 summarizes the section and material properties needed to define the superstructure elements shown in Figure 4-30.

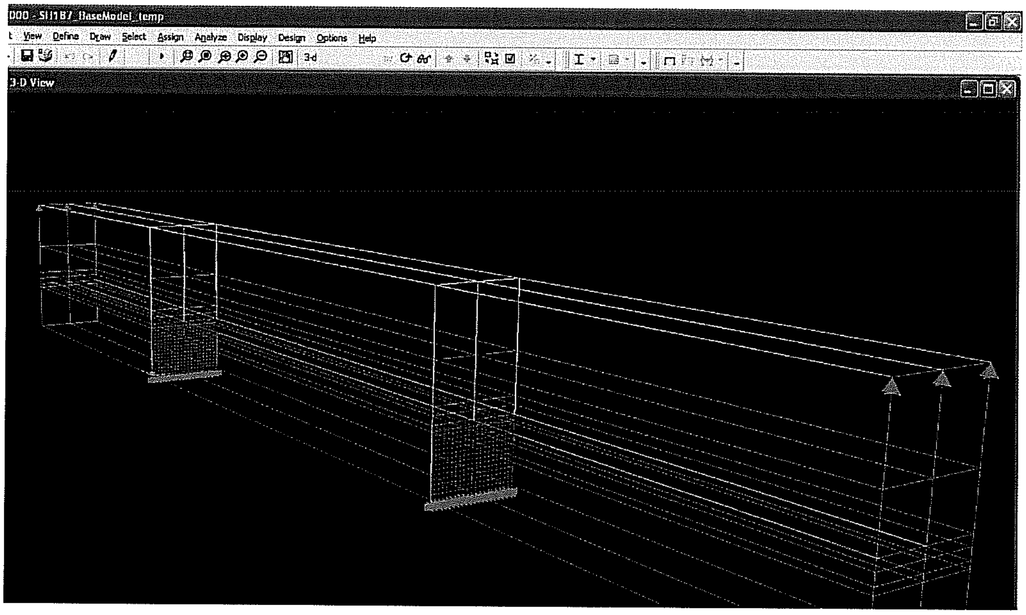


Figure 4-30. SH87 Bridge Model in SAP 2000

Table 4-4. SH87 Superstructure Properties

Member	Description	Modulus of Elasticity (ksi)	Cross Section Area (in^2)	Moment of Inertia (in^4)		Dead Load Contribution to Pier (kips)
				xx	yy	
Deck	10.5" Thick Deck 50.3' Roadway Width	3600.00	6337.80	58228.54	192422452.00	1584.53
Girder	3-span Cont Plate Girder 6 Individual Girders 72" Depth, 20" Flange Width 1.5" Plate Thickness	29000.00	490.50	346885.80	6058.20	523.16
Transformed Girder	Girder Properties Transformed to Account for Difference in Modulus of Elasticity	3600.00	3924.00	2775086.40	48465.60	523.16
Total	Entire Superstructure Properties--Deck and Transformed Girder Together	3600.00	10261.80	2833314.94	192470917.60	2107.69
Total/3	3 Elements will be used to represent the deck in SAP 2000	3600.00	3420.60	944438.31	64156972.53	702.56

Step 6: Apply Load

At this stage, the model is ready for loads to be applied for analysis. Again, this step is being done in order to calculate the linear elastic response of the shell wall bridge pier model to an arbitrary load at any given load location. The lateral displacement from this load is then used to size the truss members in the truss-grid wall model. The following list describes the type and location of all the loads that are considered. The loads are also illustrated in Figure 4-31.

Linear Elastic Load Cases for Shell-Wall Model:

- Load Location 1: Point load at the top of the wall
- Load Location 2: Point load 48 inches above the top of the wall
- Load Location 3: Point load 96 inches above the top of the wall
- Load Location 4: Distributed Load 30 inches above and below the wall

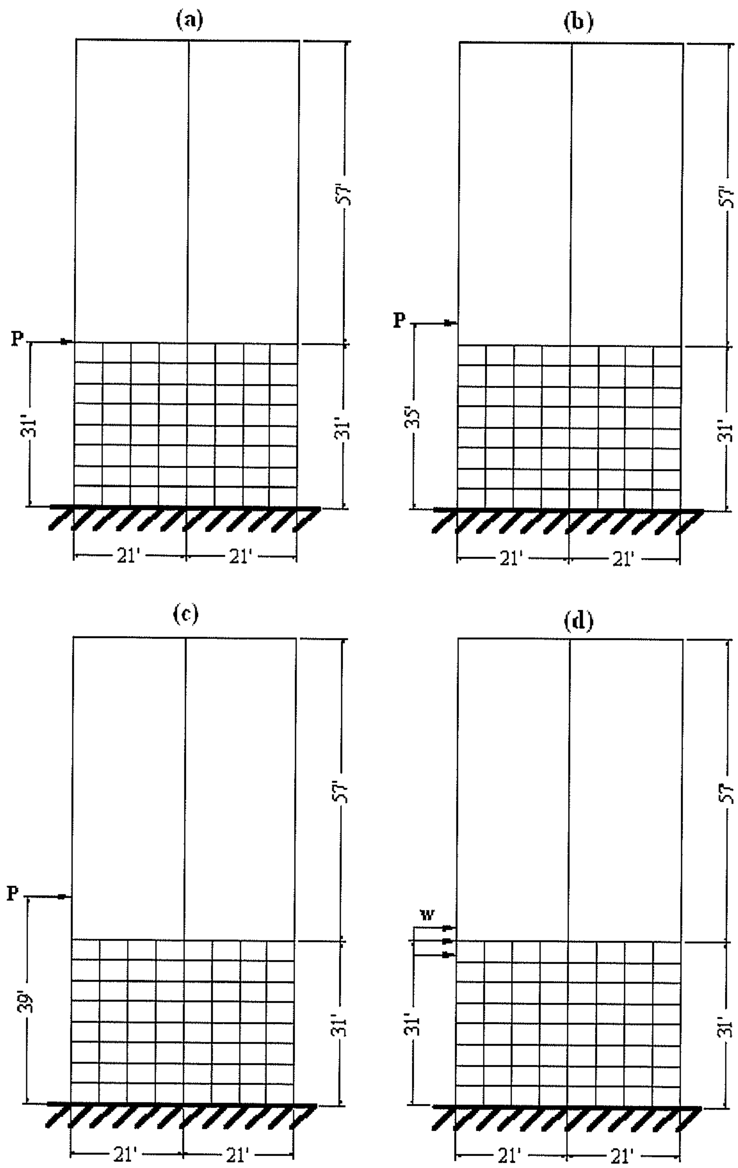


Figure 4-31. SH-87 Impact Loads: (a) Load Location 1: Point Load at Top of Wall, (b) Load Location 2: Point Load 48 inches Above Wall, (c) Load Location 3: Point Load 96 inches Above Wall, (d) Location 4: 60-inch Wide Distributed Load at Wall

It should be noted that the load cases selected here represent a set of possible cases for this bridge based on a range of water levels, not the exact cases that are required by the AASHTO specification. They have also been selected in part to provide the opportunity to make some reasonable conclusions regarding the validity of the truss-grid model. In addition, presenting multiple load cases for multiple boundary conditions emphasizes the requirement that the diagonal truss members in the truss-grid model need to be sized separately for each load and boundary condition configuration. It is important to reiterate that the load cases presented here only represent the lateral loads being used to match the response of the shell wall model and the truss-grid wall model to a linear elastic analysis. Chapter 5 discusses the loads applied to the bridge to determine the capacity to potential vessel collisions

Step 7: Run Linear Elastic Analysis

Table 4-5 summarizes the results from a linear elastic analysis on the SH87 shell-wall model built above.

Table 4-5. Shell Wall Linear Elastic Analysis Results

Load Description	Top Boundary Condition	Total Load Applied	Lateral Displacement at Point of Load
Point Load at Top of Wall	free	1000 k	0.108"
Point Load at Top of Wall	fixed	1000 k	0.103"
Point Load at Top of Wall	superstructure	1000 k	0.106"
Point Load 48" above Top of Wall	free	1000 k	0.235"
Point Load 48" above Top of Wall	fixed	1000 k	0.210"
Point Load 48" above Top of Wall	superstructure	1000 k	0.224"
Point Load 96" above Top of Wall	free	1000 k	0.472"
Point Load 96" above Top of Wall	fixed	1000 k	0.394"
Point Load 96" above Top of Wall	superstructure	1000 k	0.436"
Distributed Load 30" above and below wall	free	1000 k	0.107"
Distributed Load 30" above and below wall	fixed	1000 k	0.101"
Distributed Load 30" above and below wall	superstructure	1000 k	0.104"

4.6.2.2 Part B: Truss-Grid Wall

Step 1: Define Bridge Pier Geometry

The geometry and grid layout defined previously in Section 4.6.2.1 is also be used for the truss-grid model. The model geometry, including the wall boundary dimensions remain the same; however, the shell wall is replaced with a truss-grid wall. Replacing the shell wall with the truss-grid wall will is discussed in detail in Step 4.

Step 2: Define Material Models

The procedure to define the material properties for the various sections of the SH-87 Bridge is carried out in the same fashion as previously described for the IH-10 Bridge. In addition to performing a reinforced concrete section analysis for the beams and columns, an additional analysis is needed for the wall section. The following lists the assumed material properties for piers 21 and 22 of the SH87 Bridge:

- Concrete Compressive Strength, $f_c' = 1200\ psi$
- Modulus of Elasticity, $E = 1975\ ksi$
- Steel Reinforcing Yield Strength, $f_y = 40\ ksi$

These values were input into Response, along with the section geometry and reinforcing bar layout, to develop the material models summarized in Table 4-6.

Table 4-6. SH87 Material Properties

SH 87 Intracoastal Piers 21 &22 Top Column Section			
Basic Section Properties			
diameter (in)	48.00		
Plastic Modulus (in^3)	18432.00	Section Modulus (in^3)	10857.34
bars	24-#11	stirrups	#3 @ 6"
Response 2000 Section Analysis			
My (in-kips)	22178.40	fy (ksi)	1.20
Mp (in-kips)	27290.40	fu (ksi)	1.48
		fu/fy	1.23

SH 87 Intracoastal Piers 21 &22 Bottom Column Section			
Basic Section Properties			
width (in)	66.00	depth (in)	66.00
Plastic Modulus (in^3)	71874.00	Section Modulus (in^3)	47916.00
bars	28-#11	stirrups	# 4 @ 12"
Response 2000 Section Analysis			
My (in-kips)	46820.40	fy (ksi)	0.65
Mp (in-kips)	61653.60	fu (ksi)	0.86
		fu/fy	1.32

SH 87 Intracoastal Piers 21 &22 Cap Beam Section			
Basic Section Properties			
width (in)	51.00	depth (in)	54.00
Plastic Modulus (in^3)	37179.00	Section Modulus (in^3)	24786.00
bars	12-#11	stirrups	#5 @ 17"
Response 2000 Section Analysis			
My (in-kips)	17707.20	fy (ksi)	0.48
Mp (in-kips)	23161.20	fu (ksi)	0.62
		fu/fy	1.31

SH 87 Intracoastal Piers 21 &22 Wall Section			
Basic Section Properties			
width (in)	252.00	depth (in)	24.00
Plastic Modulus (in^3)	36288.00	Section Modulus (in^3)	24192.00
bars	12-#11	stirrups	#5 @ 17"
Response 2000 Section Analysis			
My (in-kips)	26781.60	fy (ksi)	0.74
Mp (in-kips)	39825.60	fu (ksi)	1.10
		fu/fy	1.49

Step 3: Define Element Section Properties

Section information for the beams and columns in the SH-87 Bridge is entered as previously described. For models with shear walls, two additional sections need to be defined in order to use the truss-grid wall model — a 'rigid' element and a 'truss' element. The rigid element can be defined in several ways. A general section could be used, as was done to model the bridge superstructure, and a large value for the cross-sectional area could be used. Otherwise, a simple section, such as a square or circle can be used and large section modification factors could be applied. The latter of these is shown in Figure 4-32. A 12-inch × 12-inch section is defined and called 'RIGID', then the property modification box is opened and a large value is entered for the cross-section area modifier. All of the rigid elements in the model are pinned, so it is only necessary to make certain that the model is axially rigid.

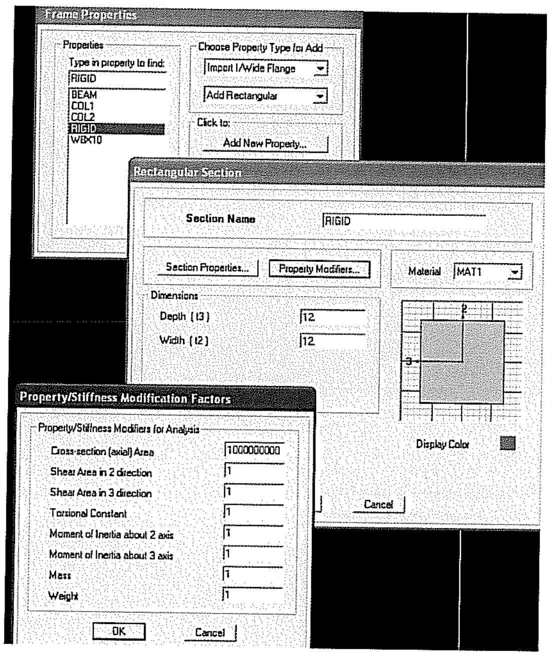


Figure 4-32. Rigid Member Section Properties

The truss members can also be defined as a general section, or by choosing a specific geometry. At this stage in building the model, the cross-sectional area of this element is not known. As an initial guess, a 12-inch × 12-inch section has been defined for the truss members. The truss section dimensions will be changed as needed to match the elastic response of the shell wall model. Once the truss and rigid sections have been defined, the truss-grid can be drawn for the model. The next step outlines this procedure.

Step 4: Draw Bridge Pier Elements

The truss-grid model is built by modifying the existing shell wall model. To start the process, the shell elements making up the shear wall are deleted. With only the pier frame composed of the beam and column elements remaining, a grid of vertical and horizontal rigid members are drawn. Each of the individual elements or panels in the grid should be approximately square. This step is important and will greatly improve the inelastic analysis results. Beyond this requirement, there are no firm rules for establishing the size of the grid. It will be easier to build the model, and the analyses performed will run quicker with fewer elements used to make up the grid. However, the grid needs to be sufficiently subdivided to properly capture the inelastic behavior of the system. It may be necessary to run several analyses to be sure that the selected grid is adequate and the SAP 2000 results are converging toward a unique solution. For the SH 87 Bridge, a 4 × 6 grid for each of the two wall segments was found to be suitable. The grid is added by drawing a rigid element across the top of the wall, and dividing it by the number of grid lines in that direction. Next, the columns are divided by the number of gridlines in the horizontal direction. Finally, the vertical and horizontal rigid elements are drawn. Pictures of the pier model, before and after the addition of the rigid elements, are shown in Figure 4-33.

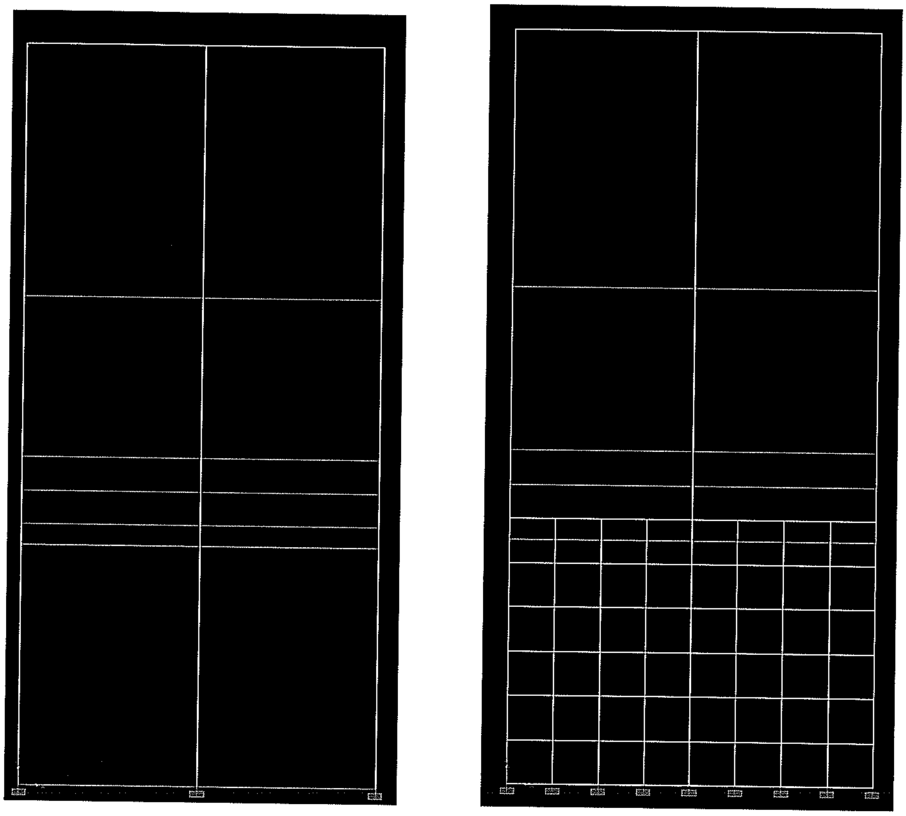


Figure 4-33. SH87 Before and After Rigid Grid Elements

Next, all of the vertical and horizontal members of the grid are divided into smaller elements at the points of intersection. Dividing elements is done under the 'Edit' menu in SAP 2000. At this stage, the rigid elements need to be classified as being pin connected to each other and to the columns, thus making the grid a truss-grid as opposed to a frame-grid. Truss elements are defined in SAP 2000 by releasing the moments at the member ends as discussed earlier.

Once the horizontal and vertical grid has been established, the diagonal truss members are drawn. Again, these members are pinned at the ends. Figure

4-34 shows the finished bridge pier geometry for the truss-grid model with the moments released on all of the interior members, as well as a 3D version of the model with shaded sections to give a better pictorial representation of the pier being modeled.

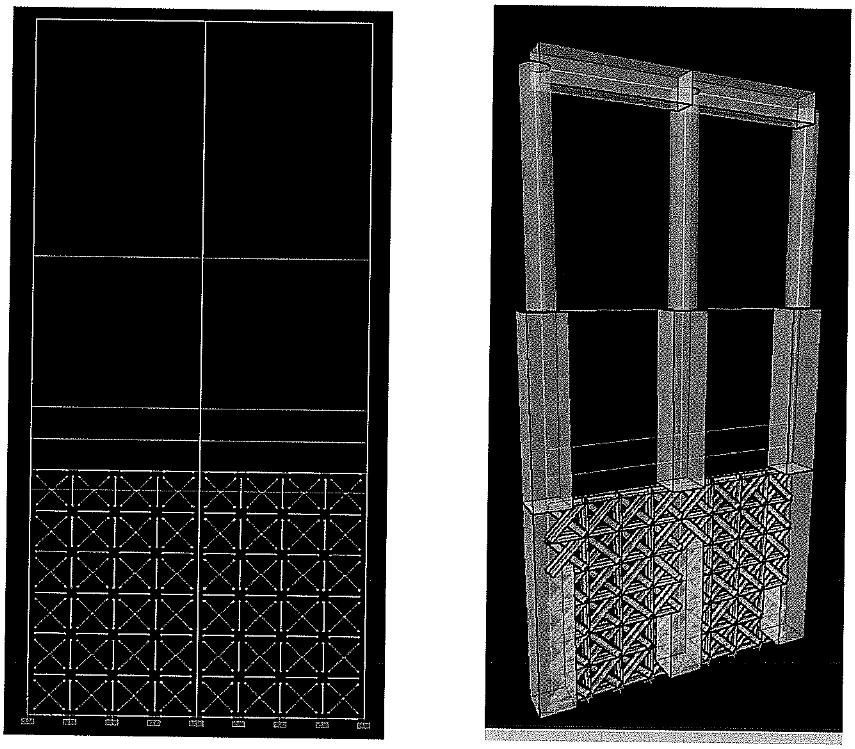


Figure 4-34. SH87 Bridge Pier with Truss-Grid Wall

Step 5: Modeling Bridge Superstructure and Adjacent Piers

The same procedure described previously in the shell wall modeling section is used to model the adjacent piers and superstructure for the truss-grid model.

Step 6: Apply Loads

The same lateral load cases described previously are applied to the truss-grid model.

Step 7: Size Truss Members

Two methods to size the diagonal truss members are presented below. The first is a more rational approach to sizing the truss members, using simple structural analysis tools. The second utilizes an iterative approach. Both methods are based on matching the stiffness of the truss-grid wall model with the initial linear elastic stiffness of the shell wall. The size of the truss members needs to be adjusted for each combination of load location and boundary conditions that one wishes to consider. For the SH-87 Bridge model, four load configurations and three top boundary conditions are being considered, so truss element sizes need to be determined for twelve cases.

Method I

The approach with this method is to determine the lateral stiffness contribution that the truss elements need to make in order to match the elastic response of the shell wall model. To start, linear elastic analysis results from the shell wall model are needed. As opposed to applying an arbitrary load, a linear elastic static 'pushover' analysis should be run, with a defined displacement limit at the load location. In this type of analysis, instead of applying a given load, SAP 2000 will increment the load up until a displacement limit is reached at a specified location. A more detailed description of a static pushover analysis is provided in Chapter 5. The SAP 2000 load output is used along with the specified displacement at the point of the load, to calculate the stiffness of the system, k = Load/Displacement.

Once the stiffness of the shell wall model is known, the stiffness of the truss-grid model needs to be determined. The stiffness of the truss-grid model depends upon the stiffness of the diagonal truss members, and the beam, column and rigid members. The size of the beams, columns and rigid members are known before the analysis begins; therefore, the stiffness of this system is known. To calculate the contribution of the beams, columns and rigid members to the overall stiffness of the system, all of the truss members are removed from the model, leaving only the columns, beams, and horizontal and vertical rigid elements. A linear elastic analysis is run for this model. Using the applied load and corresponding displacement, the stiffness of the frame and rigid elements can be determined. The difference in the stiffness of the two systems represents the required stiffness of the truss members in order for the two models to be equivalent. The required truss element stiffness is illustrated in Figure 4-35. The plot is for a typical bridge pier that is pushed so that there is a 1-inch displacement at the load location.

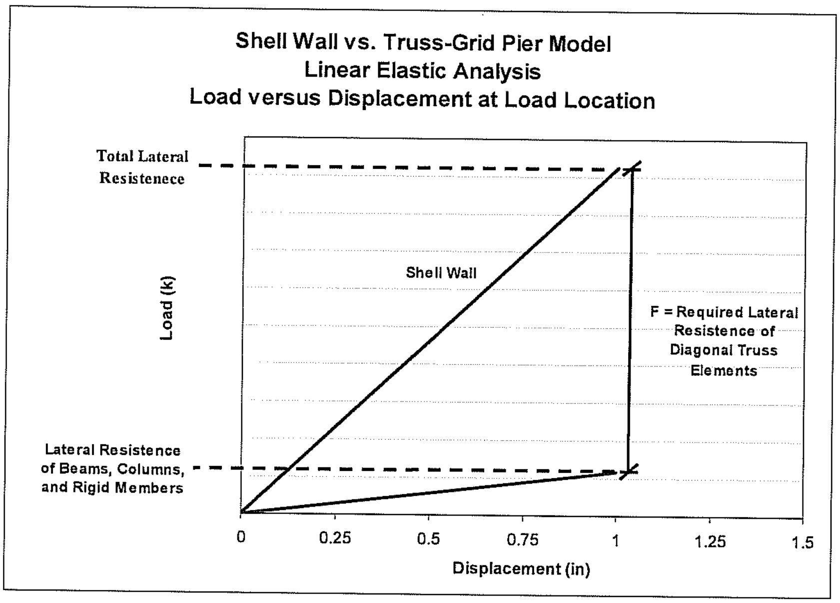


Figure 4-35. Required Stiffness of Truss Members

Once the information in Figure 4-35 is known, the required stiffness for the truss members can be calculated using known relationships between force, stiffness and displacement. For the most basic case of replacing a shell wall with a single rigid element and a single truss element, these calculations result in Equation 4-5. This equation can be used to solve for the required truss member area. Figure 4-36 illustrates the variables needed from a model to apply Equation 4-5.

$$A = \frac{F * L}{E * \Delta} \tag{4-5}$$

where

F = Total Lateral Force Truss Elements Need to Resist (see Figure 4-30)

L = Length of Wall Diagonal

E = Modulus of Elasticity of Wall Material

Δ = Displacement along Wall Diagonal

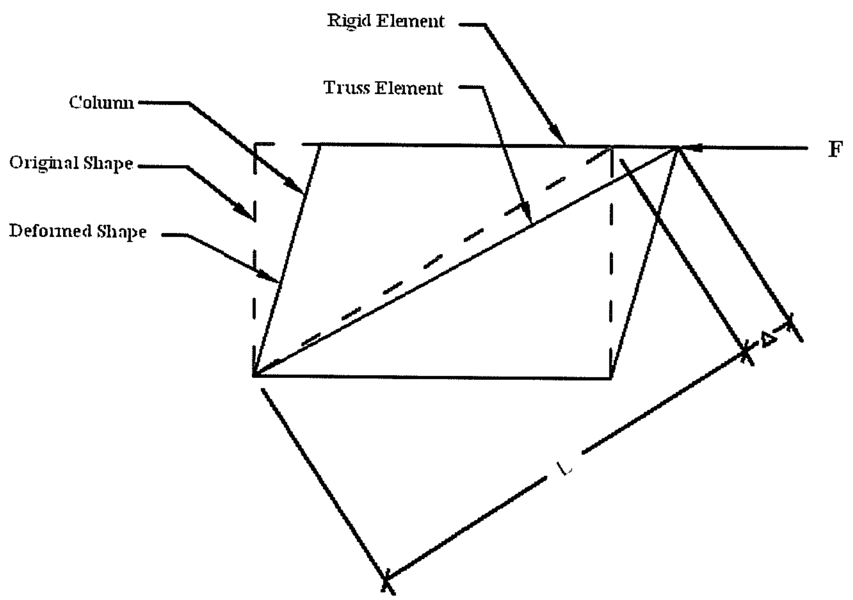


Figure 4-36. Determining Truss Size

Method I is useful in that it provides a procedure for sizing truss members that is based on basic structural analysis concepts. However, applying this method becomes more difficult when sizing truss elements that are part of a large grid, as is required for the SH-87 Bridge that is being modeled in this chapter. As the complexity of the wall and pier geometry increases, so does the difficulty of correctly sizing the truss elements. To work around this problem, a second method for sizing truss elements is presented.

Method II

Method II is essentially a guess and check approach to determining the proper size for the truss elements to match the linear elastic response of the truss-grid model to the linear elastic response of the shell wall model. This method is suggested because of its ease of use within SAP 2000. The diagonal truss members will be given an arbitrary size and an initial linear elastic analysis will be conducted for each load and boundary condition configuration. The truss member sizes will be adjusted for each load case until the linear elastic response is matched. Using this approach will result in the same size truss elements as would be found using Method I. Method II was used to size the truss elements for the SH-87 Bridge Models.

Table 4-7 summarizes the first run of linear elastic analyses on the truss-grid model. Note that the truss elements are 12-inches × 12-inches for all of the analysis cases. The size of the truss elements will be adjusted based on the results shown below, to match the analysis results presented earlier. The next step outlines this procedure.

Table 4-7. Initial Linear Elastic Analysis Results

Load Description	Top Boundary Condition	Total Load Applied	Truss Dimensions	Lateral Displacement at Point of Load
Point Load at Top of Wall	free	1000 k	12" x 12"	0.196"
Point Load at Top of Wall	fixed	1000 k	12" x 12"	0.175"
Point Load at Top of Wall	superstructure	1000 k	12" x 12"	0.184"
Point Load 48" above Top of Wall	free	1000 k	12" x 12"	0.316"
Point Load 48" above Top of Wall	fixed	1000 k	12" x 12"	0.258"
Point Load 48" above Top of Wall	superstructure	1000 k	12" x 12"	0.285"
Point Load 96" above Top of Wall	free	1000 k	12" x 12"	0.551"
Point Load 96" above Top of Wall	fixed	1000 k	12" x 12"	0.409"
Point Load 96" above Top of Wall	superstructure	1000 k	12" x 12"	0.477"
Distributed Load 30" above and below wall	free	1000 k	12" x 12"	0.196"
Distributed Load 30" above and below wall	fixed	1000 k	12" x 12"	0.175"
Distributed Load 30" above and below wall	superstructure	1000 k	12" x 12"	0.183"

Once the initial analysis results have been tabulated, the dimensions of the diagonal truss elements can be adjusted to match the previously determined linear elastic analysis results. If the results from the initial truss-grid model, with 12-inch × 12-inch truss elements, result in a smaller displacement, the truss element dimensions should be decreased. If a larger displacement is seen, the member size needs to be increased. Table 4-8 summarizes the results from this process.

Table 4-8. Adjusted Linear Elastic Analysis Results

Load Description	Top Boundary Condition	Total Load Applied	Truss Dimensions	Lateral Displacement at Point of Load	Shell Wall Model Lateral Displacement at Point of Load
Point Load at Top of Wall	free	1000 k	17.1" x 17.1"	0.108"	0.108"
Point Load at Top of Wall	fixed	1000 k	17.1" x 17.1"	0.103"	0.103"
Point Load at Top of Wall	superstructure	1000 k	17.0" x 17.0"	0.106"	0.106"
Point Load 48" above Top of Wall	free	1000 k	14.9" x 14.9"	0.235"	0.235"
Point Load 48" above Top of Wall	fixed	1000 k	14.2" x 14.2"	0.210"	0.210"
Point Load 48" above Top of Wall	superstructure	1000 k	14.5" x 14.5"	0.224"	0.224"
Point Load 96" above Top of Wall	free	1000 k	13.8" x 13.8"	0.472"	0.472"
Point Load 96" above Top of Wall	fixed	1000 k	12.1" x 12.1"	0.394"	0.394"
Point Load 96" above Top of Wall	superstructure	1000 k	13.2" x 13.2"	0.436"	0.436"
Distributed Load 30" above and below wall	free	1000 k	17.2" x 17.2"	0.107"	0.107"
Distributed Load 30" above and below wall	fixed	1000 k	17.1" x 17.1"	0.101"	0.101"
Distributed Load 30" above and below wall	superstructure	1000 k	17.1" x 17.1"	0.104"	0.104"

Step 10: Define Plastic Hinge Properties

The process within SAP 2000 to define hinge properties is the same as previously described. The truss-grid model, however, requires the use of two types of hinges. In addition to the moment-axial interaction hinges used for models without shear walls, the truss-grid model requires axial hinges. The axial hinges are placed at the ends of the diagonal truss members, and are used to capture the inelastic deformation in the wall. Moment-axial interaction hinges are used as they were before, on the beam and column members in the pier.

While the basics of defining and using plastic hinges for models with shear walls remains the same, a slightly different definition for the plastic hinge properties is used. Again, in SAP 2000, hinges are defined based on strength and deformation capacity. In Section 4.6.1, when modeling of the IH-10 Bridge (a bridge without shear walls) was discussed, the strength characteristics of the moment-axial hinges were based on the yield and plastic moment, M_y and M_p, determined by a section analysis. It is suggested that both the moment-axial hinges and axial hinges for a truss-grid model be based solely on yield moment. The hinge profile is then elastic, perfectly plastic, as opposed to the previously defined hinge, which had an area of transition from the yield to ultimate capacity. Figure 4-37 shows the two hinge definitions on the same plot, with the previous definition drawn as a dashed line.

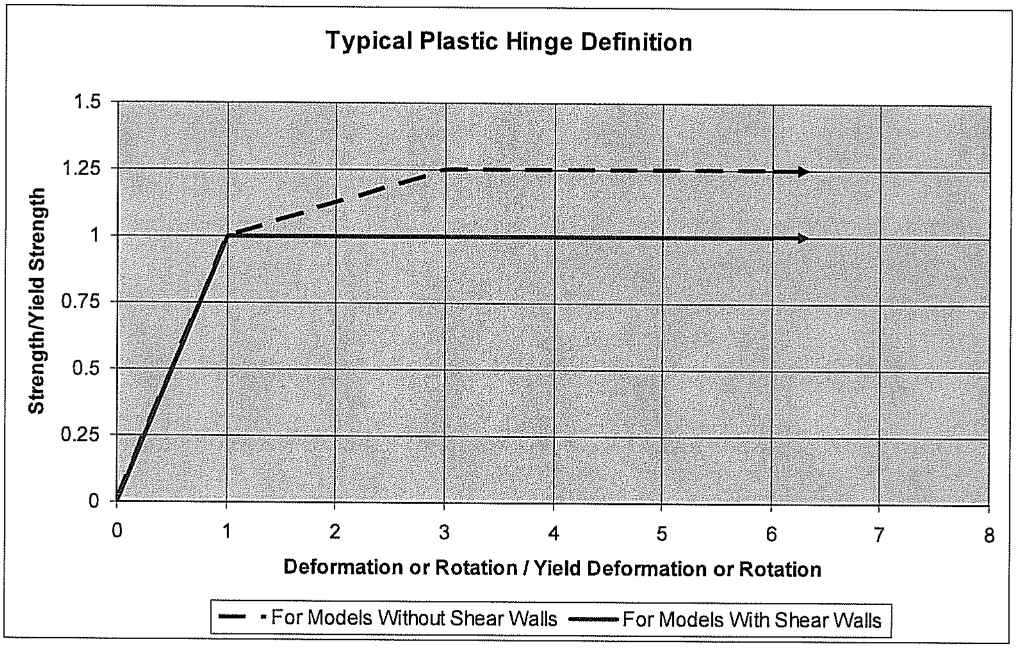


Figure 4-37. Plastic Hinge Definition Comparison

By using rigid elements in the truss-grid wall model, the forces that get transferred into the wall are distributed in a relatively even fashion throughout the diagonals in the grid. In reality, much of the inelastic behavior due to a large vessel impact force would be concentrated in a smaller local area near the point of impact. Because of the rigid members, the truss-grid model does not capture this behavior very well, and the truss-grid model is an inherently stiffer model when inelastic behavior is considered. Initial analysis results from SAP 2000 of the truss-grid pier model showed a 20-30% greater ultimate strength when compared to a finite element model analysis of the same pier with shell elements capable of plastic behavior. Coincidently, the ratio of M_p/M_y for most of the reinforced concrete sections used in the bridge piers that were examined was between 1.2 and 1.3. While the inelastic response of the truss-grid pier models result in higher (unconservative) ultimate strength results due to the modeling method, adjusting the material model provides a simple way to compensate for the error.

Step 11: Assign Plastic Hinges to Pier Elements

Plastic hinges should be assigned in the same manner as discussed in Section 4.6.1. Axial Hinges should be assigned at each end of all of the diagonal truss members. Moment-axial interaction hinges should be assigned at the ends of column and beam members, as well as at key locations along the length, such as at a load location or change in section.

4.7 Modeling Reduced Section Capacity in Area of Vessel Impact

It is likely that during a vessel collision event the area of the bridge being struck will be subject to some local crushing and spalling of the concrete due to the dynamic nature of the impact. At a minimum, it is expected that the cover concrete will be lost, and the possibility exists that some of the confined concrete could crush as well. It has been found that merely losing the cover concrete does note have a significant effect on the capacity of the section or the strength of the bridge as a whole. However, if any of the confined concrete core is lost, or if the longitudinal reinforcing bars are lost, the section capacity and bridge strength could be affected significantly. This section outlines a procedure to account for reduced section capacity within the context of the modeling approach outlined earlier. Further investigation on local behavior in the impact zone is needed in order to better estimate what kind of section loss should be considered in bridge ultimate lateral strength calculations.

For the purposes of the models and analyses contained in this report, two reduced sections are considered, a 10% loss and a 20% loss of cross-sectional area. These values are somewhat arbitrary, but are believed to be reasonable. They are being used primarily to illustrate the effect reduced sections have on overall strength if loss of section is considered. Chapter 5 examines the analysis results for models with and without reduced sections in the impact zone, and assesses the effect of a reduced section on the ultimate lateral strength. It is assumed that the concrete will be lost as shown in Figure 4-38. Examples are shown for an arbitrary loss of cross-sectional area for both circular and square column sections. The straight line assumption shown is made for ease of analysis, although, concrete is not likely to crush and spall in such a fashion.

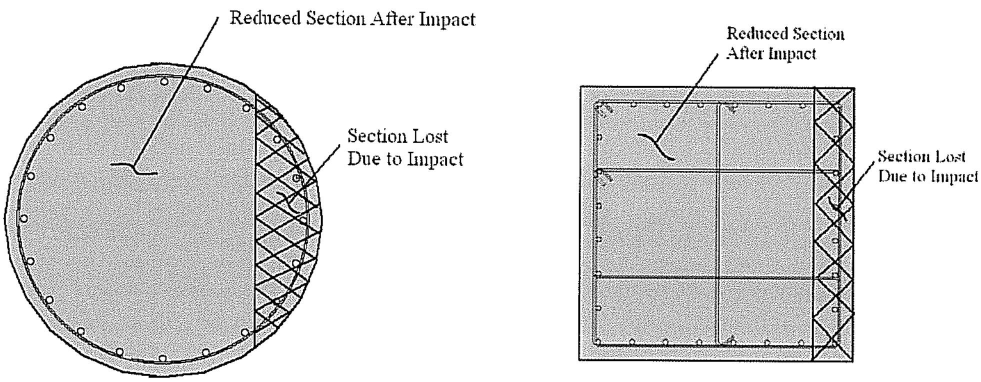


Figure 4-38. Reduced Section Shapes

To account for the reduced cross-section areas in the regions of impact, only slight modifications need to be made to the analysis steps outlined earlier.

Rather than trying to determine a reduced section shape in SAP 2000, the approach taken is to use a modified material model for elements near the impact area. The modified material properties are developed using Response-2000. Using Response-2000, the geometry of the section is modified to reflect the reduced cross section. This process involves removing any concrete and steel that falls within the lost area. New values of M_y and M_p are taken from the section analysis output. Figure 4-39 shows screen captures from Response-2000 that illustrate this procedure. Based on these values and the original section properties, f_y and f_u are determined in the same manner as previously described. Table 4-9 shows the original and the reduced section properties for the top column section of the IH-10 Bridge.

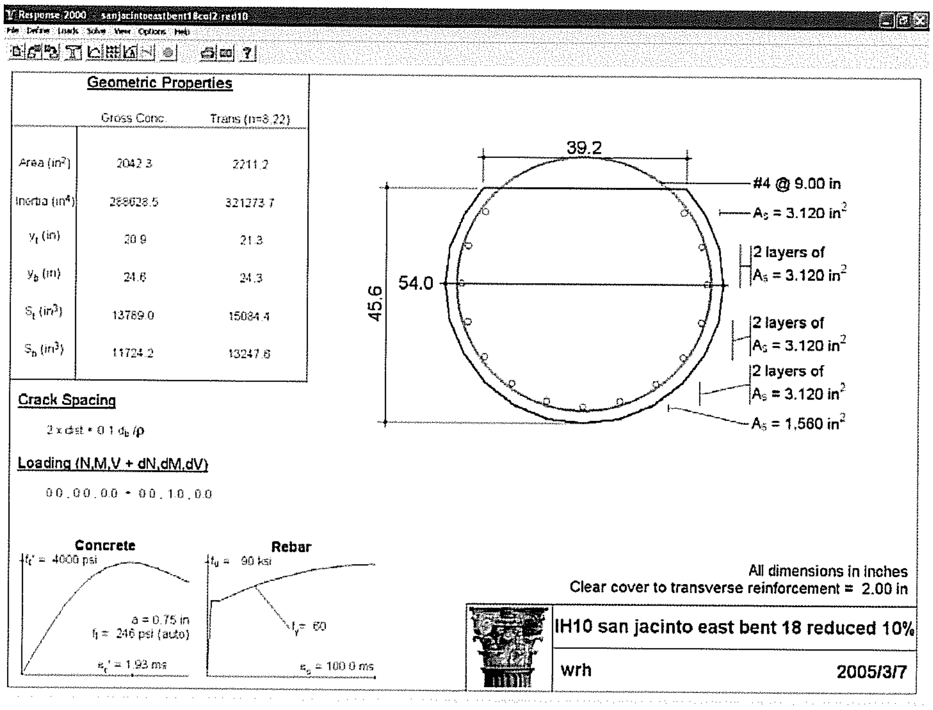


Figure 4-39. Reduced Section Input Response-2000

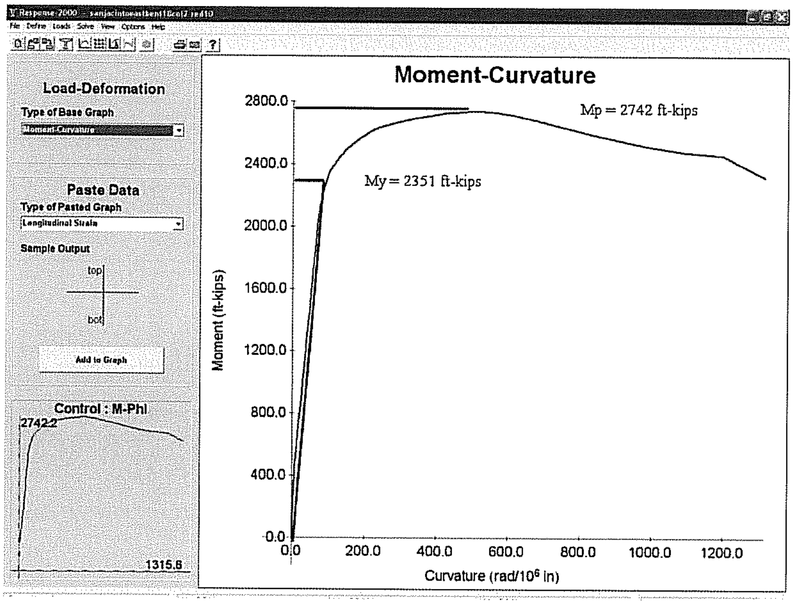


Figure 4-40. Reduced Section Analysis Results in Response-2000

Table 4-9. Full and Reduced Section Material Properties

IH10 San Jacinto Eastbound Pier 18 Top Column - Full Section Properties			
Basic Section Properties			
diameter (in)	54.00		
Plastic Modulus (in^3)	26244.00	Section Modulus (in^3)	15458.99
bars	20-#11	stirrups	#4-9" pitch
Response 2000 Section Analysis			
My (in-kips)	34572.00	fy (ksi)	1.32
Mp (in-kips)	42576.00	fu (ksi)	1.62
		fu/fy	1.23

IH10 San Jacinto Eastbound Pier 18 Top Column - 10% Section Reduction			
Basic Section Properties			
diameter (in)	54.00		
Plastic Modulus (in^3)	26244.00	Section Modulus (in^3)	15458.99
bars	20-#11	stirrups	#4-9" pitch
Response 2000 Section Analysis			
My (in-kips)	28212.00	fy (ksi)	1.07
Mp (in-kips)	32906.40	fu (ksi)	1.25
		fu/fy	1.17

IH10 San Jacinto Eastbound Pier 18 Top Column - 20% Reduced Section			
Basic Section Properties			
diameter (in)	54.00		
Plastic Modulus (in^3)	26244.00	Section Modulus (in^3)	15458.99
bars	15-#11	stirrups	#4-9" pitch
Response 2000 Section Analysis			
My (in-kips)	22950.00	fy (ksi)	0.87
Mp (in-kips)	26493.60	fu (ksi)	1.01
		fu/fy	1.15

After the reduced section material properties have been computed, new section properties, material models and hinge properties are defined in SAP 2000. Table 4-9 shows that the section geometry is not changing, but an identical section is defined with the new material model that reflects the reduced section properties. Figure 4-40 shows the new material model definition in SAP 2000 and Figure 4-41 shows the new section definition.

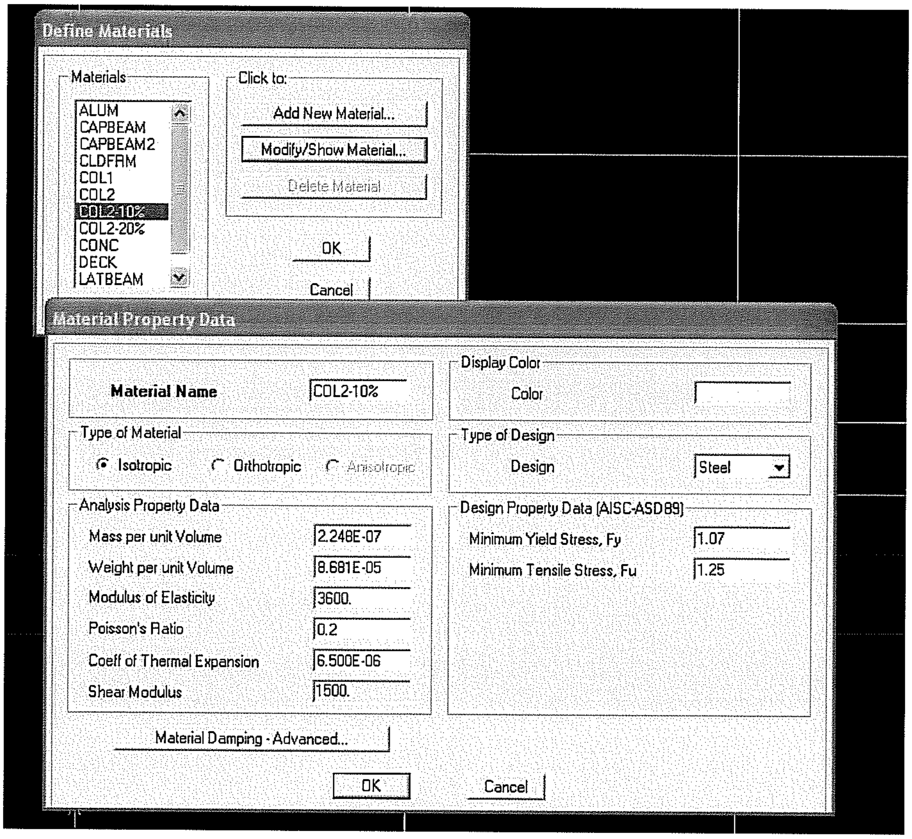


Figure 4-41. Defining Reduced Section Material Properties in SAP 2000

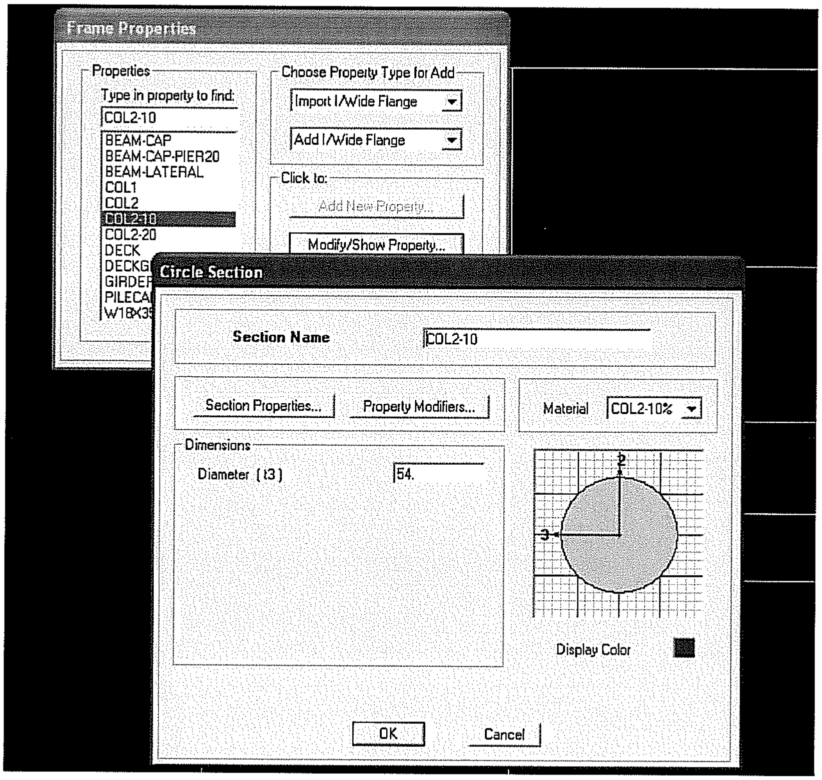


Figure 4-42. Reduced Section Definition in SAP 2000

Next, the new material and hinge properties are assigned to the elements adjacent to the area where the impact load is being applied. The location of impact loads are discussed in greater detail in Chapter 5. It is assumed that impact will result in a lost section over a depth that is equal to the length of the vessel bow. The elements within this region are assigned the reduced section properties. After section properties have been assigned, reduced section hinge properties need to be added as well. The last two steps can be seen in Figure 4-42.

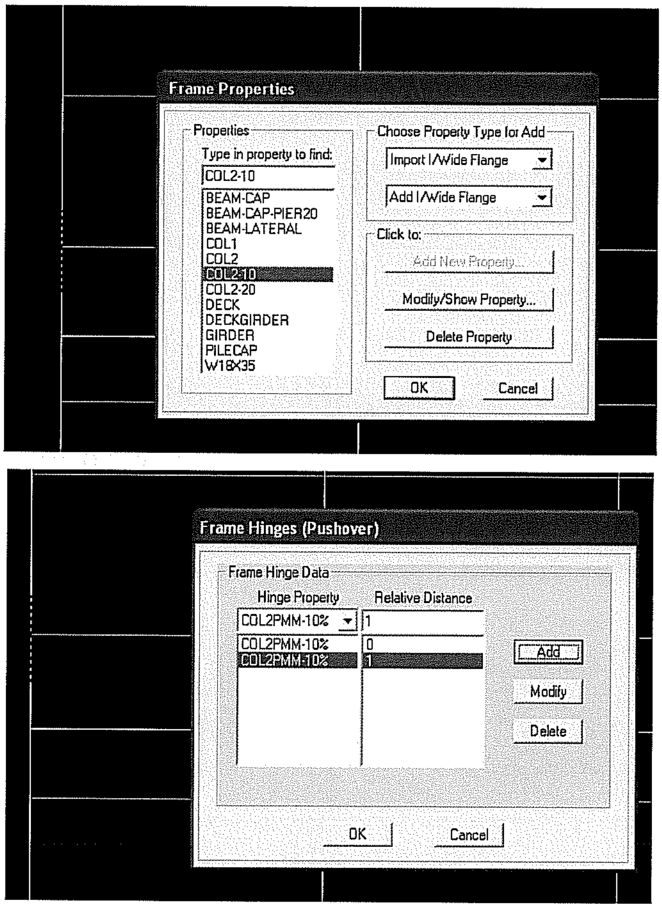


Figure 4-43. Assigning Reduced Sections and Hinges to Elements in SAP 2000

If a column or other structural member loses material due to impact, it is expected that there will be a gradual transition from the area with the most severe damage to an area where the full section is still intact. Therefore, when considering the effect of a larger loss of section, it may be necessary to phase this effect in over several elements. For example, if it is estimated that 20% of the section will be lost due to impact, the appropriate material, section and hinge properties for a 20% section reduction are applied to the elements that are immediately adjacent to the impact point. The next two elements on either side are given properties associated with a smaller reduction in section, for example, 10%. The two elements beyond this location are assumed to have the full section present. This approach can easily be implemented in SAP 2000 by dividing the member of the pier being struck into several elements. Figure 4-43 illustrates the idea of gradually changing the properties of elements around the point of impact.

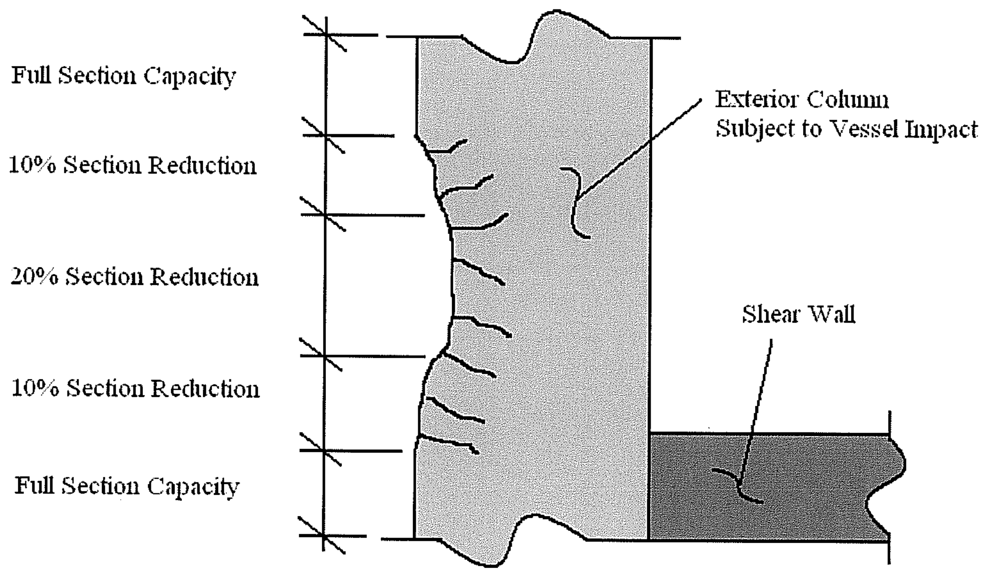


Figure 4-44. Gradual Change in Properties for Reduced Section

4.8 LIMITATIONS

In using the approximate methods for bridge modeling presented in this chapter, it is important to understand the limitations that these models have. By recognizing when these models will provide an accurate assessment of bridge lateral ultimate strength, and more importantly, situations where the models' response fails to capture the actual behavior of the system being analyzed, they can be used effectively. The following list describes some of the important assumptions and limitations of the models outlined in this chapter:

- *Assumed fixed base condition.* This assumption ignores the behavior of the bridge pier foundation and the soil-structure interaction and results in a model that has a greater stiffness then the actual structure.
- *Truss-grid model forces inelastic behavior to be evenly distributed through the wall.* It does not capture local response of the wall in the area of an impact as well as a non-linear finite element analysis would. This issue is dealt with indirectly by changing the hinge definition as shown in Step 10 of Section 4.6.2.
- *Hinges are defined as being nearly infinitely plastic.* A plastic hinge region in a real structure will have a rotational or deflection limit. This issue will be addressed in the post-analysis phase.
- *Use of smeared material properties.* While the guidelines presented are for reinforced concrete bridge piers, the material is not being modeled directly. The smeared material model approach, based on a section analysis and the conservative assumption of zero axial load, should provide a reasonable representation of a real reinforced concrete pier, but this assumption does represent a possible source of error.

In many cases, the limitations in these models can be overcome by making some slight modifications or making additional assessments in the post-analysis phase. Additional investigation could also serve to eliminate the some of the potential issues presented in the list above. For example, soil-structure interaction could be captured in SAP 2000 by using frame elements to represent the foundation and springs to represent the surrounding soil. However, such an investigation is beyond the scope of this document.

4.9 SUMMARY

This chapter has outlined general modeling techniques that can be used in accordance with simple structural analysis programs to calculate the ultimate lateral strength of bridge piers, both with and without shear walls, subject to vessel impact loads. The guidelines presented allow for strength calculations based on both the individual pier and the entire bridge system. In addition to the general modeling guidelines, step-by-step procedures for two representative bridges from the state of Texas were presented. Chapter 5 outlines the necessary loads and analysis cases for these models and presents the ultimate strength analysis results, along with finite element verification models.

CHAPTER 5

Analysis of Bridge Ultimate Strength Models

5.1 INTRODUCTION

The bridge models constructed in Chapter 4 are intended to be analyzed using a nonlinear static analysis in SAP 2000. This chapter outlines the analyses needed to determine the ultimate lateral strength of the SH-87 and IH-10 bridge models. A discussion of how to assess the analysis results follows, with an emphasis on determining which parameters control the limit state of the bridge models. The focus is on strength and ductility limit states with some additional discussion of structural stability. Finally, the analysis results for the IH-10 Bridge and the SH-87 Bridge are presented. The primary goals of the analyses are to evaluate the validity of the truss-grid model, and to determine the effect of considering system-wide response in analyzing a bridge for ultimate lateral strength. Additional results consider the effects of a reduced section in the area of impact as well as the loss of an exterior column in a multi-column bridge pier.

All of the goals outlined above fall within one of the primary objectives of this document, which is to provide bridge engineers with the necessary tools to accurately assess the ultimate lateral strength of a bridge pier, both as an individual element or as part of a larger bridge system, for use within the existing AASTHO Vessel Collision Design Specification. Currently, AASHTO does not provide any guidance in calculating the capacity of a bridge element and this report seeks to address that limitation. In addition, by improving confidence in the calculation of bridge ultimate lateral strength, the opportunity exists to examine critically the probability of collapse term in the current AASHTO Method II vessel collision annual frequency of collapse equation.

5.2 Applied Loads

The structural analyses for the models built in Chapter 4 need to be carried out in multiple steps. The initial analysis considers the effects of loads that are already on the structure prior to vessel collision. The existing, or in-place loads, come primarily from the self weight of the bridge itself. Once the effects of these loads are known, a lateral load representing vessel impact is applied as a static load case. Both point load and distributed load configurations are considered. The distributed load is applied over a length that is intended to represent the contact area dimensions of a vessel striking a bridge element.

5.2.1 Existing Loads on the Structure

As discussed previously, AASHTO currently defines bridge ultimate lateral strength as the strength of an individual element, either a pier or span. In analyzing a single element, little consideration is likely to be given to the existing loads on the structure and AASHTO makes no mention of these loads. In analyzing the bridge system as whole, it makes sense to consider the in-place loads. At a minimum, the self weight of the bridge superstructure is present during a vessel collision event, and it should be accounted for when determining the ultimate lateral strength of a bridge system. For the IH-10 and SH-87 Bridge models built in Chapter 4, the self weight of the bridge deck and girders were determined from construction drawings. An additional 20% of this load was added to account for any other superimposed loads on the structure that could not be estimated from the bridge plans. This estimate of the additional load is believed to be conservative. A lower value could be used if a more detailed investigation were conducted. Table 5-1 summarizes the existing loads on piers 18 & 19 of the IH-10 Bridge and piers 21 & 22 of the SH-87 Bridge.

Table 5-1. Existing Loads on Bridge Models

IH-10 Piers 18 & 19	
Cap Beam Geometry	
Width (in)	48
Depth (in)	84
Length (in)	654
Volume (in^3)	2,636,928
Existing Pier Loads	
Cap Beam Self Weight (kips)	229
Superstructure Self Weight (kips)	1799
Superstructure Load to Each Pier (kips)	645
Additional Loads (kips)	129
Total Load (kips)	1003
Material Properties	
Old Unit Weight (k/in^3)	8.681E-05
New Unit Weight (k/in^3)	3.803E-04

SH-87 Piers 21 & 22	
Cap Beam Geometry	
Width (in)	51
Depth (in)	54
Length (in)	504
Volume (in^3)	1,388,016
Existing Pier Loads	
Cap Beam Self Weight (kips)	120
Superstructure Self Weight (kips)	2108
Superstructure Load to Each Pier (kips)	723
Additional Loads (kips)	145
Total Load (kips)	988
Material Properties	
Old Unit Weight (k/in^3)	8.681E-05
New Unit Weight (k/in^3)	7.116E-04

Using the in-place loads calculated in Table 5-1, the unit weight for the cap beam of the pier is changed so that the entire in-place load is accounted for in this member. As a result, the load is evenly distributed across the member, as it would likely be in the real structure. A slightly more accurate representation could be achieved by distributing the load through the superstructure elements by changing the unit weight for these members. The load would then be transferred to the bridge pier at the points where the general elements connect to the pier. However, this task is more difficult than assigning the load directly to the cap beam because of the general section type that is being used in SAP 2000 to model the superstructure. Table 5-1 shows both the old and new unit weight values for the cap beam.

5.2.2 Impact Loads

Impact loads are applied as static loads with a small arbitrary load value, usually 1 k or 1 k/ft, assigned initially. This load is increased during the static nonlinear analysis by SAP 2000 until a limit state is reached. When performing a vessel collision design using the Method II procedure, AASHTO specifies in Section 3.14.14 that vessel impact forces should be applied either as a point load at the mean high-water level or as a uniform distributed load with a length equal to the depth of the vessel bow at the point on the pier where impact is expected given the draft of the vessel (AASHTO, 2004). When using the analysis and modeling guidelines outlined in the current chapter of this report and in Chapter 4 for use in an actual AASHTO Method II analysis, these locations should be considered for vessel impact. However, for purposes of this chapter, several impact load distributions are considered. The lateral load cases were previously described and shown in Chapter 4 (Section 4.6.1 and Figure 4-31) for the SH-87 Bridge and are reviewed below. Also listed are the loads that will be considered for the IH-10 Bridge. Figure 5-1 and Figure 5-2 show the load locations for each bridge.

For the IH-10 Bridge over the San Jacinto River:

- Load Location 1: Point load at the lateral beam
- Load Location 2: Point load at normal-water level
- Load Location 3: Point load at high-water level
- Load Location 4: 60-inch wide distributed load centered on the lateral beam

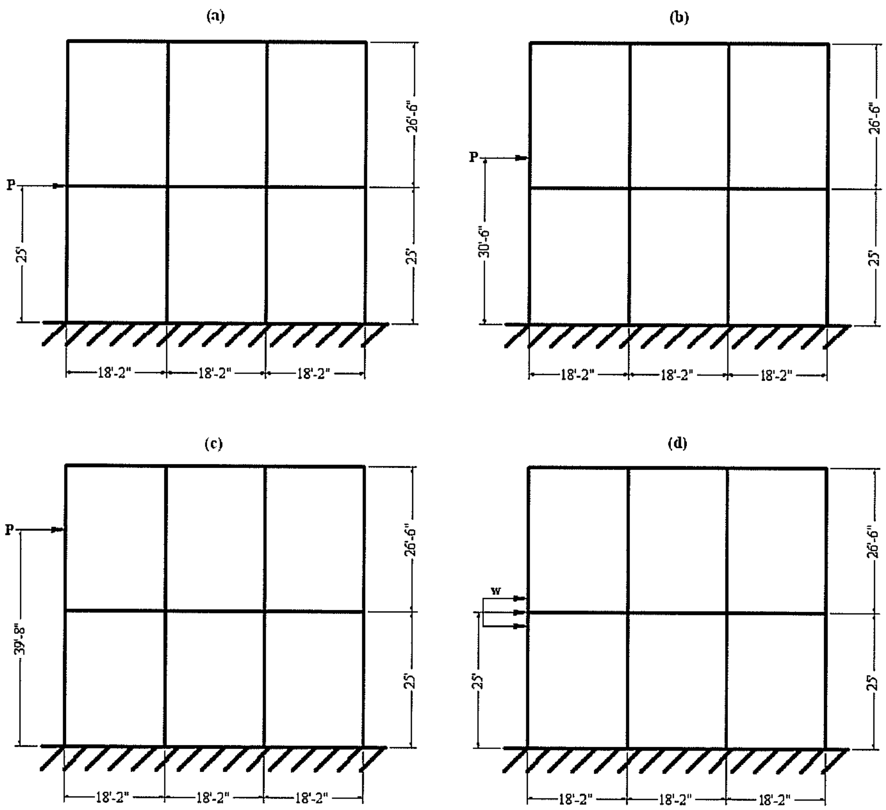


Figure 5-1. IH-10 Impact Load Locations: (a) Load Location 1: Point Load at Beam, (b) Load Location 2: Point Load at Normal Water Level, (c) Load Location 3: Point Load at High Water Level, (d) Location 4: 60-inch Wide Distributed Load at Beam

For the SH-87 Bridge over the GIWW:

- Load Location 1: Point load at the top of the shear wall
- Load Location 2: Point load 48 inches above the top of the shear wall
- Load Location 3: Point load 96 inches above the top of the shear wall
- Load Location 4: 60-inch wide distributed load centered at the top of the shear wall

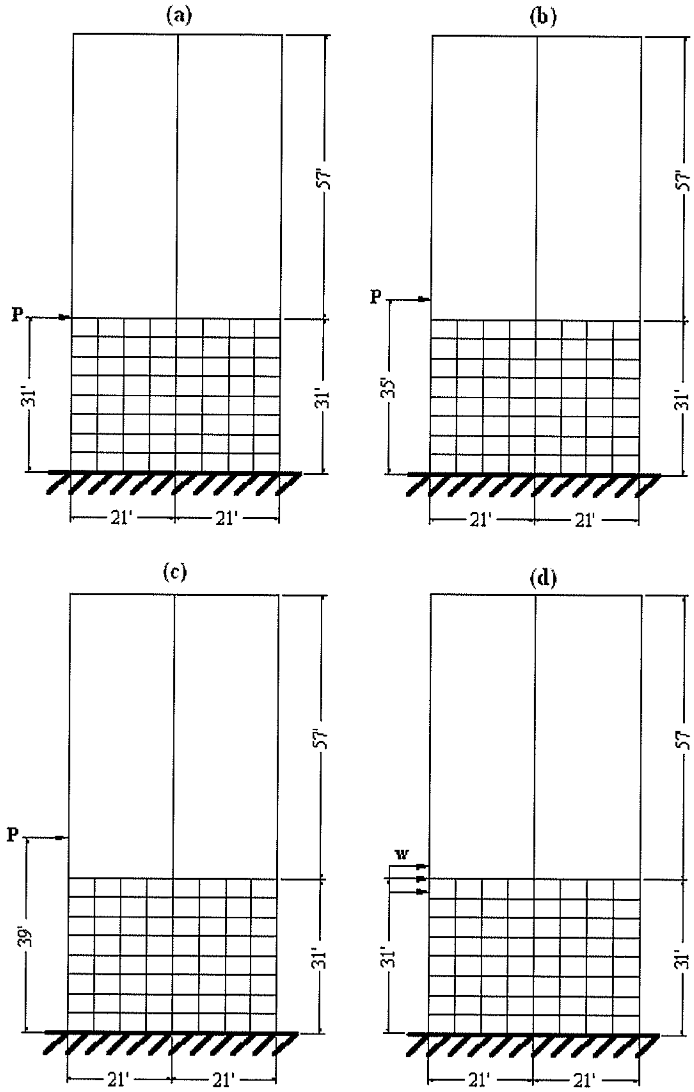


Figure 5-2. SH-87 Impact Load Locations: (a) Load Location 1: Point Load at Top of Wall, (b) Load Location 2: Point Load 48 inches Above Wall, (c) Load Location 3: Point Load 96 inches Above Wall, (d) Location 4: 60-inch Wide Distributed Load at Wall

The cases that were chosen represent a reasonable range of possible water levels for those specific bridges. By considering load situations outside of those that are explicitly defined by AASHTO, a greater understanding of how to better design bridges can be gained. In addition, considering a range of load locations could yield information on when it is appropriate to use simplified models that do not require representing the entire bridge system for acceptably accurate results.

5.3 Analysis Cases

The primary analysis case for assessing the ultimate lateral strength of the bridges modeled in Chapter 4 is a nonlinear static pushover analysis. The basics of a pushover analysis are straightforward. The load on a structure is increased in user-defined increments, and the displacement at a specified point is calculated for each of the load increments. In general, the displacement is tracked at the point where the load is applied, although any point of interest could be used. The analysis stops when a specified load or displacement limit is reached.

There are a wide range of possible outputs from a SAP 2000 pushover analysis. Of greatest interest is a load-displacement curve, which plots the total lateral load versus the displacement at a user-defined point. The bridge models built in Chapter 4 are being 'pushed' into the inelastic range. Thus, the resulting load-displacement curves show an initial slope for the linear elastic range, and as different areas of the model reach their strength limit, plastic hinges form and the slope of the curve decreases. If a structure has sufficient ductility, the curves eventually plateau after a mechanism has formed. The load value at which the curve plateaus is defined as the ultimate lateral strength. A load versus displacement curve for a structure with a clearly defined ultimate strength plateau is shown in Figure 5-3. It is also possible that some sort of structural instability could occur before a mechanism has formed. Recall that the hinges in Chapter 4 were defined as being nearly infinitely plastic. In order to determine if a structure has adequate ductility to reach the strength plateau, additional assessments need to be made. It is necessary to consider strength, stability and ductility when assessing the ultimate strength of a bridge pier or bridge system.

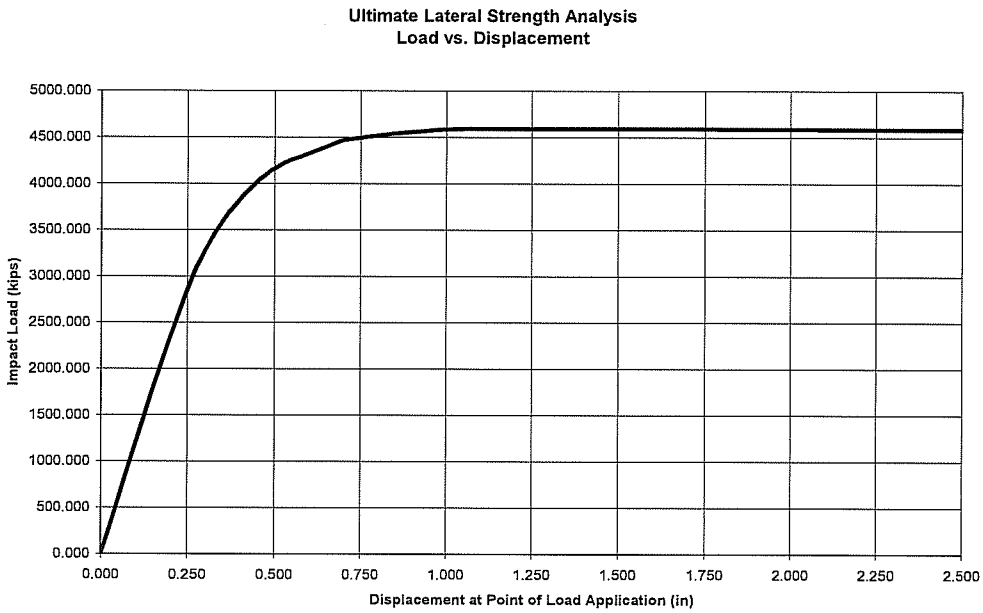


Figure 5-3. Determining Ultimate Strength from a Load versus Displacement Plot

Additional analysis cases take the initial pushover analysis a step further by assessing if a bridge can redistribute forces if a single column in a multi-column bent is destroyed. This analysis is carried out using some of the special features of a SAP 2000 static nonlinear analysis. The procedure for setting up the various analysis cases for the models built in Chapter 4 are presented in the next section.

5.4 Ultimate Lateral Strength Analysis in SAP 2000

The following section outlines the necessary steps to set up a nonlinear static pushover analysis in SAP 2000. As an example, the analysis setup for the SH-87 Bridge modeled in Chapter 4 is shown. The steps shown outline the process for considering various load configurations and make use of most of the options available within the pushover analysis feature of SAP 2000. Note that terms that appear in **bold** type represent option headings that are shown on the SAP 2000 screens, and terms that appear as *italic* type represent user input or selections.

5.4.1 Define Load Cases

A two-step process is required to set up any structural analysis in SAP 2000. First, load cases need to be defined, and then the load cases need to be assigned to an analysis case. Any number or type of loads, in any direction, can be applied for each load case. It is possible to have all of the loads on a structure applied under one load case. When load cases are assigned to an analysis case, however, only a single scale factor can be applied for all the loads in the load case. Therefore, it is often easier to define multiple load cases based on the type (dead load, live load, wind load, etc.) of load that is applied. For the bridge models analyzed for the current study, two load cases are considered. They are shown in the **Define Loads** box in SAP 2000 shown in Figure 5-4. The first case considers loads that are likely to be present during vessel impact. This case is called *DEAD*, because it is primarily dead load from the superstructure. Note that the self weight multiplier is set to *1* for this load, because all of the in-place loads have been captured by adjusting the material property for the cap beam of the pier. If the self weight factor were zero, the in-place loads would be ignored. The second load case, called *IMPACT,* contains the static lateral load that represents vessel impact.

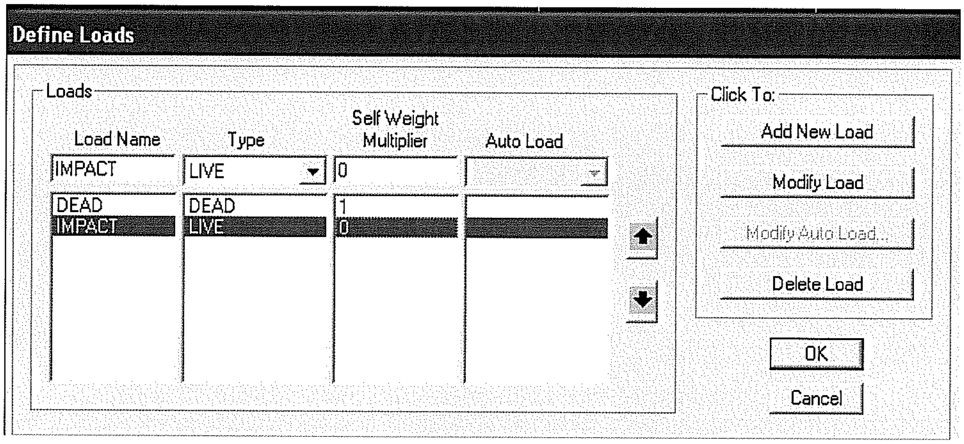


Figure 5-4. Defining Load Cases in SAP 2000

Once all of the load cases are defined, individual loads can be assigned to their appropriate case. For this problem, no loads are directly applied to the *DEAD* case, because the in-place loads are captured by the beam self weight. Loads need to be assigned to the *IMPACT* case however. As an example, Figure 5-5 shows a 100-kip point load being applied to the top of the wall in the SH-87 Bridge model.

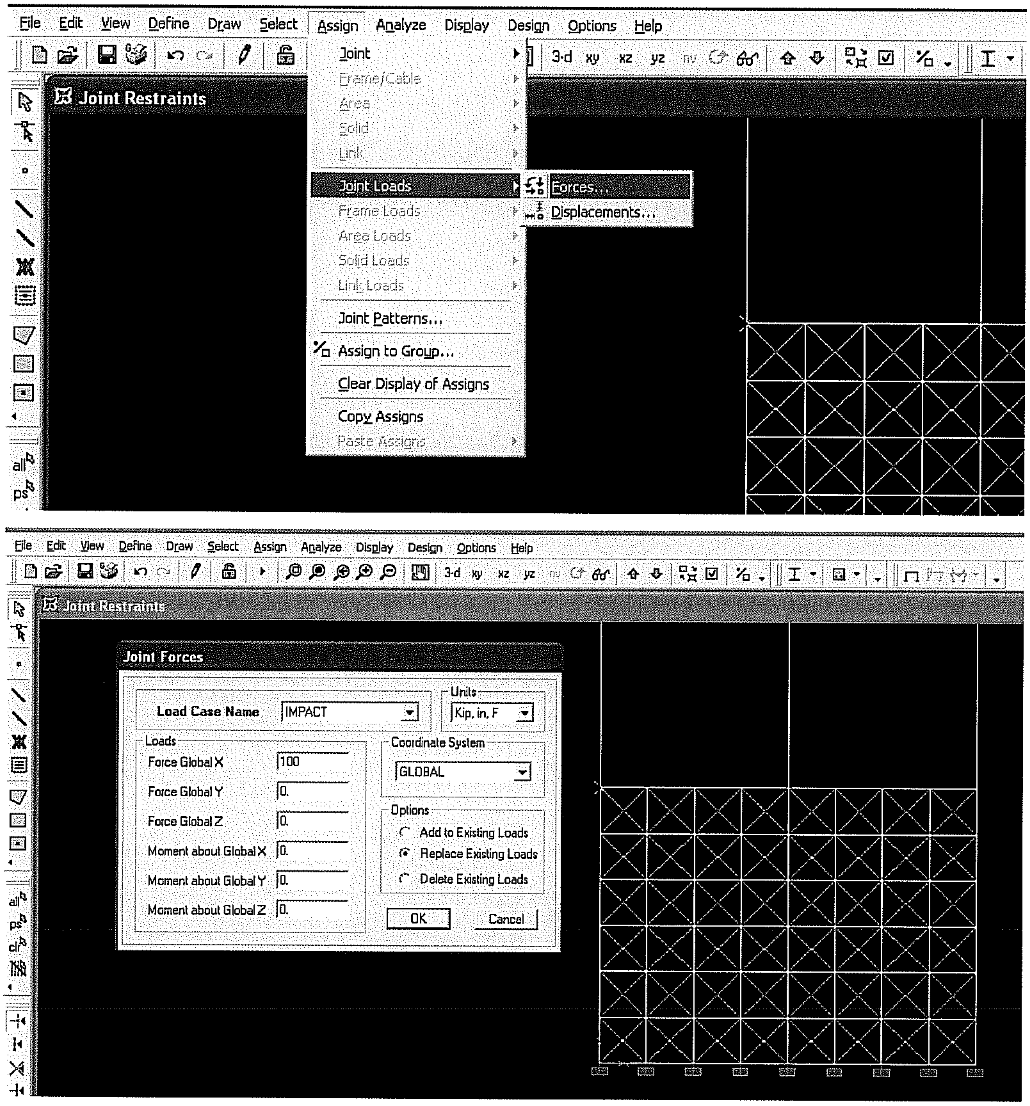


Figure 5-5. Assigning Loads to Load Cases in SAP 2000

5.4.2 Define Analysis Cases

After loads have been applied to a model in SAP 2000, analysis cases need to be defined. SAP 2000 is capable of a variety of analysis types, including dynamic analyses, as well as buckling and modal analyses. In keeping with the goal of providing a simple, user-friendly approach to determining the ultimate lateral strength of bridges, only static load cases are used with the bridge models built in Chapter 4. This approach also fits within the framework of the existing AASHTO design specifications, which uses a series of equations to express dynamic impact loads as equivalent static loads. Calculating the ultimate strength of a bridge element or system based on a static analysis provides a consistent basis on which to compare the strength of the structure being analyzed to the impact load calculated in the AASTHO Method II design procedure.

If a more detailed investigation, outside the parameters of the AASHTO Method II procedure, were desired, a dynamic response factor could be applied to a static load solution for prediction of ultimate lateral strength under impact loads. Another approach to make a consistent comparison between applied impact loads and ultimate lateral strength of a bridge would be to take a dynamic load profile from an impact event and convert that loading to a static equivalent load. Detailed discussions of those options are beyond the scope of this thesis.

For the SH-87 and IH-10 Bridge Models, both static linear and static nonlinear analysis cases are needed to determine the ultimate lateral strength. Figure 5-6 shows the analysis cases for the SH-87 Bridge. Note the options to add, delete, or modify analysis cases on the right side of the pop-up window.

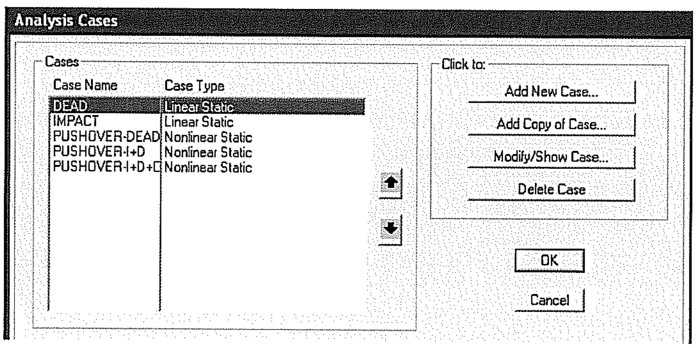


Figure 5-6. Defining Analysis Cases in SAP 2000

The analysis cases shown in Figure 5-6 are the actual cases that are used to determine the ultimate lateral strength of the SH-87 Bridge model from Chapter 4. A total of five cases are shown, two linear static cases, and three nonlinear static, or pushover cases. The linear static cases are automatically created by SAP 2000 for each of the load cases that were defined earlier. In addition, SAP 2000 requires a linear elastic analysis for each of the load cases that are included in a nonlinear static pushover, as is the case with the *DEAD* and *IMPACT* load cases. The next two sections outline the options that are available within each type of analysis case that is run.

5.4.2.1 Linear Static Analysis Options

A linear static analysis is the least complicated case to run in SAP 2000. Because of this fact, there are limited options a user can change. Figure 5-7 shows how to define a linear static analysis in SAP 2000. The options shown on the right side define the type of analysis being run. Under **Analysis Case Type,** *Static* is selected, and under **Analysis Type,** *Linear* is chosen. The left side of the box shows the options that can be modified based on the selections made for **Analysis Case Type** and **Analysis Type**. For a linear static analysis, only the **Loads Applied** options can be modified by changing which loads are applied and the corresponding scale factor for each load. All of the loads applied to Chapter 4 bridge models are selected to have a scale factor of *1*.

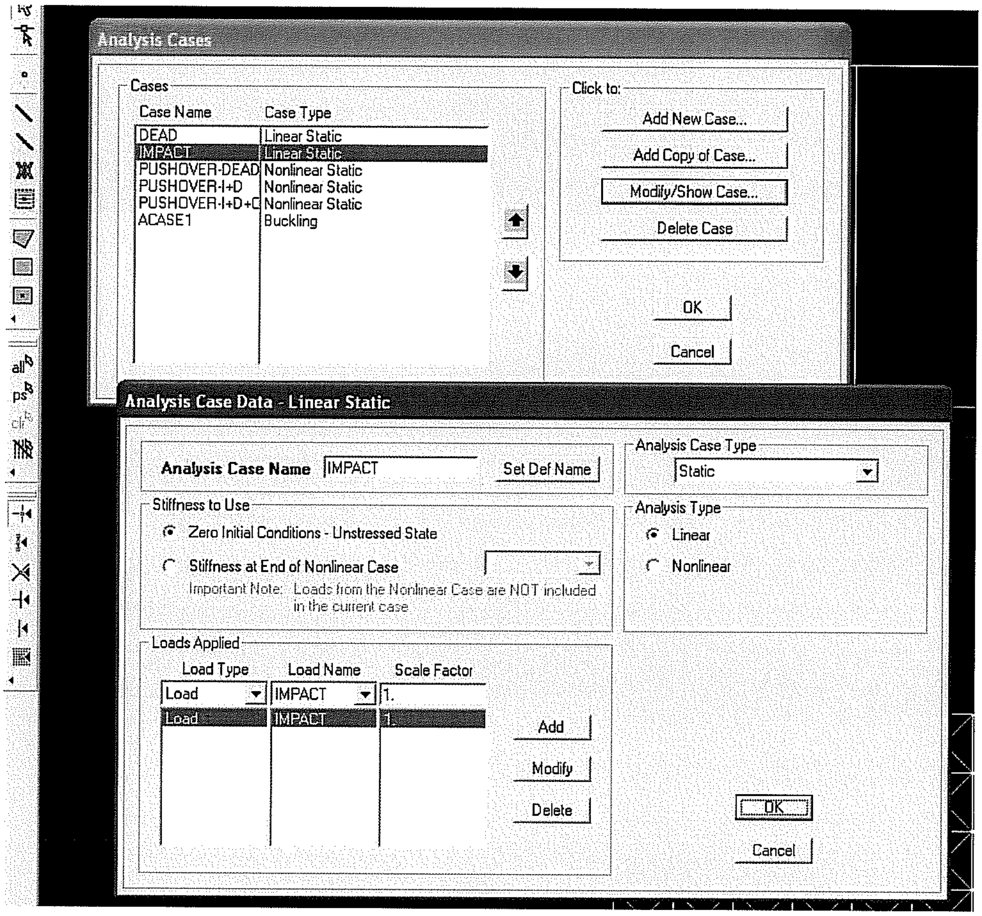


Figure 5-7. Defining Linear Static Analysis in SAP 2000

5.4.2.2 Nonlinear Static Pushover Analysis Options

Defining a nonlinear static analysis requires the specification of significantly more input parameters than a linear static analysis. The basic settings that are used for analysis of the SH-87 and IH-10 Bridges are presented in SAP 2000 screen shots below, but they only represent the settings that were used for those particular models. The values and options selected may need to be adjusted for each individual model. In addition to the screen shots, a brief explanation of the analysis settings is presented, but that discussion is somewhat limited. For more detailed descriptions of the nonlinear static pushover analysis options, it is recommended that the SAP 2000 user manual be consulted (SAP 2000, 2002).

Figure 5-8, a screen shot from the SH-87 Bridge model, shows the settings for a nonlinear static analysis case called 'PUSHOVER I+D'. This analysis case is defined to capture the effects of both the dead and impact loads on the SH-87 Bridge. The 'PUSHOVER I+D' case is described in greater detail in Section 5.4.3.2 of this report. Note on the right side of the box that the **Analysis Case type** is set to *Static,* and the **Analysis Type** is set to *Nonlinear*. The left side of the box in Figure 5-8 shows the options that can be changed for a static nonlinear analysis. Three groups of options can be adjusted, **Initial Conditions, Loads Applied** and **Other Parameters**.

Two options exist under the **Initial Conditions** box. The first option is to run the analysis from an unstressed state, or with *Zero Initial Conditions*. The second option is to run an analysis that continues from the end of a previous nonlinear analysis case. Selecting the second option allows a user to perform an analysis on a structure that has already been stressed in some fashion. For example, the dead load on a bridge could be applied before impact is considered. Once the analysis with the dead load has been run, a second case can be run with the impact loads. Figure 5-8 shows that for the 'PUSHOVER I+D' analysis case, the **Initial Conditions** box is set to *Continue from State at End of Nonlinear Case—'PUSHOVER-DEAD'*.

Below the **Initial Conditions** settings are the **Loads Applied** options. The settings here are the same as previously described for a static linear analysis. The desired loads and corresponding scale factors need to be specified. This step is also shown in Figure 5-8.

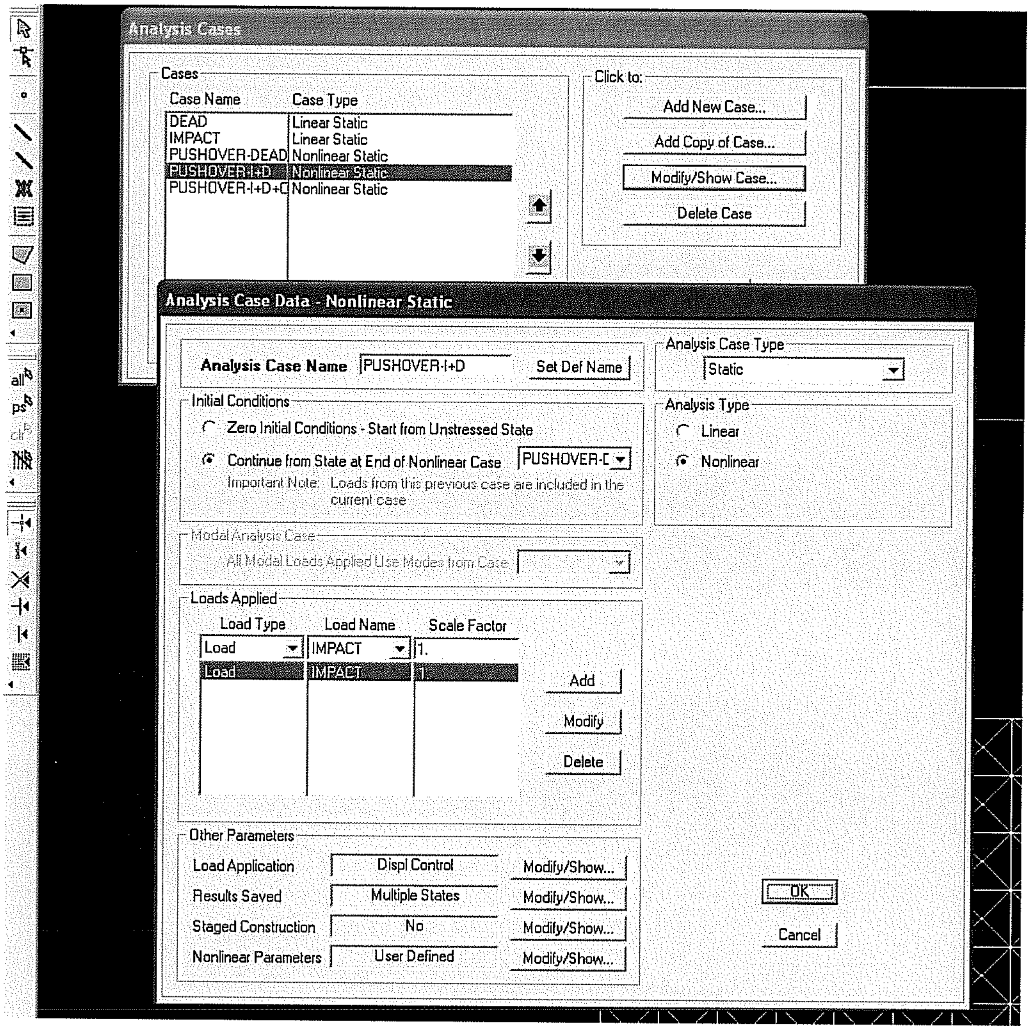


Figure 5-8. Nonlinear Static Analysis Options

The third group of settings for a nonlinear static analysis are listed under the heading **Other Parameters**, which is broken into four sets of options: **Load Application, Results Saved, Staged Construction** and **Nonlinear Parameters**. There are separate input boxes for each of these options which are accessed by selecting the *Modify/Show* buttons shown in Figure 5-8. Each of the four options listed above is described in more detail, along with additional screen captures, in the paragraphs below.

Figure 5-9 shows the **Load Application** options for a nonlinear static analysis. Nonlinear static pushover analyses can be controlled in one of two ways: by specifying either a maximum load to be applied to a model, or by specifying a maximum displacement at a given point that a model can reach. Using the *Full Load* option, SAP 2000 takes the applied loads, sub-divides them and applies them in user-specified increments until the entire load has been applied to the model. With the *Displacement Control* option, the load applied to the model is automatically increased until a specified displacement is reached at a specified location.

If the *Displacement Control* option is used, the displacement limit needs to be entered. The limits are set under the **Control Displacement** option shown in Figure 5-9. The models in this report use the *Monitored Displacement* option. When using the monitored displacement option, it is also necessary to specify which joint in the model the displacement should be tracked and in what direction the displacement limit should be enforced. For the example case shown in Figure 5-9, the analysis is set to run until a *4*-inch displacement limit is reached at joint *221* in the *U1* direction. This case represents a 4-inch lateral displacement at the point of impact for this model. In general, for the analysis cases that consider impact loads, tracking the lateral displacement at the point of impact is of the most interest, although other cases may be considered as well. The appropriate joint number is determined by looking at the model drawing in the regular SAP 2000 window.

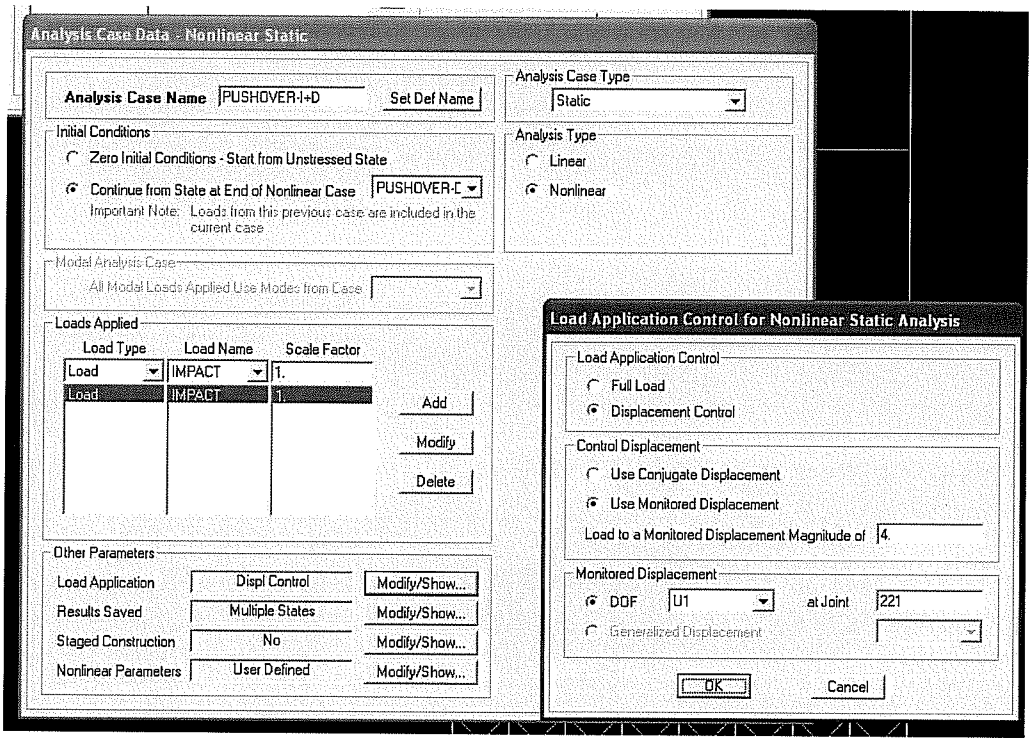


Figure 5-9. Nonlinear Static Analysis Load Application Option

Next, the **Results Saved** options need to be set (Figure 5-10). Two primary options are available: saving only the final results of the nonlinear static analysis, or saving the results for each step of the analysis. If *Multiple States* is selected as is shown in Figure 5-10, the minimum and maximum number of analysis steps must be specified. These values indicate the smallest and largest increment of load or displacement that each step can be. The initial increment that SAP 2000 uses is based on the minimum number of steps specified by the user. For example, if a 4-inch displacement limit is set with a 100-step minimum and 400-step maximum, SAP 2000 saves the analysis results at increments of 0.04-inch to start and adjusts the increments based on the computed results. In carrying out the analysis, it uses at least 100 steps and no more than 400 steps.

Using larger values for the number of saved steps results in greater solution accuracy, but it also results in longer analysis run times.

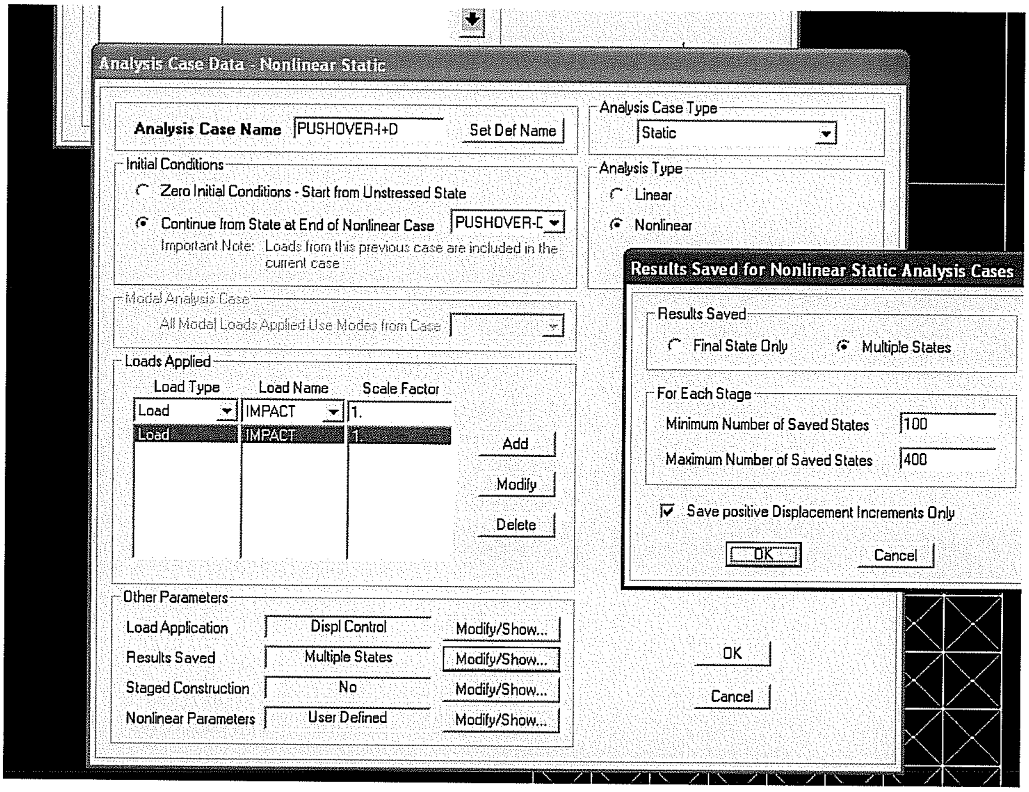


Figure 5-10. Nonlinear Static Analysis Results Saved Option

Figure 5-11 shows the **Staged Construction** options for a nonlinear static analysis. The staged construction option allows a user to add or remove specific groups of elements in a model. This feature is used to consider the effect on the bridge as a whole of losing an exterior column in a multi-column bridge pier. This option is used by selecting whether elements are to be added or removed from the model. Next, the group of elements to add or remove is selected. Groups are defined from the *Assign* menu in the main SAP 2000 window.

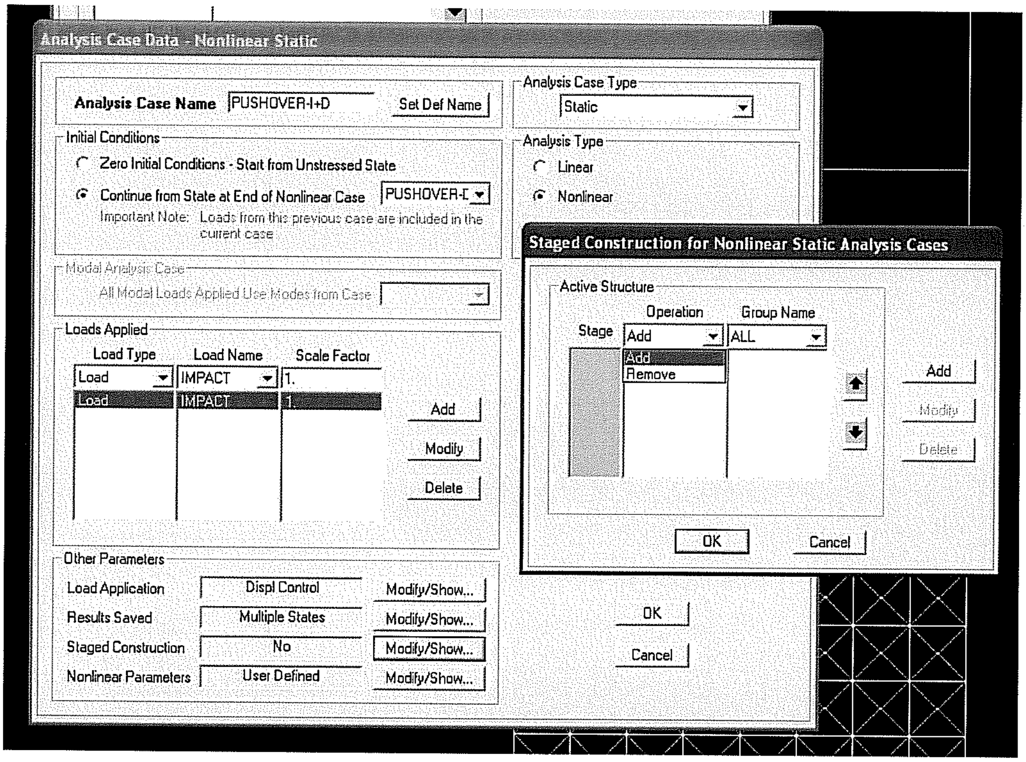


Figure 5-11. Nonlinear Analysis Staged Construction Options

The final group of options to set in a SAP 2000 nonlinear static analysis is the **Nonlinear Parameters**. In the **Nonlinear Parameters** box, the **Solution Control** settings, **Geometric Nonlinearity Parameters** and the **Hinge Unloading Method** parameters are set. Example input data are shown in Figure 5-12. Simple descriptions of the settings in Figure 5-12 are provided below. A detailed description of the items in the nonlinear parameter box can be found in the SAP 2000 user manual (SAP 2000, 2002).

Under the **Geometric Nonlinearity Parameters**, the *P-Delta* option is selected. This option indicates that, in the analysis of a model, the equilibrium equations are set up and solved in the deformed shape. Considering this effect usually results in larger member forces and displacements. There are several guidelines that can be used to determine whether geometric nonlinearity or 'P-Delta' effects need to be considered in an analysis. Generally, this determination can be made based on the magnitude of the lateral displacement relative to the overall length of the structure from a first-order analysis. For the specific bridges modeled in this report, geometric nonlinearity was not found to have a significant effect on the results. It is quite possible, however, that for other bridge geometries, P-Delta effects could be important. Therefore, it is recommended that this option be considered in a SAP 2000 analysis. The *P-Delta plus Large Displacements* option is intended primarily for SAP 2000 models that use frame elements to model cables (SAP 2000, 2002).

The **Hinge Unloading Method** is set to *Unload Entire Structure*. This setting is recommended by SAP 2000. The **Solution Control** inputs are used to set the tolerance for solution convergence and the maximum number of steps that can be used to get a model to converge at any point of the analysis. If the solution at any point does not converge, the analysis will terminate before the displacement or load limit is reached. There are a variety of reasons for the analysis not converging, ranging from some instability in the structure to a numerical solution problem. The input values shown in Figure 5-12 were found to produce good results for most of the models being analyzed in this chapter. It is strongly recommended, however, that the SAP 2000 user manual (SAP 2000, 2002) or other reference be consulted to learn more about nonlinear analysis settings.

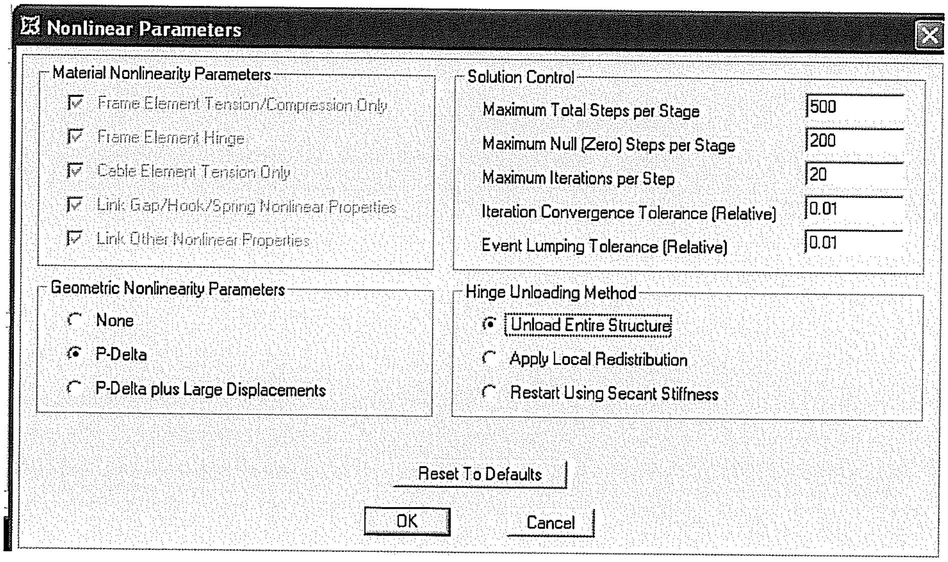


Figure 5-12. Nonlinear Static Analysis Nonlinear Parameter Options

5.4.3 Analysis Cases for Determination of Bridge Ultimate Lateral Strength

This section describes the three nonlinear static analysis cases that are used to analyze the bridge models from Chapter 4. Each of the three analysis cases builds on the previous case. The specifics of the nonlinear static analysis settings, described in detail in Section 5.4.2.2, are outlined for each of the three nonlinear static analysis cases.

5.4.3.1 Nonlinear Static Pushover Analysis – Dead Load

The first analyses of the SH-87 and IH-10 Bridge models capture the effects of the in-place or existing loads on the structure. This analysis case is called 'PUSHOVER-DEAD' and can be seen in Figure 5-6. Figure 5-13 shows the basic nonlinear static analysis settings for this case. The analysis is run with zero initial conditions. The only load applied is the dead load case, which captures the in-place loads on the structure. The analysis is run until the full load has been applied. The staged construction feature is not used. Similar nonlinear parameters as those shown in Figure 5-12 are used. Notice that this case is a nonlinear static analysis case even though it is not expected that the structure will behave inelastically. It is necessary to run the analysis as a nonlinear static case, however, in order to use the results from this analysis for later cases.

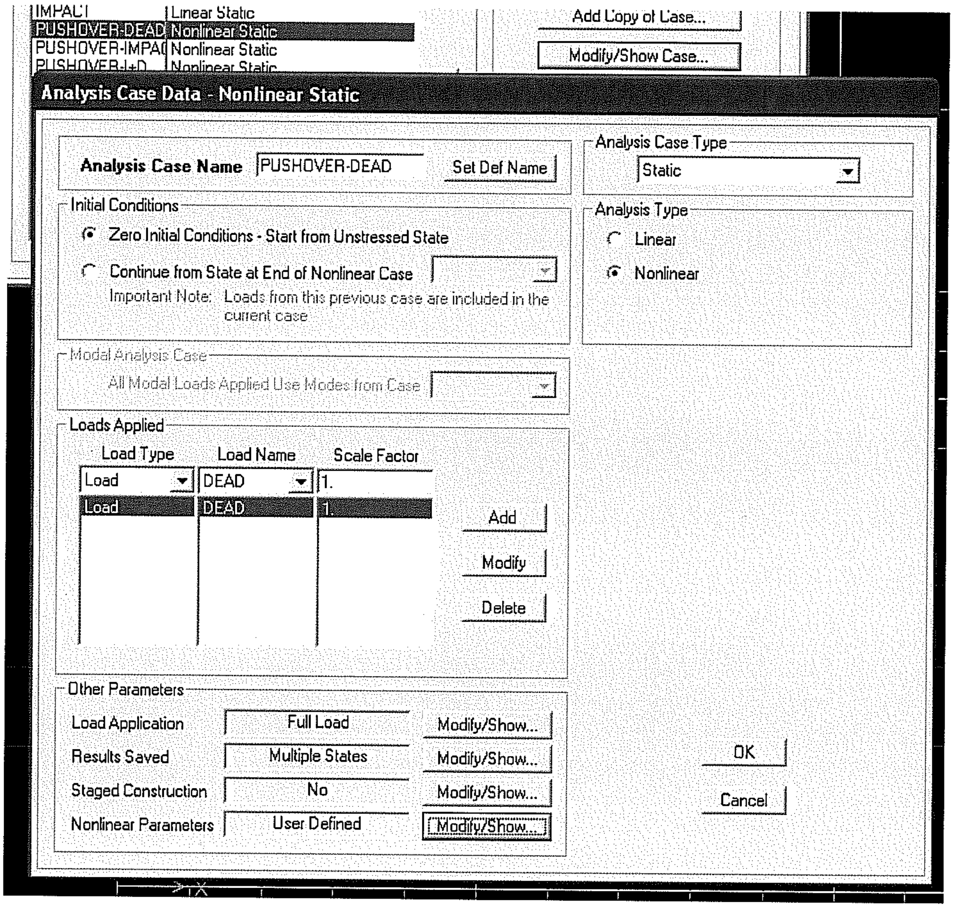


Figure 5-13. Nonlinear Static Pushover Analysis – Dead Load

5.4.3.2 Nonlinear Static Pushover Analysis – Impact + Dead Load

The second analysis case, called 'Pushover-I+D', starts with the conditions at the end of the 'PUSHOVER-DEAD' analysis. A lateral load is applied to represent vessel impact, and the analysis is run. Figure 5-14 shows the nonlinear static analysis settings for this case. This analysis is controlled by a specified displacement at the point that the load is applied. The displacement limit needs to be entered by the user. There are no specific rules for determining this value, so some adjustments may be required. The displacement limit needs to be large enough so that the load versus displacement plot reaches a plateau, representing a mechanism in the model. If the limit is too large though, the analysis may not yield an accurate solution, or may not be able to converge to a solution. Several iterations on the displacement limit may be required. For the impact load case, multiple steps are saved in order to plot the load versus displacement at the end of the analysis. The staged construction feature is not used. Similar nonlinear parameters as those shown in Figure 5-12 are used.

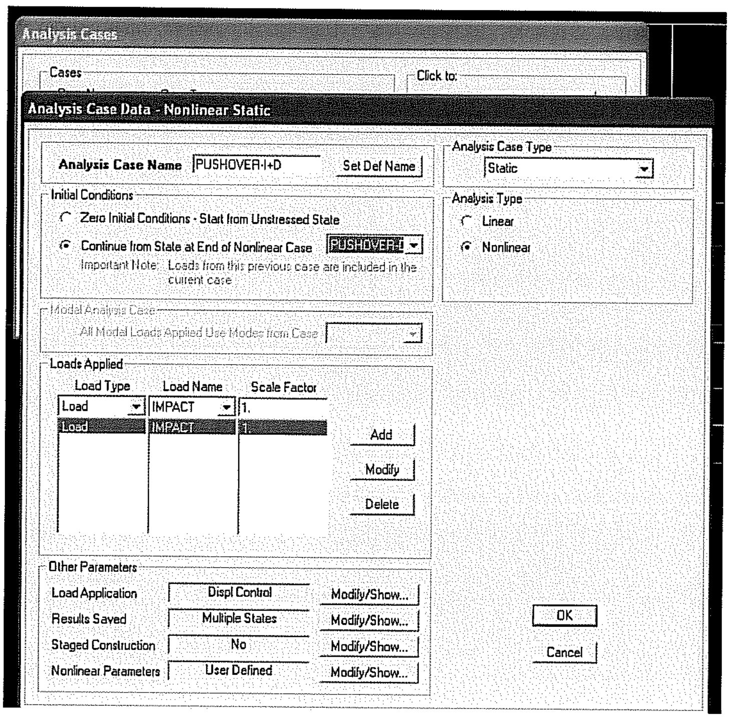


Figure 5-14. Nonlinear Static Pushover Analysis – Impact + Dead Load

5.4.3.3 Nonlinear Static Pushover Analysis – Impact + Dead Load + Column Removed

The final analysis case continues from the end of the previous case (described in Section 5.4.3.3) and considers the effect of losing a single column due to vessel impact in a multi-column bridge pier. While the analysis of a bridge pier after a single column has been lost does not clearly fit into the current AASHTO Method II design procedure, understanding this type of analysis could prove to be useful in the design of multi-column bridge piers. The results of a column removal analysis could help an engineer to better design the other elements in a pier so that the failure of a single column does not result in a more catastrophic failure of the entire pier. An in-depth investigation into this analysis case has not been conducted. The analysis steps and results presented in this chapter for a column-removal analysis are intended to provide an introduction to the topic. Further research in this area is needed in order to draw any wide ranging conclusions.

A column removal analysis was run for a single load case for both the SH-87 and IH-10 Bridge models, which are comprised of three- and four-column bridge piers, respectively. It is assumed that a two-column bridge pier will not be able to sustain the loss of a column, so this analysis is not necessary. In addition, the column removal analysis should only be run if the impact analysis determines that a mechanism has formed in the column, which is only likely for load configurations where impact occurs directly on the column.

To consider the effects of removing a failed column from a bridge model, the staged construction feature of a SAP 2000 nonlinear static analysis is used. Figure 5-15 shows the nonlinear static analysis settings for the IH-10 Bridge model.

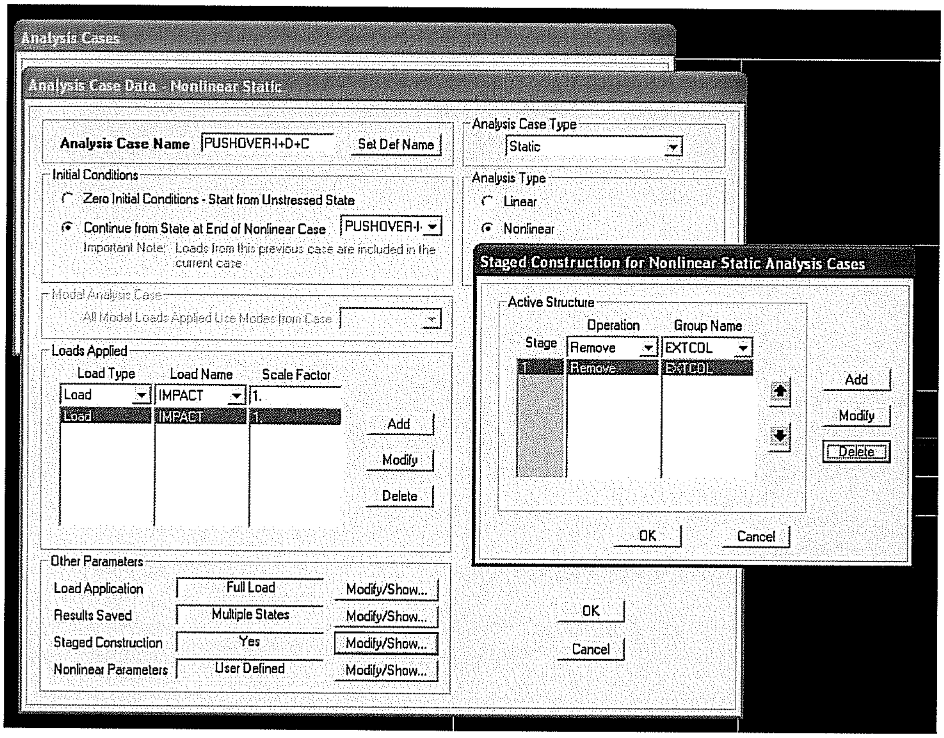


Figure 5-15. Nonlinear Static Pushover Analysis – Impact + Dead Load + Column Removed Settings

Notice that this analysis starts from the end of the PUSHOVER-I+D analysis described in Section 5.4.3.2. At this point, the in-place loads and impact loads have already been applied to the structure. For the current example, a lateral point load has been applied at mid-height of the top column in the IH-10 Bridge (Load Location 3 in Figure 5-1). Figure 5-16 shows the (exaggerated) deformed shape of the model after the nonlinear static analysis has been run for the in-place and impact loads. Notice that a mechanism has formed in the top column. At this stage the column removal analysis begins.

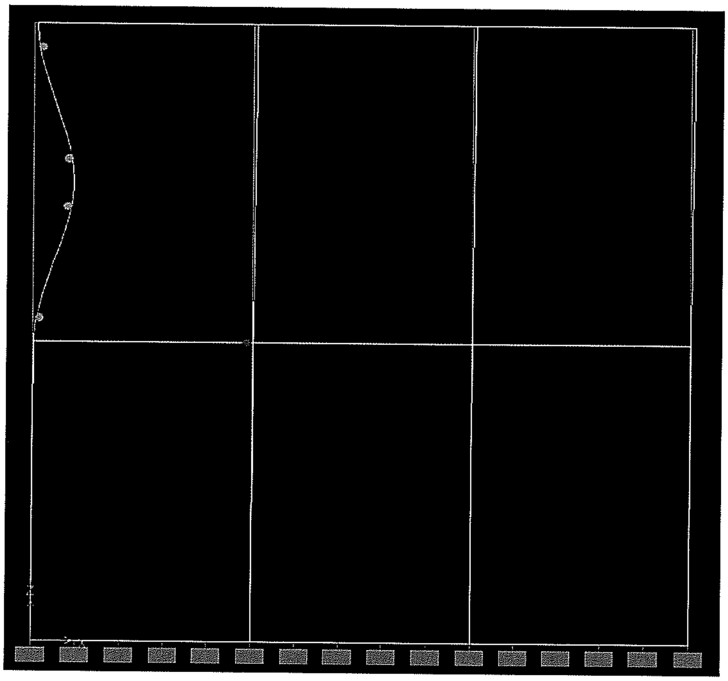


Figure 5-16. IH-10 Bridge Column Failure

The column removal analysis is run as a load control analysis, which is the only option permissible for the staged construction feature (SAP 2000 Analysis Reference, 2002). The elements from the failed exterior column in the IH-10 model were selected and assigned to a group called *extcol.* Under the **Staged Construction** option, the *extcol* group was assigned to be removed. These settings can be seen by looking at Figure 5-15. When the staged construction option is used and a group of elements is removed, SAP 2000 removes the stiffness and mass of these elements and replaces them with equivalent forces, which are reduced to zero as they get distributed through the remaining elements in the structure (SAP 2000 Analysis Reference, 2002).

5.4.4 Run Analysis

After all of the necessary load and analysis cases have been defined, the model is ready to be analyzed. Figure 5-17 shows the run options available in SAP 2000. The user can specify which analysis cases to run or can simply run all of the cases. To initiate the analysis, the *Run Now* button must be selected.

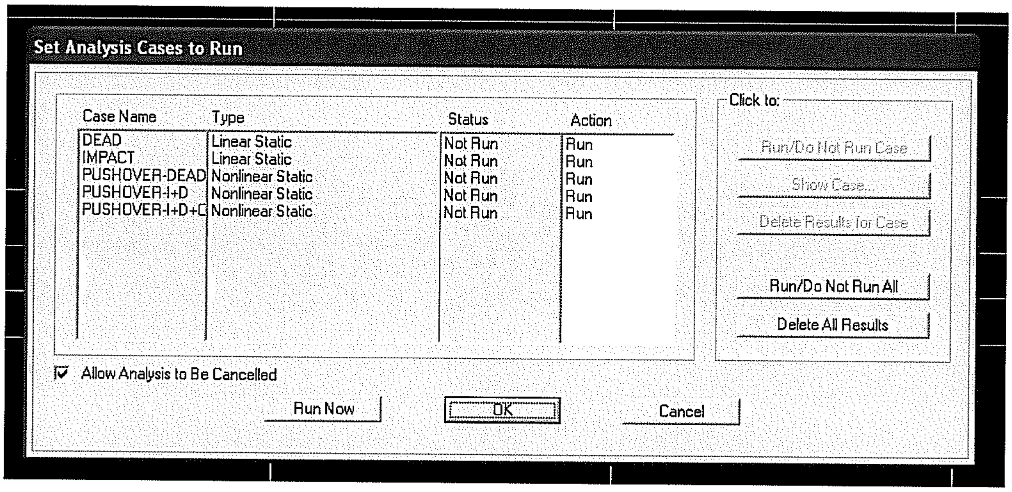


Figure 5-17. Running an Analysis in SAP 2000

5.5 ASSESSING ANALYSIS RESULTS

One of the key modeling issues discussed in Chapter 4 was the decision to define plastic hinges as being essentially infinitely plastic. A discussion of this choice can be found in Section 4.4.5. Real structures, of course, are not capable of infinite rotation or deformation after yield. A wide range of guidelines exist on ductility and displacement limits for structures subject to large lateral forces, such as earthquake, blast, and impact loads, and this information can be used to assess the rotational or deformational capacities of the bridge systems that are being analyzed in this chapter. The first method presented focuses on the overall ductility of the system and is based on the Federal Emergency Management

Agency (FEMA) NEHRP Guidelines (FEMA-302, 1997). The second method considers rotational or deformational limits for specific members and is based on guidelines published by the American Society of Civil Engineers (ASCE, 1999).

5.5.1 Limit State Based on Ductility Ratio

One possible approach is to consider a system-wide ductility limit state by assuming that the structure is capable of a set level of deformation beyond the point of first yield. This approach is appropriate for structures or systems where inelastic response is evenly distributed throughout the structure (Moehle, 1992). While vessel impact is likely to cause significant localized damage, the analysis results presented later in Section 5.6 shows that, for an impact near a wall or beam providing lateral support for a bridge pier, there is significant redistribution of forces throughout the entire system. For typically reinforced concrete structures, a ductility limit of four times the deformation at first yield is a reasonable assumption (FEMA-273, 1997). Making this assumption means that if a structure reaches a plateau in the load versus displacement curve at a displacement beyond four times the yield deformation, then the structure does not have sufficient ductility to reach that strength. Consequently, a ductility limit state controls the strength of the structure. Figure 5-18, a typical load versus displacement plot for a bridge pier, illustrates this point.

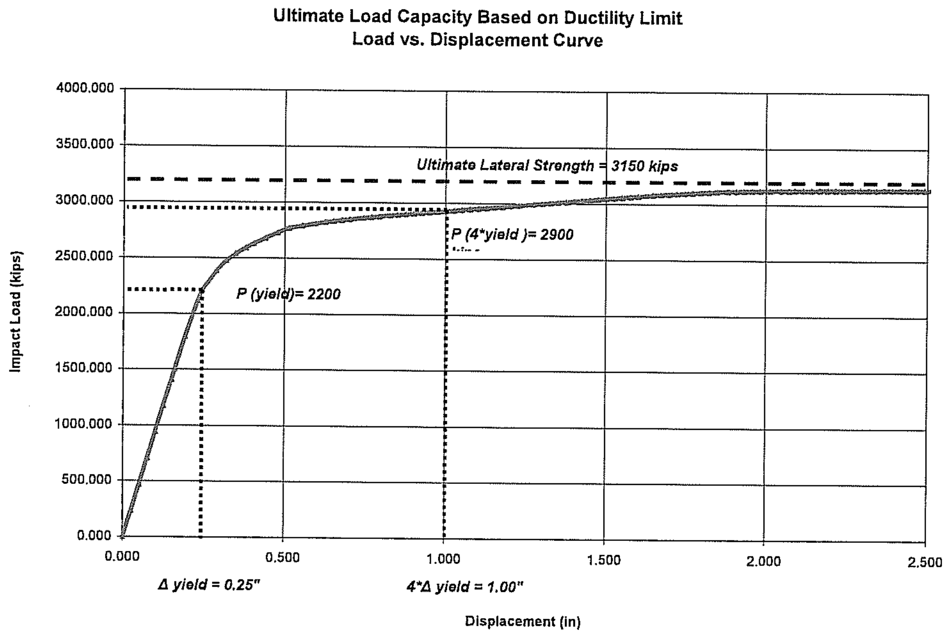


Figure 5-18. System Ductility Limit State

Figure 5-18 shows a point of first yield at a deformation of approximately 0.25 inch. At a ductility ratio of 4, the applied load is 2900 kips, which is less than the peak load of 3150 kips. Therefore, the limit state and ultimate lateral strength of this structure are controlled by the ductility of the system.

5.5.2 Member Ductility Limit

A second approach to assess the ultimate lateral strength analysis results would be to consider the rotational or deformational capacity of an individual element or member in the structure. This method is useful for situations where vessel impact is being considered at some point along the length of a column as opposed to impact at a wall or other lateral support element. In this situation, inelastic behavior is more likely to be contained within the column. The results presented in Section 5.6 illustrate this point. For typical reinforced concrete members, a mid-span displacement limit of 4% of the span length is a reasonable assumption (ASCE, 1999). The mid-span displacement limit corresponds to a rotational limit at the ends of the member of 4.57 degrees. This value can used to determine the displacement limit at a distance, x, along the length of a column by Equation 5-1 and is illustrated in Figure 5-19.

$$\Delta_{\max} = x * \sin(4.57^{\circ}) \tag{5-1}$$

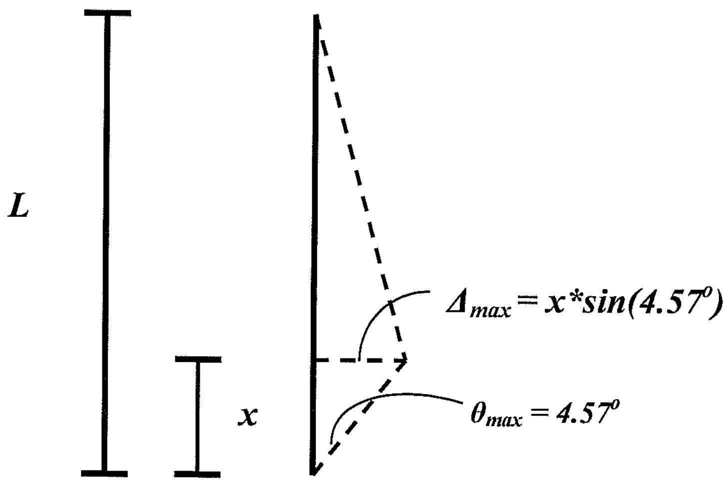


Figure 5-19. Column Displacement Limits

5.6 ANALYSIS RESULTS

This section presents the analysis results for the IH-10 and SH-87 Bridge models presented in Chapter 4. Additional results are presented to determine the effect of considering a reduced section size in the vessel impact area of a model and the effect of losing one column in a multi-column bridge pier.

5.6.1 Truss-Grid Wall Model Verification

The focus of this section is to confirm the validity of the truss-grid wall model used to capture the shear wall behavior in the SH-87 Bridge. A finite element model constructed in ANSYS provides the basis for comparison. A full description of this model follows below. The SAP 2000 truss-grid wall model and the ANSYS model are compared against each other for a range of load configurations and boundary conditions.

5.6.1.1 Finite Element Verification Models

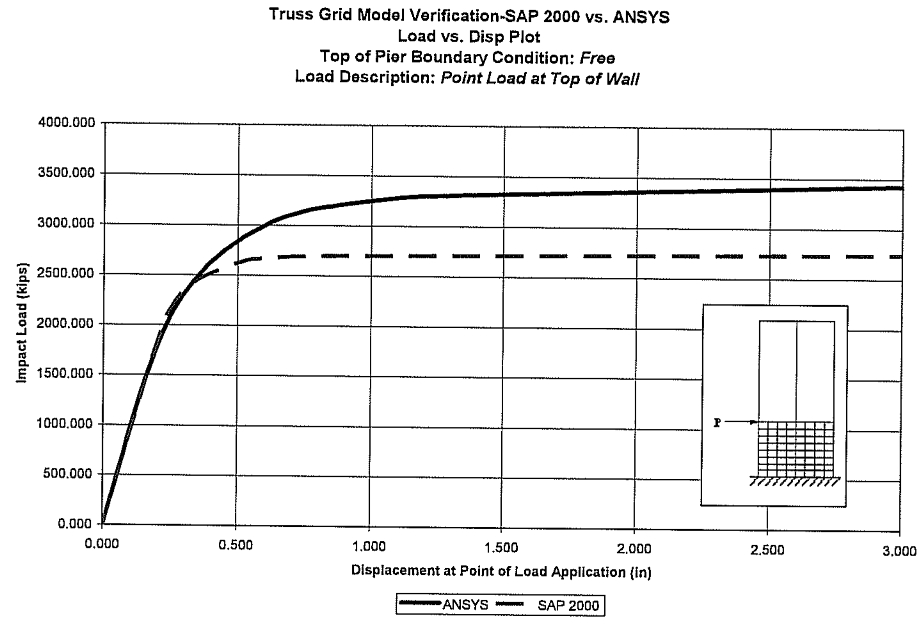
To verify the accuracy of the SAP 2000 models presented in Chapter 4, finite element models were constructed, and comparable analyses were run using ANSYS. Specifically, models were developed for the piers of the SH87 Bridge, which contain shear walls. ANSYS has the ability to capture inelastic behavior of shell elements, a feature that SAP 2000 and many other typical structural analysis programs lack. While ANSYS and other finite element analysis programs have the ability to model the response of a bridge pier or bridge system to vessel collision, they are not practical for most design situations, primarily due to their cost, both for the software package and in terms of computational time. In addition, ANSYS is not tailored directly for structural engineering use and is not as user friendly when compared to SAP 2000 or other common structural analysis programs.

The ANSYS models were used strictly to verify the inelastic behavior of the truss-grid model developed in Chapter 4 for use within SAP 2000. Other aspects of the ANSYS model were defined in a similar manner to the Chapter 4 SAP 2000 models. Columns and beams were defined using frame elements that appear as lines in ANSYS, just as they do in SAP 2000. The pier geometry, section properties and material properties were defined as they were in Chapter 4.

Identical boundary conditions were used for both sets of analyses. The base of the pier was assumed fixed for all analysis runs, and two boundary conditions at the top of the pier were considered (free and fixed). These two cases provide the range of possible strengths for the pier. Comparing the analysis results for the two extreme cases provides a clear assessment of the accuracy of the truss-grid wall model for the wide range of support conditions at the top that may be seen in real bridges. A third condition, using elements that would accurately reflect the properties of the deck and girders, was not considered No ANSYS models were constructed for the IH-10 Bridge piers because they do not contain a shear wall.

5.6.1.2 SAP 2000 vs. ANSYS Bridge Pier Model Results

The following plots compare the analysis results from SAP 2000 and ANSYS for pier 18 of the SH-87 Bridge. Results for four load configurations, with two different boundary conditions at the top of the pier for each load, are presented for a total of eight plots. They are shown below in Figure 5-20 through Figure 5-23. The title of each individual plot describes the exact load and boundary conditions for those results. Table 5-2 summarizes the plot results. A consistent approach to compare the SAP 2000 truss-grid wall model and the ANSYS shell wall model was used by comparing the ultimate lateral strength from each at the same value of displacement.



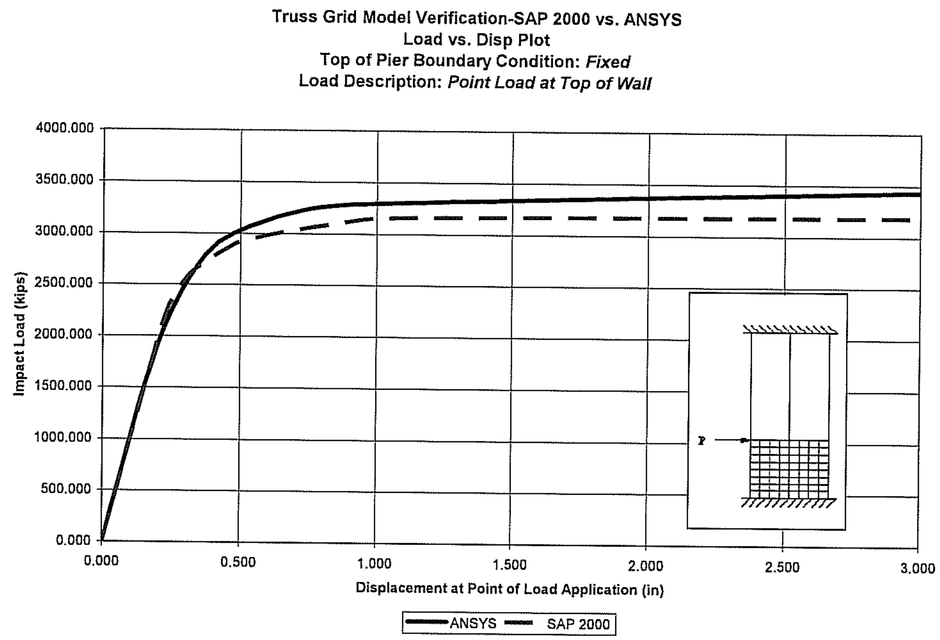
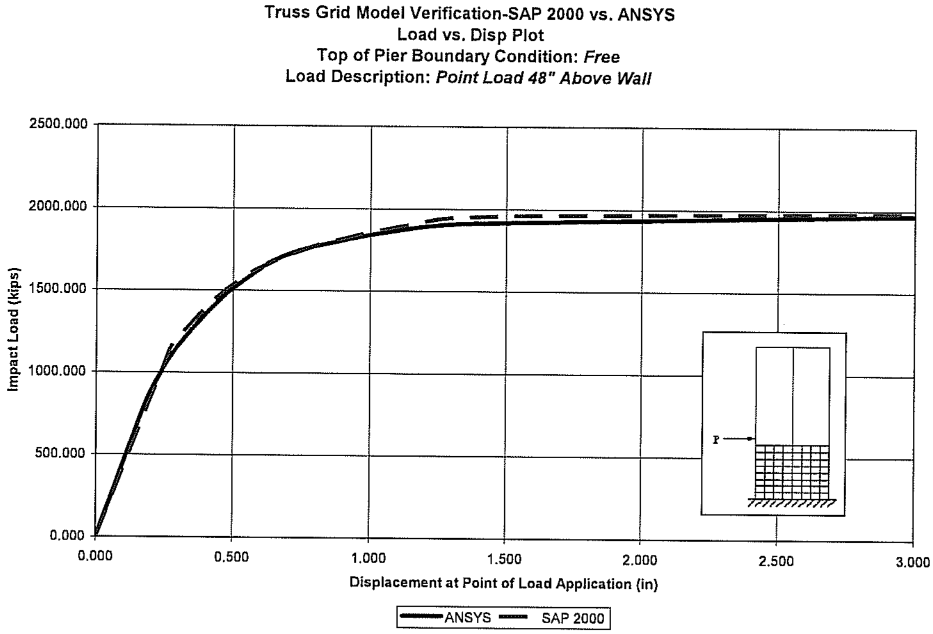


Figure 5-20. Wall Model Comparison SAP 2000 vs. ANSYS – Load Location 1



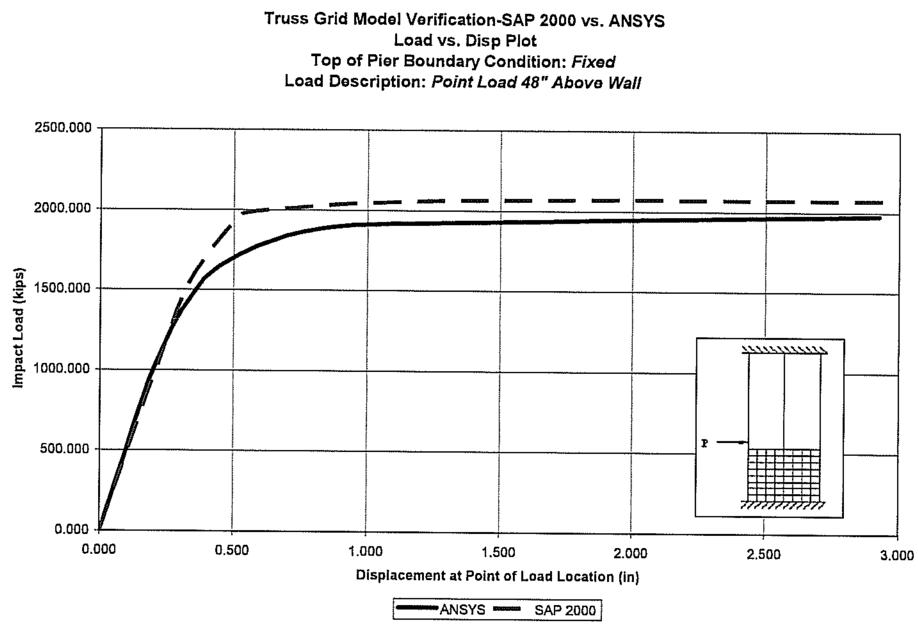
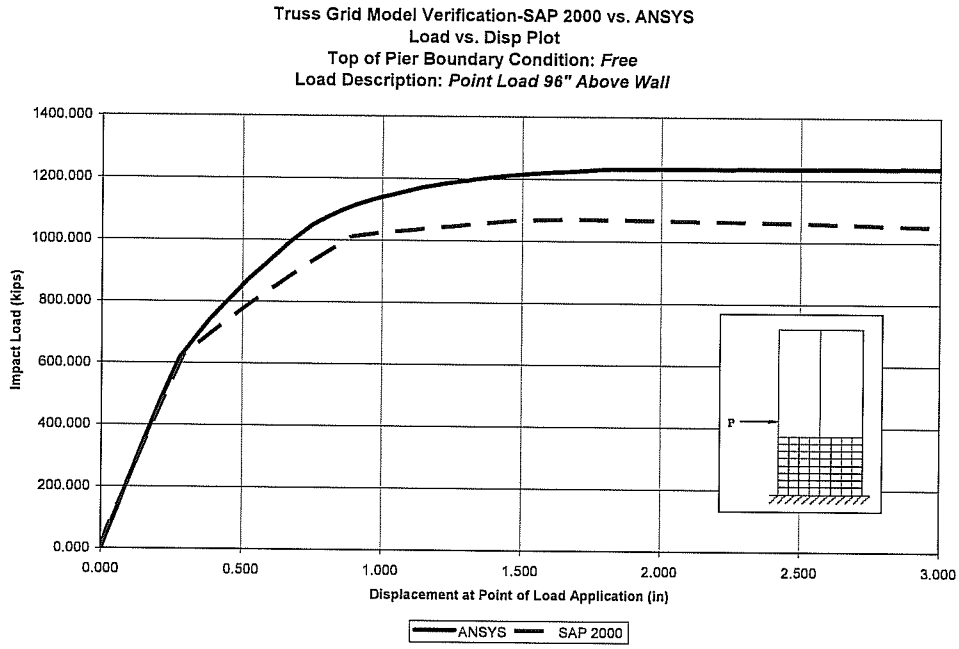


Figure 5-21. Wall Model Comparison SAP 2000 vs. ANSYS – Load Location 2



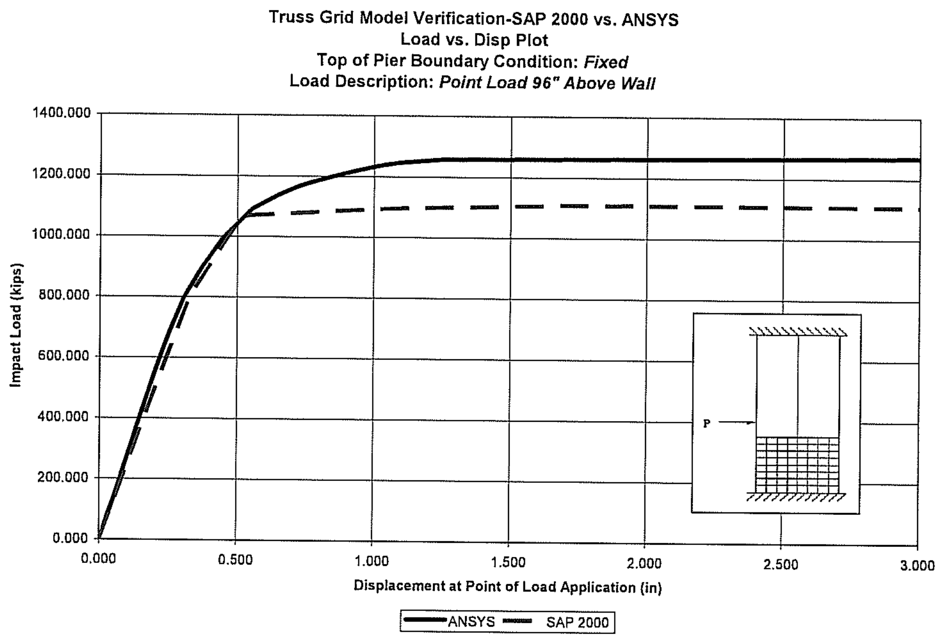
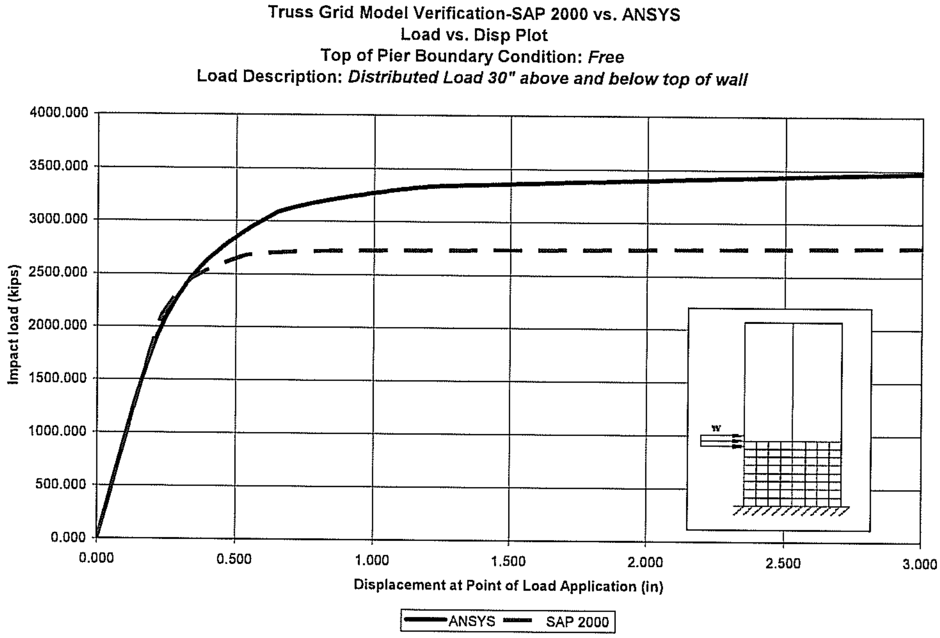


Figure 5-22. Wall Model Comparison SAP 2000 vs. ANSYS –Load Location 3



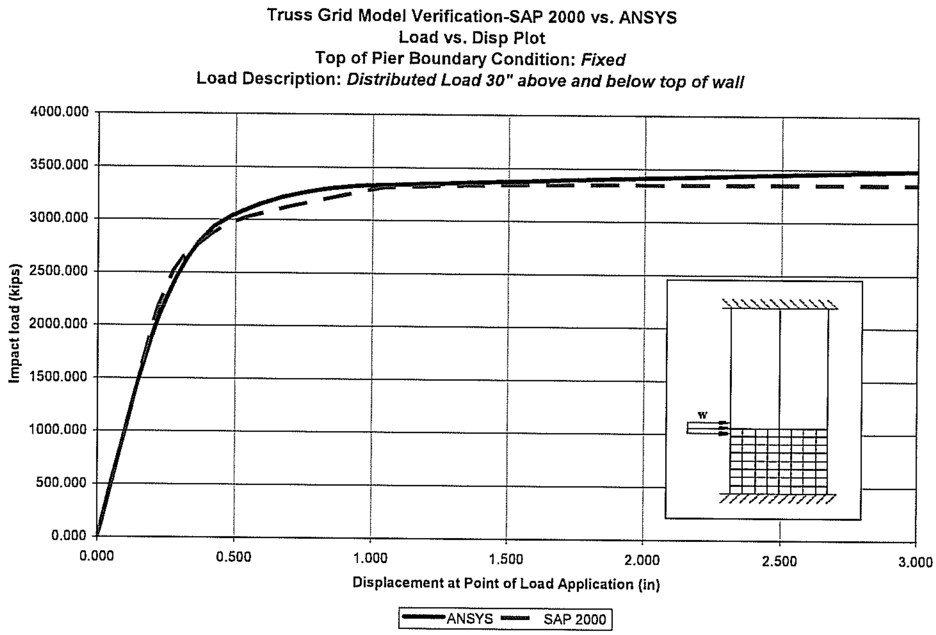


Figure 5-23. Wall Model Comparison SAP 2000 vs. ANSYS – Load Location 4

Table 5-2. Wall Model Comparison Results

SH-87 Bridge Ultimate Lateral Strength Wall Model Comparison Results					
Load Location	***Load Description***	***Boundary Condition at Top of Pier***	***ANSYS Shell Wall Model Lateral Strength (kips)***	***SAP 2000 Truss Grid Model Lateral Strength (kips)***	***% Error***
1	Point Load at Top of Wall	Free	3425.0	2742.0	19.94
2	Point Load 48" above Top of Wall	Free	1968.0	1982.0	0.71
3	Point Load 96" above Top of Wall	Free	1238.0	1050.0	15.19
4	Distributed Load 30" above and below wall	Free	3472.0	2765.0	20.36
1	Point Load at Top of Wall	Fixed	3433.0	3185.0	7.22
2	Point Load 48" above Top of Wall	Fixed	1975.0	2070.0	4.81
3	Point Load 96" above Top of Wall	Fixed	1268.0	1107.0	12.70
4	Distributed Load 30" above and below wall	Fixed	3485.0	3353.0	3.79

Figure 5-20 through Figure 5-25 and Table 5-2 show a mixed range of results. Clearly, the initial linear portion of the ANSYS load versus displacement curve matches the SAP 2000 load versus displacement curve. This observation verifies that the dimensions of the truss elements, which were sized in Chapter 4 specifically to match the linear elastic response of an equivalent model with a shell wall, were determined correctly. As the plots move into the inelastic range, however, differences between the SAP 2000 and ANSYS models begin to develop. Several of the SAP 2000 results show very good correlation to the ANSYS results, while others have errors of up to 20%. It is worth noting that for the load cases with higher error, the SAP 2000 values are conservative.

Close examination of the results reveal some interesting observations about the variation in the results based on the configuration of the load. Therefore, it is useful to separate the discussion of the results based on where the load is applied. It makes sense to compare the results from Load Locations 1 and 4 independent from the results of Load Locations 2 and 3. See Table 5-2 or Section 5.2.2 for clarification on load locations.

Loads at or Centered on the Top of the Shear Wall (Load Locations 1 & 4)

Load Locations 1 and 4 are located at or centered on a point at the top of the shear wall. Table 5-3 shows the analysis results for just these two load configurations. The values in Table 5-3 are taken directly from Table 5-2. They are separated only for ease of comparison. The plots for these cases are shown in Figure 5-20 and Figure 5-23, respectively. Table 5-3 indicates that the ultimate lateral strength results for Load Locations 1 and 4 are very similar, which is expected given that they are applied in the same area near the wall. Interestingly, the accuracy of the SAP model is very sensitive to the boundary condition at the top for these two models. With a fixed boundary condition, the SAP 2000 truss-grid wall model results match quite well with the ANSYS shell wall model results. However, the largest errors for any of the load locations are seen in the results from the same models with a free top.

Table 5-3. Wall Model Comparison Results Load Locations 1 and 4

SH-87 Bridge Ultimate Lateral Strength Wall Model Comparison Results Load Locations 1 and 4					
Load Location	*Load Description*	*Boundary Condition at Top of Pier*	*ANSYS Shell Wall Model Lateral Strength (kips)*	*SAP 2000 Truss Grid Model Lateral Strength (kips)*	*% Error*
1	Point Load at Top of Wall	Free	3425.0	2742.0	19.94
1	Point Load at Top of Wall	Fixed	3433.0	3185.0	7.22
4	Distributed Load 30" above and below wall	Free	3472.0	2765.0	20.36
4	Distributed Load 30" above and below wall	Fixed	3485.0	3353.0	3.79

Loads Applied on the Column (Load Locations 2 & 3)

Load Locations 2 and 3 are located at points along the exterior column of the pier and do not have any contact with the wall. Table 5-4 shows the comparison between the ANSYS and SAP 2000 models for these two load locations. The values from Table 5-4 are also taken directly from Table 5-2.

Table 5-4. Wall Model Comparison Results Load Locations 2 and 3

SH-87 Bridge Ultimate Lateral Strength Wall Model Comparison Results Load Locations 2 and 3					
Load Location	Load Description	Boundary Condition at Top of Pier	ANSYS Shell Wall Model Lateral Strength (kips)	SAP 2000 Truss Grid Model Lateral Strength (kips)	% Error
2	Point Load 48" above Top of Wall	Free	1968.0	1982.0	0.71
2	Point Load 48" above Top of Wall	Fixed	1975.0	2070.0	4.81
3	Point Load 96" above Top of Wall	Free	1238.0	1050.0	15.19
3	Point Load 96" above Top of Wall	Fixed	1268.0	1107.0	12.70

The results from Table 5-4 show that there is greater error in the Load Location 3 model when compared to the Load Location 2 model. For both cases, however, there is little difference between the results when the top boundary condition is changed, for both the SAP 2000 and the ANSYS models.

Also note that the SAP 2000 load versus displacement curves in Figure 5-21 and Figure 5-22 for loads 2 and 3 show sharp changes in the slope, whereas Figure 5-20 and Figure 5-23 for Load Locations 1 and 4 show a relatively smooth change in the slope. These observations can be explained by examining how plastic hinges were defined in Chapter 4, how hinges form in SAP 2000 and where hinges are forming for the particular load being applied.

Recall that the hinges for the SH-87 model were defined as being nearly elastic perfectly plastic, with little hardening after the hinge formed (see Figure 4-37). Also recall that Load Locations 2 and 3 were applied to a column away from elements of lateral support. Thus, there is a strong likelihood that hinges are forming at the column ends and at the point where the load is being applied (this observation will be verified later in this chapter). The inelastic response of the structure is being concentrated in just a few locations. Furthermore, SAP 2000 does not consider gradual yielding of a section in determining when a hinge forms. In reality, plasticity starts at the extreme fiber in a section and gradually yields through the depth of the section. In SAP 2000, when the yield moment or force has been reached, the hinge forms instantly. Taking all of these facts into consideration, the sharp changes in the load versus displacement plot for Load Locations 2 and 3 are reasonable. When the load is applied at or near the wall, plasticity is likely to spread through many elements in the wall, as opposed to a single column element, and the change in stiffness in the structure is much more gradual. Figure 5-20 and Figure 5-23 show smooth load versus displacement curves for the SAP 2000 models. As analysis results are presented throughout this chapter, it is essential to keep this discussion in mind. The trend of sharp changes in the load versus displacement curves for bridges with loads applied along the column is seen in the results for both the SH-87 and IH-10 Bridges.

Summary of Truss-Grid Verification Models

The plots and tables shown above verify that the truss-grid wall model captures the nonlinear strength and deformation characteristics of the shell wall model with reasonable accuracy. The greatest error is around 20%, and the mean error for all of the load configurations and boundary conditions is approximately 11%. Given the simplifying assumptions that were made in developing the truss-grid model, this error is considered acceptable. In addition, while there is a significant spread in the error depending on the load location and boundary conditions at the top of the pier, nearly all of the SAP 2000 models resulted in conservative estimates of the ultimate lateral strength, with only two exceptions, which only slightly exceeded the ANSYS estimate of ultimate lateral strength.

5.6.2 System-Wide Response Analysis Results

One of the goals in analyzing the IH-10 and SH-87 Bridge models is to compare system-wide response to individual element response. The AASHTO

LRFD Design Specification currently requires that ultimate lateral strength be calculated for single elements, which it defines as a bridge pier or a bridge span. This report has focused on calculating the ultimate lateral strength of bridge piers, and the results presented in this section compare the analyses of the main piers of the IH-10 and SH-87 Bridges. In the models for these two bridges, system-wide response is captured by adjusting the boundary conditions at the top of the bridge pier. In assessing the individual pier response, the pier is left free at the top. To consider system-wide response, elements representing the bridge superstructure are used, and the adjacent piers in the bridge are included as well. Recall from Chapter 4 that the superstructure elements for both the IH-10 and SH-87 represent 3-span continuous steel plate girders and a concrete deck. Therefore, for both bridges, the superstructure is continuous over the bridge piers that are subject to vessel impact. The connection at the other adjacent pier is pinned.

This section compares the ultimate lateral strength of the IH-10 and SH-87 Bridges for three top boundary conditions: free, fixed, and with the superstructure in-place. The free and fixed top cases provide a range of possible strengths for the system. The results from the system-wide models will fall somewhere in this range. Furthermore, by comparing all three of the top boundary conditions described, insight into when it is necessary to model the entire superstructure can be gained.

5.6.2.1 IH-10 Bridge Analysis Results

Figure 5-24 through Figure 5-27 show the ultimate lateral strength analysis results for the IH-10 Bridge model. The load locations under consideration have been described previously in Chapter 4 and in Section 5.2.2 of this chapter and were shown in Figure 5-1. Diagrams on the plots show where the load is being considered and what the top boundary condition is for each curve.

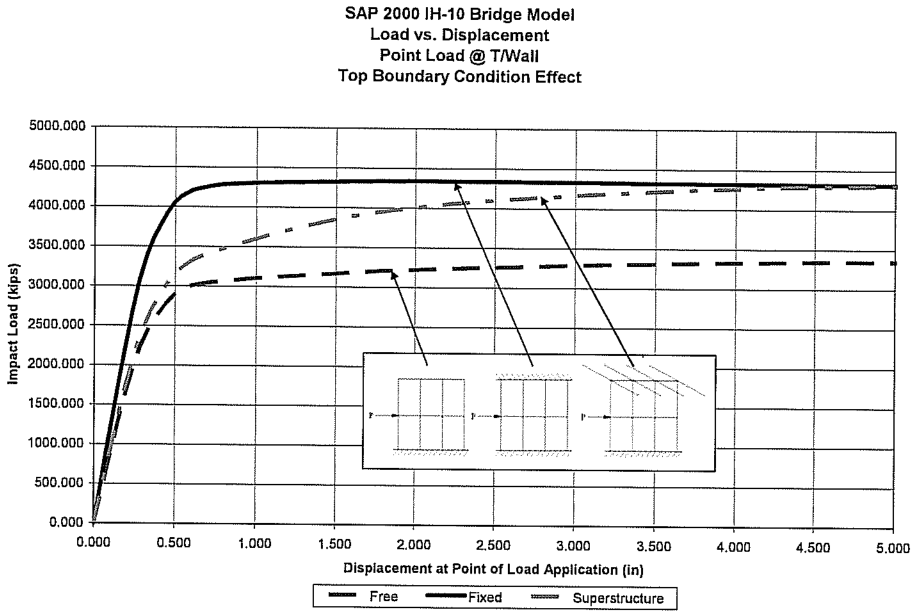


Figure 5-24. IH-10 Bridge Ultimate Strength Analysis Results-Load Location 1

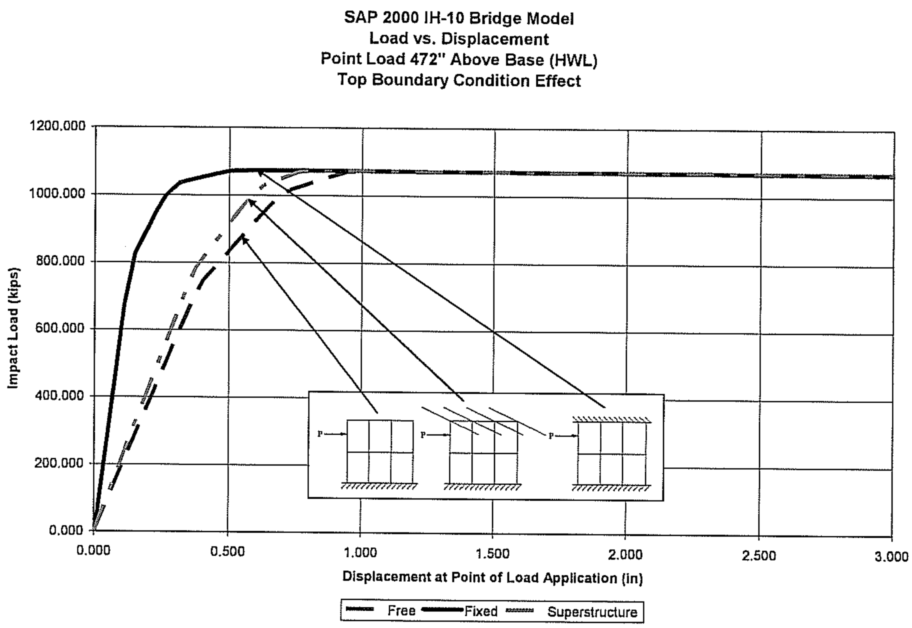


Figure 5-25. IH-10 Bridge Ultimate Strength Analysis Results-Load Location 2

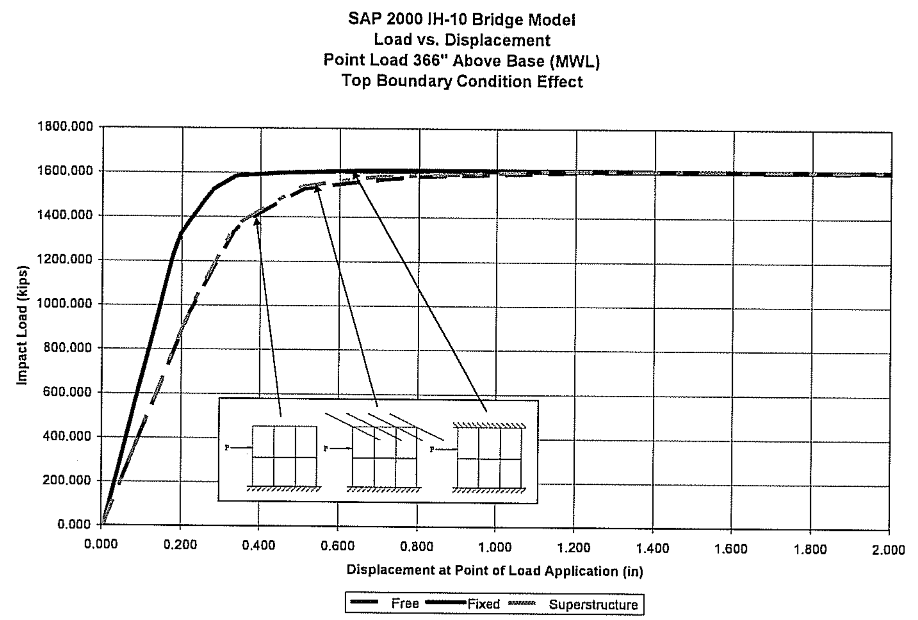


Figure 5-26. IH-10 Bridge Ultimate Strength Analysis Results-Load Location 3

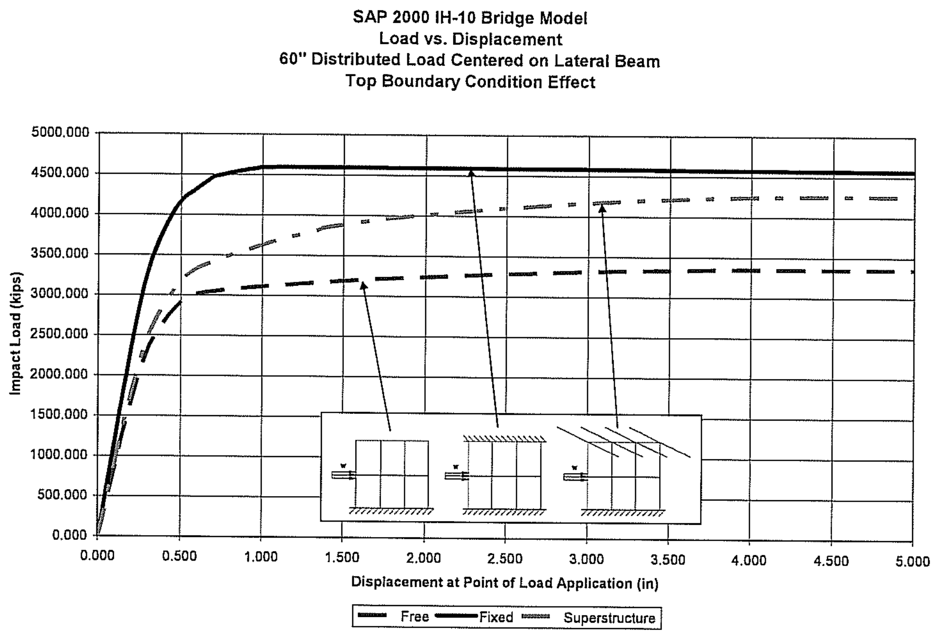


Figure 5-27. IH-10 Bridge Ultimate Strength Analysis Results-Load Location 4

The load versus displacement plots for the IH-10 Bridge model show interesting trends, and several conclusions can be drawn regarding the effect of including the superstructure when modeling a bridge for ultimate lateral strength calculation. First, there are some clear differences in the results between the models with loads applied at or around the lateral beam in the pier and those models with loads applied at some point along the column away from the beams. Results from Load Locations 1 and 4 (Figure 5-24 and Figure 5-27) have loads applied directly at or around the location where the lateral beam frames into the column. Results from Load Locations 2 and 3 (Figure 5-25 and Figure 5-26) have loads applied along the column, away from the beams. Separate discussions for Load Locations 1 & 4 and Load Locations 2 & 3 can be found below, but first it is necessary to outline the trends that are consistent for all of the load configurations.

Considering the effect of the superstructure, all load cases show very little effect on the response in the initial linear elastic portion of the load versus displacement plots. Including the superstructure slightly increases the stiffness of the system, but it still remains close to the free top condition. In addition, the superstructure does appear to have a large effect the point of first yield. Again, a slight increase in the yield point is seen, but the superstructure results are still closer to the results from the free top case than the fixed case. The effect of the superstructure appears to increase as the model moves further into the inelastic range. For all of the results presented, including the superstructure allows the model to reach, or nearly reach, the same ultimate lateral strength as a fixed top condition. Keep in mind that adequate ductility is required for this result to occur. Table 5-5 summarizes the analysis results of the IH-10 Bridge. Results for the initial stiffness, yield strength and ultimate strength are shown along with the percent increase as compared to the free top case.

Table 5-5. IH-10 Ultimate Strength Analysis Results

IH-10 Bridge Ultimate Lateral Strength Analysis Results Top Boundary Condition Comparison								
			Initial Stiffness (kips/in)		Yield Strength (kips)		Ultimate Strength (kips)	
Load Location	Load Description	Boundary Condition at Top of Pier	Value	% Increase (Relative to Free)	Value	% Increase (Relative to Free)	Value	% Increase (Relative to Free)
1	Point Load at Beam	Free	8523.0	-	2915.0	-	3349.0	-
1	Point Load at Beam	Superstructure	8981.0	5.4	3171.0	8.8	4301.0	28.4
1	Point Load at Beam	Fixed	11501.0	34.9	4055.0	39.1	4316.0	28.9
2	Point Load at MWL	Free	4027.0	-	1365.0	-	1608.0	-
2	Point Load at MWL	Superstructure	4447.0	10.4	1378.0	1.0	1608.0	0.0
2	Point Load at MWL	Fixed	6057.0	50.4	1525.0	11.7	1608.0	0.0
3	Point Load at HWL	Free	1985.0	-	748.0	-	1073.0	-
3	Point Load at HWL	Superstructure	2164.0	9.0	782.0	4.5	1073.0	0.0
3	Point Load at HWL	Fixed	5461.0	175.1	827.0	10.6	1073.0	0.0
4	60" Dist'd Load Centered at Beam	Free	8491.0	-	2921.0	-	3349.0	-
4	60" Dist'd Load Centered at Beam	Superstructure	8945.0	5.3	3166.0	8.4	4326.0	29.2
4	60" Dist'd Load Centered at Beam	Fixed	11509.0	35.5	4206.0	44.0	4566.0	36.3

Loads Applied Near the Beam (Load Locations 1 & 4)

Figure 5-24 and Figure 5-27 show the load versus displacement plots for the IH-10 Bridge with loads applied at or near the location of the beam. The analysis results of these models indicate that the superstructure has a significant effect on the strength of the pier, but as previously indicated, there is little change in the initial stiffness and yield point. Table 5-5 shows that for Load Location 1, there is less than a 10% change in the initial stiffness and yield point, while the increase in ultimate strength is nearly 30%. The results from the analysis with Load Location 4 are similar.

Loads Applied on the Column (Load Locations 2 & 3)

Figure 5-25 and Figure 5-26 show the load versus displacement plots for the IH-10 Bridge with loads applied away from the pier beams at some point along the column above the beam. The top exterior column is 318-inches long, measured from the beam near the middle of the pier frame up to the cap beam. Load Location 2 is 66 inches above the beam, or about one-fifth of the way up the column. Load Location 3 is 176.4 inches above the beam or just beyond the midpoint of the column. The results for these cases are summarized in Table 5-5. The results indicate that it is not necessary to consider the effect of the top boundary condition, or to even model the entire bridge pier. The ultimate lateral strength is governed almost entirely by the strength of the individual column being struck. The boundary condition at the top affects the initial stiffness and point of first yield, but the ultimate strength plateaus at the same value for all of the models.

5.6.2.2 SH-87 Bridge Analysis Results

The results presented below for the SH-87 Bridge show trends similar to the IH-10 Bridge analysis results. Figure 5-28 through Figure 5-31 show the ultimate lateral strength analysis results for four different load configurations, each with three different boundary conditions at the top of the model. The load locations under consideration have been described and shown previously in Figure 5-2. Table 5-6 summarizes the results of the analyses. Discussion of the results can again be broken down into two groups — loads applied at or around the top of the wall, and loads applied along the column.

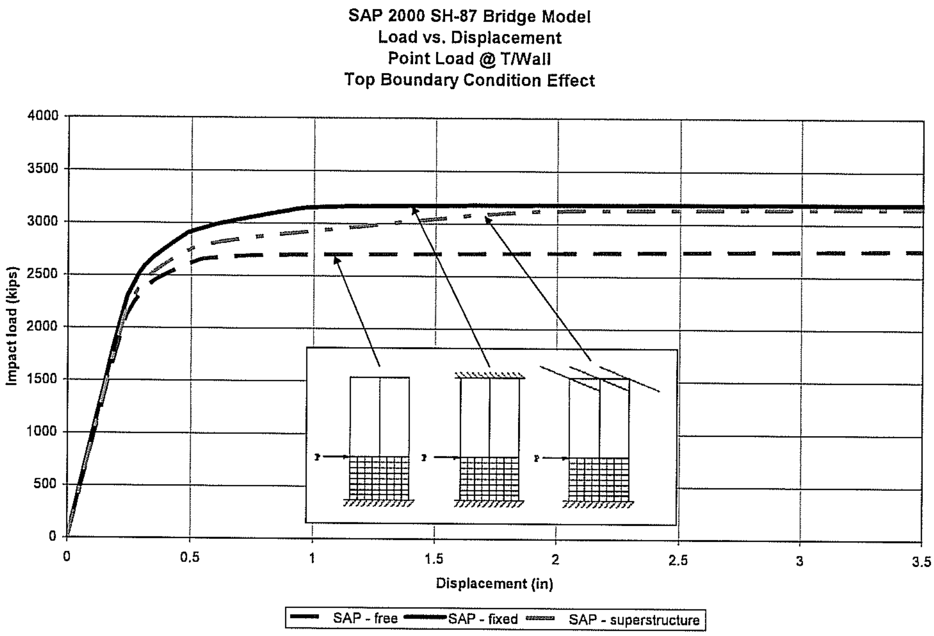


Figure 5-28. SH-87 Bridge Ultimate Strength Analysis Results 1

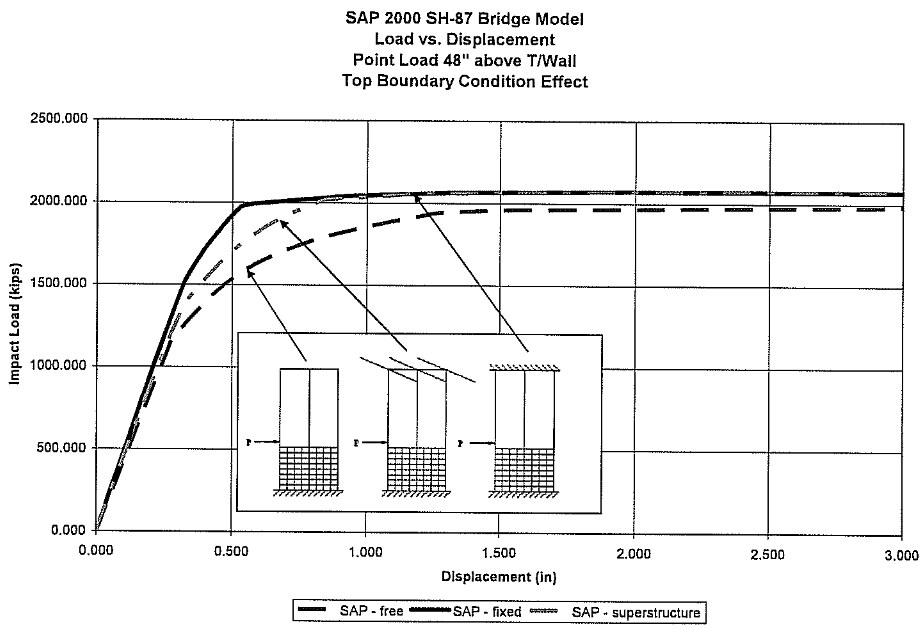


Figure 5-29. SH-87 Bridge Ultimate Strength Analysis Results 2

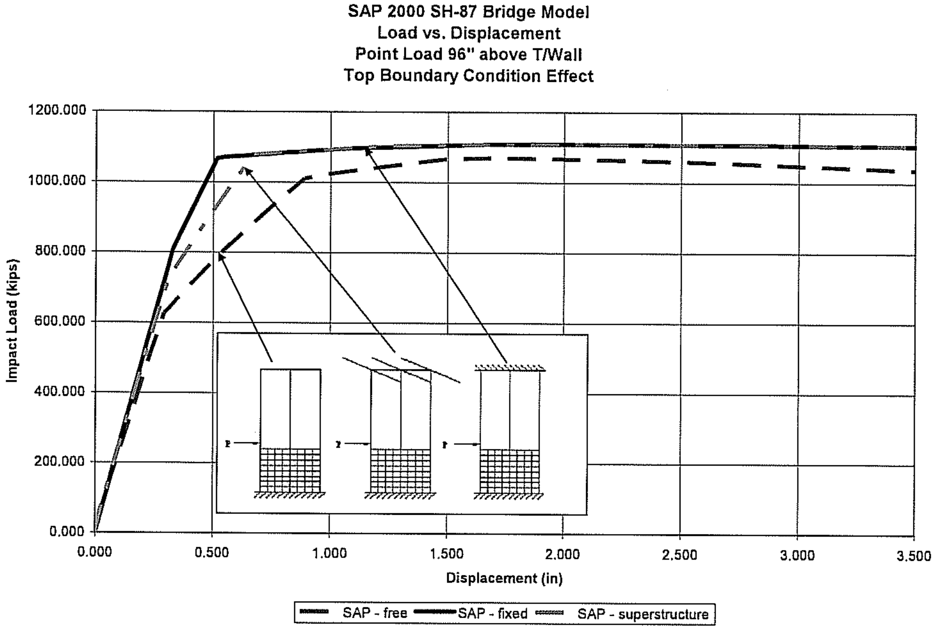


Figure 5-30. SH-87 Bridge Ultimate Strength Analysis Results 3

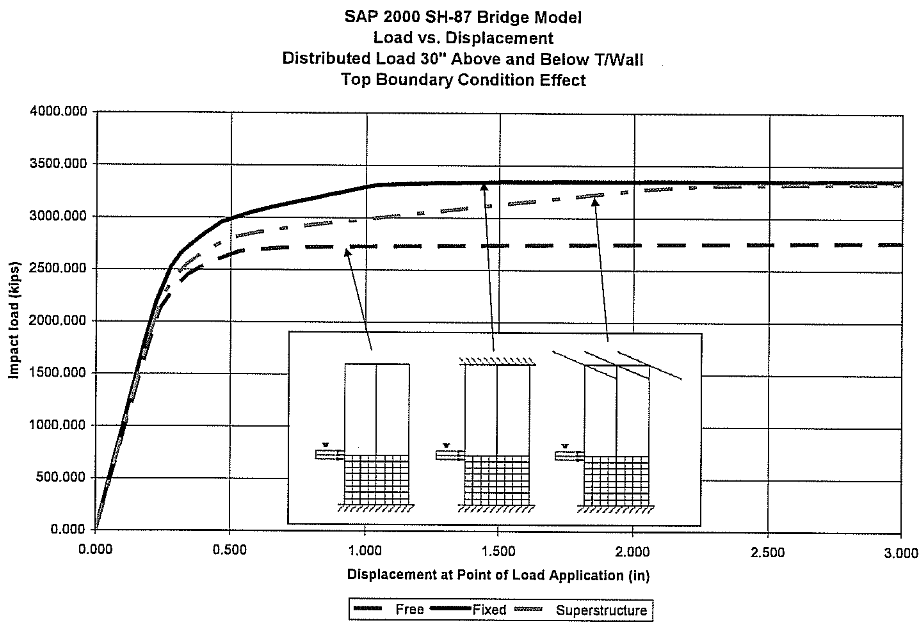


Figure 5-31. SH-87 Bridge Ultimate Strength Analysis Results 4

Table 5-6. SH-87 Ultimate Strength Analysis Results

SH-87 Bridge Ultimate Lateral Strength Analysis Top Boundary Condition Comparison								
Load Location	Load Description	Boundary Condition at Top of Pier	Initial Stiffness (kips/in)		Yield Strength (kips) (estimated)		Ultimate Strength (kips)	
			Value	% Increase (Relative to Free)	Value	% Increase (Relative to Free)	Value	% Increase (Relative to Free)
1	Point Load at Top of Wall	Free	9203.0	-	2335.0	-	2742.0	-
1	Point Load at Top of Wall	Superstructure	9410.0	2.2	2386.0	2.2	3137.0	14.4
1	Point Load at Top of Wall	Fixed	9803.0	6.5	2504.0	7.2	3185.0	16.2
2	Point Load 48" above Top of Wall	Free	4264.0	-	625.0	-	1974.0	-
2	Point Load 48" above Top of Wall	Superstructure	4450.0	4.4	724.0	15.8	2074.0	5.1
2	Point Load 48" above Top of Wall	Fixed	4849.0	13.7	806.0	29.0	2074.0	5.1
3	Point Load 96" above Top of Wall	Free	2174.0	-	1152.0	-	1070.0	-
3	Point Load 96" above Top of Wall	Superstructure	2353.0	8.2	1341.0	16.4	1107.0	3.5
3	Point Load 96" above Top of Wall	Fixed	2483.0	14.2	1523.0	32.2	1107.0	3.5
4	Distributed Load 30" above and below wall	Free	9291.0	-	2122.0	-	2765.0	-
4	Distributed Load 30" above and below wall	Superstructure	9557.0	2.9	2232.0	5.2	3322.0	20.1
4	Distributed Load 30" above and below wall	Fixed	10007.0	7.7	2518.0	18.7	3353.0	21.3

Loads Applied at, or Centered on the Top of the Wall (Load Locations 1 & 4)

The analysis results for the SH-87 models with loads at or around the wall are shown in the load versus displacement plots in Figure 5-28 through Figure 5-31 and are summarized in Table 5-6. The results show similar trends when compared to the IH-10 Bridge results for loads applied near the beam. While the geometries of the two bridges are quite different, the load locations are both located at or near the main lateral support elements in the pier. As with the IH-10 Bridge, the SH-87 results show little increase in the initial stiffness and yield strength when the superstructure is included in the model, but there is a significant increase in the ultimate strength.

Examining the displaced shapes of the SH-87 Bridge pier at the limit state also provides insight into the behavior of the system. The displaced shape for Load Locations 1 and 4 are shown in Figure 5-32. The small dots at member ends represent locations where hinges have formed. Both models show extensive inelastic behavior throughout the system. Clearly forces are being redistributed throughout the pier, and pier-wide response is dominating.

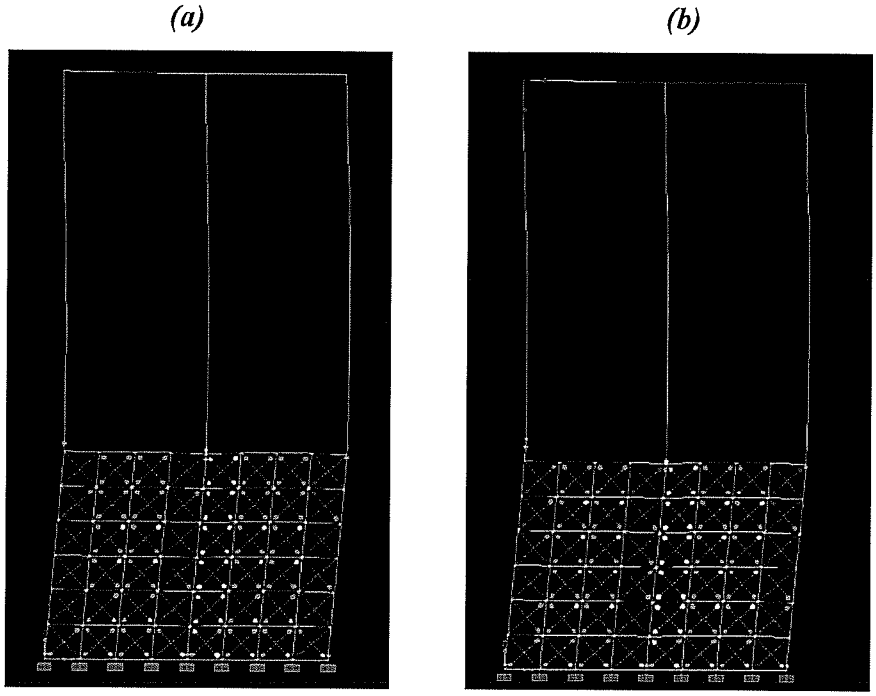


Figure 5-32. SH-87 Displaced Shape at Limit State: (a) Load Location 1; (b) Load Location 4

Loads Applied on the Column (Load Locations 2 & 3)

The results from an analysis of the SH-87 Bridge with loads applied to the column at 48 inches and 96 inches above the wall are shown in Figure 5-29 and Figure 5-30, respectively. These plots and the results summarized in Table 5-6 show patterns that are somewhat similar to the IH-10 Bridge results for loads applied to the column. The results from both bridges show little increase in the initial stiffness of the system when the superstructure is included. Unlike the IH-10 Bridge, however, the SH-87 results show that the superstructure affects both the yield strength and the ultimate strength. This trend was also found for loads applied to the SH-87 Bridge at points higher than 96 inches above the wall. These results indicate that it is not possible to draw a conclusion on the effect of including the superstructure on the yield strength for loads applied to a column based on the analysis results presented. Further investigation into this matter is required.

Figure 5-33 and Figure 5-34 show the model setup and displaced shape (after the ultimate lateral strength has been reached) for the SH-87 Bridge pier model for Load Locations 2 and 3, respectively. The small dots represent locations where plastic hinges have formed. In both models, a mechanism has formed in the exterior column to which the load is being applied. There are hinges at each end of the column, as well as two hinges along the length. The hinges along the length are on each side of the point load and physically represent a single hinge location.

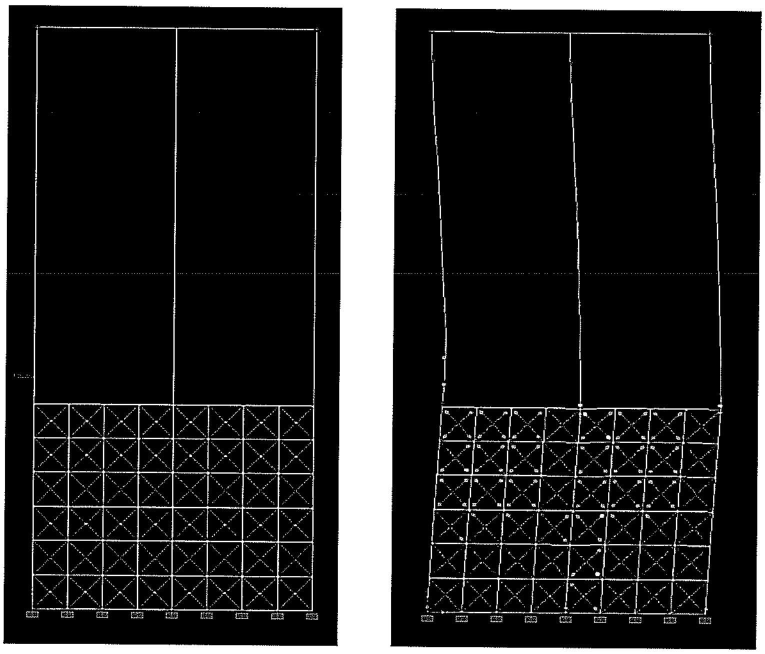


Figure 5-33. Load Location 2 Model and Displaced Shape

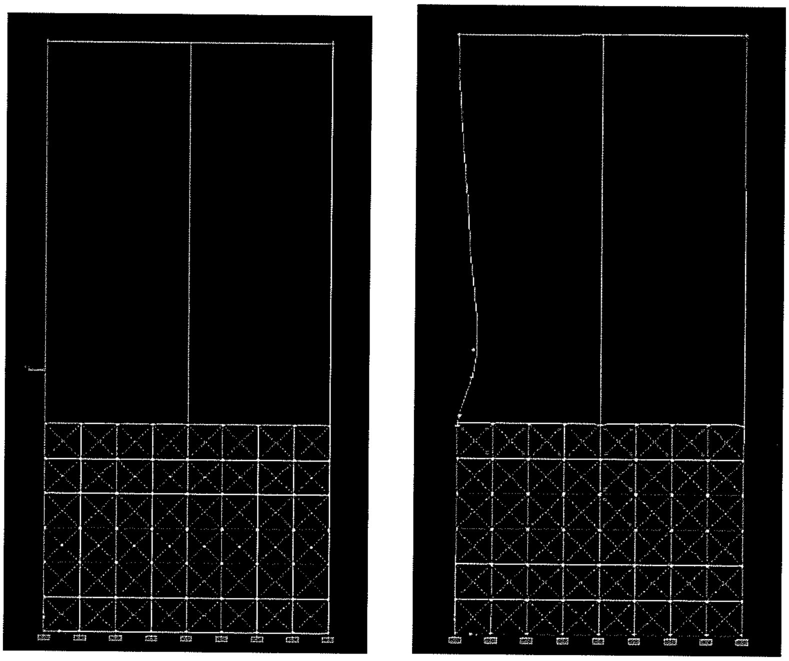


Figure 5-34. Load Location 3 Model and Displaced Shape

Load Location 2 is a point load 48 inches above the top of the wall, or about 7% up the length of the column, and Load Location 3 is a point load 96 inches above the wall, or about 14% up the length of the column. Despite the small difference in load location relative to the overall length of the column (684 inches), there is a significant difference in the displaced shapes. Figure 5-33 shows that with a point load 48 inches above the wall, there is still noticeable deformation in the wall, and the wall is clearly having a significant contribution to the response of the system. Figure 5-34 shows smaller deformations in the wall. Only a few of the truss elements have yielded by the time a mechanism has formed in the column. Figure 5-34 also suggests that it may not be necessary to consider the wall if this point is where vessel impact is expected. The response of the column clearly dominates for Load Location 3, and a simple plastic analysis of this member should provide a reasonably accurate estimate of the ultimate lateral strength of the pier. The displaced shape at the limit state for Load Location 2 looks very similar to Figure 5-32, which shows the displaced shape for Load Locations 1 and 4. The response of the system is less localized than with Load Location 3.

Summary of System-Wide Response Analyses

Despite significantly different bridge pier geometries, the IH-10 and SH-87 Bridge models show similar trends based on the load location when the effect of system-wide response is considered in an ultimate lateral strength analysis. Based on the results presented in this chapter, the following conclusions can be drawn about the effect of including the superstructure to capture system-wide response.

- *Conclusion 1:* Modeling the superstructure has little impact on the initial stiffness for all load cases considered.
- *Conclusion 2:* If impact occurs at a distance greater than 10% of the column length away from the lateral support element on a single column, local response dominates. The rest of the bridge pier has little effect on the analysis results, and a simple plastic analysis of the column would produce a reasonable estimate of the ultimate lateral strength. This conclusion assumes that there is adequate stiffness in the lateral support elements to allow a column mechanism to form, which is believed to be a reasonable assumption
- *Conclusion 3:* If impact occurs in close proximity to a lateral support element in a pier (i.e., a wall or beam), system-wide response dominates, and accurately modeling and analyzing the entire bridge pier is necessary.
- *Conclusion 4:* If impact occurs in close proximity to a lateral support element in a pier, including the superstructure results in an increase in the

ultimate lateral strength. For some cases, this increase can be significant (up to a 30% increase).

5.6.3 Reduced Section Analysis Results

Section 4.7 covered the modeling procedure to capture the effect of some loss of cross-sectional area in the regions of a bridge pier where impact is being considered. This section outlines the analysis results using the Chapter 4 guidelines for section loss modeling on the IH-10 Bridge. The same four load cases that have been used throughout this chapter are used. The superstructure and adjacent piers have been included in the model. A 10% and 20% loss of cross sectional area in the impact region are considered. Use of these values is not to suggest that they represent the expected section loss due to impact; instead they merely represent possible section losses. Determining these values exactly would require a detailed finite element model and dynamic analysis of the bridge pier and vessel, or some sort of physical testing, both of which are beyond the scope of this document. This section is presented to show how the ultimate lateral strength would change if the modeling procedure described in Section 4.7 was used and a section loss of 10% or 20% was assumed. Table 5-7 shows the ultimate lateral strength analysis results, including the effect of cross section loss, for the IH-10 Bridge.

Table 5-7. IH-10 Bridge Reduced Cross Section Analysis Results

IH-10 Bridge Ultimate Lateral Strength Analysis Results Affect of Cross Section Loss due to Impact					
Load Location	**Load Description**	**% of Cross-Section Area Lossed**	**Boundary Condition at Top of Pier**	**Ultimate Strength (kips)**	
				Value	**% Decrease**
1	Point Load at Beam	Full Section	Superstructure	4301.0	-
1	Point Load at Beam	10% Loss	Superstructure	4197.0	2.4
1	Point Load at Beam	20% Loss	Superstructure	4154.0	3.4
2	Point Load at MWL	Full Section	Superstructure	1608.0	-
2	Point Load at MWL	10% Loss	Superstructure	1437.0	10.6
2	Point Load at MWL	20% Loss	Superstructure	1168.0	27.4
3	Point Load at HWL	Full Section	Superstructure	1073.0	-
3	Point Load at HWL	10% Loss	Superstructure	956.0	10.9
3	Point Load at HWL	20% Loss	Superstructure	872.0	18.7
4	60" Dist'd Load Centered at Beam	Full Section	Superstructure	4326.0	-
4	60" Dist'd Load Centered at Beam	10% Loss	Superstructure	4262.0	1.5
4	60" Dist'd Load Centered at Beam	20% Loss	Superstructure	4101.0	5.2

The results shown in Table 5-7 reinforce some of the conclusions that have already been made regarding both the IH-10 and SH-87 Bridge. Table 5-7 shows that for the loads applied near the lateral support element in the pier (Load Locations 1 and 4), considering a loss of cross sectional area due to impact results in little change in the ultimate lateral strength. Because the load is applied near a lateral support element, forces can be redistributed through the system with a minimal decrease in the overall strength. System-wide behavior dominates the response. For Load Locations 2 and 3, modeling a cross-sectional area loss in the area of impact causes a significant decrease in the ultimate lateral strength. The local behavior of the column dominates the response for these two load cases, and assigning reduced section properties has a noticeable effect on the strength of the column.

5.6.4 Column Removal Analysis Results

For cases where vessel impact is expected to occur at a point along the length of the column, it may be useful to consider the effect of losing that single column on the response of the rest of the bridge. This section presents an analysis of the IH-10 Bridge and SH-87 Bridge with an exterior column removed for one of the previously outlined load cases. The results presented here are intended to be an introductory example into this type of analysis. An in-depth investigation has not been conducted on the effect of losing a single column in a multi-column bent. While the analysis procedure and results produced in this chapter for a column removal analysis are limited, they are still important. Gaining a better understanding of system behavior after failure of a single element could allow engineers to design bridge structures that can withstand the loss of individual elements without catastrophic failure of the entire system.

5.6.4.1 IH-10 Bridge Column Removal Analysis

For the IH-10 Bridge model, a column removal analysis was performed for Load Location 3 (a point load located 476 inches above the base of the pier at the high water level). For this load case, hinges form at the column ends and at the point where the lateral load is applied. The displaced shape of the IH-10 Bridge with a mechanism formed at the point of impact was shown previously in Figure 5-16. Figure 5-35 shows the bridge pier after the column removal analysis has been performed, using the staged construction option. Notice that no hinges (represented by small dots) have formed at the end of the now cantilevered cap beam. Based on the analysis results presented here, the response suggests the remaining portion of the structure will not fail.

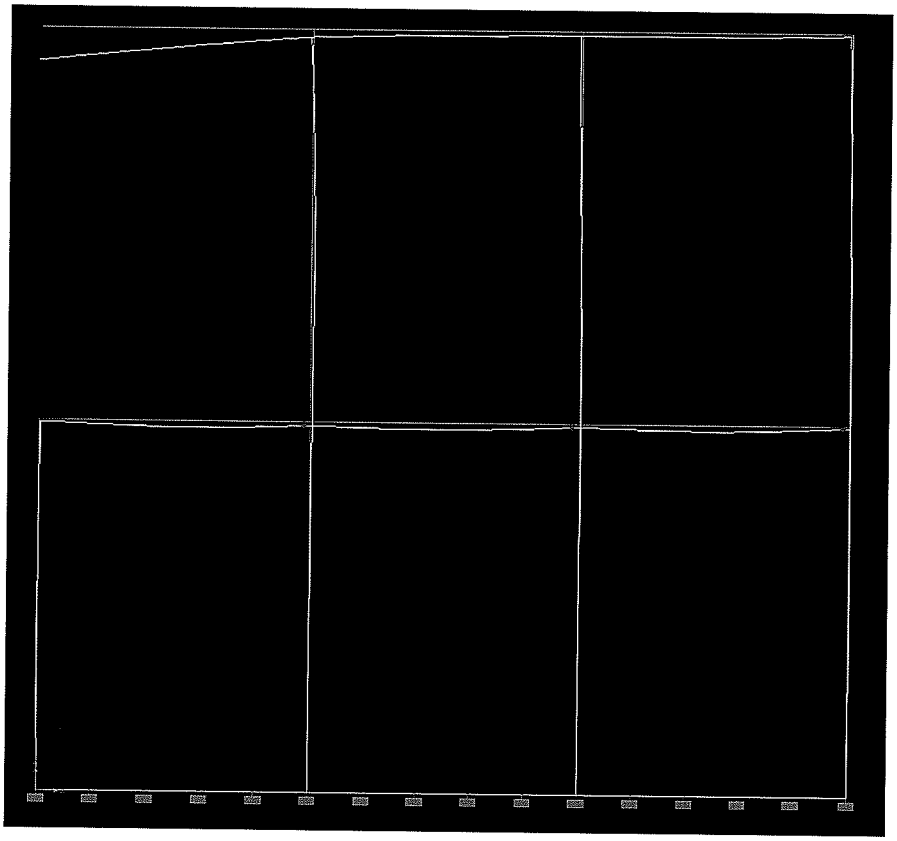


Figure 5-35. IH-10 Bridge Column Removal Analysis

5.6.4.2 SH-87 Bridge Column Removal Analysis

For the SH-87 Bridge model, a column removal analysis was also performed for Load Location 3 (a point load located 96 inches above the shear wall). For this load case, hinges form at the column ends and at the point where the lateral load is applied, creating a failure mechanism. Figure 5-36 shows the SH-87 bridge pier after the column removal analysis is performed. Notice that a hinge (represented by a small dot) has formed at the end of the now cantilevered cap beam. Figure 5-37 shows a three-dimensional view of the SH-87 Bridge after the failed column has been removed from the structure. For this case, the column removal analysis was not able to complete due to excessive deformations at the end of cap beam. Thus, it is possible to conclude that this bridge has insufficient strength and ductility to support the deck elements framing into the cap beam, and the loss of a single exterior column leads to a progressive failure in the bridge system.

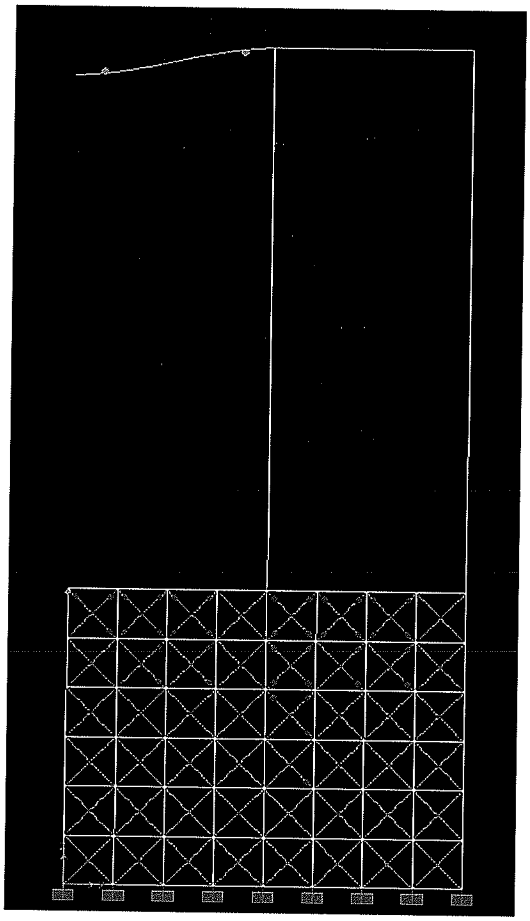


Figure 5-36. SH-87 Bridge Column Removal Analysis-1

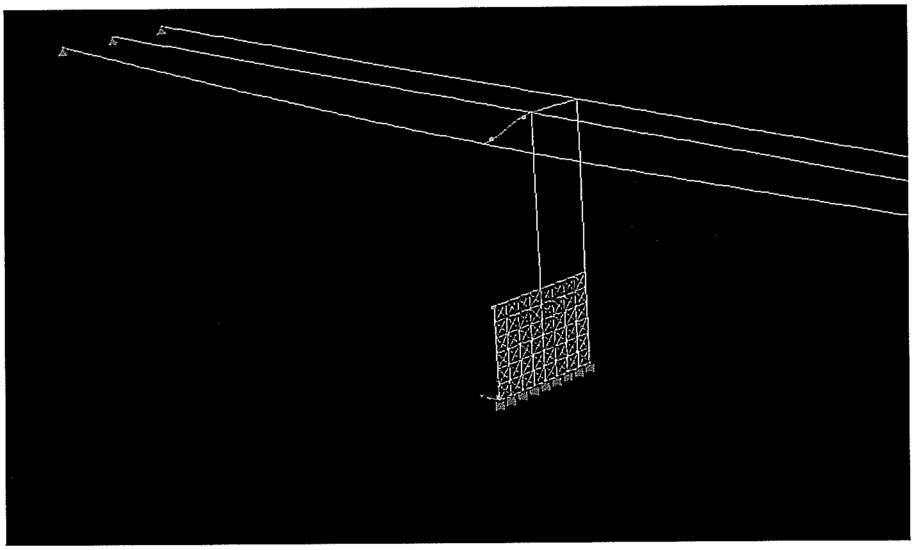


Figure 5-37. SH-87 Bridge Column Removal Analysis-2

5.7 Analysis and Results Summary

This chapter presented the analysis results of the IH-10 and SH-87 Bridge models that were constructed in Chapter 4. Several static nonlinear analysis cases, for use within SAP 2000, were outlined as a means to calculate the ultimate lateral strength of bridge elements and bridge systems. Tools for assessing analysis results were also presented. Research findings presented in this chapter demonstrated that a truss-grid model is adequate for modeling the response of bridge piers with shear walls. In addition, the effect of modeling system-wide behavior on the ultimate lateral strength was illustrated. Additional guidelines were presented to account for reduced cross-sectional areas due to impact and the failure of a single member in a multi-column bridge pier. Chapter 6 summarizes the work in this document and reviews the major conclusions that were drawn based on the research completed for TxDOT Project 0-4650.

CHAPTER 6
Summary and Conclusions

6.1 INTRODUCTION

This document has investigated the problem of vessel collision with bridges and has attempted to provide engineers with useful tools to deal with this issue by examining the calculation of the ultimate lateral strength of bridges subject to impact loads. The importance of considering vessel impact loads for bridges over navigable waterways was illustrated by showing the consequences of vessel collision accidents with bridges using specific examples from the last twenty-five years. A thorough review of the events, groups, and research work that lead to the development of the *AASHTO Guide Specification for Vessel Collision Design of Bridges* was provided. In addition, a detailed review and critical examination of the AASHTO Guide Specification was made. Several areas in need of improvement for the AASHTO recommended Method II design procedure were identified. Calculation of the Probability of Collapse term was emphasized as an area in need of examination. This calculation is based on the impact load from a vessel and the ultimate lateral strength of a bridge element. AASHTO provides little guidance in the calculation of the ultimate lateral strength and does give any consideration to the strength of bridge system as a whole.

6.2 SUMMARY OF WORK

The research in this report has focused on the modeling and analysis of bridge piers and bridge systems subject to impact loads. Chapter 4 focused on the modeling of these systems, and Chapter 5 presented results from the analyses of those models. One of the primary goals of this report has been to provide easy-to-use guidelines and procedures to calculate the ultimate lateral strength of these structures using typical structural analysis software packages. Reinforced concrete bridge piers, both with and without shear walls, were investigated. Guidelines for modeling reinforced concrete using a smeared material model based on a sectional analysis were presented. The use of plastic hinges to capture the inelastic behavior in bridge systems was discussed. A truss-grid model was introduced to capture the inelastic response of shear walls in a bridge pier. Modeling examples for two representative bridges from Texas, the IH-10 Bridge over the San Jacinto River and the SH-87 Bridge over the Gulf Intracoastal Waterway, were shown using SAP 2000, a typical structural analysis program.

Additional work focused on investigating the effect of system-wide bridge response to impact loads. Further guidelines outlined a procedure to account for a reduced section size in the regions of a bridge pier where vessel impact occurred. Analysis methods to capture the response of a multi-column bridge pier, given the failure of a single column, were also introduced.

Several conclusions were reached based on the results presented in Chapter 5, and they are summarized below. For a more detailed discussion refer to Chapter 5.

Important Conclusions Based on Ultimate Lateral Strength Analysis Results

Modeling Conclusions

- *Truss-Grid Model.* A comparison of two models of the same SH-87 Bridge pier, one built in SAP-2000 using a truss-grid model for the wall and the other built in ANSYS using shell elements for the wall, showed similar load versus displacement responses for a range of load locations and boundary conditions. Based on these results, it is believed that the truss-grid model outlined in Chapter 4 can capture the inelastic response of a shear wall with acceptable accuracy for the purposes of design.

- *Localized Response.* Results from Chapter 5 showed that, for loads applied along the length of a column away from lateral support elements, the response of an individual column controlled the ultimate lateral strength, and extensive modeling of the entire structure is not required. Based on the IH-10 and SH-87 Bridge results, it is suggested that a simple plastic analysis on a column is appropriate if the impact load is being applied at a point greater than 10% of the column length away from a lateral support. For both bridges, the lateral support elements at the top and bottom of the column provide enough stiffness so that a mechanism can form in the column. Given the differences in geometry between the piers of the IH-10 and SH-87 Bridges, it is believed that this rule could be applied across a variety of bridge piers
- *Pier-Wide Response.* When impact loads were considered at or near locations of beams or walls providing lateral support for a pier, forces were distributed throughout the entire bridge pier and inelastic response spread through the structure.

System-Wide Response Conclusions

- *Initial Stiffness.* Including elements to represent the bridge superstructure and modeling adjacent bridge piers had little effect on the initial response of a bridge system subject to lateral loads.
- *Point of First Yield.* The effect of including elements to represent the bridge superstructure and of modeling adjacent bridge piers on the initial strength of the bridge system varied. For the IH-10 Bridge, little change

in the initial strength was found when the superstructure was modeled; however, a slight increase in the yield point was seen in the SH-87 Bridge.

- *Ultimate Lateral Strength.* For cases where impact loads acted at or near lateral support elements, including the bridge superstructure and adjacent bridge piers had a noticeable effect on the ultimate lateral strength of the bridge systems being studied. For cases where impact loads were applied along the length of a column, the top boundary condition had little impact on the ultimate lateral strength of the systems considered.

It is also important to note the limitations of the models and procedures presented in Chapters 4 and 5. These issues have been addressed previously and are summarized below:

- *Local Response of Wall.* The truss-grid model has been shown to accurately capture the overall response of a shear wall in a bridge pier. However, because of the rigid grid, inelastic behavior is spread through the truss elements more evenly than in a real wall, which would see more localized damage near the point of impact.
- *Base Boundary Condition.* A fixed condition was assigned to the base of the bridge pier models. In reality, these structures have some flexibility at the base. By accurately modeling the soil-structure interaction, a better representation of the base condition can be made.
- *Material Model.* This report focused on reinforced concrete bridge piers. Instead of modeling the concrete and steel directly, a smeared material model approach was taken. In doing so, a level of ductility was assumed for all of the members, which in turn assumed that the bridge pier elements were properly detailed. This assumption greatly simplified

modeling and analysis of the bridge piers, but a more accurate representation of the material is possible.

- *Limited Results.* One of the clear limitations of the research contained in this report is that results have been presented for only two bridges. The IH-10 and SH-87 Bridges were selected because they are representative of bridge piers found in Texas. While they did provide the opportunity to examine two different bridge pier geometries, investigating more bridges would likely allow further and more distinct conclusions to be drawn on the issues addressed in this report.

6.3 Integration with Other Research to Improve AASHTO Vessel Collision Design Specifications

This report is one part of a three-part study at the University of Texas at Austin. Another aspect of this research, conducted by Adam Cryer under the supervision of Dr. Loukas Kallivokas, is investigating the actual loads imparted to a bridge from barge impact. The probability of collapse term in the AASHTO Method II is based on the impact load and the ultimate lateral strength. With the improved understanding of bridge strength provided in this report and a more accurate determination of the loads imparted during vessel impact by the Cryer report, a critical examination of the probability of collapse term can be made. This work is part of the third aspect of the research and is being completed by Kenny Berlin under the supervision of Dr. Lance Manuel.

6.4 Future Research Opportunities

Vessel collision design is an evolving field. The first design code in the United States was not introduced until the early 1990s. Implementation of the AASHTO recommended Method II design procedure has been slow, and many states lack access to the necessary information to effectively use this procedure in designing and analyzing bridges subject to vessel collision. Furthermore, little physical testing has been done to investigate the mechanics of vessel impact on bridges, mostly due to the impractical nature of testing full-scale vessel-bridge impact. In short, there exists a wide range of future research opportunities that could be explored in order to improve bridge design for vessel collision in the United States. This section outlines some of these areas, with a focus on how the research presented in this document could be expanded.

One area for future research would be to repeat the same analyses presented in Chapter 5 over a wider range of bridge types. Different pier geometries should be considered. In addition, a more thorough investigation into the effect of including superstructure elements in an analysis could be conducted by considering a range of deck and girder types and configurations. Loading types and locations could also be carried out across a wider range. A more accurate model for reinforced concrete could also be developed for use in the analyses presented in this report. In addition, future research could focus on determining the effect of soil-structure interaction and behavior during vessel impact, in order to better model bridge pier base conditions. The research possibilities outlined here could lead to more detailed guidelines and rules for calculating ultimate lateral strength of bridges.

Several other areas of possible research were discussed in this report, but not investigated fully. For example, the basics of a column removal analysis were outlined, but this research was limited. A more detailed look into progressive collapse of bridge elements or bridge systems could yield valuable information. Research into the mechanics of vessel-bridge impact, through physical testing or detailed finite element modeling, could lead to a better understanding of the dynamic effects of vessel impact and could produce guidelines on how much damage impact causes on bridge pier sections. This research would also provide information for dynamic load cases that could be applied to bridge models.

6.5 Final Remarks

This report has outlined modeling and analysis procedures that can be used to calculate the ultimate lateral strength of bridge elements and systems. It is the hope of the author that the methods and tools presented in this document will assist engineers in designing or analyzing bridge piers and bridge systems that are subject to vessel collision so that the catastrophic accidents of the last twenty-five years are not seen in the future.

References

AASHTO, (1991). *Guide Specification and Commentary for Vessel Collision Design of Highway Bridges*, American Association of State Highway and Transportation Officials, Washington, D.C.

AASHTO, (2004). *LRFD Bridge Design Specifications*, American Association of State Highway and Transportation Officials, Washington, D.C.

ACI 318-02, (2002). *Building Code Requirements for Structural Concrete and Commentary*, American Concrete Institute, Farmington Hills, MI.

ASCE, (1999). *Structural Design for Physical Security State of Practice,* American Society of Civil Engineers, Reston, VA.

Bentz, E.C., Collins, M.P., (2001). *RESPONSE-2000 User Manual*, University of Toronto, Toronto, Ontario Canada.

Consolazio, G.R., Cowan, D.R., (2003). "Nonlinear Analysis of Barge Crush Behavior and its Relationship to Impact Resistant Bridge Design", Computers and Structures, Vol. 81, No. 8-11, 547-557.

Consolazio, G.R., Cowan, D.R., Biggs, A., Cook, R.A., Ansley, M., Bollmann, H.T., (2005). "Full-Scale Experimental Measurement of Barge Impact Loads on Bridge Piers", Transportation Research Board (TRB) Annual Meeting CD-ROM, Washington, D.C.

FEMA 302, (1997). *NEHRP Recommended Provisions for Seismic Regulations for New Buildings and Other Structures*, Federal Emergency Management Agency, Washington, D.C.

Fujii, Y., (1978). "The Estimation of Losses Resulting from Marine Accidents", Journal of Navigation, Vol. 31, No. 1, 117-125.

Knott, M., Prucz, Z., (2000). *Vessel Collision Design of Bridges: Bridge Engineering Handbook*, CRC Press LLC, Washington D.C.

Liu, C. and Wang, T., (2001). "Statewide Vessel Collision Design for Bridges", Journal of Bridge Engineering, ASCE, Vol. 6, No. 3, 213-219.

Moehle, J.P., (1992). "Displacement-Based Design of RC Structures Subjected to Earthquakes", Earthquake Spectra, Vol. 8, No. 3, 403-428.

Modjeski and Masters Consulting Engineers, (1984). *Criteria For: The Design of Bridge Piers with Respect to Vessel Collision in Louisiana Waterways*, Prepared for the Louisiana Department of Transportation and Development and the Federal Highway Administration, Harrisburg, PA.

Patev, R.C., (2005). "Development of USACE Engineering Guidance for the Barge Impact Design of Navigation Structures", Transportation Research Board (TRB) Annual Meeting CD-ROM, Washington, D.C.

SAP 2000, (2002). *SAP 2000 Analysis Reference Manual,* Computers and Structures, Inc., Berkeley, CA.

TxDOT, (2001). *Bridge Design Manual*, Texas Department of Transportation, Austin, TX.

TxDOT, (1969). Intracoastal Canal Bridge State Highway 87 Construction Plans, Texas Department of Transportation.

Whitney, M.W., Harik, I.E., Griffin, J.J., Allen, D.L., (1996). "Barge Collision Design of Highway Bridges", Journal of Bridge Engineering, ASCE, Vol. 1, No. 2, 47-58.

VITA

Wyatt Robert Henderson was born in Minneapolis, MN on March 25, 1980 to Robert Johnson Henderson III and Karen Swanson Henderson. He completed his undergraduate degree with a B.S. in Civil Engineering from The University of Wisconsin-Madison in December, 2002. He started graduate school at The University of Texas at Austin in September of 2003. While attending graduate school he worked as a Graduate Research Assistant in the Department of Civil Engineering. He completed his graduate degree requirements in May, 2005, earning a Master of Science in Civil Engineering.

Permanent Address: 5035 Garfield Ave
Minneapolis, MN 55419

This thesis was typed by the author.