University of Southern Queensland

Faculty of Engineering and Surveying

Finite Element Analysis of a Composite Bridge Deck

A dissertation submitted by

Lindsay Edward Klein

in fulfilment of the requirements of

Courses ENG4111 and 4112 Research Project

towards the degree of

Bachelor of Civil Engineering

Submitted: November, 2006

Abstract

The complex nonlinear properties of concrete make it difficult to determine its true strength and response under loading conditions. This has often lead to many reinforced concrete structures such as bridge decks being over designed. Finite element analysis techniques such as the smeared crack approach have now been developed to model concrete structures with considerable accuracy.

This dissertation uses the general purpose finite element software ABAQUS to model a composite bridge deck that comprises of a reinforced concrete slab and longitudinal steel girders. Three separate models were produced with different girder spacings. Loading conditions were determined from the Australian Standard for Bridge Design and used on the structure to produce the worst effects. From this the response of the bridge deck was determined and an optimum girder spacing chosen. It was found that a four girder bridge deck would provide the optimum design for a two lane bridge to meet Australian Standards.

University of Southern Queensland

Faculty of Engineering and Surveying

<div style="border:1px solid black;">

ENG4111 Research Project Part 1 &

ENG4112 Research Project Part 2

</div>

Limitations of Use

The Council of the University of Southern Queensland, its Faculty of Engineering and Surveying, and the staff of the University of Southern Queensland, do not accept any responsibility for the truth, accuracy or completeness of material contained within or associated with this dissertation.

Persons using all or any part of this material do so at their own risk, and not at the risk of the Council of the University of Southern Queensland, its Faculty of Engineering and Surveying or the staff of the University of Southern Queensland.

This dissertation reports an educational exercise and has no purpose or validity beyond this exercise. The sole purpose of the course pair entitled "Research Project" is to contribute to the overall education within the student's chosen degree program. This document, the associated hardware, software, drawings, and other material set out in the associated appendices should not be used for any other purpose: if they are so used, it is entirely at the risk of the user.

Professor R Smith
Dean
Faculty of Engineering and Surveying

Certification

I certify that the ideas, designs and experimental work, results, analyses and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.

I further certify that the work is original and has not been previously submitted for assessment in any other course of institution, except where specifically stated.

Lindsay Edward Klein
Student Number: 0050009763
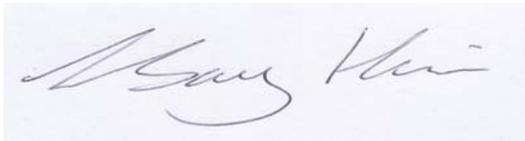
_____
Signature

1 November 06
_____
Date

Acknowledgements

The author would like to sincerely thank Dr Amar Khennane for his guidance and support throughout the course of this project.

Many thanks also has to go out to CESRC (Computational Engineering and Science Research Centre) at USQ for the use of their computers for the finite element modelling aspects of this project.

The author also wishes to thank his parents for their support and assistance in proofreading the dissertation.

Mr Lindsay Klein
November, 2006

Contents

List of Figures

List of Tables

Nomenclature

a	=	depth of rectangular stress block in concrete
ALF_i	=	accompanying lane factor
A_{st}	=	cross section area of torsional reinforcement
b	=	width of rectangular cross section
c	=	cover to reinforcing steel
C	=	compressive force in concrete
d	=	depth to resultant force in tensile steel at M_u
d_n	=	depth to neutral axis in a cross-section, measure from extreme compression fibre
D	=	overall depth of section
f'_c	=	characteristic compressive cylinder strength of concrete at 28 days
f_y	=	yield strength of steel
f_{sy}	=	yield strength of reinforcing steel
g	=	acceleration due to gravity, 9.81 m/s^2
k_u	=	ratio of neutral axis depth to depth of resultant force in tensile steel
L	=	length
M	=	mass
M	=	bending moment
M_u	=	ultimate strength in bending
T	=	tensile force in steel
V	=	volume
Z_x	=	section modulus about x-axis
w	=	weight per unit length
α	=	dynamic load allowance
γ	=	ratio of the depth of the assumed rectangular compressive stress block to d_n at M_u
$\gamma_{concrete}$	=	density of concrete, 2400 kg/m^3

γ_g =   dead load factor

σ =   stress

ϕ =   strength reduction factor / capacity factor

Chapter 1

Introduction

1.1   General Background Information

Bridges are a necessity in the transport network. They account for only short sections of road however they provide a convenient way of joining two inaccessible areas. For this reason they are used to cross rivers, creeks and other roads such as in the construction of overpasses on freeways. Bridges can range from small unnoticeable structures to large impressive man made engineering wonders that are recognised worldwide. Common examples of these are the Sydney Harbour Bridge in Australia and the Golden Gate Bridge in the United States of America. These structures are not built this way just to look impressive. All their components are needed to support the large loads that are imposed on them.

Versatile and strong materials are needed in the construction of bridges. The most common of these materials today are reinforced concrete, prestressed concrete and steel. Reinforced concrete in particular is very popular in the construction of small bridges since it is very simple to construct a concrete deck that is supported by girders.

Reinforced concrete does not act like typical materials when loaded. Concrete by itself is very strong in compression, however its tensile strength is very low in comparison. A standard concrete mix could have a compressive strength of 32 MPa and a tensile

strength of only 2 – 3 MPa. To overcome the lack of tensile strength steel bars are added to the concrete. Therefore in a concrete section the compression forces are taken by the concrete while the tensile forces are taken by the area of steel that is provided in the tensile part of the section. This combination of two different materials with very different properties make the analysis of reinforced concrete more complex than that of a steel structure. In addition concrete does not have a linear relationship between stress and strain. Once the concrete is loaded to about 40 % of its compressive strength the stress-strain relationship becomes increasingly nonlinear by the formation of microcracks in the interaction between the aggregate and the mortar [1]. Another property of concrete that makes accurate structural analysis more difficult is that under biaxial compression, its compressive strength increases. From test results, maximum gain in compressive strength is obtained when the ratio of the perpendicular stresses applied is 0.6 [1]. Due to the complex nature of these properties, simplified design methods were produced that do not accurately predict the true behaviour of concrete structures.

By applying the finite element method, it is possible to model concrete a lot more accurately. The finite element method was first developed by Richard Courant and applied in 1943 in his work on solving a torsion problem [2]. It was not until the 1960's and 70's that it was given the finite element name and further developed to solve a host of problems, from heat transfer to fluid dynamics and structural analysis [3]. A finite element model consists of a mesh of elements defined by nodes that describe the geometrical properties of an object. Section properties are added to describe the behaviour of the elements. Forces are applied to the nodes, from this the elements reaction can be determined by parameters such as stresses and displacements. Now with the increase in the popularity of the finite element method, more accurate models of material behaviours have been included in finite element software. Concrete can now be accurately modelled and the analysis can even accounting for the cracking that occurs.

There are various ways of modelling reinforced concrete with the finite element method. One way of defining reinforced concrete is by superimposing the steel reinforcement in the generated mesh of plain concrete. This way the concrete is able to be defined separately from the steel. However in this situation the bond between the

steel and concrete is not accounted for. To overcome this tension stiffening is added to the concrete to account for the steel and concrete bond. The finite element analysis software ABAQUS [4] is able to effectively model reinforced concrete by the method described. It is also able to apply a nonlinear analysis to the concrete to derive an accurate prediction of the stresses and deformation that will occur under loading. This makes ABAQUS [4] the ideal finite element analysis tool to model the effect that girder spacing has on a reinforced concrete bridge deck.

1.2   Aims and Objectives

This project aims to use the finite element method to determine the optimum girder spacing on a composite bridge deck. The bridge under analysis will be comprised of a reinforced concrete bridge deck supported by steel I-beam girders. A cross section of the design is shown in Figure 1.1.
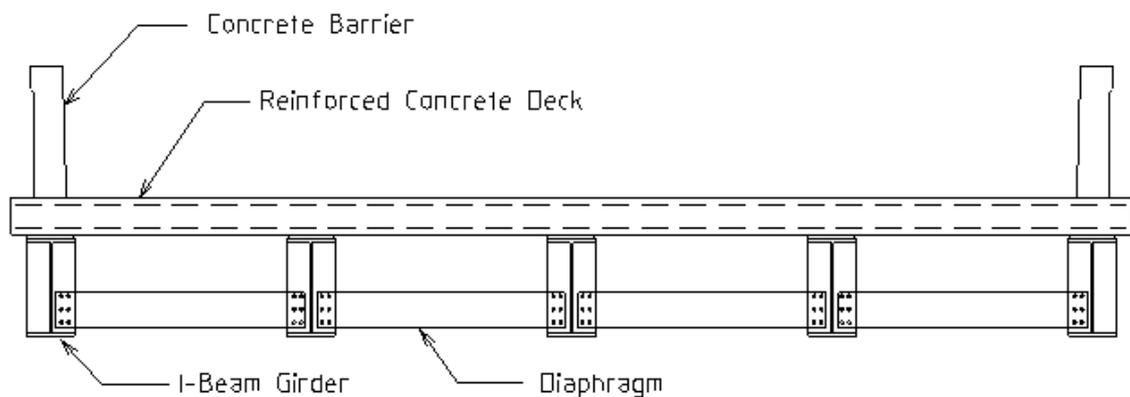
Figure 1.1: Cross section of composite bridge deck.

In order to achieve this aim the following objectives have to be met:

1. Research background information of the non-linear behaviour of concrete and how it is applied in a finite element model.

2. Determine the loading requirements of the bridge from the relevant Australian Standards.

3. Conduct a preliminary design of the bridge.

4. Learn how to effectively use the ABAQUS [4] finite element software.

5. Model the bridge deck using the finite element method.

6. Employ the finite element model to determine the effect of girder spacing on the deck response.

1.3   Structure of Dissertation

Chapter 1 gives background information on the topic and describes the objectives needed to achieve the aim of the project.

Chapter 2 gives some background on the use of finite element technique in the analysis of reinforced concrete structures. It also explains how the finite element method is used to effectively model the nonlinear behaviour of concrete.

Chapter 3 contains the loading requirements set out by AS 5100.2 [5]. These loads are then used to determine a preliminary sizing of members for the bridge deck to be used in the finite element models.

Chapter 4 contains the methodology relating to production of a finite element model. This starts off with the basics of the ABAQUS [4] software then continues on to the methodology behind a preliminary model that was produced with the student edition of ABAQUS [4]. The chapter finishes with the methodology behind the three finite element models that were used for analysis.

Chapter 5 involves the analysis and discussion of results for the three separate bridges with different girder spacings. From these results the optimum girder spacing can be determined.

Chapter 6 provides a summary of the analysis undertaken and the conclusions drawn. This chapter will also give details on future work that can be conducted in relation to the finite element.

The appendices provide additional material to the project work. These include the project specification and the input files for the finite element models.

Chapter 2

Background Research

2.1 Introduction

This chapter will provide information on the properties of concrete and how they are applied in a finite element model. Reinforced concrete is a composite material containing a mix of a number of different materials including sand, aggregate, cement and steel bars. This mix of materials means concrete does not have well defined constitutive properties like steel. The finite element model then has to apply these nonlinear properties to produce results that will match the true behaviour of the reinforced concrete.

2.2 Uniaxial Stress-Strain Relationship

The uniaxial compressive stress-strain relationship for concrete is linear for small stress values. However once stresses reach approximately 40% of the compressive strength of the concrete the relationship becomes increasingly nonlinear [1]. This is shown in Figure 2.1.
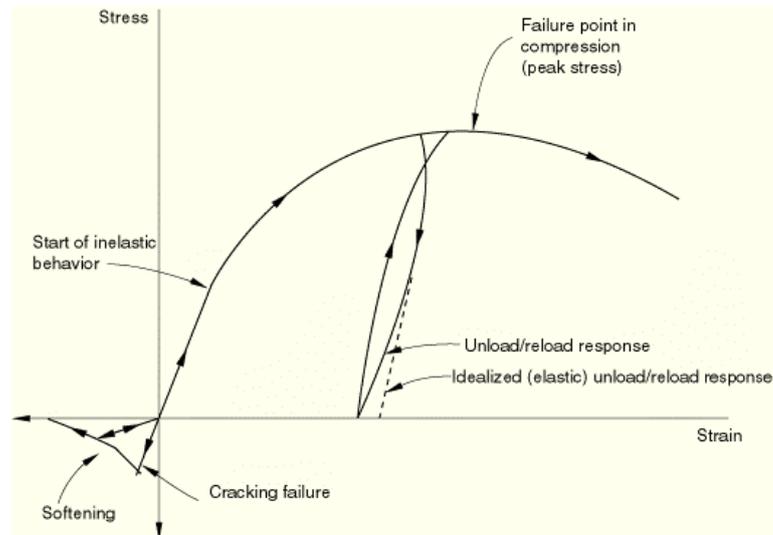
Figure 2.1:   Uniaxial stress-strain relationship for concrete [6].

If the load applied increases, the stress value will reach the peak of the nonlinear curve. At this point the concrete is at its maximum compressive strength and any additional loads will cause failure of the bond between the aggregate and the cement paste. ABAQUS [4] models the compressive stress-strain relationship as elastic and plastic. Elastic properties are given for the initial linear behaviour. Then specific concrete properties are used to state the initial yield point and the stress and strain values at which the concrete fails.

When the concrete has been loaded into the inelastic range and is then unloaded some elasticity will be lost. While this loss of elasticity is accounted for in the model the unload/reload response is idealised as a straight line.

The stress-strain relationship in tension is considered linear to its failure point. This is considered to be 8-10 % of the total compression stress [6]. Once cracks form the concrete undergoes softening where it is still able to withstand small tensile stresses as can be seen in Figure 2.1. The effect of this in relation to reinforced concrete will be explained in the section on tension stiffening.

2.3 Combined Stresses

The compressive strength of concrete increases when under biaxial stress. This increase depends on the magnitude of the lateral compressive stress. Test results have shown the maximum strength gain is achieved when σ_2 / σ_1 is approximately 0.6 [1]. The failure envelope representing biaxial stress is shown in Figure 2.2. ABAQUS [4] uses this relationship in its analysis of concrete and also determines a crack detection surface to account for biaxial compression-tension.
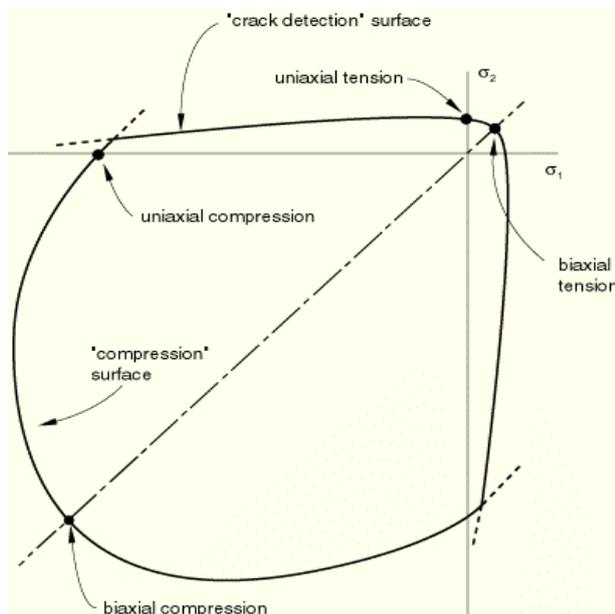
Figure 2.2: Failure envelope for concrete biaxial stress [6].

2.4 Tension Stiffening

When concrete reaches its maximum tensile strength cracks begin to form perpendicular to the direction the stress has been applied. However the concrete between these cracks still has the ability to carry stress. Tension stiffening is a term used when the cracked concrete provides stiffness in conjunction with the reinforcement. It describes the strain softening behaviour of the concrete after it has cracked and is used to account for the interaction between the steel reinforcement and the concrete. ABAQUS [4] allows the

user to define this softening behaviour for concrete models being analysed. The stress-strain relationship of a tension stiffening model is shown in Figure 2.3.
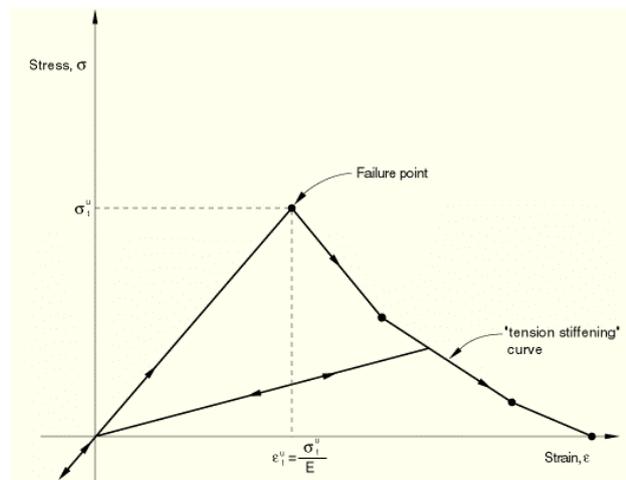
Figure 2.3: Tension stiffening model for concrete [6].

2.5 Smeared Crack Approach

ABAQUS [4] uses the smeared crack approach in its analysis of plain concrete. For this analysis individual cracks are not analysed. Instead when the concrete reaches its failure surface the stiffness properties of the material are changed in the direction orthogonal to the crack. This then acts like a crack has formed. As the material properties are only evaluated at the integration points the alteration of the stiffness properties of the material effect the region from which these properties are evaluated, therefore smearing the crack over a whole region. Also there is no permanent strain associated with the cracking. This allows the cracks to close again if a compressive load is applied.

The smeared crack approach is used in the analysis of reinforced concrete by adding a layer of steel reinforcement within the concrete. In this situation the steel acts independently from the concrete. To describe the interaction between the two materials the tension stiffening model is used.

2.6   Finite Element Analysis Accuracy for Concrete

When using finite element analysis on a composite material such as reinforced concrete there is a chance that the analysis will not produce suitable results. For this reason research has been undertaken into the accuracy of finite element methods for reinforced concrete.

There are other analysis techniques such as the discrete method that can be used in place of the smeared crack approach. It has been stated however that the smeared crack approach is often the more attractive method to define the cracking of the concrete [7]. The advantage is that there is no need to continuously redefine the element mesh as in the discrete method, which significantly slows down the analysis.

There have been concerns that the smeared crack approach introduces mesh sensitivity into the solutions so the finite element results do not converge to a unique result [6]. ABAQUS [4] is able to deal with this to some extent by using Hilleborg's [8] approach that can deal with this problem for practical purposes. However the ABAQUS User Manual [6] states that by having each element contain steel reinforcement, the mesh sensitivity is reduced. This depends on an adequate amount of tension stiffening being added to simulate the interaction between the concrete and the steel reinforcement.

Previous research has been conducted into the accuracy of using the finite element method in the analysis of reinforced concrete bridge decks. Biggs et al. [9] used the smeared crack approach in comparing experimental test results of a slab to those produced from the finite element method. Results proved accurate enough for Biggs et al. [9] to recommend the method to the Virginia Department of Transportation for the analysis of reinforced concrete bridges. Huria et al. [10] compared their finite element analysis results with a decommissioned bridge deck that had been tested to failure. The results obtained from the smeared crack approach closely matched those from the experimental testing.

2.7 Summary

The complex nonlinear properties of concrete such as the uniaxial stress-strain relationship and the increase in compressive strength under biaxial stress has made it difficult to predict its true behaviour. Finite element analysis techniques such as the smeared crack approach have now been developed to closely match these properties. However as with any theoretical modelling there are always questions about the accuracy of the results. Research has been conducted comparing experimental testing results with those produced from the finite element analysis. The conclusions from such work have shown that concrete can be accurately modelled with the finite element method.

Chapter 3

Preliminary Bridge Design

3.1   Introduction

This project involves the design of a bridge using the finite element software ABAQUS [4]. The bridge is comprised of a concrete deck supported by longitudinal steel girders. All bridges in Australia must meet Australian Standards to ensure their safety. In particular, Part 2 of the Bridge Design Code [5] is important as it states what loads the bridge will be required to support. Lane and shoulder widths also must meet the Australian Standards to ensure safe passages of vehicles across the bridge.

Once the loading and overall dimensions of the bridge are accounted for a rough design is obtained from traditional design methods. This is done for a number of reasons. A starting point is needed with the finite element model in terms of girder sizes and slab thicknesses. The author had no previous experience in designing bridges. Therefore the approximate depth of the concrete slab and the girder sizes are not known. The design can also be used for a rough comparison with the results from the finite element analysis. If the finite element analysis produces results that are greatly different from those calculated in this section then there is a good chance a mistake has been made.

The following sections of this chapter outline the dimensions of the bridge, the loading requirements and present a preliminary design of member sizes required to meet these loads.

3.2   Bridge Dimensions and Features

A typical cross section of a bridge consisting of steel girders supporting a reinforced concrete slab is shown in Figure 3.1.
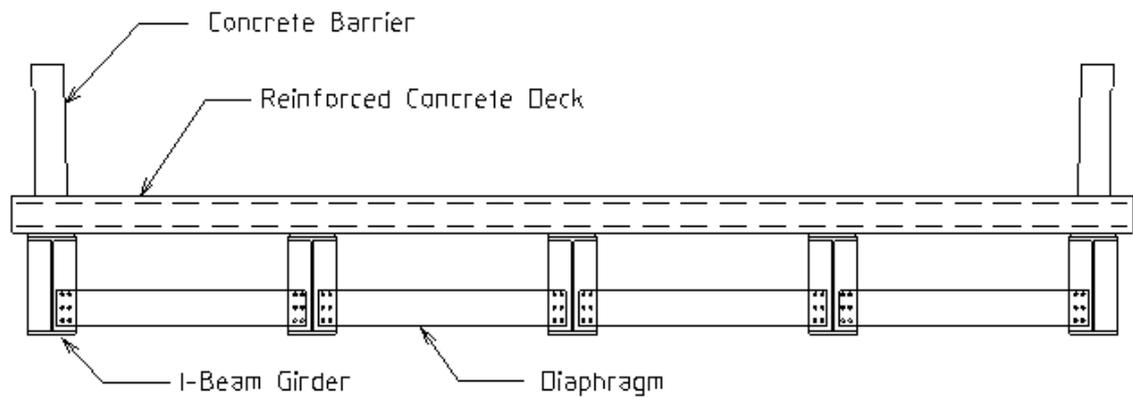
Figure 3.1: Typical cross section of bridge deck.

3.2.1   Bridge Features

The steel I-beam girders run longitudinally between the supports. Sitting on top of these is the reinforced concrete slab with reinforcement in the top and bottom for negative and positive bending moments respectively. The concrete slab is attached to the top flange of the girders with shear studs. These transmit shear loads along the plane of the slab. Diaphragms run between the girders and are used to provide lateral restraint and counteract buckling of the girders. Concrete safety barriers are placed on each side for vehicle crash safety.

3.2.2   Bridge Dimensions

The first item to determine for the bridge is its dimensions. It is assumed that the bridge runs over a small creek and therefore does not have a great span. The only limit on the span chosen is the maximum moment that can be supported by a standard I-section. The

maximum span of a Welded I-Section produced by One Steel (a producer of structural sections in Australia) is 18 m. These welded sections range in depth from 700 mm to 1200 mm [11]. To ensure these beams can take the moments applied, the distributed load that the maximum size section can withstand over an 18 m span is calculated.
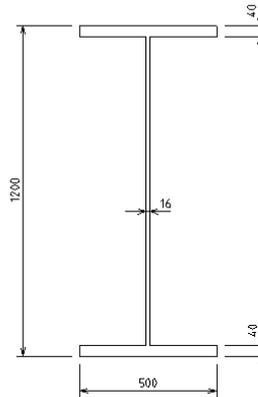
Figure 3.2: 1200WB455 cross section.

1200WB455 Section properties:

$$f_y = 280 MPa$$
$$Z_x = 25600 \times 10^3 mm^3$$

Maximum moment girder can support:

$$M = Z_x \sigma$$
$$M = 25600 \times 10^3 \times 280$$
$$M = 7168 kNm$$

Maximum distributed load 18 m long girder can support:

$$M = \frac{wL^2}{8}$$
$$w = \frac{8M}{L^2}$$
$$w = \frac{8 \times 7168}{18^2}$$
$$w = 177 kN/m$$

The result determined the girder could support 177 kN/m, which is over 17 tonne per metre for the entire length of the girder. Therefore it can be assumed that an 18 m long girder can easily support the weight of traffic above it as long as the spacing between girders is adequate. This quick check does not account for buckling. However if buckling is a problem in the finite element design then extra diaphragms can be added to counteract the buckling effect.

The width of the bridge depends on the Australian Standards for lane and shoulder widths. In particular it is dependant on the amount of traffic the bridge will carry. It is assumed the bridge spans a creek in the country and therefore will not experience high traffic flows. It is also assumed that there is no need for a pedestrian walkway. Table 9.5 of AS5100.1 [12] states edge clearances from the edge of the traffic lane to the face of the safety barrier for bridges without walkways. From this table with a traffic volume of 500 – 5000 vehicles per day the edge clearance required is 1000 mm. As for lane widths the Austroads publication, Rural Road Design [13] states that the desirable lane width on rural roads should be 3.5 m for traffic volumes of 1000 – 3000 vehicles per day.

The last item to determine the exact geometric requirements of the bridge is the concrete barriers placed on either side. A standard barrier was chosen from AS/NZS 3845 Road Safety Barrier Systems [14]. The 1100 High VCB Road Safety Barrier System chosen complies with Test Level 3. This is an Australian Standard test level that will restrain all passenger cars and four wheel drives in the event of an impact. The barrier is shown in Figure 3.3.
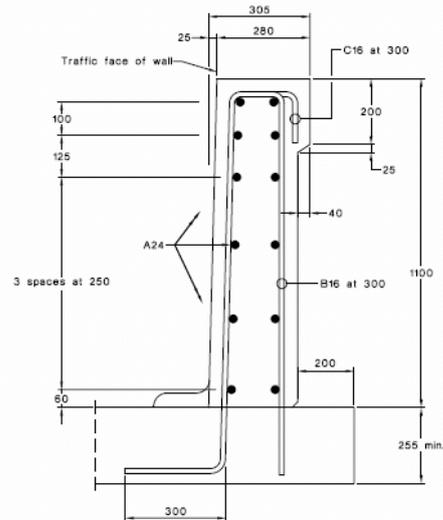
Figure 3.3:   1100 High VCB road safety barrier system [14].

Taking into account the values above, the geometric layout of the bridge is shown in the plan view, Figure 3.4.
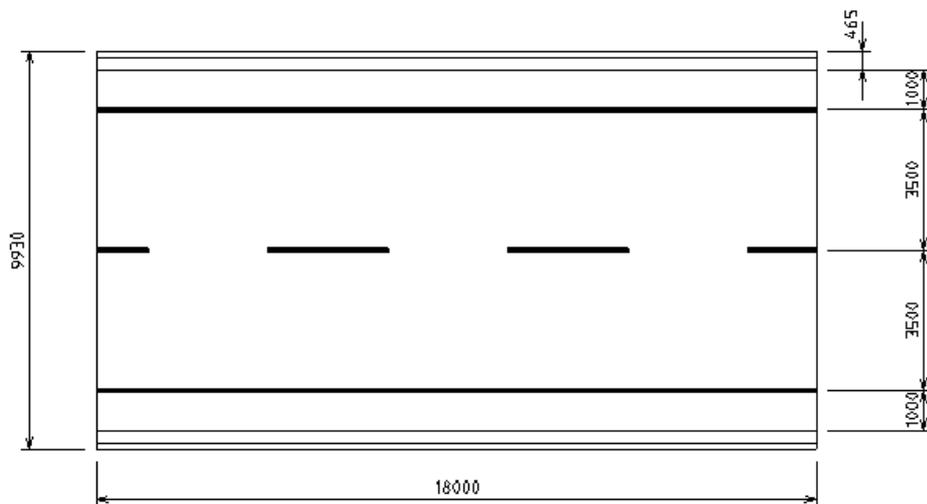
Figure 3.4:   Plan view of bridge.

3.3   Loading Requirements

To provide a realistic analysis the loading on the bridge must comply with AS 5100.2 [5]. To determine the most adverse effects on the bridge there are a number of different

loads that are required to be applied by the Australian Standard. Since the aim of this dissertation is to find the optimum girder spacing, braking forces will be ignored.

The loads defined by AS 5100.2 [5] that are applied to the bridge are listed below:

- Dead load.
- W80 wheel load.
- A160 axle load.
- M1600 moving traffic load.
- S1600 stationary traffic load.

These loads must have factors applied to determine the design loads for ultimate and serviceability limit states. These are listed below:

- (γ_g) dead load factor.
- (α) dynamic load allowance.
- (ALF_i) accompanying lane factor.

3.3.1   Load Factors for Dead Load of Structure

Clause 5.2 of AS 5100.2 [5] states the load factors required for the dead load of the structure. The factors that are relevant to this bridge are listed in Table 3.1.

Table 3.1: Load factors for dead load of structure [5].

Type Of Construction	Ultimate Limit States Where Dead Load		Serviceability Limit States
	Reduces Safety	Increases Safety	
Steel	1.1	0.9	1
Concrete	1.2	0.85	1

3.3.2   Dynamic Load Allowance

The dynamic load allowance is a factor accounting for dynamic and vibratory effects of the bridge under loading conditions. It is in essence the static equivalent of the dynamic effects of vehicles moving over the bridge with road surface irregularities [5]. It is applied to the load in question with the following expression:

$$\text{design action} = (1 + \alpha) \times \text{load factor} \times \text{action under consideration}$$

The value of the dynamic load allowance for the appropriate loading is shown in Table 3.2.

Table 3.2: Dynamic Load Allowance [5].

Loading	Dynamic Load Allowance (α)
W80 wheel load	0.4
A160 axle load	0.4
M1600 tri-axle group	0.35
M1600 load	0.3
S1600 load	0

3.3.3   Accompanying Lane Factor

When more than one lane is loaded then the loading applied to additional lanes must be multiplied by the accompanying lane factors. These are given in Table 3.3.

Table 3.3: Accompanying Lane Factors [5].

Standard Design Lane Number	Accompanying Lane Factor (ALFi)
1 lane loaded	1.0
2 lanes loaded	1.0 for first lane; and
	0.8 for second lane

3.3.4   Load Factors

Clause 6.10 of AS 5100.2 [5] states the load factors required for the ultimate and serviceability loads on the structure. The factors that are relevant to the bridge being designed are listed in Table 3.4

Table 3.4: Load factors for design road traffic loads [5].

Traffic Load	Limit State	
	Ultimate	Serviceability
W80 Wheel Load	1.8	1
A160 Axel Load	1.8	1
M1600 Moving Traffic Load	1.8	1
S1600 Stationary Traffic Load	1.8	1

3.3.5   W80 Wheel Load

This models an individual heavy wheel load and consists of an 80 kN load distributed over a rectangular contact area 400 mm × 250 mm. It is placed in any position on the bridge that will give the most adverse effect.

3.3.6   A160 Axle Load

The A160 Axle Load is similar to the W80 wheel load. However it models a single heavy axle so there are two W80 wheel loads placed at 2000 mm centres. The application of the load in a standard design lane is shown in Figure 3.5.
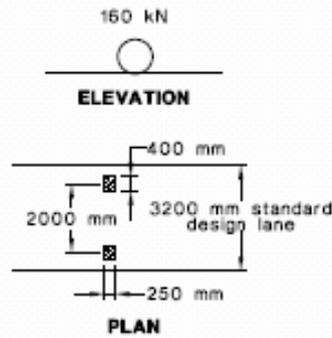
Figure 3.5: A160 Axle Load [5].

3.3.7   M1600 Moving Traffic Load

This load models a moving stream of traffic across the bridge. The entire load is placed in a standard 3.2 m wide design lane and continues along the length of the bridge. The M1600 load consists of a uniformly distributed load placed over the width of the design lane plus a number of tri-axial groups to represent a constant stream of trucks passing over the bridge. To achieve the most adverse effects the spacing of the tri-axial groups may be modified and the distributed load may be continuous or discontinuous for any length that is necessary. The application and value of the M1600 design loads are defined in Figure 3.6.
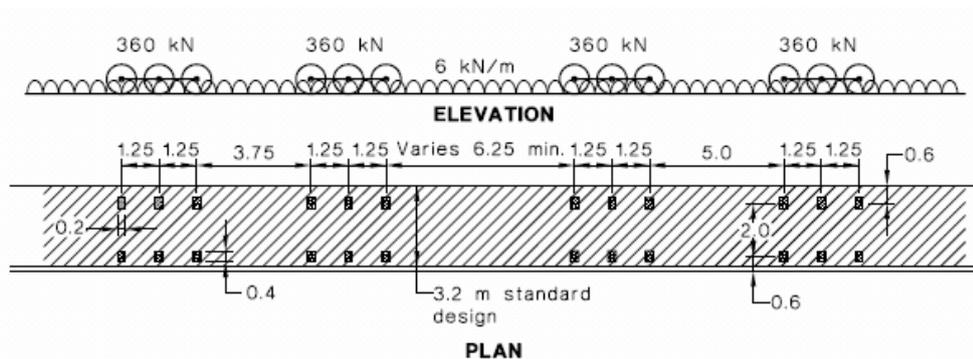
Figure 3.6: M1600 Moving Traffic Load [5].

3.3.8   S1600 Stationary Traffic Load

The S1600 stationary traffic load models a stream of stationary traffic on the bridge. Like the M1600 load it consists of a uniformly distributed load over a standard design lane plus a series of tri-axial groups. To achieve the most adverse effects the spacing of the tri-axial groups may be modified and the distributed load may be continuous or discontinuous for any length that is necessary. The application and value of the S1600 design loads are defined in Figure 3.7.
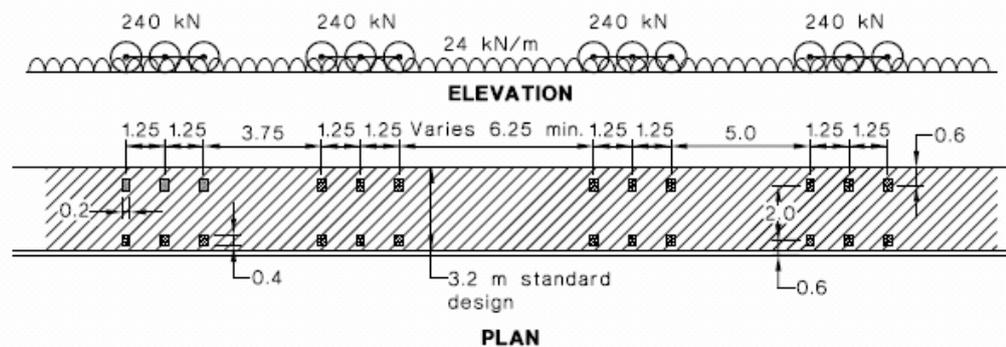
Figure 3.7: S1600 Stationary Traffic Load [5]

3.4 Preliminary Sizing of Structural Members

The preliminary sizing of members is determined from the worst effect produced by the design loads defined above. Only a rough size for the structural members is needed. Bending moments are analysed from the design loads to determine rough section sizes. To determine the respective bending moments the structural design software Multiframe [15] is used.

3.4.1   Preliminary Design of the Concrete Slab

The slab has a total width of 9.93 m with a length of 18 m. The dead loads of the slab are first determined and then combined with the design loads to calculate the maximum

bending moments. From these moments the slab thickness and the amount of steel reinforcement are determined. For this preliminary design the slab will be supported by five girders, the spacing of which is shown in Figure 3.8.
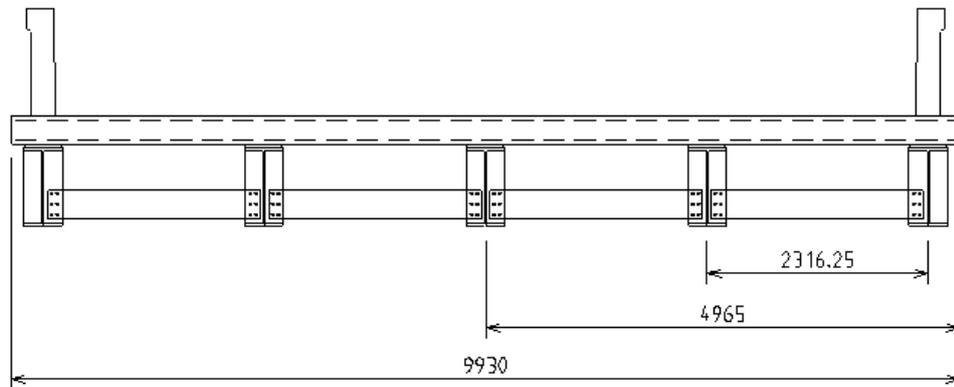
Figure 3.8: Girder spacing for preliminary five girder bridge.

To design the slab a free-body diagram is produced for the cross section of the bridge deck where the girders are represented as supports. This produces primary bending moments in one direction because the design is for a one-way slab. All calculations therefore, will involve a 1 m design strip.

3.4.1.1   Dead Loads

It is initially assumed the thickness of the concrete slab is 250 mm. In addition to the dead load of the slab there are permanent concrete barriers on each side of the bridge. These impose a permanent distributed load along each side of the slab. The calculations of the dead loads are shown below.

Density of concrete:

$$\rho_{concrete} = 2400 kg/m^3$$

Dead load factor:

$$\gamma_g = 1.2$$

Volume of 1 m wide design strip of slab:

$$V = 1 \times 1 \times 0.25 = 0.25m^3$$

Volume of 1 m length of concrete barrier using dimensions from Figure 3.3:

$$V = 0.265 \times 1.1 - \frac{0.025 \times 1.1}{2} + 0.2 \times 0.04 = 0.286m^3$$

Dead load for 1 m design strip of slab:

$$W = \gamma_g V \rho_{concrete} g$$
$$W = 1.2 \times 0.25 \times 2400 \times 9.81$$
$$W = 7.06kN/m$$

Dead load for 1 m length of concrete barrier:

$$W = \gamma_g V \rho_{concrete} g$$
$$W = 1.2 \times 0.286 \times 2400 \times 9.81$$
$$W = 8.08kN/m$$

3.4.1.2 W80 Wheel Load

The design action for the W80 wheel load is as follows:

$$\text{Design action} = (1 + \alpha) \times \text{load factor} \times \text{action}$$
$$\text{Design action} = (1 + 0.4) \times 1.8 \times 80$$
$$\text{Design action} = 201.6kN$$

The slab is designed as a cross sectional strip 1 m wide and the design action acts over an area 0.4 m × 0.25 m. Assuming this area is orientated so the length of 0.4 m runs across the deck and the length of 0.25 m runs the length of the deck the distributed load in a 1 m wide design cross section is:

$$\frac{201.6kN}{0.4m} = 504kN/m$$

The wheel area is spread out over an area 0.4 m × 1 m. This does not produce the exact bending moment generated from the wheel load however it provides a good

approximation. The distributed load is placed on a continuous beam in Multiframe [15] along with the dead load of the deck to determine the maximum bending moment. The bending moment produced is shown in Figure 3.9.
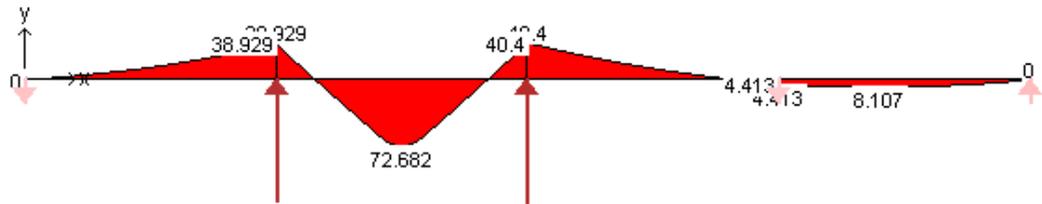
Figure 3.9: Bending moment diagram produced from W80 load.

3.4.1.3   A160 Axle Load

The design action for the A160 axle load is two W80 wheel loads placed two meters apart. The load is imposed so one wheel is placed in the centre of two supports to generate the maximum bending moment possible. The maximum bending moment from this load plus the dead load is shown in Figure 3.10.
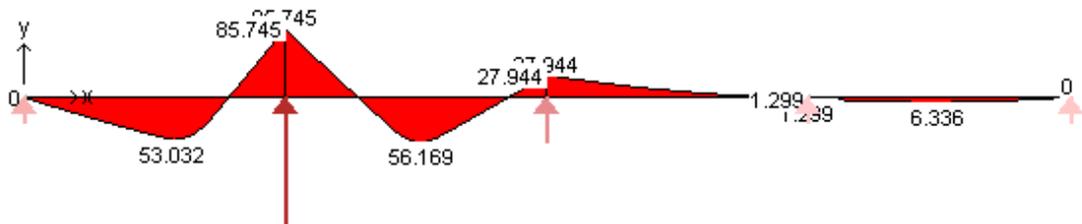
Figure 3.10: Bending moment diagram produced from A160 axle load in one lane.

The bridge being modelled contains two lanes so the A160 axle load is also imposed on the second lane. The loads in the second lane are multiplied by the accompanying lane factor which is 0.8. Therefore each distributed wheel load in the second lane is 403.2 kN/m. The bending moment diagram produced is shown in Figure 3.11.
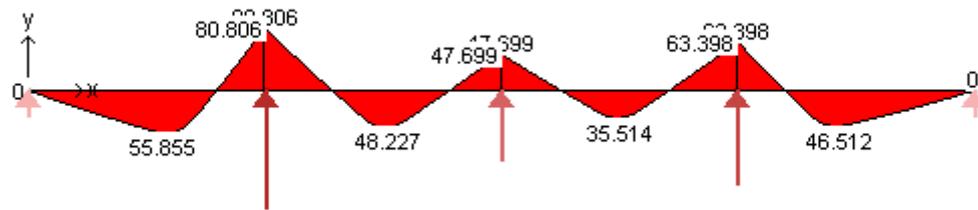
Figure 3.11: Bending moment diagram produced from A160 axle load in both lanes.

3.4.1.4   M1600 Moving Traffic Load

The M1600 load is comprised of 360 kN imposed over six separate areas that represent the wheel footprints of a truck plus a 6 kN/m distributed load over a 3.2 m lane. The design action on one wheel area is:

$$\text{Design action} = (1+\alpha)\times \text{load factor} \times \text{action}$$
$$\text{Design action} = (1+0.35)\times 1.8 \times 60$$
$$\text{Design action} = 145.8 kN$$

The design action of 145.8 kN is distributed over an area 0.4 m × 0.2 m. This area is orientated so the length of 0.4 m runs across the deck and the length of 0.2 m runs the length of the deck. The distributed load in a 1 m wide design cross section is:

$$\frac{145.8 kN}{0.4m} = 364.5 kN/m$$

The 6 kN/m distributed load is spread over a 3.2 m design lane. All calculations are based on a 1 m design strip so the distributed load for this design strip is calculated as follows:

$$\frac{6kN/m}{3.2m} = 1.875 kN/m^2$$

The design action is:

$$
\begin{aligned}
\text{Design action} &= (1 + \alpha) \times \text{load factor} \times \text{action} \\
\text{Design action} &= (1 + 0.3) \times 1.8 \times 1.875 \\
\text{Design action} &= 4.39 kN / m^2
\end{aligned}
$$

These loads are combined with the dead load of the deck to produce the bending moment diagram for one lane loaded. This is shown in Figure 3.12.
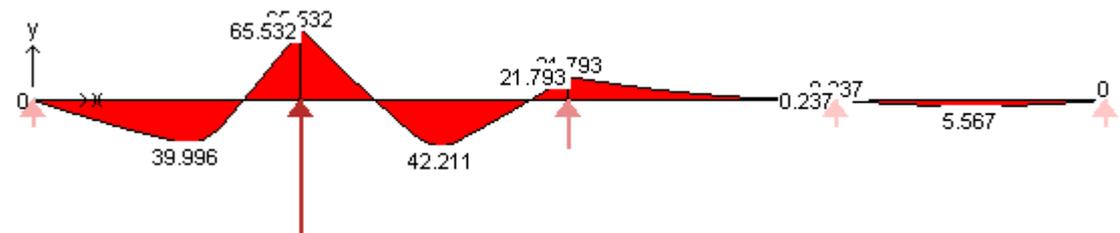
Figure 3.12: Bending moment diagram produced from M1600 traffic load in one lane.

The second lane is loaded the same as the first however all loads are multiplied by the accompanying lane factor of 0.8. The bending moment diagram is shown in Figure 3.13.
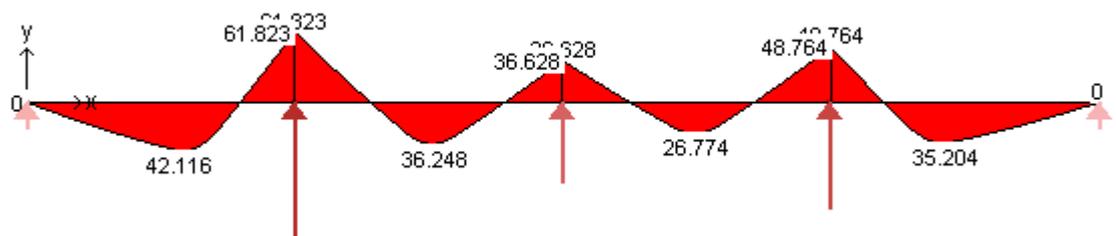
Figure 3.13: Bending moment diagram produced from M1600 traffic load in both lanes.

3.4.1.5 S1600 Stationary Traffic Load

The S1600 load is defined the same way as the M1600 load with loads over separate wheel areas and a distributed load over a design lane. In this case there is 240 kN

distributed over six wheel areas and 24 kN/m distributed over the 3.2 m design lane. The design action on one wheel area is:

$$\text{Design action} = (1+\alpha) \times \text{load factor} \times \text{action}$$
$$\text{Design action} = (1+0) \times 1.8 \times 40$$
$$\text{Design action} = 72kN$$

The design action of 72 kN is distributed over the same area as the M1600 load and is orientated the same way. The load is therefore:

$$\frac{72kN}{0.4m} = 180kN/m$$

The distributed load over a 1 m design strip:

$$\frac{24kN/m}{3.2m} = 7.5kN/m^2$$

The design action is therefore:

$$\text{Design action} = (1+\alpha) \times \text{load factor} \times \text{action}$$
$$\text{Design action} = (1+0) \times 1.8 \times 7.5$$
$$\text{Design action} = 13.5kN/m^2$$

These loads applied to one lane plus the dead load of the deck gave the following bending moment diagram, Figure 3.14.
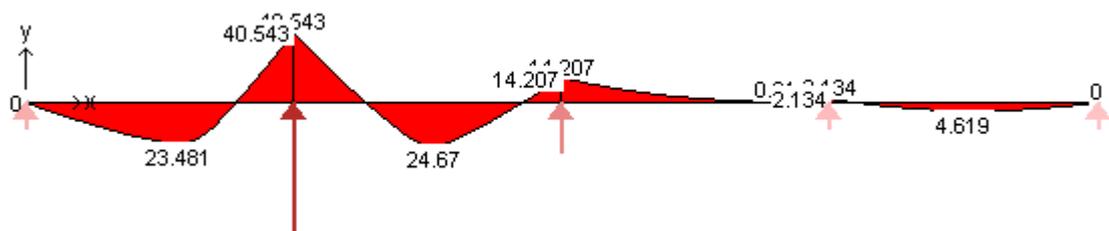
Figure 3.14: Bending moment diagram produced from S1600 traffic load in one lane.

Again the second land is loaded the same as the first and all loads multiplied by the accompanying lane factor of 0.8. Results of this are shown in Figure 3.15.
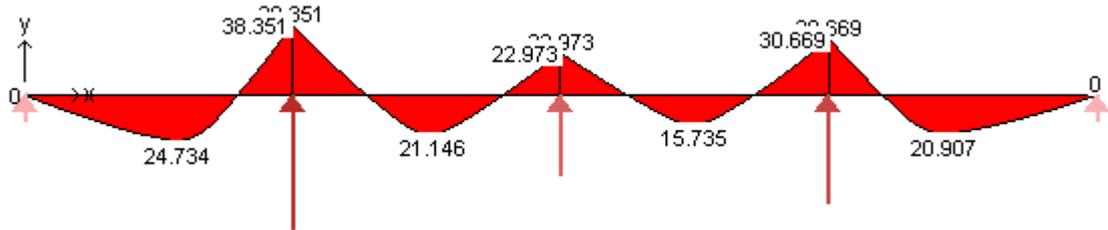
Figure 3.15: Bending moment diagram produced from S1600 traffic load in both lanes.

The maximum moments from the analysis are tabulated in Table 3.5. The W80 wheel load produced the greatest positive moment of 72.7 kNm. The greatest negative moment is 85.7 kNm. This is produced from the A160 axle load when only one lane is loaded.

Table 3.5: Maximum moments produced from loading combinations on deck.

Loading	Positive Moment (kNm)	Negative Moment (kNm)
W80	72.7	40.4
A160 One Lane	56.2	85.7
A160 Two Lanes	55.9	80.8
M1600 One Lane	40.0	65.5
M1600 Two Lanes	42.1	61.8
S1600 One Lane	24.7	40.5
S1600 Two Lanes	24.7	38.4

It is noted that the weight of the concrete barriers are not added into the calculations for the moments. This is because the barriers run directly on top of the outside girders and therefore do not impose any bending loads on the deck.

3.4.1.6   Concrete Slab Sizing

In this section a rough design of the concrete slab is undertaken. It sole purpose is to give approximate dimensions of the concrete slab to be used in the model, such as the amount of steel reinforcement and the depth of the slab. Therefore this design is based solely on the stresses imposed by the moments calculated in the previous sections.

The first area considered in the design is the durability requirements. It was assumed the bridge is in a location in South East Queensland and is within 50 km of the coastline. Therefore from Table 4.3 in AS 5100.5 [16] the exposure classification is B1 and from Table 4.5 the minimum characteristic strength is 32 MPa. Abrasion requirements are required to be met in accordance with AS 5100.5 [16]. The minimum compressive strength for abrasion for pneumatic-tyred traffic heavier than 3 tonne gross mass is 32 MPa. From this it has been decided to use 40 MPa concrete. This was done because the bridge deck is a major structural element therefore concrete stronger than the minimum required can only improve the design.

With the strength of the concrete decided the cover required for the steel reinforcement is determined. Standard compaction and formwork is used for a situation like this, as the slab would be poured on site. Therefore from Table 4.10.3(A) in AS 5100.5 [16] the nominal cover to the reinforcement is 40 mm.

With durability requirements accounted for the thickness of the slab and reinforcement is determined. The rectangular stress block approach is used to determine the ultimate strength in bending of the slab. Figure 3.16 shows a diagram of the rectangular stress block approach. This states the maximum strain in the extreme compression fibre of concrete is 0.003.
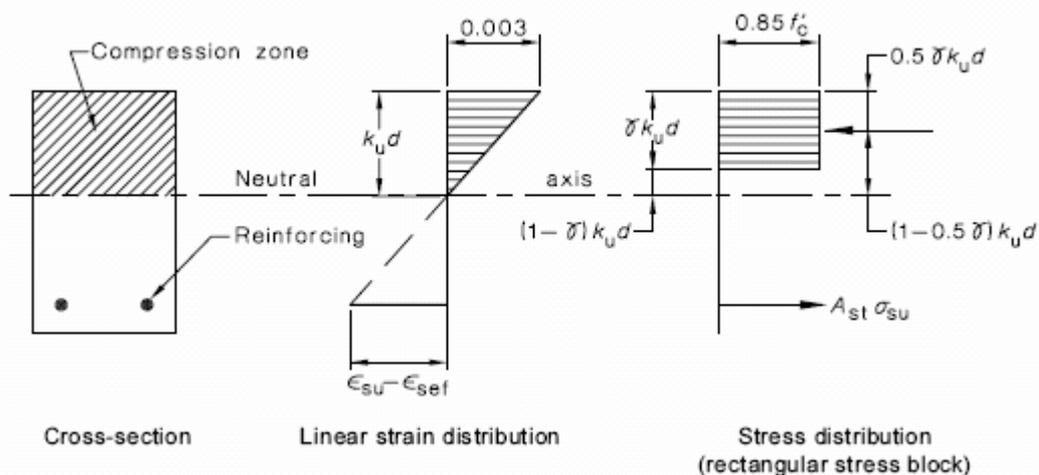
Figure 3.16: Rectangular stress block at ultimate strength [16].

The value of k_u must be equal to or less than 0.4 to ensure the section has a ductile failure. In this situation the steel yields before the concrete in compression fails, therefore if the structure is overloaded it will not catastrophically collapse.

Reinforcement is run in both the top and bottom of the slab because there are positive and negative moments. However as this is a basic design the section sizes will be determined by only accounting for the steel in tension. The slab thickness is initially assumed to be 250 mm deep with 12 mm reinforcement bars placed at 100 centres.

Predefined values:

$$\text{reo size} = 12mm \text{ dia bars}$$
$$\text{reo spacing} = 100 \text{ mm crs}$$
$$c = 40mm$$
$$D = 250mm$$
$$f'_c = 40MPa$$
$$f_{sy} = 500MPa$$

Calculating γ which is ratio of the depth of the assumed rectangular stress block to $k_u d$ when the structure is under bending or bending and compression:

$$\gamma = 0.85 - 0.007(f'_c - 28)$$
$$\gamma = 0.85 - 0.007(40 - 28)$$
$$\gamma = 0.766$$

Assuming the reinforcement bar diameter is 12 mm the effective depth of the cross-section is:

$$d = D - c - \tfrac{1}{2} bar \ dia.$$
$$d = 250 - 40 - 6$$
$$d = 204mm$$

The reinforcement used is 12 mm bars run at 100 mm centres, therefore the total area of steel over a 1 m design strip is equal to 1130 mm^2. The ultimate moment capacity of the section is now determined.

Compression force in concrete:

$$C = 0.8bf_{c}d_{n}\gamma$$
$$C = 0.85 \times 1000 \times 40 \times 0.766 \times d_{n}$$
$$C = 26044d_{n} \ N$$

Tension force in steel:

$$T = f_{sy}A_{st}$$
$$T = 500 \times 1130$$
$$T = 565kN$$

Applying equilibrium conditions where $C=T$ to determine the neutral axis depth:

$$C = T$$
$$26044d_{n} = 565000$$
$$d_{n} = 21.69mm$$

Determining the depth of the stress block:

$$a = d_{n}\gamma$$
$$a = 21.69 \times 0.766$$
$$a = 16.62mm$$

Calculating the ultimate moment capacity of the section by applying a lever arm from the centre of the steel to the centre of the rectangular stress block:

$$M = T\left(d - \frac{a}{2}\right)$$
$$M = 565 \times \left(204 - \frac{16.62}{2}\right)$$
$$M = 110.6kNm$$

Checking k_u to ensure the section is ductile:

$$k_u = \frac{d_n}{d}$$
$$k_u = \frac{21.69}{204}$$
$$k_u = 0.1$$

The value of k_u is less than 0.4 therefore the section is considered ductile.

From AS 5100.5 Table 2.2 [16] the capacity reduction factor is 0.8 for bending without axial tension or compression where $k_u \leq 0.4$. Appling this to the maximum moment capacity of the section:

$$\phi M_u = 0.8 \times 110.6$$
$$\phi M_u = 88.48kNm$$

The design capacity of the section is 88.48 kNm. This value is greater than the design moment of 85.7 kNm therefore adopt a 250 mm thick slab with 12 mm reinforcement bars at 100 mm centres. For simplicity of the design it is assumed that this reinforcement runs in the top and bottom of the slab in both directions.

3.4.2   Preliminary Design of Steel Girders

In the preliminary model the slab is supported by five girders spaced at 2316.25 mm centres. The girders are I-beam sections with a span of 18 m. The dead load imposed on the girders is first determined and then combined with the design loads to determine the maximum bending moments. The size of the I-beam required is calculated from the bending moments.

The free-body diagram for the girders consists of a simply supported span between two supports. It is assumed that the centre girders will support a 2.316 m width of slab. This is used for all calculations to give the greatest bending moments.

3.4.2.1   Dead Loads

The dead loads imposed on the girders are from the concrete slab and the self weight of the girder. Each one-meter length along the girder supports a slab area of 2.316 m × 1 m. The calculations for the dead load due to the slab and the girders are shown below.

Volume of slab supported by 1 m length of girder:

$$V = LBD$$
$$V = 2.316 \times 1 \times 0.25$$
$$V = 0.579 m^3$$

The dead load from the concrete slab on the girder is therefore:

$$W = \gamma_g V \rho_{concrete} \ g$$
$$W = 1.2 \times 0.579 \times 2400 \times 9.81$$
$$W = 16.36 kN/m$$

Assume a 900WB282 I-beam is used. This has a depth of 900 mm and a mass of 282 kg/m. The dead load of this beam per metre is:

$$W = \gamma_g mg$$
$$W = 1.2 \times 282 \times 9.81$$
$$W = 3.32 kN/m$$

Therefore the total dead load for every metre length of girder is:

$$W = 16.36 + 3.32$$
$$W = 19.68 kN/m$$

3.4.2.2   W80 Wheel Load

The design action for the W80 wheel load was determined previously for the loading on the deck. It was calculated as 201.6 kN over an area 0.25 m × 0.4 m. As the girders run perpendicular to the deck this load will be orientated so it is distributed over a length of 0.25 m on the girder. The distributed load is:

$$\frac{201.6 kN}{0.25 m} = 806.4 kN/m$$

This load is placed in the centre of an 18 m span of a simply supported beam to generate the maximum bending moment. The results are shown in Figure 3.17.
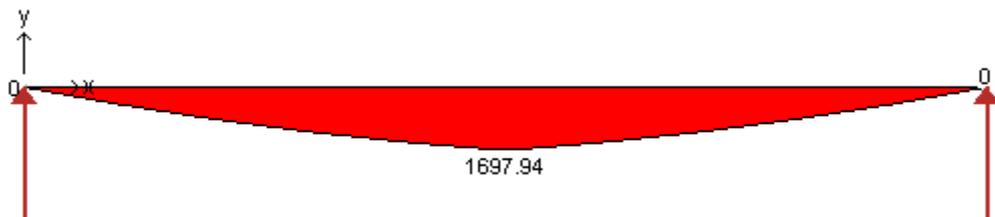
Figure 3.17: Bending moment diagram produced from W80 wheel load centrally placed on girder.

3.4.2.3   A160 Axle Load

The distance between girders is 2.316 m and the total distance between the outside of the wheel footprints representing the A160 load is 2.4 m. It is therefore assumed that the greatest loading on one single girder will be when one of these wheels is directly above the girder and placed in the centre. This gives the same bending moment as the W80 wheel load shown in Figure 3.17.

3.4.2.4   M1600 Moving Traffic Load

Like the A160 axle load the distance between the outside of the two wheel footprints is 2.4 m. However there is also a distributed load that acts over a 3.2 m width. It is assumed that the maximum load on the girder will occur when one row of wheels is directly above it. It is also assumed that half of the M1600 distributed load will act on the girder. The spacing of the wheel footprints is in accordance with Figure 3.6 to produce the greatest bending moment. Due to the wheel footprint sizes each wheel load is distributed over a 0.2 m length along the girder. The design action for the M1600 load was calculated in section 3.4.1.4. Applying this over a 0.2 m length gives:

$$\frac{145.8kN}{0.2m} = 729kN/m$$

Half of the distributed M1600 load will act on the girder, this is calculated below:

$$4.39 \times 1.6 = 7.02kN/m$$

There is only need to design for one lane, as the loading from the second lane will act on a different girder. Also due to the accompanying lane factor the loading is less, producing a smaller bending moment. The diagram of the resulting bending moment from the M1600 load is shown in Figure 3.18.
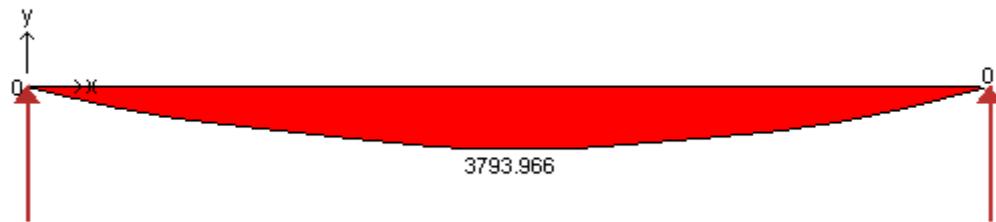
Figure 3.18: Bending moment diagram produced from M1600 traffic load placed on girder.

3.4.2.5   S1600 Stationary Traffic Load

The S1600 stationary traffic load is placed on a girder in the same way as the M1600 load. The design actions for the A1600 load were determined in section 3.4.1.5. The distributed loads for the girder are calculated below.

The wheel load distributed over a 0.2 m length of girder:

$$\frac{72kN}{0.2m} = 360kN/m$$

Half of the S1600 load distributed on the girder:

$$13.5 \times 1.6 = 21.6kN/m$$

Again like the M1600 load there is no need to calculate both lanes loaded at once because a different girder will support the second lane. Applying the dead loads and the live loads in the same way as the M1600 load gives the bending moment diagram shown in Figure 3.19.
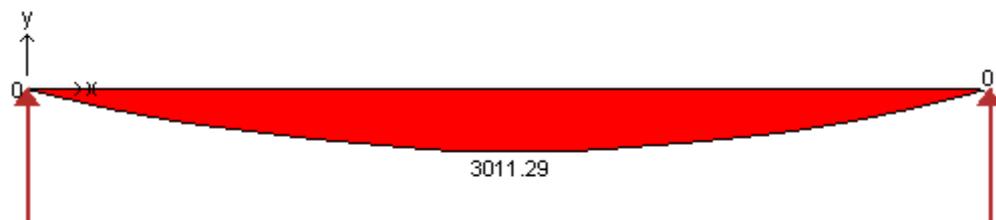
Figure 3.19: Bending moment diagram produced from S1600 traffic load placed on girder.

The maximum moments from the analysis are tabulated in Table 3.6. The M1600 moving traffic load produced the greatest bending moment of 3794 kNm.

Table 3.6: Maximum moments produced from loading combinations on girder.

Loading	Positive Moment (kNm)
W80	1698
A160	1698
M1600	3794
S1600	3011

3.4.2.6   Girder Sizing

In this section the girder size is determined from the bending moments produced above. It sole purpose is to give an approximate size of the girders to be used in the finite element model.

From HRSSP [11] the yield stress in a welded I-beam section is around 280 MPa for the flanges and 300 MPa for the web. The exact yield stresses depend on the size of the section. To be on the cautious side it is assumed the yield stress through the entire section is 280 MPa. The elastic section modulus required to withstand this bending moment is calculated below:

$$Z_x = \frac{M}{\sigma}$$
$$Z_x = \frac{3794kNm}{280MPa}$$
$$Z_x = 13550 \times 10^3 mm^3$$

The welded beam with a section modulus that closest matches the answer above is a 1000WB322. As the section modulus of this beam is larger at 14600×10^3 mm^3 the maximum stress in the girder is less than the yield stress. Therefore adopt a 1000WB322 girder to be used in the finite element model.

3.5   Summary

This chapter defined the loading requirements needed for a road bridge built in Australia. It then applied these requirements in the preliminary sizing of bridge members to be used in the finite element analysis. A 250 mm thick concrete slab comprising of 40 MPa concrete was chosen. From this the required cover to reinforcement was 40 mm. To withstand the ultimate bending loads the reinforcement was 12 mm bars spaced at 100 centres in the top and bottom of the slab in both directions. Five girders were chosen to support the concrete deck. From the loads applied the girder size required to meet the maximum bending moment was a 1000WB322 I-beam.

Chapter 4

Model Methodology

4.1   Introduction

Three bridge deck configurations, each with a different number of girders ranging from three to five are analysed using the finite element software ABAQUS [4]. Initially, the ABAQUS [4] student version, with a limitation of 1000 nodes, is used since only limited access is available to the full version at the CESRC (Computational Engineering and Science Research Centre) at USQ. This is to help the author to familiarise himself with the software and modelisation techniques before attempting to produce models on the full version with an increased number of nodes.

4.2   ABAQUS Software

ABAQUS [4] is one of the world leaders in advanced Finite Element Analysis. It provides complete and powerful solutions for routine and sophisticated linear and nonlinear engineering problems. It contains different analyses modules such as ABAQUS Standard for general nonlinear solid mechanics, ABAQUS Explicit for dynamic problems, and ABAQUS Aqua for fluid mechanics, CAE and Viewer for pre- and post-processing.

ABAQUS CAE, which stands for ABAQUS Computer Aided Engineering, is a graphical interface, which allows the user to input all the model data, run the analysis,

and view the results. During the analysis, a basic ASCII file with the extension ".inp" is produced. The file contains all the model information and can be simply opened with a text editor. This file is very useful since information regarding certain aspects of the model that cannot be defined within CAE can be added to the input file. The modified input file can then be submitted as an analysis job using the command line within the operating system shell.

During an analysis, ABAQUS [4] generates a large number of output files, among them are the output database file ".odb", the data file ".dat", and the message file ".msg". The first here is a binary file that provides the information required by the Viewer to provide the graphical representation of the results. The data file is a text file that contains information about the model and its history definition generated by the analysis. It also contains an output of results that have been requested, and any error or warning messages that were detected. All diagnostic and informative messages about the progress of the solution are contained within the message file. All errors and warnings are detailed in the message file with the results of this tabulated at the end of the message file.

If the analysis of a model is based around an input file then the Viewer section of ABAQUS [4] comes into use. It reads the output database file and provides a graphical representation of all requested results. This can be seen by a contour diagram drawn over the part, a displacement diagram of the part or an x-y plot of the requested output.

4.3   Layout of an ABAQUS Model

To effectively produce a model with ABAQUS [4], specific information must be given so the finite element analysis can take place. There are a number of steps to producing a finite element model and these all have to be correctly accounted for before the model will run and valid answers can be produced. An ABAQUS [4] model consists of two parts: model geometry, and model history.

4.3.1   Model geometry

The model geometry contains different parts of the model that are assembled together, the material properties of these parts and in some situations how they are tied together.

A mesh of nodes and elements defines each part with each node given a location in a Cartesian coordinate system and each element being defined by a number of these nodes. During this process the type of element is also stated. All the information regarding the part is contained within a part option. A section option is used within the part option to give the section properties of the particular part. This is used to define the dimensions of the elements that constitute the part, the material used and any other parameters or data that are deemed necessary.

Once all the parts have been defined they can be assembled as a number of part instances. For example there is more than one girder in the bridge but they are all identical so only a mesh of one girder is needed. When it comes to assembling the bridge the first girder is defined as Girder – Instance-1 and the second as Girder – Instance-2. This continues in the same fashion until all the girders are defined in the assembly. Even though there is one deck it still has to be defined as an instance as the visualisation section of ABAQUS [4] runs off part instances. Each instance is positioned in the global coordinates by a translation in the x, y and z directions and by a rotation defined by an axis of rotation and an angle.

Material definitions provide the model with the information needed for the analysis of the material behaviour. Steel is an elastic material so the inputs for the model will be Young's modulus and Poisson's ratio. However as ABAQUS [4] can undertake a nonlinear analysis the yield stress of the steel can also be input under the material properties. This means that the plastic strain of the material can be measured if the stresses are high enough. ABAQUS [4] has a wide range material definitions that allow modelling of most materials. In terms of this project they have a number of plastic models for concrete with one suited to the analysis of reinforced concrete. This is the smeared crack approach.

Contact and interaction is used to join parts together and define friction properties between them. This is often defined as part of the model geometry but sometimes interaction properties can be applied as model history. Nodes, elements and surfaces are used to define particular contact and interaction restraints. For example various amounts of friction can be applied between two surfaces or two nodes can be tied together so their global displacements and rotations are the same.

4.3.2   Model history

The model history contains the steps in the analysis, loads, boundary conditions and output requests.

The analysis of an ABAQUS [4] model is run as a series of steps. There is always a minimum of two steps. The first step is automatically built in to the analysis and is used to apply the boundary conditions. Following steps define the loading conditions and outputs required. If contact has been defined as part of the model history then a separate step is required before the load step to define this contact.

Numerous loading situations can be modelled with ABAQUS [4], from point loads applied on nodes to distributed loads on elements or surfaces.

A boundary condition is produced by restraining various degrees of freedom at a node. The number of degrees of freedom for a particular node will depend on the element used. Although in structural situations such as this each node will nearly always have six degrees of freedom, three displacement and three rotation. The boundary conditions fix certain parts of the model so there will be support reactions to counteract the loads applied to the structure.

When it comes to producing results ABAQUS [4] automatically has a standard list of outputs if none are defined. These contain most of the properties needed for a general structural analysis such as stress, strain and displacement. If other outputs are required

these can be defined. These results can be output to the output database file for use in the Viewer or written to the data file as a list of results.

4.4   The Input File

The entire finite element model is defined in the input file. All information displayed in the input file is given in terms of keywords. Each keyword starts with an asterisk and is then often followed by a parameter. For example the keyword ***element** is used to define the elements of a part. It is then followed by the required parameter **type** to define what type of element it will be. Keywords are often followed by data lines which give additional information required. In the example given above for the ***element** keyword the data lines contain the element number and the nodes that make up the element. All information that relates to a particular keyword is referred to as an option. Therefore all the information given above is described as an element option.

Often when defining options, the elements or nodes that the option acts upon must be previously stated. This is done by grouping nodes or elements into sets by using either the ***nset** or ***elset** option respectively. This ends up being the bulk of the input file as once a node set or element set is established it only requires an option that may be one or two lines to impose the contact, boundary conditions and other details that are required.

4.5   Initial Modelling – Student Edition

4.5.1   Background

Modelling was initially started by using ABAQUS CAE, as it has a graphical user interface and makes it much easier to produce a working finite element mesh. Once the mesh is produced, all the relevant data is added to the input file using a text editor.

4.5.2   Preliminary Model Details

Due to the node limitation of the student version, initially a model of a bridge deck with only two girders and three diaphragms running between the girders is produced. This model is not realistic, and is only produced as a learning process on how to effectively use ABAQUS [4].

4.5.2.1   Elements Used

The deck and the girders are modelled with three dimensional shell elements while the diaphragms are modelled using truss elements. Shell elements are used when one dimension, the thickness, is significantly less than the other two dimensions. Conventional shell elements have six degrees of freedom, three being displacement degrees with the other three defining rotation.

S4R elements are used to discretise the deck. This is a conventional four node, quadrilateral, stress/displacement shell element with reduced integration and a large strain formulation. It is a rectangular shaped element with a node at each corner. The properties of the section are calculated by using the shell section option. This uses numerical integration through the thickness of the shell and is suited to solving nonlinear problems, in this case the analysis of a reinforced concrete deck. In this option the thickness of the shell and the number of integration points are defined. The ABAQUS [4] documentation recommends at least nine integration points when analysing concrete.

The girders also use S4R elements. There are beam elements in ABAQUS [4] however these are only defined as a line. The shape of the cross section is then given in the section definition for the part. Therefore when it comes to analyse the part an accurate stress distribution cannot be obtained through the section. This is why the I-beam girders are produced with shell elements. The user is then able to see the shape of the girder and how the stresses and displacements vary between the flanges and web of the girder.

To model the steel reinforcement in the concrete, rebar elements are used. These are elements within ABAQUS [4] specifically designed to model steel reinforcement. They are added as layers to existing elements in a smeared layer where the thickness is equal to the area of reinforcement bar divided by the reinforcement bar spacing. To define this in ABAQUS [4] the rebar layer option is used. For this option the following information must be stated: the name of the reinforcement layer; the cross sectional area and spacing of the reinforcement; the location in the thickness direction measured from the midpoint of the shell and the angle of the reinforcement in relation to the x-direction.

The diaphragms that run between the girders are modelled with truss elements. These are long slender elements that can only transmit axial forces. They cannot transfer moments. Truss elements were chosen because the diaphragms sole purpose is to transmit axial forces to prevent buckling of the girders. The T3D2 truss element was used in the model. This is a two-node 3-D truss element. Section details for the element are defined by the solid section parameter. The only inputs needed are the cross sectional area of the section and the name of the material it is made from.

4.5.2.2   Connections

Once all the parts have been assembled and the section properties for each have been defined the next step is to provide connections between these separate parts. This allows forces and moments to transfer between parts just as they would in the real world.

When a bridge such as this is constructed the concrete deck is tied to the girders by the way of shear connectors. These are studs than protrude from the girders and are cast into the concrete. They are spaced along the length of the girder at constant intervals and can either be multiple rows of studs or just a single line down the centre. This situation is modelled using the ***contact pair** option with the **tied** parameter. To apply this interaction, the first step is to define the top surface of each girder and the bottom surface of the deck plus list the ***surface interaction** and ***surface behaviour** options. Once this is done the ***contact pair option** ties the two surfaces together when they come in contact with each other. A **hard** pressure overclosure relationship is chosen for

the surface behaviour. This does not allow the slave nodes to penetrate into the master surface. The top of the girders are defined as the slave surface and the bottom of the deck is defined as the master surface. ABAQUS [4] attempts to find the closest point on the master surface that matches up with a node on the slave surface. This is where the master surfaces normal passes through the node on the slave surface and the interaction is then discretised between a point on the master surface and a slave node. An **adjust** parameter is also used for this option. This adjusts the position of the nodes on the slave surface at the start of the simulation so they precisely contact the master surface. Even though this way of modelling shear connectors is not exactly the same as the real situation it was determined that it would be an accurate representation as the vertical loads being transferred between the deck and the girders would be the same.

The diaphragms have to be attached to the girders. As the diaphragms are modelled with truss elements pinned restraints are chosen. This is done by attaching the node at the end of the diaphragm to a node on the girder. The ***equation** option could be used in this situation as it is designed to define linear multi-point restraints. The restraint requires a linear combination of nodal variables to be equal to zero. In this situation both nodes must move together so the three displacement degrees of freedom for the two nodes must be tied together. This is defined by the equation shown where u is the nodal variable at node D or G and 3 is the degree of freedom in question.

$$u_3^D = u_3^G$$

Rearranging so the variables are equal to zero gives:

$$u_3^D - u_3^G = 0$$

The properties of the equation are defined in the input file as two data lines. The first line states the number of terms in the equation, in this case there are two. The second line states the first node in question, the degree of freedom for that node and the coefficient for that node which is one. This is repeated for the second node with the coefficient being negative one so the equation equals zero. Only one degree of freedom

can be restrained per equation so three equations are used for each constraint to tie all three degrees of freedom

4.5.2.3   Boundary Conditions

The boundary conditions are set up to represent a simply supported structure. One end of the structure is be pinned while the opposite end is a pinned/sliding restraint. This is the closest way to model how the bridge would act when sitting on bearing pads without actually modelling them. Bearing pads have very little shear support so they will take any deformation in the long direction of the bridge.

The ***boundary** option is used to apply the boundary conditions. Data required for this option is that the nodes be given boundary conditions, and the degrees of freedom to be restrained. The nodes chosen are on the bottom side of the girders at each end as this is where the supports are. At one end the nodes are restrained for the three displacement directions. At the opposite end the nodes are only restrained in two directions. The direction the nodes are not supported in is the direction that runs the length of the bridge. This gives a sliding restraint for the simply supported condition.

4.5.2.4   Materials

The model contains only two main materials, steel and concrete. The ***material** option is used to define these. Within this option other options are used to define the material properties.

For steel the elastic properties are defined with Young's modulus as 200 GPa and Poisson's Ratio as 0.3. Plastic properties are not used at this stage because the main aim is to produce a working finite element model. The density of the steel is listed as 7850 kg/m^2 [11] so it can be used to determine the loading from the self-weight of the bridge.

The concrete is defined with elastic and specific concrete properties to define its behaviour outside the elastic range. These values are taken from Gilbert and Warner [17]. The ABAQUS Example Problems Manual [6] used these values in the analysis of

a concrete slab and some of their assumed values are also been used in this analysis, as they are not available from Gilbert and Warner [17]. The properties outside the elastic range are listed as yield stress with respect to plastic strain. The first stress-strain point is used to list the initial yield point of the concrete so the plastic strain value must be equal to zero. This yield stress is assumed in the ABAQUS Example Problems Manual [6] and this value of 20.68 MPa is used in the model. The next stress strain point is the failure ratio for the concrete, this is taken from Gilbert and Warner [17] and listed as 37.92 MPa with an assumed plastic strain at failure of 1.5×10^{-3}. Elastic properties are again from Gilbert and Warner [17] with a Young's Modulus of 28.6 GPa and Poisson's Ratio of 0.15. Finally for self weight loads the density of the concrete is listed as 2400 kg/m^2 [16].

4.5.2.5   Analysis Steps

For the bridge being modelled a general static analysis step is chosen. This effectively models the nonlinear behaviour of both the concrete and the steel. The ***static** option is used to define this and because it is a nonlinear analysis details have to be given on the time step. The default values that ABAQUS [4] recommends are used. These are an initial time increment of 1, a step period of 1, 1×10^{-5} for the minimum time increment allowed and 1 for the maximum time increment allowed.

The loads for this basic model are just the self-weight of the girders and deck. These are defined by a gravity load. The option ***dload** is used with the inputs being the elements for each of the girders and the deck, the parameter **grav** to define the loading as being gravity, the acceleration due to gravity which is 9.81 m/s and the direction in which it acts. In the model this is the negative z-direction. ABAQUS [4] uses the density of each material that is defined in the material property definitions and the acceleration due to gravity and then multiplies these by the volume of each element making up the part. This gives the distributed loading that results from the self-weight of the structure.

The ***output** option is used to state the results requested from the simulation. The default outputs are chosen for the model by the **preselect** parameter. No specific results are required as this model is used as a learning experience. There are two different types

of output, field output and history output. Field outputs are the results that are written at fairly low frequencies from a large portion of the entire model. These include variables such as stress and strain. History outputs are results that are written at fairly high frequencies from a small portion of the model. These include variables such as the displacement of a single node. An x-y plot of history results can also be produced. This can be used to show the displacement of a node over the time of the simulation.

4.6   Three, Four and Five Girder Models

After the initial trial model had been produced with the student edition, and an understanding of how to create a model of the bridge-deck was obtained, the next step is to move onto the full version of ABAQUS [4]. New models with more degrees of freedom are created. These are a three, four and five girder deck and from this the optimum girder spacing will be obtained. The input file for each model can be found in Appendix B.

4.6.1   Member Sizes

The member sizes used in the three separate models remained the same even though the number of girders would vary. This is done to limit the variables encountered when it came to determining an optimum girder spacing. Sizes used are similar to those determined in Chapter 3 with the Preliminary Bridge Design.

The girder size used is the largest 1000 welded beam produced by OneSteel. This is an I-beam 1024 mm deep with a flange width of 400 mm and a weight of 322 kg/m [11]. The maximum available length of this beam is 18 m and is what is used in all models. A cross section of this beam is shown in Figure 4.1.
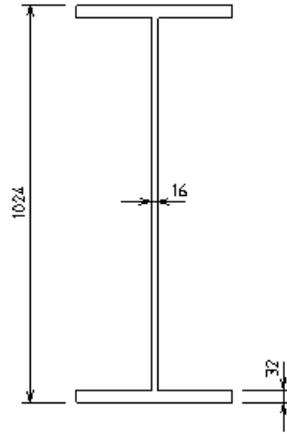
Figure 4.1: 1000 WB 322 I-Beam

The concrete slab used in all three models is a 250 mm deep slab with 10 mm reinforcement bars at 100 mm spacing. This spacing is run in both directions in the top and bottom of the slab. The reinforcement is reduced from the 12 mm bars designed in Chapter 3 so the nonlinear response of the deck would be more pronounced. Cover is 40 mm top and bottom to adhere to Australian Standards. Details of this are explained in Chapter 3.

As there is no preliminary design for the diaphragms the sizes for these are assumed values. However Biggs et al. [9] in their analysis of a similar bridge used channel sections 380 mm deep. For the analysis 300 × 32 mm plates are chosen. These are attached to web stiffeners at the centre and ends of the girders.

4.6.2   Elements

The elements that are used have already been determined in the preliminary model, the difference here being that all the elements are smaller so more accurate results will be obtained. S4R shell elements are used for the deck and girders and B31 beam elements for the diaphragms. Specific rebar elements are added to the deck to model the reinforcement.

The deck is meshed with a total of 2940 elements. This is produced as a rectangular mesh with 70 elements spanning the 18 m length of the deck and 42 elements spanning the 9.93 m width. The element size is therefore 0.257 m × 0.236 m respectively. When meshing the girders the element length is kept the same as the deck with 70 elements spanning the length of the girder on the both the flange and web of the beam. Both the flanges and the web of the beam have 8 elements spanning their width and depth respectively. Each of the web stiffeners has elements sizes so that they matched the web and flange elements of the girders. A total of 1872 elements are used in the girders. Unlike the girders and the deck the number of elements change for the diaphragms depending on the model. This is due to their different lengths caused by the varying girder spacing in the models. The diaphragms in the five girder bridge contains six elements while ten elements are used for the four girder model and eighteen elements are used for the three girder bridge deck. The diaphragms are modelled using B31 beam elements that are able to take bending moments and axial loads. These are chosen since the diaphragms are fixed to the girders with more than one bolt. Such a connection creates full restraint and allows bending moments to be transferred from the girders to the diaphragms.

The beam element B31 is a two node linear beam with the standard six degrees of freedom. It is defined as a one-dimensional element in three-dimensional space. The section is defined with the ***beam section** option. This accounts for the nonlinear response of the material. ABAQUS [4] has predefined sections available as parameters, and a rectangular section is chosen to model the plates. The data needed for this option is the dimensions of the plate and the direction of the plates n_1 vector. In the case of the model this is in the z-direction.

4.6.3   Connections

The deck is tied to the girders using the ***tie** option. Contact surfaces of the girders and deck need to be pre-defined for this option. The only parameters that are to be stated is the type of tie, which is a **surface to surface** interaction, and to set the **adjust** parameter

to yes. This initially moves all tied nodes on the slave surface onto the master surface without causing any strain.

A MPC (Multi Point Constraint) is used to attach the diaphragms to the girders. This is defined with the ***mpc** option. The **tie** parameter is used to restrain all six degrees of freedom for the node on the diaphragm to the node on the girder. This enabled moments to be transferred between the two parts as they would under real conditions.

4.6.4   Boundary Conditions

The boundary conditions for the bridge deck are applied to model a simply supported condition. One end of the girders is pinned in all three directions and at the opposite end a pinned/sliding arrangement is defined. The length of the deck would subsequently be free to change when loaded.

The supporting area of the boundary conditions is changed between models as each contains a different number of girders. If the supporting area is kept the same between models then the reaction forces at the boundary conditions in the three girder model would be a lot higher than that in the five girder model. For this reason the five and four girder models have boundary conditions spanning over two rows of elements at the end of each girder. This is increased to three rows of elements for the three girder model.

4.6.5   Materials

The material definition for concrete is defined with a number of options. The density is specified as 2400 kg/m^3 [16] so the self-weight of the structure can be applied as a load. The elastic properties are defined with Young's Modulus of 28.6 GPa and Poisson's Ratio of 0.15. The specific concrete properties are defined with an initial yield stress of 20.68 MPa* and a failure stress of 37.92 MPa at a plastic strain of 1.5×10^{-3}*. Tension stiffening properties are used in concrete model to account for the stress taken between the cracks that have formed. It is defined as the fraction of remaining stress to stress at

cracking and the absolute value of the direct strain minus the direct strain at cracking. In the models, for zero remaining stress the value of direct strain is listed between 0.001 to 0.01. The tension stiffening value is purely theoretical and is changed between models to ensure the solution will converge to a result. All of these values are taken from Gilbert and Warner [17] or where marked with an asterisk taken from the ABAQUS Example Problems Manual [6].

Three separate material properties for steel with different yield stresses are defined in the model. The girders consist of two types of steel according to HRSSP [11]. The web of the girders is 300 MPa steel while the flanges are 280 MPa steel. The reinforcement modelled is standard N class bars with a yield stress of 500 MPa. These yield points are listed in the input file by using the ***plastic** option and stating the yield stress of the steel. The elastic properties are the same for all steel and listed as with Young's modulus as 200 GPa and Poisson's Ratio as 0.3. A density of 7850 kg/m^2 [11] is listed so the self-weight of the structure can be applied as a load.

4.6.6   Analysis Steps

A general static analysis step is chosen for the analysis. This will effectively model the nonlinear behaviour of the materials. The default values that ABAQUS [4] recommends are used for the time step. This is an initial time increment of 1, a step period of 1, 1×10^{-5} for the minimum time increment allowed and 1 for the maximum time increment allowed.

Loading from AS 5100.2 [5] is applied to model the design loads. The exact loads and their positions on the bridge are defined in Chapter 2. All the loads defined in AS 5100.2 [5] are either footprints of wheel loads or a distributed load over the deck. These are all applied by the ***dsload** option in ABAQUS [4]. This is a loading option to define a distributed load over an element or surface. The elements that accounted for the wheel footprints are gathered into an element set and then defined as a surface. The elements that account for the distributed load over the deck are defined as a separate surface. The element sizes in the model meant that the wheel footprints are not exactly the same area

as those defined in the code. However the overall load applied is still exactly the same. Once the elements constituting the surfaces are selected the type of distributed load is set as a pressure and the magnitude of this is defined. This same procedure is used for the self-weight of the concrete barriers that are applied to the deck. The loads due to the self-weight of the parts are defined by gravity loading.

The default outputs are used for the analysis. However a few other output requests are included to obtain all the results that are required. The stress for the reinforcement in the concrete is requested to be written to the data file. This gives a table of the stress and force in the reinforcement at each element in the bridge deck plus maximum and minimum values. The ***el print** option is used where the elements requested for output need to be defined and the output variables for stress and force are stated. The stress in the diaphragms is written to the data file in the same way. ABAQUS [4] is also able to state if cracks have formed in the concrete and in what direction. This is done by using the same ***el print** option as above and by stating the crack output variable. The results for this are in tabular form that show what elements have cracked, which surface they are cracked on and the direction of the normal to each crack. A history output request is used so a plot of a node displacement can be produced for the time period of the simulation. This will show if the deck is loaded enough for the concrete to be effected by nonlinear behaviour. It is defined in the model by the ***node output** option where the node in question was listed plus the direction in which the displacement will be measured.

4.7 Conclusion

A number of finite element models were produced of the bridge deck for various loading combinations and girder spacings. The results from the analysis of these models will be presented in the next chapter and from this the optimum girder spacing will be determined.

Chapter 5

Results

5.1 Introduction

This chapter presents and explains the results obtained from the finite element analysis. Three bridge-decks were modelled with ABAQUS [4]. These ranged from a three girder bridge deck to a five girder bridge deck. Each has the same girder size, 1000WB322 I-beams and the same slab. This is a 250 mm thick 37.92 MPa concrete slab with 10 mm reinforcement bars top and bottom running in both directions.

All bridges were modelled using nonlinear material behaviour laws. This is particularly important for the slab, as cracking occurs at small tensile stresses in the concrete causing a shift in the neutral axis of the section. ABAQUS [4] is able to analyse the nonlinear behaviour and give results of stress, strain and deflection. Stress shows if the parts can withstand the loads applied. If the stress is greater than the failure or yield stress of the concrete or steel respectively then it is known that the part has failed. Plastic strain is important in the analysis of the girders, as it shows if any of the steel is overloaded and yielded. Displacement results from the analysis are important as they are used to check serviceability requirements. The midspan deflection of the deck must not exceed the limit defined by the AS 5100.2 [5].

Fatigue load effects have not been analysed due to the requirement by AS 5100.5 [16].
It states that when a road bridge has an effective number of stress cycles less than
500 000 the design does not need to consider fatigue.

Geometric nonlinearity has not been taken into account. This means that the structure
has not been analysed for buckling. However the results from the analysis should still
give a good indication of the strength of the bridge deck. Buckling would only be
expected when the members are under high stresses.

An example of the visualisation output ABAQUS [4] produces from the analysis is
shown in Figure 5.1. Displacements are shown as the deformation in the deck and the
stress gradients are shown by the different colours.
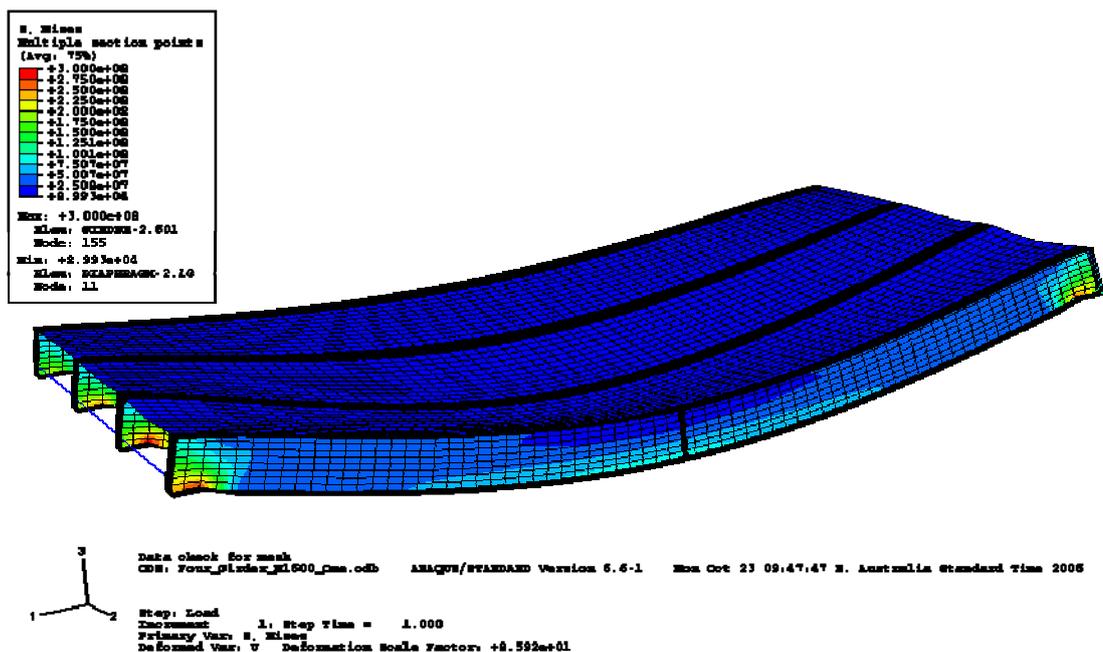
Figure 5.1: Visualisation output of analysis.

5.2 Five Girder Deck

This section deals with the results obtained from the bridge deck comprising five
girders. This describes the stresses and strains within the members and the displacement

of the entire deck. Every loading combination defined by AS 5100.2 [5] is analysed
with this model.

5.2.1   Concrete Slab Stresses

As the slab is reinforced concrete, the results of interest are the compressive stress in the
concrete and tensile stress in the steel reinforcement. The maximum tensile and
compressive stresses for the concrete deck produced from the various loading
combinations are shown in Table 5.1.

Table 5.1: Maximum in plane stresses for the top and bottom surfaces of the five girder bridge deck.

Loading Condition	Deck Maximum In-Plane Tensile Stress (MPa)		Deck Maximum In-Plane Compressive Stress (MPa)	
	Top	Bottom	Top	Bottom
A160	1.97	2.7	4.03	2.13
M1600 One Lane	3.55	3.5	8.09	5.88
M1600 Both Lanes	3.54	3.49	6.65	5.8
S1600 One Lane	3.53	3.47	8.15	5.03
S1600 Both Lanes	3.52	3.55	6.69	5.21
W80	1.53	2.88	4.22	1.16

The results show both tensile and compressive stresses in the concrete. Tensile stresses
are provided to show if the concrete reached its cracking strength. It can be seen from
Table 5.1 that the concrete has a maximum tensile stress of approximately 3.5 MPa
before it cracks. The A160 and W80 wheel loads do not impose large enough stresses
for cracks to form. Maximum compressive forces are produced from the M1600 and
S1600 wheel loads when accounting for only one lane of traffic. These values are 8.15
MPa for the top surface of the slab and 5.88 MPa for the bottom surface of the slab.
This is well below the failure point of 37.92 MPa for the concrete.

It is noted that greater stresses are produced when only one lane of traffic is positioned
on the deck. The cause of this is due to torsion. When one lane is loaded the whole
bridge deck twists causing large stresses to be transmitted through the slab. This is
where the finite element method is very useful because normal analysis techniques are
not able to account for torsion.
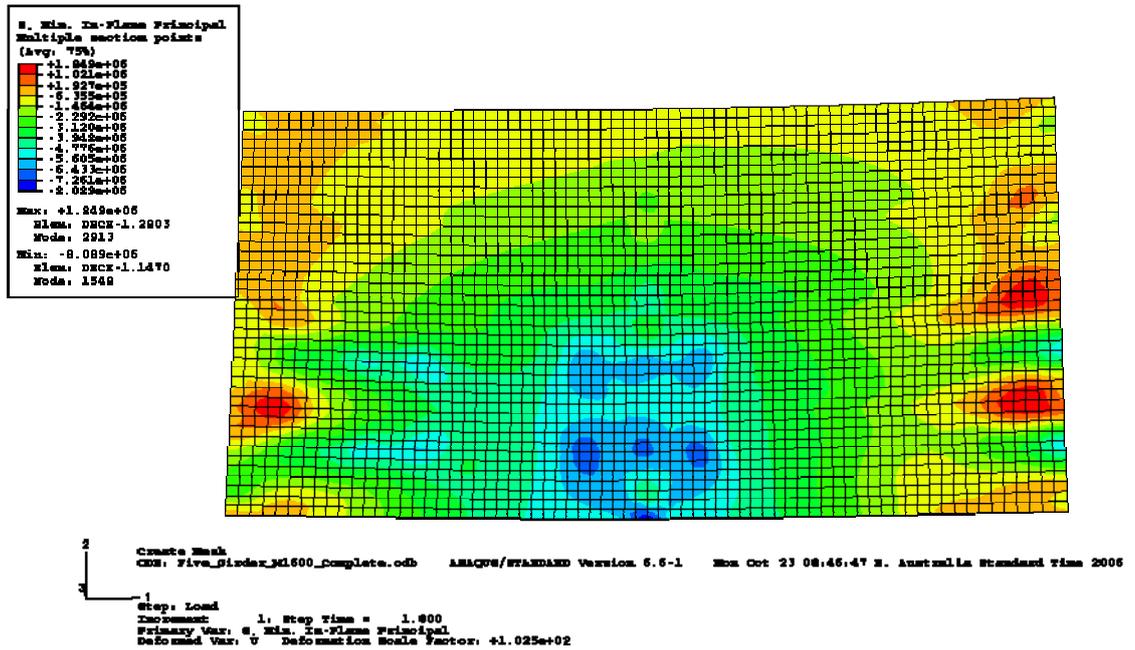
Figure 5.2: Maximum in-plane compressive stress for the top surface of the slab supporting the M1600 single lane traffic load.

The stresses vary considerably across the slab with high localised stress points in areas that would not normally be considered. Figure 5.2 shows the maximum in-plane compressive stress for the top slab surface with a M1600 single lane traffic load. This output from ABAQUS [4] gives the greatest negative stress at each element, and where there is no negative stress it gives the smallest positive stress.

The areas under compressive stress are clearly visible by blue, green, yellow and brown areas with blue representing the areas of highest compressive stress. The highest stresses are in the areas expected, directly where the loading is applied. However this image only shows the maximum negative stress values for each element so it does not give a complete picture on the stresses in the top face of the slab. Figure 5.3 shows the maximum tensile stresses recorded at each element in the top surface of the slab.
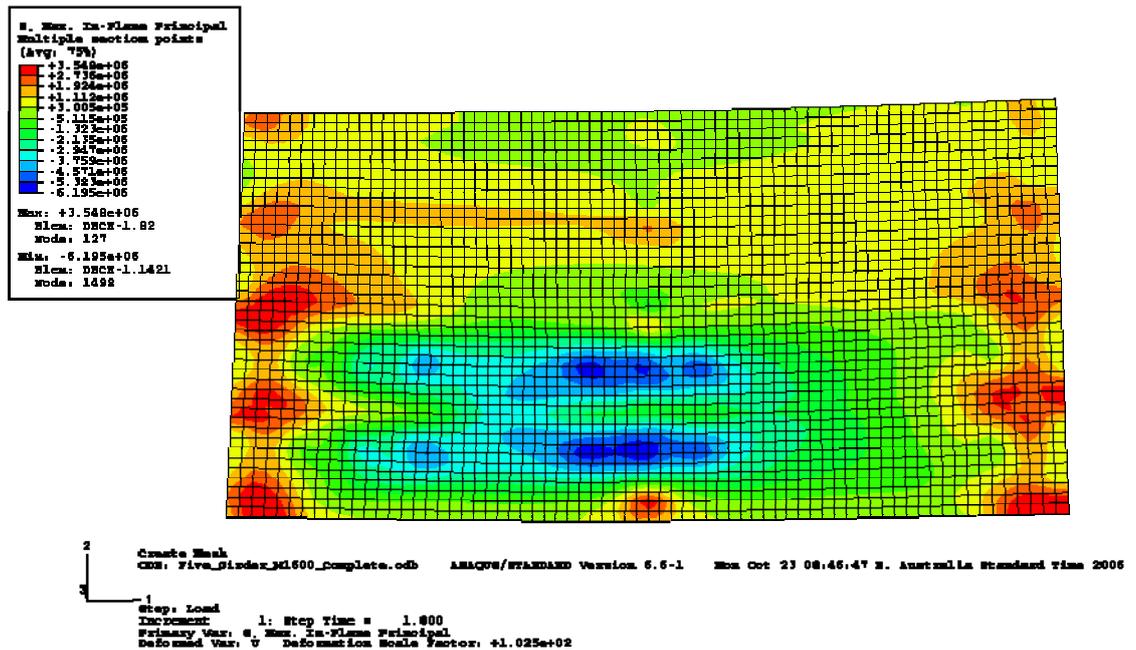
Figure 5.3: Maximum in-plane tensile stress for the top surface of the slab supporting the M1600 single lane traffic load.

The red, yellow and orange areas in Figure 5.3 are all in tension. Comparing Figure 5.2 to 5.3 shows how some elements are in compression and tension. In Figure 5.2 the majority of the slab was seen to be in compression, however in Figure 5.3 there is a tension zone highlighted by yellow and orange running the length of the slab. In this area the concrete is in compression in the x-direction (length direction of deck) and tension in the y-direction (width direction of deck). A continuous support condition is provided for the slab by the girders. This allows negative bending moments to be produced over the girders that result in tensile stresses in the top face of the slab. However the girders are affected by a positive bending moment that results in compressive stresses in the concrete. This produces an element that is in both tension and compression on the same face.

Figure 5.3 also has a small area high in tension at the midspan point along the bottom edge of the slab. This is caused by the girders web stiffener. The stiffener prevents the inboard flange of the outer girder from deflecting and as a consequence a stress concentration results in that area.

The areas above each end of the girders show high tensile stresses in Figures 5.2 and 5.3. These high stress areas can also be found on the bottom surface of the slab as large compressive stresses. Figure 5.4 gives the best representation of these stresses, which are highlighted by the blue areas at each end of the slab.
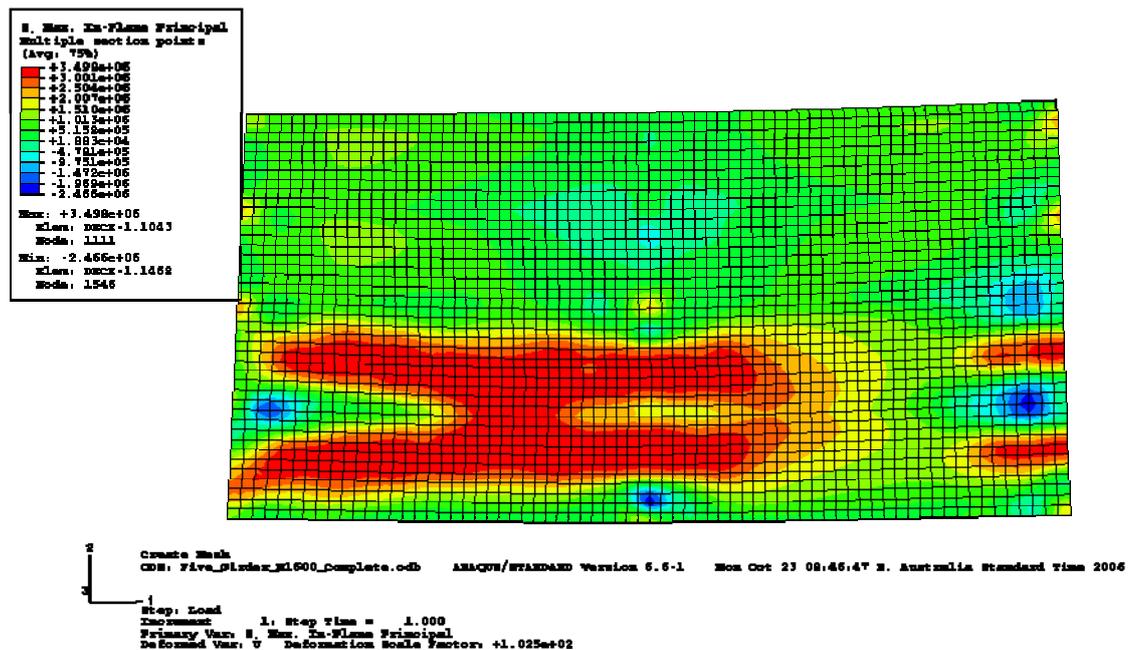
Figure 5.4: Maximum in-plane tensile stress for the bottom surface of the slab supporting the M1600 single lane traffic load.

The high stresses in the slab at the end of the girders are due to the support conditions and the stiffness of the girders. The boundary conditions were applied over a series of elements at the end of each girder and restrained movement in the vertical direction. This causes a sharp bend in the girder at the edge of each support and because the girders are stiffer than the slab stress concentrations formed.

The tensile forces in the steel reinforcement must be assessed to ensure they are within the yield limit of the steel. The stress carried by each rebar element was written as a table to the output data file. The maximum tensile and compressive stress for all loading combinations with the five girder bridge deck is displayed in Table 5.2.

Table 5.2: Maximum reinforcement stress for the five girder bridge deck.

Loading Condition	Maximum Stress in Steel Reinforcement (MPa)	
	Tension	Compression
A160	15.23	21.67
M1600 One Lane	30.77	45.82
M1600 Both Lanes	42.19	36.9
S1600 One Lane	28.15	46.17
S1600 Both Lanes	42.3	37.06
W80	16.77	19.48

Stresses in the steel reinforcement are very low for the five girder bridge deck with a maximum tensile value of only 42.19 MPa. This is much smaller than the steel yield stress of 500 MPa. The maximum tensile stresses are due to the M1600 and S1600 loads when two lanes of traffic are imposed on the deck. The reinforcement containing high tensile stresses is located on the top surface of the slab above the supports for the girders. The same high stress areas previously discussed for the concrete slab.

The maximum compressive stresses in the steel are also shown in Table 5.2 because a majority of the values are higher than the tensile stresses. The maximum compressive stresses are located where the slab was completely in compression through its cross section or where tensile stresses are very small in comparison to compressive stresses. An example of where this has occurred is above the web stiffener on the outside girder. Normally this would not produce the highest reinforcement stresses in the model but in this case the slab is well over designed. Small stress concentrations are therefore giving the maximum recorded values.

Results provided by the analysis show that the reinforced concrete slab for the five girder bridge deck can withstand all loading requirements defined by AS 5100.2 [5].

5.2.2   Girder Stresses

The most important aspect with the girders is to ensure the stresses do not reach the yield point of the steel. The maximum girder stresses recorded for each of the loading combinations are shown in Table 5.3.

Table 5.3: Maximum girder stress for the five girder bridge deck.

Loading Condition	Maximum Stress in Steel (MPa)	
	Tension	Compression
A160	66.62	129.1
M1600 One Lane	177	321.6
M1600 Both Lanes	219.3	317.3
S1600 One Lane	176.6	321.3
S1600 Both Lanes	218.5	314.6
W80	53.63	105.8

The welded I-beams used for the girders are comprised of two steel strengths. The flanges use 280 MPa steel and the web uses 300 MPa steel. Table 5.3 shows that the steel has yielded in the four major load cases. The M1600 load case where both lanes are loaded has the highest tensile stress therefore it will be used to explain why the steel is yielding. To identify where the yielding is occurring Figure 5.5 shows the girder elements that have undergone plastic strain.

The only girder elements that have undergone plastic strain are shown in red in Figure 5.5. These are directly above the inner edge of the supports and can be explained by the boundary conditions that have been applied to the model. The girders are currently fixed at each end to a rigid surface where in reality they would be mounted on bearing pads. These are essentially a large elastomer block that is used to dampen vibrations from the deck and allow a smooth transition of forces from the girders to a fixed support such as a concrete footing. This would eliminate the high localised stresses causing the steel to yield in the web of the girders. Web stiffeners could also be used to remove the localised stress point by adding them at the inner edge of the support. These distribute the loads over the entire web of the girder and would be used in conjunction with the bearing pads to ensure a smooth stress gradient.
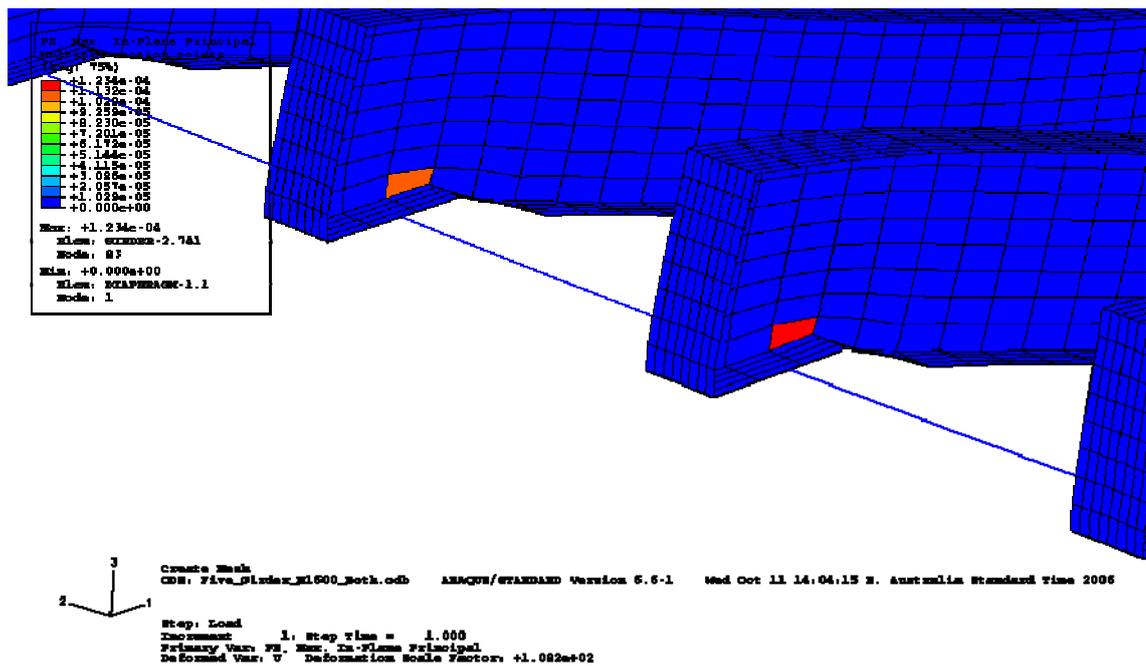
Figure 5.5: Plastic strain in girders for two lanes of the M1600 load on the five girder bridge deck.

If the stress concentrations at the supports are not accounted for the girders can easily take the bending loads applied. The maximum stress recorded on the bottom flange of a girder for the five girder bridge deck is 125 MPa. The compressive stress on the top flange is only 36 MPa. The differences in stresses is due to the concrete deck is acting in conjunction with top flange of the girder to share the compressive forces.

Ignoring the stresses at the supports, the results provided by the analysis show that the steel girders for the five girder bridge deck can withstand all loading requirements defined by AS 5100.2 [5].

5.2.3   Diaphragm Stresses

The stresses in the diaphragms have to be checked for yielding. If high compressive stresses are found in the diaphragms then they should be checked for buckling, as nonlinear geometry was not specified in this analysis. The diaphragm stresses recorded for each loading combination is shown in Table 5.4.

Table 5.4: Maximum and minimum stress values for diaphragms in five girder bridge deck.

Loading Condition	Stress in Diaphragms (MPa)	
	Maximum	Minimum
A160	10.29	0.193
M1600 One Lane	23.51	-0.821
M1600 Both Lanes	22.32	0.606
S1600 One Lane	23.84	-0.853
S1600 Both Lanes	22.71	0.732
W80	8.5	0.384

The results for the loading combinations show the diaphragms are all in tension. The smallest stress values recorded are in compression, however this was because of the connection applied between the diaphragms and girders. It is a completely fixed connection so the diaphragms are able to bend producing positive and negative stresses. The diaphragms at the midspan of the deck always produced the highest stresses. At each end of the deck the diaphragms did very little work as the girders were restrained by both the boundary conditions and by the concrete deck. The maximum stress recorded is 23.84 MPa under the S1600 traffic load. Although all of the loading conditions representing traffic produced similar results with less than 2 MPa difference.

Results provided by the analysis show that the diaphragms for the five girder bridge deck can withstand all loading requirements defined by AS 5100.2 [5].

5.2.4   Deck Displacements

In order to satisfy requirements by AS 5100.2 [5] the deflection must not exceed 1/600 of the span under serviceability conditions. For bridge design this is the same as ultimate load conditions. The girders are 18 m long however the span of the bridge is 16.97 m because the supports on each end of the girder run the length of two elements equal to 0.514 m. The deflection limit for the bridge is:

$$\frac{16.97}{600} = 28.3mm$$

The deflections produced from the finite element analysis for each loading condition is shown in Table 5.5.

Table 5.5: Maximum deflection values for all loading combinations on the five girder bridge deck.

Loading Condition	Displacement (mm)
A160	8.28
M1600 One Lane	17.56
M1600 Both Lanes	16.63
S1600 One Lane	17.82
S1600 Both Lanes	16.9
W80	6.81

The maximum deflection recorded is 17.82 mm for one lane of the S1600 stationary traffic load. This is only 0.26 mm different from one lane of the M1600 moving traffic load. All values are less than the limit of 28.3 mm. The results show greater displacement when only one lane is loaded. As with the deck stress analysis this is due to torsion being applied to the bridge deck.

A history output was requested of the node under maximum deflection. A plot of this is shown in Figure 5.6.
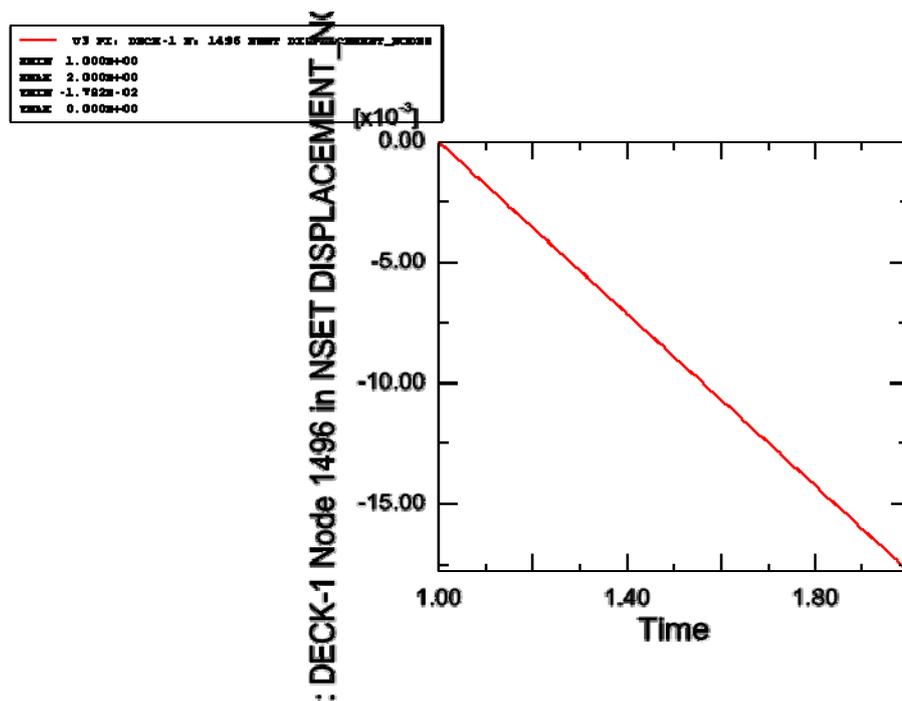
Figure 5.6: Displacement plot of node under maximum deflection for a single lane S1600 load.

In Figure 5.6 the time scale represents the load applied. At time 1 the load has not been applied and at time 2 the load is fully applied. The displacement of this node is linear which shows that the stresses within the concrete are not great enough to cause nonlinear behaviour.

Results provided by the analysis show that the five girder bridge deck is within deflection limits for all loading requirements defined by AS 5100.2[5].

5.3   Four Girder Deck

The four girder bridge deck is analysed in the same way as the five girder bridge deck. However the amount of load combinations have been reduced. The A160 axle load and the W80 wheel load are not analysed because the results from these loads were never a limiting factor in the design.

5.3.1   Concrete Slab Stresses

The maximum compressive stresses produced from the various loading combinations are shown in Table 5.6.

Table 5.6: Maximum in-plane stress for the top and bottom surface of the four girder bridge deck.

Loading Condition	Deck Maximum In-Plane Compressive Stress (MPa)	
	Top	Bottom
M1600 One Lane	8.41	9.84
M1600 Both Lanes	9.84	10.71
S1600 One Lane	8.66	7.79
S1600 Both Lanes	9.9	8.78

The maximum compressive stresses produced in the slab are only slightly greater than those produced from the five girder deck. However this time the maximum stresses

occur when both lanes are loaded with the M1600 load producing the maximum compressive stress of 10.71 MPa. Figure 5.7 shows the maximum in-plane compressive stress for the top slab surface with two lanes of an M1600 moving traffic load.
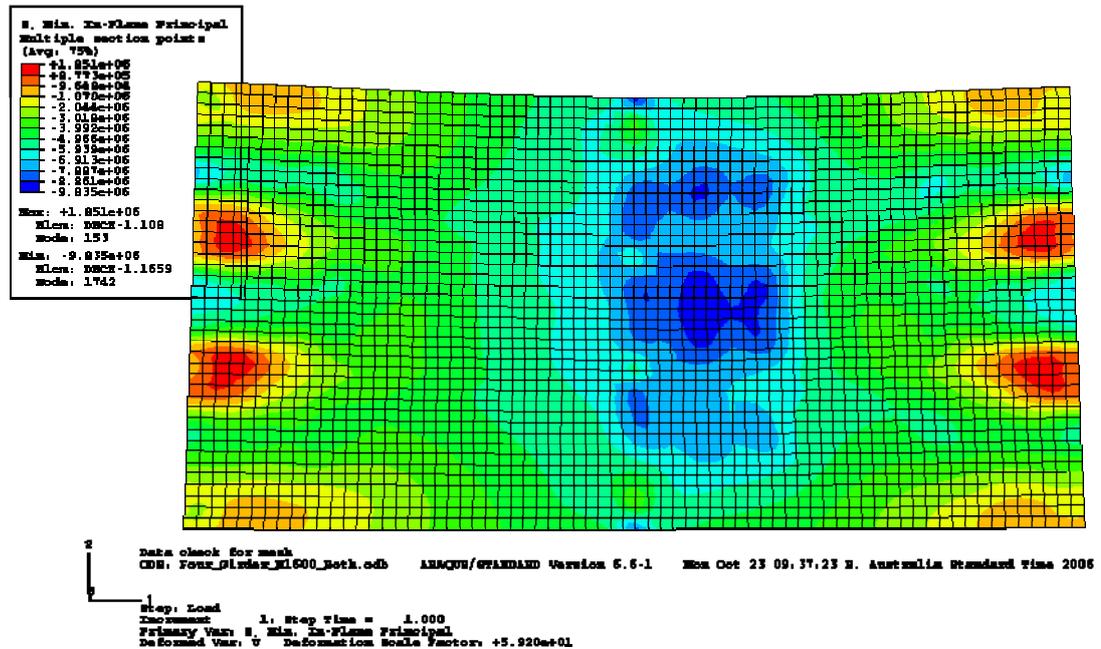
Figure 5.7: Maximum in-plane compressive stress for the top surface of the slab supporting the M1600 single lane traffic load.

Blue, green, yellow and orange all show areas that are in compression. The maximum compressive load is spread out around the centre of the slab in blue. From this it can be seen that the web stiffeners on the outer girders are again causing stress concentrations. The red areas show that above each girder high tensile stresses are being produced.

The largest compressive stress is produced on the bottom surface of the slab. In Figure 5.8 it can be seen that this is located at the end of the girders above the supports. These are the areas in dark blue.
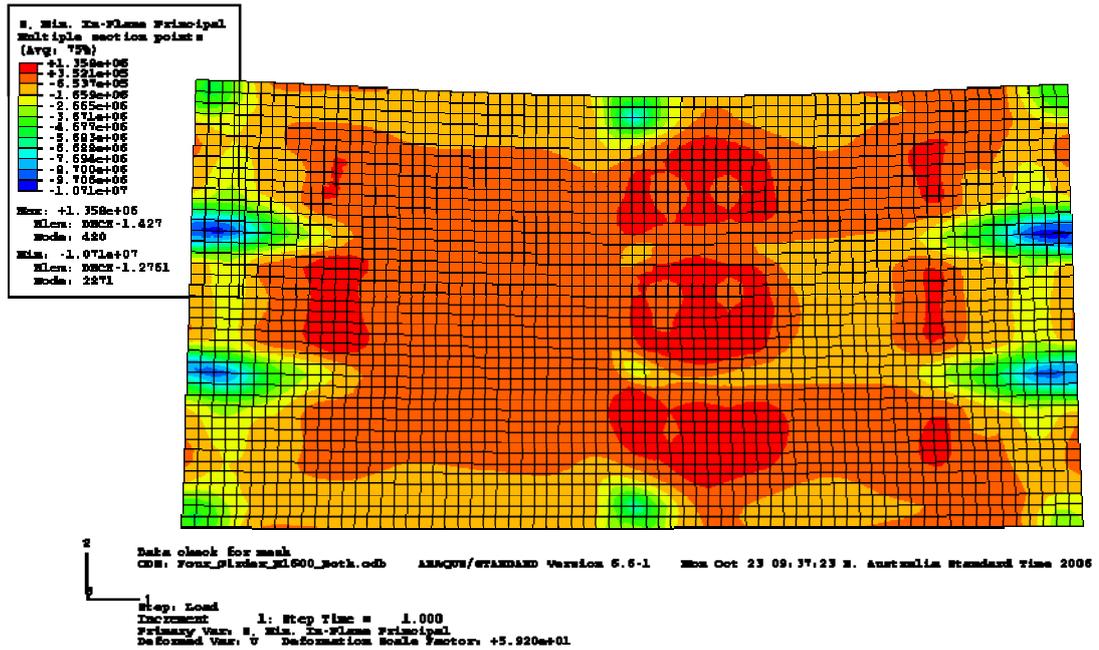
Figure 5.8: Maximum in-plane compressive stress for the bottom surface of the slab supporting the M1600 single lane traffic load.

The steel reinforcement stresses in the slab were only slightly greater than those produced in the five girder deck. The maximum tensile and compressive stresses for the reinforcement in the four girder bridge deck is displayed in Table 5.7.

Table 5.7: Maximum reinforcement stress for the four girder bridge deck.

Loading Condition	Stress in Steel Reinforcement (MPa)	
	Tension	Compression
M1600 One Lane	59.23	45.81
M1600 Both Lanes	54.48	51.09
S1600 One Lane	45.13	47.57
S1600 Both Lanes	41.94	53.2

The single lane M1600 load produced the highest tensile stress in the reinforcement of 59.23 MPa. This was on the far right edge of the deck centred between two girders. The wheel load placed at that location was not the sole cause of the maximum stress value. As the edge of the deck is loaded it is unable to transfer the stress in both directions.

This results with the reinforcement in the y-direction having to respond by taking more stress than it would if it was located at the centre of the deck.

The compressive stresses in the reinforcement are not as pronounced as they were in the five girder model. The increased span between the girders produced larger bending moments, which in turn increased the tensile stresses in the reinforcement. With the four girder model these stresses are becoming larger than the compressive stresses caused by small stress concentrations.

Results provided by the analysis show that the reinforced concrete deck for the four girder bridge deck can withstand the loading requirements defined by AS 5100.2 [5].

5.3.2   Girder Stresses

The maximum girder stresses recorded for each loading combination are shown in Table 5.8.

Table 5.8: Maximum girder stress for the four girder bridge deck.

Loading Condition	Maximum Stress in Steel (MPa)	
	Tension	Compression
M1600 One Lane	181.7	328.9
M1600 Both Lanes	255.5	340.2
S1600 One Lane	195.3	331.5
S1600 Both Lanes	272.1	341.1

The results produced from the analysis show that the steel is yielding with all compression values larger than the yield stress of 300 MPa. This is due to the same cause pointed out with the five girder model. The higher tensile stresses are due to the support conditions in the model not allowing a smooth transition of stress to the supports. The high tensile stress points are located on the web stiffeners at each end of the girders.

The maximum stresses at the midspan of the girder give a true indication of the strength of the section. The maximum tensile stress recorded in the bottom flange is 146 MPa and the maximum compressive stress recorded in the top flange is 42 MPa. These are well below the 280 MPa yield stress of the girder.

Ignoring the stresses at the supports results provided by the analysis show that the steel girders for the four girder bridge deck can withstand the loading requirements defined by AS 5100.2 [5].

5.3.3   Diaphragm Stresses

The diaphragm stresses recorded for the various loading combinations is shown in Table 5.9.

Table 5.9: Maximum and minimum stress values for diaphragms in four girder bridge deck.

Loading Condition	Stress in Steel (MPa)	
	Maximum	Minimum
M1600 One Lane	24.99	-0.143
M1600 Both Lanes	36.21	1.44
S1600 One Lane	26.55	-0.880
S1600 Both Lanes	38.01	1.29

All the diaphragms are primarily in tension with very small compression values due to the small bending that is occurring. The maximum stress value of 38.01 MPa is still small in comparison to the yield stress of the steel. Even though the diaphragms are plates there is still no chance of them buckling as they are all in tension.

Results provided by the analysis show that the diaphragms for the four girder bridge deck can withstand the loading requirements defined by AS 5100.2[5].

5.3.4   Deck Displacements

The maximum deflection limit for the bridge deck required by AS 5100.2 [5] is 28.3 mm. The results recorded for the different loading combinations on the four girder bridge deck are shown in Table 5.10.

Table 5.10: Maximum deflection values for loading combinations on the four girder bridge deck.

Loading Condition	Displacement (mm)
M1600 One Lane	20.95
M1600 Both Lanes	30.41
S1600 One Lane	21.72
S1600 Both Lanes	31.61

From the results obtained it can be seen that when both lanes of the bridge are loaded the deflection recorded exceeds the maximum limit. This was expected in a situation such as this because serviceability requirements are often the defining factor in structural design situations.

A deflection plot of the node under maximum displacement is shown in Figure 5.9.
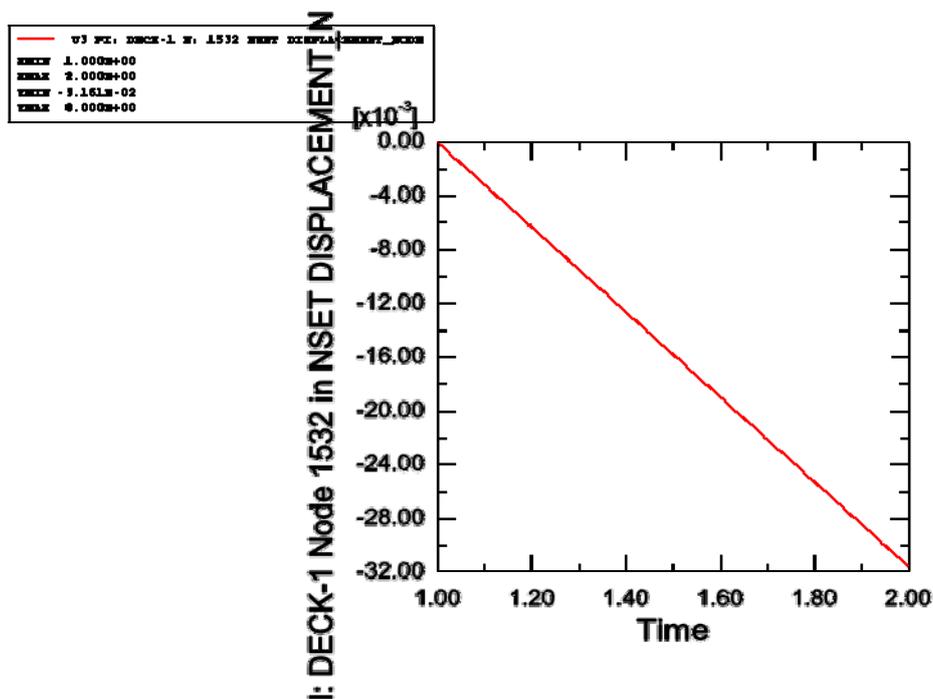
Figure 5.9: Displacement plot of node under maximum deflection for a dual lane M1600 load.

The plot of displacement with respect to the load applied produced a linear relationship. This shows that the concrete is not loaded enough for the nonlinear stress-strain relationship to come into effect.

Results provided by the analysis show that the four girder bridge deck is not within deflection requirements defined by AS 5100.2 [5]. Even though the stresses in the deck and girders are well within their limits the four girder model is not an acceptable design.

5.4   Three Girder Deck

Only the four major load combinations are applied to the three girder bridge deck. These are the M1600 and S1600 loads with one and two lanes of traffic. It is already known that the deflection with this model will be greater than that of the four girder bridge deck and therefore be an unacceptable design. However the three girder bridge

deck is analysed to see if the concrete will show nonlinear behaviour due to the increased stresses.

5.4.1   Concrete Slab Stresses

The compressive stresses produced from the various loading combinations are shown in Table 5.11.

Table 5.11: Maximum in-plane stress for the top and bottom surface of the three girder bridge deck.

Loading Condition	Maximum In-Plane Compressive Stress (MPa)	
	Top	Bottom
M1600 One Lane	20.77	22.49
M1600 Both Lanes	19.8	24.88
S1600 One Lane	17.42	20.89
S1600 Both Lanes	16.38	20.45

The stresses are now significantly larger than those produced with the four and five girder decks with a maximum recorded stress of 24.88 MPa. This is still less than the failure stress of 37.92 MPa. The maximum compressive stresses were located on the bottom surface of the slab at each end of the centre girder. These are seen in Figure 5.10 by the dark blue areas. These resulted from the negative bending moments produced in the girder and slab at this location.
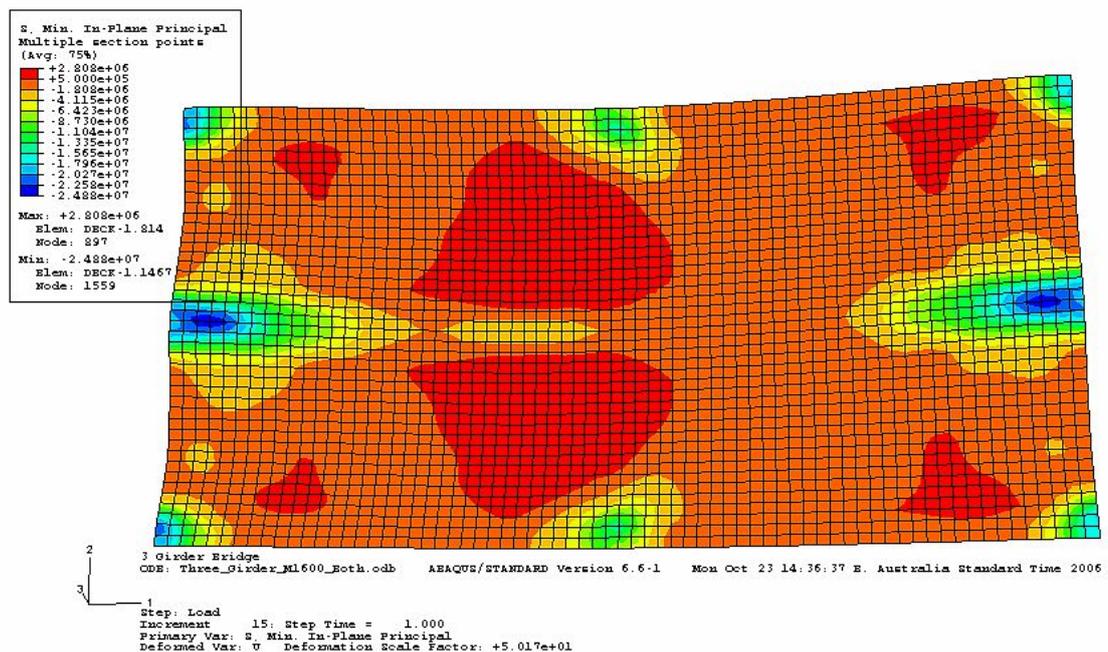
Figure 5.10: Maximum in-plane compressive stress for the bottom surface of the slab supporting the M1600 dual lane traffic load.

The green areas at the midpoint of the top and bottom of edges of Figure 5.10 show that the web stiffeners are still causing significant stress concentrations. The compressive stresses are approximately 10 MPa higher in these areas than the surrounding slab.

The compressive forces in the top concrete face are highest at each end of the slab, centred between the girders. The blue areas in Figure 5.11 show these compressive forces. As was explained in the four girder model the loads have been applied so some of the wheel loads from the M1600 and S1600 traffic loads are placed at the very end of the slab. The stresses are higher in these areas because the loads cannot be transferred in both the x and y-directions.
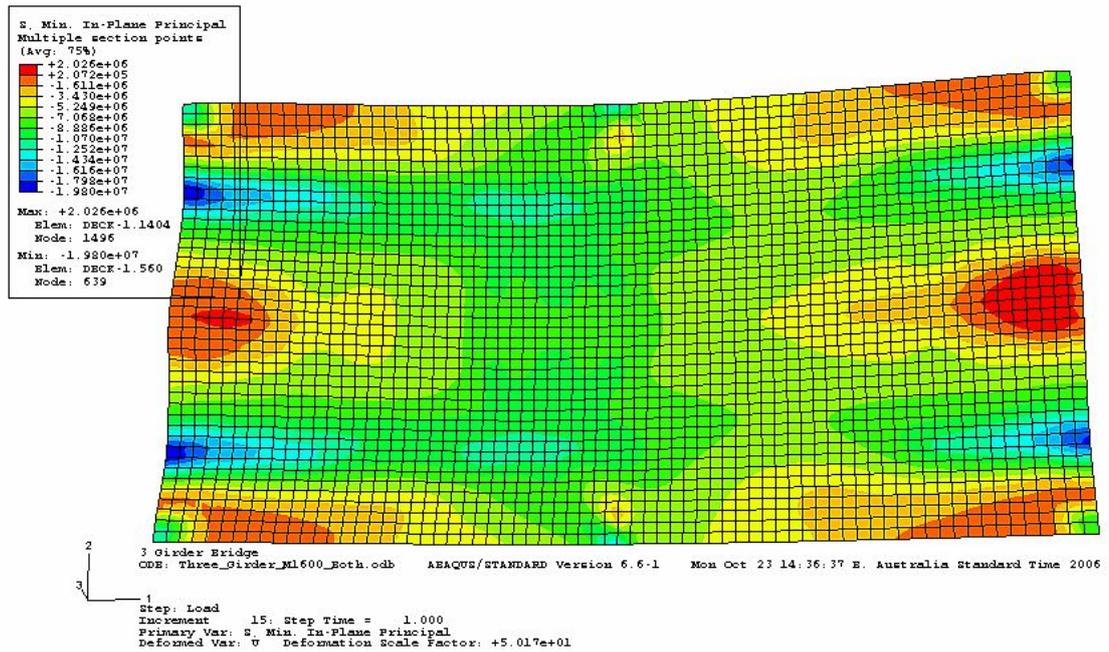
Figure 5.11: Maximum in-plane compressive stress for the top surface of the slab supporting the M1600 dual lane traffic load.

With the increased stress recorded in the concrete the tensile stress in the reinforcement would have to increase. This was the case with the results shown in Table 5.12.

Table 5.12: Maximum reinforcement stress for the three girder bridge deck.

Loading Condition	Stress in Steel Reinforcement (MPa)	
	Tension	Compression
M1600 One Lane	251.6	104.1
M1600 Both Lanes	211.3	107.9
S1600 One Lane	186.7	89.7
S1600 Both Lanes	169.8	86.4

The stresses recorded are much greater than those in the four and five girder models with a maximum recorded stress 251.6 MPa for the M1600 single lane load. This is located on the bottom layer of reinforcement midway between two girders at the end of the deck. This is the same area where the maximum compressive forces in the concrete are found. Even though the stress values are much higher than previous models they are still well under the failure stress of the steel at 500 MPa. It is also noted that the

difference between the ultimate compression and tension stresses are now much greater as the bending moments across the deck are significantly greater than those produced in the other models.

Results provided by the analysis show that the reinforced concrete slab for the three girder bridge deck can withstand the loading requirements defined by AS 5100.2 [5].

5.4.2   Girder Stresses

The maximum girder stresses recorded for each loading combination are shown in Table 5.13.

Table 5.13: Maximum girder stress for the three girder bridge-deck.

	Maximum Stress in Steel (MPa)	
Loading Condition	**Tension**	**Compression**
M1600 One Lane	282.2	338.8
M1600 Both Lanes	337.8	345.3
S1600 One Lane	281.7	339.1
S1600 Both Lanes	336.7	345.0

From Table 5.13 it is seen that the stresses in both tension and compression are exceeding the yield point of the steel. However these yield points are again at the supports. Figure 5.12 shows the contour diagram of plastic strain for the centre girder with both lanes under the M1600 load.

The contour plot clearly shows that the only areas that have yielded are the areas above the supports. It has been stated previously that the high stresses in these areas would be reduced if the bearing pads were modelled in the analysis. The steel is still within its limits at the midpoint of the girder where a true indication of its strength can be assessed. The maximum stress value recorded at this point in the bottom flange is 215 MPa and the stress at the top flange is 156 MPa. Both are within the yield limits for the steel.
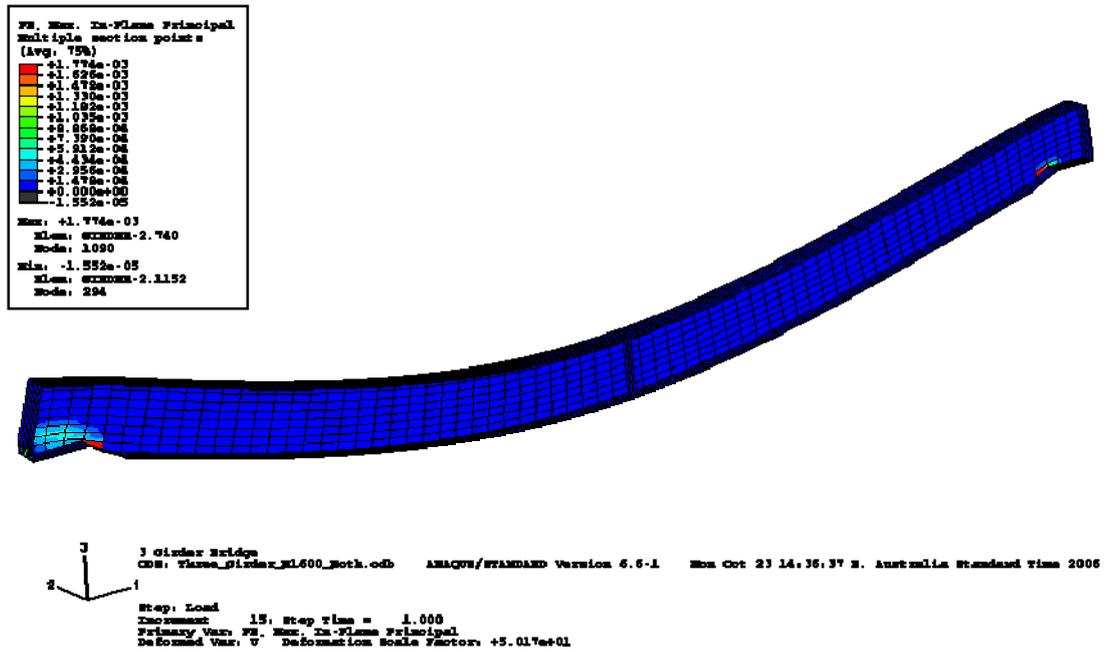
Figure 5.12: Plastic strain of the centre girder in the three girder bridge deck under M1600 dual lane traffic load.

Ignoring the stresses at the supports results provided by the analysis show that the steel girders for the three girder bridge deck can withstand the loading requirements defined by AS 5100.2 [5].

5.4.3   Diaphragm Stresses

The diaphragm stresses recorded for the various loading combination are shown in Table 5.14.

Table 5.14: Maximum and minimum stress values for diaphragms in three girder bridge deck.

Loading Condition	Maximum Stress in Steel (MPa)	
	Maximum	**Minimum**
M1600 One Lane	35.37	-4.31
M1600 Both Lanes	47.18	3.27
S1600 One Lane	36.95	-3.87
S1600 Both Lanes	46.11	1.01

All the diaphragms are primarily in tension with low compression values due to the small bending that is occurring. The maximum stress value of 47.18 MPa is still small in comparison to the yield stress of the steel. The compression stresses have risen in comparison to the other models however the values are still low. In future work the bridge deck could be analysed with nonlinear geometry to ensure that buckling will not occur.

Results provided by the analysis show that the diaphragms for the three girder bridge deck can withstand the loading requirements defined by AS 5100.2 [5].

5.4.4   Deck Displacements

The deflection results recorded for the different loading combinations on the three girder bridge deck are shown in Table 5.15.

Table 5.15: Maximum deflection values for loading combinations on the three girder bridge deck.

Loading Condition	Displacement (mm)
M1600 One Lane	24.36
M1600 Both Lanes	35.88
S1600 One Lane	24.9
S1600 Both Lanes	34.23

The results show that when both lanes are loaded the deflection exceeds the limit of 28.3 mm. A plot of the maximum deflection from the M1600 load in both lanes is shown in Figure 5.13.

Unlike the four and five girder bridges the displacement plot for the three girder bridge deck is not linear. It can be seen that it is initially linear however as the load increases the deflection starts to increase at a greater rate. This is due to the nonlinear stress-strain relationship in the concrete. Once the stress has reached a certain point microcracks begin to develop between the mortar and course aggregate which changes the relationship between stress and strain in the concrete [1].
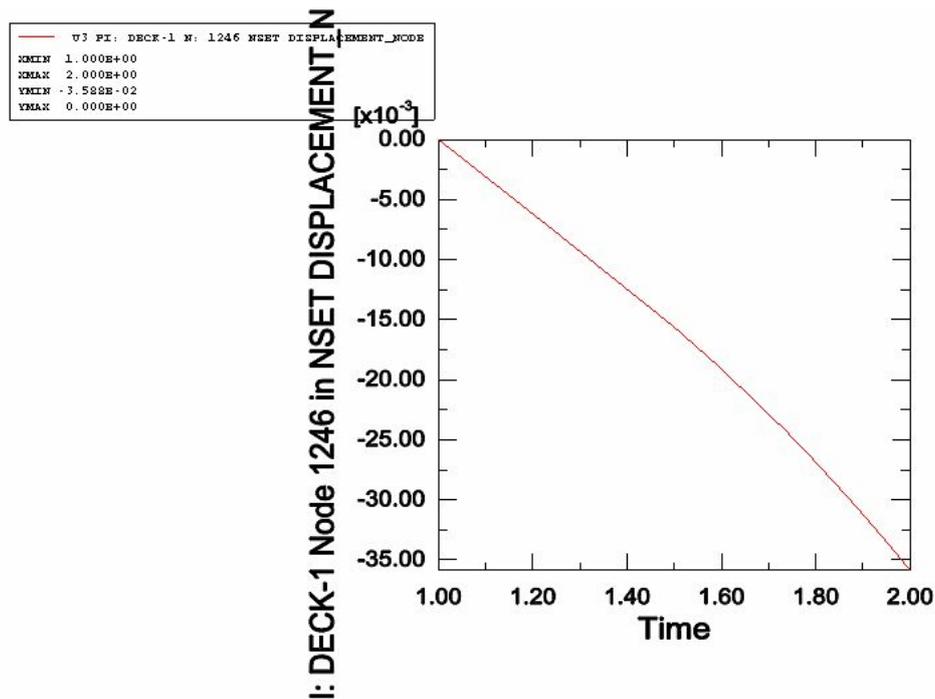
Figure 5.13: Displacement plot of node under maximum deflection for a dual lane M1600 load.

Results provided by the analysis show that the three girder bridge deck is not within deflection requirements defined by AS 5100.2 [5]. Even though the stresses in the slab and girders are within their limits the three girder model is not an acceptable design.

5.5   Optimum Girder Spacing

Results from the finite element analyses of three different girder spacings have been analysed. It has been determined that the four girder model would provide an optimum spacing for a two lane bridge to meet Australian Standards.

All stresses in the members of the four girder bridge deck were well within the yield and failure limits. In fact they are low enough that the thickness of the concrete slab and the amount of steel reinforcement could be reduced while still meeting strength requirements. However the bridge deck did not meet the deflection requirements defined by AS 5100.2 [5]. Deflection values were only slightly larger than the limit therefore this was considered a minor problem because a welded I-beam one size larger could be used. This would allow the bridge deck to meet serviceability requirements.

The three girder bridge deck also meet strength parameters and failed with deflection requirements. It has been dismissed as being the optimum girder spacing for a number of reasons. The slab has shown some nonlinear behaviour that is the result of excessive cracking. The bridge could last for a few months or longer however the cracks would allow corrosion of the steel reinforcement significantly reducing the strength of the slab. In addition the members could be subjected to buckling, as the stresses recorded in the members are fairly high.

The five girder bridge deck was rejected as the optimum because it is over designed. The stresses in all the members are very small and deflections are well below the limits.

5.6   Summary

This chapter analysed the results from three different bridge decks containing different girder spacings. Loading combinations were applied to produce the most adverse effects on the bridge deck. From this an optimum girder spacing was chosen.

All models were able to withstand the stresses generated in their members. The exception was the girders at the supports. The boundary conditions modelled did not represent a true support and therefore caused excessive stresses in the girders at these locations.

Serviceability requirements defined by AS 5100.2 [5] stated that the maximum deflection could not exceed span/600. Both the three and four girder models exceeded this limit therefore making both models unacceptable designs.

Buckling was not considered in the analysis. However the low stresses generated in both the four and five girder bridge decks gave a good indication that buckling will not occur in these models. To be certain that this is the case it is recommended that nonlinear geometry be specified with the finite element analysis.

The optimum girder spacing chosen was the four girder model for a two lane bridge to meet Australian Standards. This would make the most use of its materials while still meeting strength and serviceability requirements.

Chapter 6

Conclusions

6.1   Summary

This research project has effectively used the finite element method to analyse a bridge deck consisting of a reinforced concrete slab and steel girders. Three separate models were analysed with a different number of girders and from this the optimum girder spacing was determined.

6.2   Conclusions

In order to achieve the aim of this project a number of objectives had to be met. The completion of these objectives is described below.

- As concrete displays complex nonlinear properties it is difficult to predict its true behaviour. Finite element analysis techniques such as the smeared crack approach have now been developed, however as with any theoretical modelling there are always questions about the accuracy of the results. Research has shown that experimental testing results have closely matched those produced from the finite element method.

- To ensure all bridges constructed in Australia can withstand all vehicular traffic they must meet particular loading requirements. These were determined from AS 5100.2 [5].

- A preliminary sizing of bridge members were determined as a starting point for the finite element model. By basing this design purely on bending moments a 250 mm thick concrete slab with 12 mm reinforcement was chosen for the deck and a 1000WB322 I-beam was determined for the girders.

- The ABAQUS [4] multi purpose finite element software was used for the analysis. A number of finite element models of the bridge deck were produced for various loading combinations and girder spacings. These were then used to determine the maximum stresses and deflections imposed on the deck.

- Results obtained showed that all models were able to withstand the stresses generated in their members. The exception was the girders at the supports as the boundary conditions did not represent an elastomer bearing pad. This caused excessive stress concentrations. Both three and four girder bridges exceed the deflection limits for serviceability requirements making them unsuitable designs.

- The optimum girder spacing chosen was the four girder model for a two lane bridge to meet Australian Standards. Even though this did not meet deflection requirements it proved the most efficient design with the use of its materials. The maximum deflection was only slightly larger than the limit so it could be easily improved by increasing the girder size.

6.3   Further Work

There are a number of areas of this project where further work could be conducted. These are listed below.

- The supports for the girders were modelled as if they were fixed to a completely rigid structure. This caused excessive stresses in the girders at these locations. Elastomer bearing pads could be modelled at these locations. This would remove the localised stresses on the girders and it would also be useful to find out how effective a typical bearing pad is at providing a smooth transition of stress from the girders to the supports.

- The nonlinear geometry of the structure was not analysed with the models, therefore buckling effects were not considered. Nonlinear geometry could be used to ensure the separate models analysed are not subjected to these buckling effects.

- The four girder bridge deck was chosen as being the optimum girder spacing. However the deck was well over designed with very small stress values recorded in the reinforcement and in the concrete. Additional trial models could be produced with varying deck thicknesses and reinforcement areas. This would allow the optimum deck thickness for the four girder bridge-deck to be determined.

References

[1]   Warner, RF, Rangan, BV, Hall, AS, & Faulkes, KA 1998, *Concrete Structures*, Addison Wesley Longman Australia Pty Ltd, Melbourne.

[2]   O'Connor, JJ & Robertson, EF 2000, *Richard Courant*, School of Mathematics and Statistics University of St Andrews, Scotland, viewed 6 June 2006, <http://www-history.mcs.st-andrews.ac.uk/Biographies/Courant.html>.

[3]   Widas, P 1997, *Introduction to Finite Element Analysis*, Virginia Tech Material Science and Engineering, viewed 6 June 2006, <http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/num/widas/history.html>

[4]   'ABAQUS', version 6.6, Hilbert, Karlsson & Sorenson Inc. Pawtucket, RI, USA, 2006. see www.abaqus.com

[5]   *AS 5100.2 Bridge Design Part 2: Design Loads* 2004, Standards Australia International Ltd, Sydney.

[6]   'ABAQUS Documentation (including User Manual, Theory Manual & Example Problems Manual)', version 6.5, Hilbert, Karlsson & Sorenson Inc. Pawtucket, RI, USA, 2004. see www.abaqus.com

[7]   Petrangeli M & Ozbolt J 1996, 'Smeared Crack Approaches – Material Modelling', *Journal of Engineering Mechanics*, vol. 122, no. 6, pp. 545-554.

References

[8] Hilleborg, AM, Modeer, and PE, Petersson, 'Analysis of Crack Formation and Crack Growth in Concrete by Means of Fracture Mechanics and Finite Elements', *Cement and Concrete Research*, vol. 6, pp. 773–782, 1976.

[9] Biggs, MR, Barton, FW, Gomez, JP, Massarelli, PJ, McKeel, WT 2000, *Finite Element Modelling and Analysis of Reinforced –Concrete Bridge Decks*, Virginia Transportation Research Council, Virginia.

[10] Huria, V, Lee, K L, Aktan, A E 1993, Nonlinear Finite Element Analysis of RC Slab Bridge. *ASCE Journal of Structural Engineering*, vol. 119, no. 1, pp 88-107.

[11] 'HRSSP Hot Rolled Structural Steel Products' 3rd edition 2003, OneSteel, Australia. see www.onesteel.com

[12] *AS 5100.1 Bridge Design Part 1: Design Scope and General Principles* 2004, Standards Australia International Ltd, Sydney.

[13] *Rural Road Design: A Guide to the Geometric Design of Rural Roads* 2003, Austroads Inc, Sydney.

[14] *AS/NZS 3845 Road Safety Barrier Systems* 1999, Standards Australia International Ltd, Sydney.

[15] Multiframe 4D Demo, version 9.05, Formation Design Systems, 2004. see www.formsys.com/multiframe

[16] *AS 5100.5 Bridge Design Part 5: Concrete* 2004, Standards Australia International Ltd, Sydney.

[17] Gilbert, RI, and RF, Warner, 'Tension Stiffening in Reinforced Concrete Slabs,' *Journal of the Structural Division, American Society of Civil Engineers*, vol. 104, ST12, pp. 1885–1900, 1978.

Appendix A – Project Specification

University of Southern Queensland
Faculty of Engineering and Surveying

ENG 4111/2 Research Project
PROJECT SPECIFICATION

For: Lindsay Klein

Topic: Finite element analysis of a composite (steel and reinforced concrete) bridge deck using ABAQUS.

Supervisor: Dr Amar Khennane

Project Aim: The aim of this project is to use the finite element method to determine the optimum
girder spacing on the response of a composite (steel and reinforced concrete) bridge deck.

Program: Issue A, 24 March 2006

1. Preliminary design of the bridge.

2. Literary review on the non-linear behaviour of concrete.

3. Learn how to effectively use the ABAQUS finite element software.

4. Determine the loading on the bridge from the relevant Australian Standards.

5. Model the bridge deck using the finite element method.

6. Employ the finite element model to determine the effect of girder spacing on the deck response.

AGREED: _____ (Student) _____ (Supervisor)

__ / __ / __   __ / __ / __

Appendix B – Model Input Files

B.1 - Five Girder Model

```
*Heading
 5 Girder Bridge, M1600 Loading in One Lanes
** Job name: Five Girder name: M1600 Loading
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Deck
*INCLUDE,INPUT=Deck_Mesh.inp
**
***************Section Properties Deck****************
**
** Section: Concrete_Deck
*Shell Section, elset=Deck_Elements, material=Concrete
0.25, 9
*Rebar Layer
xx_bottom, 7.854e-5, 0.1, -0.08, Steel_500, 0., 1
yy_bottom, 7.854e-5, 0.1, -0.08, Steel_500, 90., 1
xx_top, 7.854e-5, 0.1, 0.08, Steel_500, 0., 1
yy_top, 7.854e-5, 0.1, 0.08, Steel_500, 90., 1
**
*End Part
**
*Part, name=Diaphragm
*INCLUDE, INPUT=Diaphragm_Mesh.inp
**
*BEAM SECTION, elset=Diaphragm_Elements, section=rect, material=Steel_280
.032, .3
0, 0, 1
*End Part
**
*Part, name=Girder
*INCLUDE, INPUT=Girder_Mesh.inp
**
****************Select Flange Nodes****************
**
*Nset, nset=Girder_Flange_Nodes, generate
1, 873, 1
*Nset, nset=Girder_Flange_Nodes, generate
1112, 1501, 1
*Nset, nset=Girder_Flange_Nodes, generate
1740, 1943, 1
**
****************Select  Flange Elements*******************
**
*Elset, elset=Girder_Flange_Elements, generate
1, 484, 1
*Elset, elset=Girder_Flange_Elements, generate
765, 1312, 1
*Elset, elset=Girder_Flange_Elements, generate
1593, 1872, 1
**
****************Select Web Nodes***********************8
**
*Nset, nset=Girder_Web_Nodes
     1,     2,     7,     9,    11,    14,    19,    20,    21,    22,    23,    24,    25,
  52,    53,    54
    55,    56,    57,    58,    59,    60,    61,    62,    63,    64,    65,    66,    67,
  68,    69,    70
    71,    72,    73,    74,    75,    76,    77,    78,    79,    80,    81,    82,    83,
  84,    85,   123
   124,   125,   126,   127,   128,   129,   130,   131,   132,   133,   134,   135,   136,
 137,   138,   139
   140,   141,   142,   143,   144,   145,   146,   147,   148,   149,   150,   151,   152,
 153,   154,   155
   156,   194,   195,   196,   197,   198,   199,   200,   201,   202,   203,   204,   205,
 206,   207,   208
   209,   210,   211,   212,   213,   214,   215,   216,   217,   218,   219,   220,   221,
 222,   223,   224
   225,   226,   227,   265,   266,   267,   268,   269,   270,   271,   309,   310,   311,
 312,   313,   314
```

```
 315,  356,  357,  358,  359,  360,  361,  362,  363,  364,  365,  366,  367,
 368,  369,  370
 371,  372,  373,  374,  375,  376,  377,  378,  379,  380,  381,  382,  383,
 384,  385,  386
 387,  388,  389,  874,  875,  876,  877,  878,  879,  880,  881,  882,  883,
 884,  885,  886
 887,  888,  889,  890,  891,  892,  893,  894,  895,  896,  897,  898,  899,
 900,  901,  902
 903,  904,  905,  906,  907,  908,  909,  910,  911,  912,  913,  914,  915,
 916,  917,  918
 919,  920,  921,  922,  923,  924,  925,  926,  927,  928,  929,  930,  931,
 932,  933,  934
 935,  936,  937,  938,  939,  940,  941,  942,  943,  944,  945,  946,  947,
 948,  949,  950
 951,  952,  953,  954,  955,  956,  957,  958,  959,  960,  961,  962,  963,
 964,  965,  966
 967,  968,  969,  970,  971,  972,  973,  974,  975,  976,  977,  978,  979,
 980,  981,  982
 983,  984,  985,  986,  987,  988,  989,  990,  991,  992,  993,  994,  995,
 996,  997,  998
 999, 1000, 1001, 1002, 1003, 1004, 1005, 1006, 1007, 1008, 1009, 1010, 1011,
1012, 1013, 1014
1015, 1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026, 1027,
1028, 1029, 1030
1031, 1032, 1033, 1034, 1035, 1036, 1037, 1038, 1039, 1040, 1041, 1042, 1043,
1044, 1045, 1046
1047, 1048, 1049, 1050, 1051, 1052, 1053, 1054, 1055, 1056, 1057, 1058, 1059,
1060, 1061, 1062
1063, 1064, 1065, 1066, 1067, 1068, 1069, 1070, 1071, 1072, 1073, 1074, 1075,
1076, 1077, 1078
1079, 1080, 1081, 1082, 1083, 1084, 1085, 1086, 1087, 1088, 1089, 1090, 1091,
1092, 1093, 1094
1095, 1096, 1097, 1098, 1099, 1100, 1101, 1102, 1103, 1104, 1105, 1106, 1107,
1108, 1109, 1110
1111, 1502, 1503, 1504, 1505, 1506, 1507, 1508, 1509, 1510, 1511, 1512, 1513,
1514, 1515, 1516
1517, 1518, 1519, 1520, 1521, 1522, 1523, 1524, 1525, 1526, 1527, 1528, 1529,
1530, 1531, 1532
1533, 1534, 1535, 1536, 1537, 1538, 1539, 1540, 1541, 1542, 1543, 1544, 1545,
1546, 1547, 1548
1549, 1550, 1551, 1552, 1553, 1554, 1555, 1556, 1557, 1558, 1559, 1560, 1561,
1562, 1563, 1564
1565, 1566, 1567, 1568, 1569, 1570, 1571, 1572, 1573, 1574, 1575, 1576, 1577,
1578, 1579, 1580
1581, 1582, 1583, 1584, 1585, 1586, 1587, 1588, 1589, 1590, 1591, 1592, 1593,
1594, 1595, 1596
1597, 1598, 1599, 1600, 1601, 1602, 1603, 1604, 1605, 1606, 1607, 1608, 1609,
1610, 1611, 1612
1613, 1614, 1615, 1616, 1617, 1618, 1619, 1620, 1621, 1622, 1623, 1624, 1625,
1626, 1627, 1628
1629, 1630, 1631, 1632, 1633, 1634, 1635, 1636, 1637, 1638, 1639, 1640, 1641,
1642, 1643, 1644
1645, 1646, 1647, 1648, 1649, 1650, 1651, 1652, 1653, 1654, 1655, 1656, 1657,
1658, 1659, 1660
1661, 1662, 1663, 1664, 1665, 1666, 1667, 1668, 1669, 1670, 1671, 1672, 1673,
1674, 1675, 1676
1677, 1678, 1679, 1680, 1681, 1682, 1683, 1684, 1685, 1686, 1687, 1688, 1689,
1690, 1691, 1692
1693, 1694, 1695, 1696, 1697, 1698, 1699, 1700, 1701, 1702, 1703, 1704, 1705,
1706, 1707, 1708
1709, 1710, 1711, 1712, 1713, 1714, 1715, 1716, 1717, 1718, 1719, 1720, 1721,
1722, 1723, 1724
1725, 1726, 1727, 1728, 1729, 1730, 1731, 1732, 1733, 1734, 1735, 1736, 1737,
1738, 1739
**
***************************Select Web Elements********************
**
*Elset, elset=Girder_Web_Elements, generate
485, 764, 1
*Elset, elset=Girder_Web_Elements, generate
1313, 1592, 1
```

```
**
*****************Section Properties Girder***********************
**
** Section: Girder_Flange
*Shell Section, elset=Girder_Flange_Elements, material=Steel_280
0.032, 5
** Section: Girder_Web
*Shell Section, elset=Girder_Web_Elements, material=Steel_300
0.016, 5
**
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Deck-1, part=Deck
*End Instance
**
*Instance, name=Girder-1, part=Girder
          -9., -4.728571429,      -0.496
          -9., -4.728571429,      -0.496, -8.42264973081037, -
4.15122115981037, 0.0813502691896258,         120.
*End Instance
**
*Instance, name=Girder-2, part=Girder
          -9., -2.364285714,      -0.496
          -9., -2.364285714,      -0.496, -8.42264973081037, -
1.78693544481037, 0.0813502691896258,         120.
*End Instance
**
*Instance, name=Girder-3, part=Girder
          -9.,          0.,      -0.496
          -9.,          0.,      -0.496, -8.42264973081037,
0.577350269189626, 0.0813502691896258,         120.
*End Instance
**
*Instance, name=Girder-4, part=Girder
          -9.,  2.364285714,      -0.496
          -9.,  2.364285714,      -0.496, -8.42264973081037, 2.94163598318963,
0.0813502691896258,         120.
*End Instance
**
*Instance, name=Girder-5, part=Girder
          -9.,  4.728571429,      -0.496
          -9.,  4.728571429,      -0.496, -8.42264973081037, 5.30592169818963,
0.0813502691896258,         120.
*End Instance
**
*Instance, name=Diaphragm-1, part=Diaphragm
           9., 3.5642857147, -0.743999999999999
           9., 3.5642857147, -0.743999999999999,           9., 3.5642857147,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-2, part=Diaphragm
           0., 3.5642857147, -0.743999999999999
           0., 3.5642857147, -0.743999999999999,           0., 3.5642857147,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-3, part=Diaphragm
          -9., 3.5642857147,      -0.744
          -9., 3.5642857147,      -0.744,          -9., 3.5642857147,
0.256, 90
*End Instance
**
*Instance, name=Diaphragm-4, part=Diaphragm
           9., 1.1999999997, -0.743999999999999
```

```
            9., 1.1999999997, -0.74399999999999,          9., 1.1999999997,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-5, part=Diaphragm
          0., 1.1999999997, -0.74399999999999
          0., 1.1999999997, -0.74399999999999,          0., 1.1999999997,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-6, part=Diaphragm
         -9., 1.1999999997,       -0.744
         -9., 1.1999999997,       -0.744,          -9., 1.1999999997,
0.256, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-7, part=Diaphragm
          9., -1.1642857143, -0.74399999999999
          9., -1.1642857143, -0.74399999999999,          9., -1.1642857143,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-8, part=Diaphragm
          0., -1.1642857143, -0.74399999999999
          0., -1.1642857143, -0.74399999999999,          0., -1.1642857143,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-9, part=Diaphragm
         -9., -1.1642857143,       -0.744
         -9., -1.1642857143,       -0.744,          -9., -1.1642857143,
0.256, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-10, part=Diaphragm
          9., -3.5285714283, -0.74399999999999
          9., -3.5285714283, -0.74399999999999,          9., -3.5285714283,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-11, part=Diaphragm
          0., -3.5285714283, -0.74399999999999
          0., -3.5285714283, -0.74399999999999,          0., -3.5285714283,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-12, part=Diaphragm
         -9., -3.5285714283,       -0.744
         -9., -3.5285714283,       -0.744,          -9., -3.5285714283,
0.256, 89.9999990194245
*End Instance
**
**APPLY MULITPOINT CONSTRAINT BETWEEN TRUSSES AND GIRDERS
**
****************Define nodes to be used in constraints**************
**
*Nset, nset=girder_1r, instance=Girder-1
330, 43, 344
*Nset, nset=girder_2l, instance=Girder-2
324, 34, 350
*Nset, nset=girder_2r, instance=Girder-2
330, 43, 344
*Nset, nset=girder_3l, instance=Girder-3
324, 34, 350
*Nset, nset=girder_3r, instance=Girder-3
330, 43, 344
*Nset, nset=girder_4l, instance=Girder-4
324, 34, 350
*Nset, nset=girder_4r, instance=Girder-4
330, 43, 344
*Nset, nset=girder_5l, instance=Girder-5
```

```
324, 34, 350
**
***Girder 1 right
*Nset, nset=girder_1r1, instance=Girder-1
330
*Nset, nset=girder_1r2, instance=Girder-1
43
*Nset, nset=girder_1r3, instance=Girder-1
344
***girder 2 left
*Nset, nset=girder_2l1, instance=Girder-2
324
*Nset, nset=girder_2l2, instance=Girder-2
34
*Nset, nset=girder_2l3, instance=Girder-2
350
***girder 2 right
*Nset, nset=girder_2r1, instance=Girder-2
330
*Nset, nset=girder_2r2, instance=Girder-2
43
*Nset, nset=girder_2r3, instance=Girder-2
344
***girder 3 left
*Nset, nset=girder_3l1, instance=Girder-3
324
*Nset, nset=girder_3l2, instance=Girder-3
34
*Nset, nset=girder_3l3, instance=Girder-3
350
***girder 3 right
*Nset, nset=girder_3r1, instance=Girder-3
330
*Nset, nset=girder_3r2, instance=Girder-3
43
*Nset, nset=girder_3r3, instance=Girder-3
344
***girder 4 left
*Nset, nset=girder_4l1, instance=Girder-4
324
*Nset, nset=girder_4l2, instance=Girder-4
34
*Nset, nset=girder_4l3, instance=Girder-4
350
***girder 4 right
*Nset, nset=girder_4r1, instance=Girder-4
330
*Nset, nset=girder_4r2, instance=Girder-4
43
*Nset, nset=girder_4r3, instance=Girder-4
344
***girder 5 left
*Nset, nset=girder_5l1, instance=Girder-5
324
*Nset, nset=girder_5l2, instance=Girder-5
34
*Nset, nset=girder_5l3, instance=Girder-5
350
**
*Nset, nset=diaphragm_1_left, instance=Diaphragm-1
1
*Nset, nset=diaphragm_1_right, instance=Diaphragm-1
7
*Nset, nset=diaphragm_2_left, instance=Diaphragm-2
1
*Nset, nset=diaphragm_2_right, instance=Diaphragm-2
7
*Nset, nset=diaphragm_3_left, instance=Diaphragm-3
1
*Nset, nset=diaphragm_3_right, instance=Diaphragm-3
7
```

```
*Nset, nset=diaphragm_4_left, instance=Diaphragm-4
1
*Nset, nset=diaphragm_4_right, instance=Diaphragm-4
7
*Nset, nset=diaphragm_5_left, instance=Diaphragm-5
1
*Nset, nset=diaphragm_5_right, instance=Diaphragm-5
7
*Nset, nset=diaphragm_6_left, instance=Diaphragm-6
1
*Nset, nset=diaphragm_6_right, instance=Diaphragm-6
7
*Nset, nset=diaphragm_7_left, instance=Diaphragm-7
1
*Nset, nset=diaphragm_7_right, instance=Diaphragm-7
7
*Nset, nset=diaphragm_8_left, instance=Diaphragm-8
1
*Nset, nset=diaphragm_8_right, instance=Diaphragm-8
7
*Nset, nset=diaphragm_9_left, instance=Diaphragm-9
1
*Nset, nset=diaphragm_9_right, instance=Diaphragm-9
7
*Nset, nset=diaphragm_10_left, instance=Diaphragm-10
1
*Nset, nset=diaphragm_10_right, instance=Diaphragm-10
7
*Nset, nset=diaphragm_11_left, instance=Diaphragm-11
1
*Nset, nset=diaphragm_11_right, instance=Diaphragm-11
7
*Nset, nset=diaphragm_12_left, instance=Diaphragm-12
1
*Nset, nset=diaphragm_12_right, instance=Diaphragm-12
7
**
*Nset, nset=diaphragms_g1r
diaphragm_10_left, diaphragm_11_left, diaphragm_12_left
*Nset, nset=diaphragms_g2l
diaphragm_10_right, diaphragm_11_right, diaphragm_12_right
*Nset, nset=diaphragms_g2r
diaphragm_7_left, diaphragm_8_left, diaphragm_9_left
*Nset, nset=diaphragms_g3l
diaphragm_7_right, diaphragm_8_right, diaphragm_9_right
*Nset, nset=diaphragms_g3r
diaphragm_4_left, diaphragm_5_left, diaphragm_6_left
*Nset, nset=diaphragms_g4l
diaphragm_4_right, diaphragm_5_right, diaphragm_6_right
*Nset, nset=diaphragms_g4r
diaphragm_1_left, diaphragm_2_left, diaphragm_3_left
*Nset, nset=diaphragms_g5l
diaphragm_1_right, diaphragm_2_right, diaphragm_3_right
**
*************************Define constraint equations*****************
**
***girder1-2
*MPC
TIE,diaphragm_10_left, girder_1r1
*MPC
TIE, diaphragm_10_right,girder_2l1
*MPC
TIE, diaphragm_11_left,girder_1r2
*MPC
TIE, diaphragm_11_right,girder_2l2
*MPC
TIE, diaphragm_12_left,girder_1r3
*MPC
TIE, diaphragm_12_right,girder_2l3
**
***girder2-3
```

```
*MPC
TIE,diaphragm_7_left, girder_2r1
*MPC
TIE, diaphragm_7_right,girder_3l1
*MPC
TIE, diaphragm_8_left,girder_2r2
*MPC
TIE, diaphragm_8_right,girder_3l2
*MPC
TIE, diaphragm_9_left,girder_2r3
*MPC
TIE, diaphragm_9_right,girder_3l3
**
***girder3-4
*MPC
TIE,diaphragm_4_left, girder_3r1
*MPC
TIE, diaphragm_4_right,girder_4l1
*MPC
TIE, diaphragm_5_left,girder_3r2
*MPC
TIE, diaphragm_5_right,girder_4l2
*MPC
TIE, diaphragm_6_left,girder_3r3
*MPC
TIE, diaphragm_6_right,girder_4l3
**
***girder4-5
*MPC
TIE,diaphragm_1_left, girder_4r1
*MPC
TIE, diaphragm_1_right,girder_5l1
*MPC
TIE, diaphragm_2_left,girder_4r2
*MPC
TIE, diaphragm_2_right,girder_5l2
*MPC
TIE, diaphragm_3_left,girder_4r3
*MPC
TIE, diaphragm_3_right,girder_5l3
**
**
***************************DEFINE SURFACES FOR CONTACT**********************8
**
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
345, 484, 1
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
765, 904, 1
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
1033, 1312, 1
**
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
345, 484, 1
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
765, 904, 1
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
1033, 1312, 1
**
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
345, 484, 1
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
765, 904, 1
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
1033, 1312, 1
**
*Elset, elset=girder_4_topflange, instance=Girder-4, generate
345, 484, 1
*Elset, elset=girder_4_topflange, instance=Girder-4, generate
765, 904, 1
*Elset, elset=girder_4_topflange, instance=Girder-4, generate
1033, 1312, 1
```

```
**
*Elset, elset=girder_5_topflange, instance=Girder-5, generate
345, 484, 1
*Elset, elset=girder_5_topflange, instance=Girder-5, generate
765, 904, 1
*Elset, elset=girder_5_topflange, instance=Girder-5, generate
1033, 1312, 1
**
*Elset, elset=Deck_Elements_1, instance=Deck-1
Deck_Elements
**
*SURFACE, name=top_surface_girder_1, type=element
girder_1_topflange, SNEG
*SURFACE, name=top_surface_girder_2, type=element
girder_2_topflange, SNEG
*SURFACE, name=top_surface_girder_3, type=element
girder_3_topflange, SNEG
*SURFACE, name=top_surface_girder_4, type=element
girder_4_topflange, SNEG
*SURFACE, name=top_surface_girder_5, type=element
girder_5_topflange, SNEG
**
*SURFACE, name=bottom_surface_deck, type=element
Deck_Elements_1, SNEG
**
*****************DEFINE NODES FOR BOUNDARY CONDITIONS******************
**
*Nset, nset=BC_Left, instance=Girder-1
16, 328, 327, 326, 9, 157, 158, 159, 10, 491, 1841, 1840, 1839, 156, 771, 770
769, 160, 490, 1838, 1837, 1836, 155, 768, 767, 766, 161
*Nset, nset=BC_Right, instance=Girder-1
89, 667, 668, 669, 85, 1941, 1942, 1943, 525, 8, 88, 87, 86, 7, 346, 347
348, 18, 90, 664, 665, 666, 84, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-2
16, 328, 327, 326, 9, 157, 158, 159, 10, 491, 1841, 1840, 1839, 156, 771, 770
769, 160, 490, 1838, 1837, 1836, 155, 768, 767, 766, 161
*Nset, nset=BC_Right, instance=Girder-2
89, 667, 668, 669, 85, 1941, 1942, 1943, 525, 8, 88, 87, 86, 7, 346, 347
348, 18, 90, 664, 665, 666, 84, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-3
16, 328, 327, 326, 9, 157, 158, 159, 10, 491, 1841, 1840, 1839, 156, 771, 770
769, 160, 490, 1838, 1837, 1836, 155, 768, 767, 766, 161
*Nset, nset=BC_Right, instance=Girder-3
89, 667, 668, 669, 85, 1941, 1942, 1943, 525, 8, 88, 87, 86, 7, 346, 347
348, 18, 90, 664, 665, 666, 84, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-4
16, 328, 327, 326, 9, 157, 158, 159, 10, 491, 1841, 1840, 1839, 156, 771, 770
769, 160, 490, 1838, 1837, 1836, 155, 768, 767, 766, 161
*Nset, nset=BC_Right, instance=Girder-4
89, 667, 668, 669, 85, 1941, 1942, 1943, 525, 8, 88, 87, 86, 7, 346, 347
348, 18, 90, 664, 665, 666, 84, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-5
16, 328, 327, 326, 9, 157, 158, 159, 10, 491, 1841, 1840, 1839, 156, 771, 770
769, 160, 490, 1838, 1837, 1836, 155, 768, 767, 766, 161
*Nset, nset=BC_Right, instance=Girder-5
89, 667, 668, 669, 85, 1941, 1942, 1943, 525, 8, 88, 87, 86, 7, 346, 347
348, 18, 90, 664, 665, 666, 84, 1938, 1939, 1940, 524
**
*****************DEFINE ELEMENT SETS FOR SELF WEIGHT LOADS*************
**
*Elset, elset=Girder_1_Elements, instance=Girder-1
Girder_Flange_Elements, Girder_Web_Elements
*Elset, elset=Girder_2_Elements, instance=Girder-2
Girder_Flange_Elements, Girder_Web_Elements
*Elset, elset=Girder_3_Elements, instance=Girder-3
Girder_Flange_Elements, Girder_Web_Elements
*Elset, elset=Girder_4_Elements, instance=Girder-4
Girder_Flange_Elements, Girder_Web_Elements
*Elset, elset=Girder_5_Elements, instance=Girder-5
Girder_Flange_Elements, Girder_Web_Elements
**
```

```
*******************DEFINE SURFACE FOR M1600 WHEEL LOAD**************
**
*Elset, elset=EL_Moving_Traffic, instance=Deck-1
236, 237, 245, 246, 446, 447, 455, 456, 656, 657, 665, 666, 1244, 1245, 1253,
1254
1454, 1455, 1463, 1464, 1664, 1665, 1673, 1674, 2714, 2715, 2723, 2724, 2924,
2925, 2933, 2934
**
*SURFACE, name=M1600, type=element
EL_Moving_Traffic, SPOS
**
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
25, 2923, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
26, 2924, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
27, 2925, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
28, 2926, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
29, 2927, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
30, 2928, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
31, 2929, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
32, 2930, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
33, 2931, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
34, 2932, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
35, 2933, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
36, 2934, 42
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
37, 2935, 42
**
*SURFACE, name=M1600_Dist, type=element
EL_Moving_Traffic_Dist, SPOS
**
*****************DEFINE DIAPHRAGMS FOR OUTPUT DATA FILE**************
**
*Elset, elset=diaphragms, instance=Diaphragm-1, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-2, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-3, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-4, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-5, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-6, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-7, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-8, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-9, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-10, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-11, generate
1, 7, 1
*Elset, elset=diaphragms, instance=Diaphragm-12, generate
1, 7, 1
**
**
*TIE, NAME = TIE_1, TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_1, bottom_surface_deck
```

```
*TIE, NAME = TIE_2, TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_2, bottom_surface_deck
*TIE, NAME = TIE_3, TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_3, bottom_surface_deck
*TIE, NAME = TIE_4, TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_4, bottom_surface_deck
*TIE, NAME = TIE_5, TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_5, bottom_surface_deck
**
*Nset, nset=displacement_nodes, instance=Deck-1
1453
**, 1527, 1507, 1542
*End Assembly
**
*********************MATERIALS*****************
**
*Material, name=Concrete
*Density
 2400
*Elastic
 2.86e+10, 0.15
*Concrete
2.068e+07,      0.
3.792e+07, 0.0015
*Tension Stiffening
1.,     0.
0., 0.001
**
*Material, name=Steel_280
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 2.8e+8
**
*Material, name=Steel_300
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 3e+8
**
*Material, name=Steel_500
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 5e+8
********************INTERACTION PROPERTIES*************************
**
***Surface Interaction, name=GirderDeckInteraction
**1.,
***Surface Behavior, pressure-overclosure=HARD
**
***Contact Pair, interaction=GirderDeckInteraction, adjust=0.0001
,tied,TYPE=SURFACE TO SURFACE
**top_surface_girder_1, bottom_surface_deck
***Contact Pair, interaction=GirderDeckInteraction,  adjust=0.0001
,tied,TYPE=SURFACE TO SURFACE
**top_surface_girder_2, bottom_surface_deck
***Contact Pair, interaction=GirderDeckInteraction,  adjust=0.0001
,tied,TYPE=SURFACE TO SURFACE
**top_surface_girder_3, bottom_surface_deck
***Contact Pair, interaction=GirderDeckInteraction,  adjust=0.0001
,tied,TYPE=SURFACE TO SURFACE
**top_surface_girder_4, bottom_surface_deck
***Contact Pair, interaction=GirderDeckInteraction,  adjust=0.0001
,tied,TYPE=SURFACE TO SURFACE
```

```
**top_surface_girder_5, bottom_surface_deck
**
*********************BOUNDARY CONDITIONS***************************
**
** Name: Pinned Type: Displacement/Rotation
*Boundary
BC_Left, 1, 3
**BC_Left, pinned
**BC_Left, pinned
BC_Right, 2, 3
**BC_Right, 3, 3
** Name: PinnedSliding Type: Displacement/Rotation
**
*******************APPLY STEPS*****************************
**
** STEP: Contact
**
*Step, name=Contact
**Establish Contact
*Static
1., 1., 1e-05, 1.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Load
**
*Step, name=Load
**Apply Loads
*Static
1., 1., 1e-05, 1.
**
** LOADS
**
** Name: Dead_Load_Girder_1   Type: Gravity
*Dload
Girder_1_Elements, GRAV, 9.81, 0, 0 , -1
** Name: Dead_Load_Girder_2   Type: Gravity
*Dload
Girder_2_Elements, GRAV, 9.81, 0, 0, -1
** Name: Dead_Load_Girder_3   Type: Gravity
*Dload
Girder_3_Elements, GRAV, 9.81, 0, 0, -1
** Name: Dead_Load_Girder_4   Type: Gravity
*Dload
Girder_4_Elements, GRAV, 9.81, 0, 0, -1
** Name: Dead_Load_Girder_5   Type: Gravity
*Dload
Girder_5_Elements, GRAV, 9.81, 0, 0, -1
**
** Name: Dead_Load_Deck          Type: Gravity
*Dload
Deck_Elements_1, GRAV, 9.81, 0, 0, -1
**
*DSLOAD
M1600, P, 1244082
*DSLOAD
M1600_Dist, P, 4725
** OUTPUT REQUESTS
**
```

```
***Restart, write, frequency=0
**
** OUPUT TO THE DATA FILE
**
*EL Print, elset=diaphragms
S
*El Print, elset=Deck_Elements_1
crack
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
*El Print, REBAR
S, RBFOR
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Node output, nset=displacement_nodes
U3
*End Step
```

B.2 – Four Girder Model

```
*Heading
 4 Girder Bridge, S1600 Loading in Both Lanes
** Job name: Four_Girder Model name: Model-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Deck
*INCLUDE,INPUT=Deck_Mesh.inp
**
**************Section Properties Deck****************
**
** Section: Concrete_Deck
*Shell Section, elset=Deck_Elements, material=Concrete
0.25, 9
*Rebar Layer
xx_bottom, 7.854e-5, 0.1, -0.08, Steel_500, 0., 1
yy_bottom, 7.854e-5, 0.1, -0.08, Steel_500, 90., 1
xx_top, 7.854e-5, 0.1, 0.08, Steel_500, 0., 1
yy_top, 7.854e-5, 0.1, 0.08, Steel_500, 90., 1
**
*End Part
**
*Part, name=Diaphragm
*INCLUDE, INPUT=Diaphragm_Mesh.inp
** Section: Truss_Section
**Solid Section, elset=Diaphragm_Elements, material=Steel_300
**0.0096,
**
*BEAM SECTION, elset=Diaphragm_Elements, section=rect, material=Steel_280
.032, .3
0, 0, 1

*End Part
**
*Part, name=Girder
*INCLUDE, INPUT=Girder_Mesh.inp
**
*****************Section Properties Girder************************
**
** Section: Web_Stiffeners_Section
*Shell Section, elset=Web_Stiffener_Elements, material=Steel_300
0.032, 5
** Section: Flange_Section
*Shell Section, elset=Flange_Elements, material=Steel_280
0.032, 5
** Section: Web_Section
*Shell Section, elset=Web_Elements, material=Steel_300
0.016, 5
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Deck-1, part=Deck
*End Instance
**
*Instance, name=Girder-1, part=Girder
        -9.,   4.722804878,       -0.496
        -9.,   4.722804878,       -0.496, -8.42264973081037, 5.30015514718963,
0.0813502691896258,        120.
*End Instance
**
*Instance, name=Girder-2, part=Girder
        -9.,   1.574268293,       -0.496
        -9.,   1.574268293,       -0.496, -8.42264973081037, 2.15161856218963,
0.0813502691896258,        120.
*End Instance
**
```

```
*Instance, name=Girder-3, part=Girder
        -9., -1.574268293,        -0.496
        -9., -1.574268293,        -0.496, -8.42264973081037, -
0.996918023810374, 0.0813502691896258,        120.
*End Instance
**
*Instance, name=Girder-4, part=Girder
        -9., -4.722804878,        -0.496
        -9., -4.722804878,        -0.496, -8.42264973081037, -
4.14545460881037, 0.0813502691896258,        120.
*End Instance
**
*Instance, name=Diaphragm-1, part=Diaphragm
         9., -3.522804878, -0.743999999999999
         9., -3.522804878, -0.743999999999999,         9., -3.522804878,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-2, part=Diaphragm
         0., -3.522804878, -0.743999999999999
         0., -3.522804878, -0.743999999999999,         0., -3.522804878,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-3, part=Diaphragm
        -9., -3.522804878,        -0.744
        -9., -3.522804878,        -0.744,         -9., -3.522804878,
0.256, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-4, part=Diaphragm
         9., -0.374268293000001, -0.743999999999999
         9., -0.374268293000001, -0.743999999999999,         9., -
0.374268293000001, 0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-5, part=Diaphragm
         0., -0.374268293000001, -0.743999999999999
         0., -0.374268293000001, -0.743999999999999,         0., -
0.374268293000001, 0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-6, part=Diaphragm
        -9., -0.374268293,        -0.744
        -9., -0.374268293,        -0.744,         -9., -0.374268293,
0.256, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-7, part=Diaphragm
         9.,  2.774268293, -0.743999999999999
         9.,  2.774268293, -0.743999999999999,         9.,  2.774268293,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-8, part=Diaphragm
         0.,  2.774268293, -0.743999999999999
         0.,  2.774268293, -0.743999999999999,         0.,  2.774268293,
0.256000000000001, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-9, part=Diaphragm
        -9.,  2.774268293,        -0.744
        -9.,  2.774268293,        -0.744,         -9.,  2.774268293,
0.256, 89.9999990194245
*End Instance
**
**APPLY MULITPOINT CONSTRAINT BETWEEN TRUSSES AND GIRDERS
**
*****************Define nodes to be used in constraints**************
**
**
```

```
***Girder 1 left
*Nset, nset=girder_1l1, instance=Girder-1
355
*Nset, nset=girder_1l2, instance=Girder-1
43
*Nset, nset=girder_1l3, instance=Girder-1
378
***girder 2 right
*Nset, nset=girder_2r1, instance=Girder-2
361
*Nset, nset=girder_2r2, instance=Girder-2
34
*Nset, nset=girder_2r3, instance=Girder-2
384
***girder 2 left
*Nset, nset=girder_2l1, instance=Girder-2
355
*Nset, nset=girder_2l2, instance=Girder-2
43
*Nset, nset=girder_2l3, instance=Girder-2
378
***girder 3 right
*Nset, nset=girder_3r1, instance=Girder-3
361
*Nset, nset=girder_3r2, instance=Girder-3
34
*Nset, nset=girder_3r3, instance=Girder-3
384
***girder 3 left
*Nset, nset=girder_3l1, instance=Girder-3
355
*Nset, nset=girder_3l2, instance=Girder-3
43
*Nset, nset=girder_3l3, instance=Girder-3
378
***girder 4 right
*Nset, nset=girder_4r1, instance=Girder-4
361
*Nset, nset=girder_4r2, instance=Girder-4
34
*Nset, nset=girder_4r3, instance=Girder-4
384
**
*Nset, nset=diaphragm_1_left, instance=Diaphragm-1
1
*Nset, nset=diaphragm_1_right, instance=Diaphragm-1
11
*Nset, nset=diaphragm_2_left, instance=Diaphragm-2
1
*Nset, nset=diaphragm_2_right, instance=Diaphragm-2
11
*Nset, nset=diaphragm_3_left, instance=Diaphragm-3
1
*Nset, nset=diaphragm_3_right, instance=Diaphragm-3
11
*Nset, nset=diaphragm_4_left, instance=Diaphragm-4
1
*Nset, nset=diaphragm_4_right, instance=Diaphragm-4
11
*Nset, nset=diaphragm_5_left, instance=Diaphragm-5
1
*Nset, nset=diaphragm_5_right, instance=Diaphragm-5
11
*Nset, nset=diaphragm_6_left, instance=Diaphragm-6
1
*Nset, nset=diaphragm_6_right, instance=Diaphragm-6
11
*Nset, nset=diaphragm_7_left, instance=Diaphragm-7
1
*Nset, nset=diaphragm_7_right, instance=Diaphragm-7
11
```

```
*Nset, nset=diaphragm_8_left, instance=Diaphragm-8
1
*Nset, nset=diaphragm_8_right, instance=Diaphragm-8
11
*Nset, nset=diaphragm_9_left, instance=Diaphragm-9
1
*Nset, nset=diaphragm_9_right, instance=Diaphragm-9
11
**
**
**************************Define constraint equations*****************
**
***girder1-2
*MPC
TIE,diaphragm_7_left, girder_2r1
*MPC
TIE, diaphragm_7_right, girder_1l1
*MPC
TIE, diaphragm_8_left, girder_2r2
*MPC
TIE, diaphragm_8_right, girder_1l2
*MPC
TIE, diaphragm_9_left, girder_2r3
*MPC
TIE, diaphragm_9_right, girder_1l3
**
***girder2-3
*MPC
TIE,diaphragm_4_left, girder_3r1
*MPC
TIE, diaphragm_4_right, girder_2l1
*MPC
TIE, diaphragm_5_left, girder_3r2
*MPC
TIE, diaphragm_5_right, girder_2l2
*MPC
TIE, diaphragm_6_left, girder_3r3
*MPC
TIE, diaphragm_6_right, girder_2l3
**
***girder3-4
*MPC
TIE,diaphragm_1_left, girder_4r1
*MPC
TIE, diaphragm_1_right, girder_3l1
*MPC
TIE, diaphragm_2_left, girder_4r2
*MPC
TIE, diaphragm_2_right, girder_3l2
*MPC
TIE, diaphragm_3_left, girder_4r3
*MPC
TIE, diaphragm_3_right, girder_3l3
**
*************************DEFINE SURFACES FOR CONTACT*********************
**
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
625, 904, 1
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
1033, 1312, 1
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
625, 904, 1
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
1033, 1312, 1
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
625, 904, 1
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
1033, 1312, 1
*Elset, elset=girder_4_topflange, instance=Girder-4, generate
625, 904, 1
*Elset, elset=girder_4_topflange, instance=Girder-4, generate
```

```
1033, 1312, 1
**
*Elset, elset=Deck_Elements_1, instance=Deck-1
Deck_Elements
**
*SURFACE, name=top_surface_girder_1, type=element
girder_1_topflange, SNEG
*SURFACE, name=top_surface_girder_2, type=element
girder_2_topflange, SNEG
*SURFACE, name=top_surface_girder_3, type=element
girder_3_topflange, SNEG
*SURFACE, name=top_surface_girder_4, type=element
girder_4_topflange, SNEG
**
*SURFACE, name=bottom_surface_deck, type=element
Deck_Elements_1, SNEG
**
**
******************DEFINE NODES FOR BOUNDARY CONDITIONS******************
**
*Nset, nset=BC_Left, instance=Girder-1
15, 359, 358, 357, 7, 86, 87, 88, 8, 491, 1841, 1840, 1839, 85, 669, 668
667, 89, 490, 1838, 1837, 1836, 84, 666, 665, 664, 90
*Nset, nset=BC_Right, instance=Girder-1
160, 769, 770, 771, 156, 1941, 1942, 1943, 525, 10, 159, 158, 157, 9, 382, 381
380, 18, 161, 766, 767, 768, 155, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-2
15, 359, 358, 357, 7, 86, 87, 88, 8, 491, 1841, 1840, 1839, 85, 669, 668
667, 89, 490, 1838, 1837, 1836, 84, 666, 665, 664, 90
*Nset, nset=BC_Right, instance=Girder-2
160, 769, 770, 771, 156, 1941, 1942, 1943, 525, 10, 159, 158, 157, 9, 382, 381
380, 18, 161, 766, 767, 768, 155, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-3
15, 359, 358, 357, 7, 86, 87, 88, 8, 491, 1841, 1840, 1839, 85, 669, 668
667, 89, 490, 1838, 1837, 1836, 84, 666, 665, 664, 90
*Nset, nset=BC_Right, instance=Girder-3
160, 769, 770, 771, 156, 1941, 1942, 1943, 525, 10, 159, 158, 157, 9, 382, 381
380, 18, 161, 766, 767, 768, 155, 1938, 1939, 1940, 524
*Nset, nset=BC_Left, instance=Girder-4
15, 359, 358, 357, 7, 86, 87, 88, 8, 491, 1841, 1840, 1839, 85, 669, 668
667, 89, 490, 1838, 1837, 1836, 84, 666, 665, 664, 90
*Nset, nset=BC_Right, instance=Girder-4
160, 769, 770, 771, 156, 1941, 1942, 1943, 525, 10, 159, 158, 157, 9, 382, 381
380, 18, 161, 766, 767, 768, 155, 1938, 1939, 1940, 524
**
******************DEFINE ELEMENT SETS FOR SELF WEIGHT LOADS*************
**
*Elset, elset=Girder_1_Elements, instance=Girder-1
Girder_Elements
*Elset, elset=Girder_2_Elements, instance=Girder-2
Girder_Elements
*Elset, elset=Girder_3_Elements, instance=Girder-3
Girder_Elements
*Elset, elset=Girder_4_Elements, instance=Girder-4
Girder_Elements
**
******************DEFINE SURFACE FOR S1600 WHEEL LOAD**************
**
*Elset, elset=EL_Moving_Traffic, instance=Deck-1
2848, 2847, 2839, 2838, 2643, 2642, 2634, 2633, 2438, 2437, 2429, 2428, 1864,
1863, 1855, 1854
1659, 1658, 1650, 1649, 1454, 1453, 1445, 1444, 429, 428, 420, 419, 224, 223,
215, 214
19, 18, 10, 9
**
*SURFACE, name=S1600, type=element
EL_Moving_Traffic, SPOS
**
************DEFINE SURFACE FOR DISTRIBUTED S1600 WHEEL LOAD*********
**
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
```

```
2849, 20, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2848, 19, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2847, 18, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2846, 17, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2845, 16, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2844, 15, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2843, 14, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2842, 13, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2841, 12, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2840, 11, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2839, 10, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2838, 9, -41
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
2837, 8, -41
**
*SURFACE, name=S1600_Dist, type=element
EL_Moving_Traffic_Dist, SPOS
**
***********LOADING ON SECOND LANE***************88
**
*Elset, elset=EL_Moving_Traffic_2, instance=Deck-1
2852, 2853, 2861, 2862, 2657, 2656, 2648, 2647, 2452, 2451, 2443, 2442, 1878,
1877, 1869, 1868
1673, 1672, 1664, 1663, 1468, 1467, 1459, 1458, 443, 442, 434, 433, 238, 237,
229, 228
33, 32, 24, 23
**
*SURFACE, name=S1600_2, type=element
EL_Moving_Traffic_2, SPOS
**
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
22, 2851, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
23, 2852, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
24, 2853, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
25, 2854, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
26, 2855, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
27, 2856, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
28, 2857, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
29, 2858, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
30, 2859, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
31, 2860, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
32, 2861, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
33, 2862, 41
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
34, 2863, 41
**
*SURFACE, name=S1600_Dist_2, type=element
EL_Moving_Traffic_Dist_2, SPOS
**
```

```
******************DEFINE SURFACE FOR DEAD LOAD FROM BARRIERS***********
**
*Elset, elset=Barrier_Load, instance=Deck-1, generate
40, 2869, 41
*Elset, elset=Barrier_Load, instance=Deck-1, generate
2, 2831, 41
**
*SURFACE, name=Barrier, type=element
Barrier_Load, SPOS
**
******************DEFINE DIAPHRAGMS FOR OUTPUT DATA FILE***************
**
*Elset, elset=diaphragms, instance=Diaphragm-1, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-2, generate
1,11, 1
*Elset, elset=diaphragms, instance=Diaphragm-3, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-4, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-5, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-6, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-7, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-8, generate
1, 11, 1
*Elset, elset=diaphragms, instance=Diaphragm-9, generate
1, 11, 1
**
*TIE, NAME = TIE_1,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_1, bottom_surface_deck
*TIE, NAME = TIE_2,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_2, bottom_surface_deck
*TIE, NAME = TIE_3,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_3, bottom_surface_deck
*TIE, NAME = TIE_4,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_4, bottom_surface_deck
**
*Nset, nset=displacement_node, instance=Deck-1
1532
**
*End Assembly
**
**********************MATERIALS*****************
**
*Material, name=Concrete
*Density
 2400
*Elastic
 2.86e+10, 0.15
*Concrete
2.068e+07,      0.
3.792e+07, 0.0015
*Tension Stiffening
1.,     0.
0., 0.01
**
*Material, name=Steel_280
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 2.8e+8
**
*Material, name=Steel_300
*Density
 7850
*Elastic
```

```
 2e+11, 0.3
*PLASTIC
 3e+8
**
*Material, name=Steel_500
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 5e+8
*********************BOUNDARY CONDITIONS****************************
**
** Name: Pinned Type: Displacement/Rotation
*Boundary
BC_Left, 1, 3
BC_Right, 2, 3
******************APPLY STEPS****************************
**
** STEP: Contact
**
*Step, name=Contact
**Establish Contact
*Static
1., 1., 1e-05, 1.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Load
**
*Step, name=Load
**Apply Loads
*Static
1., 1., 1e-05, 1.
**
** LOADS
**
** Name: Dead_Load_Girder_1   Type: Gravity
*Dload
Girder_1_Elements, GRAV, 9.81, 0, 0 , -1
** Name: Dead_Load_Girder_2   Type: Gravity
*Dload
Girder_2_Elements, GRAV, 9.81, 0, 0, -1
** Name: Dead_Load_Girder_3   Type: Gravity
*Dload
Girder_3_Elements, GRAV, 9.81, 0, 0, -1
** Name: Dead_Load_Girder_4   Type: Gravity
*Dload
Girder_4_Elements, GRAV, 9.81, 0, 0, -1
**
** Name: Dead_Load_Deck         Type: Gravity
**
*Dload
Deck_Elements_1, GRAV, 9.81, 0, 0, -1
**
*****LOAD FIRST LANE****
**
*DSLOAD
S1600, P, 829389
```

```
*DSLOAD
S1600_Dist, P, 18900
**
*****LOAD SECOND LANE****
**
*DSLOAD
S1600_2, P, 663511
*DSLOAD
S1600_Dist_2, P, 15120
**
*****BARRIER LOADS*****
**
*DSLOAD
Barrier, P, 31440
**
** OUTPUT REQUESTS
**
***Restart, write, frequency=0
**
** OUPUT TO THE DATA FILE
**
*EL Print, elset=diaphragms
S
*El Print, elset=Deck_Elements_1
crack
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
*El Print, REBAR
S, RBFOR
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Node output, nset=displacement_node
U3
*End Step
```

B.3 – Three Girder Model

```
*Heading
 3 Girder Bridge, M1600 Loading in Both Lanes
** Job name: 3Girder Model name: Model-1
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Deck
*INCLUDE,INPUT=Deck_Mesh.inp
**
**************Section Properties Deck****************
**
** Section: Concrete_Deck
*Shell Section, elset=Deck_Elements, material=Concrete
0.25, 9
*Rebar Layer
xx_bottom, 7.854e-5, 0.1, -0.08, Steel_500, 0., 1
yy_bottom, 7.854e-5, 0.1, -0.08, Steel_500, 90., 1
xx_top, 7.854e-5, 0.1, 0.08, Steel_500, 0., 1
yy_top, 7.854e-5, 0.1, 0.08, Steel_500, 90., 1
**
*End Part
**
*Part, name=Diaphragm
*INCLUDE, INPUT=Diaphragm_Mesh.inp
**
*BEAM SECTION, elset=Diaphragm_Elements, section=rect, material=Steel_280
.032, .3
0, 0, 1

*End Part
**
*Part, name=Girder
*INCLUDE, INPUT=Girder_Mesh.inp
**
*****************Section Properties Girder***********************
**
** Section: Web_Stiffeners_Section
*Shell Section, elset=Web_Stiffeners_Elements, material=Steel_300
0.032, 5
** Section: Flange_Section
*Shell Section, elset=Flange_Elements, material=Steel_280
0.032, 5
** Section: Web_Section
*Shell Section, elset=Web_Elements, material=Steel_300
0.016, 5
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Deck-1, part=Deck
*End Instance
**
*Instance, name=Girder-1, part=Girder
         9.,   4.728571429, -0.496000000000001
         9.,   4.728571429, -0.496000000000001, 9.57735026918963,
4.15122115981038, -1.07335026918963,         120.
*End Instance
**
*Instance, name=Girder-2, part=Girder
         9.,   1.77635683940025e-15, -0.496000000000001
         9.,   1.77635683940025e-15, -0.496000000000001, 9.57735026918963, -
0.577350269189624, -1.07335026918963,         120.
*End Instance
**
*Instance, name=Girder-3, part=Girder
         9., -4.728571429, -0.496000000000001
```

```
          9., -4.728571429, -0.496000000000001, 9.57735026918963, -
5.30592169818962, -1.07335026918963,         120.
*End Instance
**
*Instance, name=Diaphragm-1, part=Diaphragm
          9.,           1.2, -0.744000000000001
          9.,           1.2, -0.744000000000001,           9.,           1.2,
0.255999999999999, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-2, part=Diaphragm
          0.,           1.2, -0.744000000000001
          0.,           1.2, -0.744000000000001,           0.,           1.2,
0.255999999999999, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-3, part=Diaphragm
         -9.,           1.2,       -0.744
         -9.,           1.2,       -0.744,          -9.,           1.2,
0.256, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-4, part=Diaphragm
          9., -3.528571429, -0.744000000000001
          9., -3.528571429, -0.744000000000001,           9., -3.528571429,
0.255999999999999, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-5, part=Diaphragm
          0., -3.528571429, -0.744000000000001
          0., -3.528571429, -0.744000000000001,           0., -3.528571429,
0.255999999999999, 89.9999990194245
*End Instance
**
*Instance, name=Diaphragm-6, part=Diaphragm
         -9., -3.528571429,       -0.744
         -9., -3.528571429,       -0.744,          -9., -3.528571429,
0.256, 89.9999990194245
*End Instance
**
**
**APPLY MULITPOINT CONSTRAINT BETWEEN TRUSSES AND GIRDERS
**
****************Define nodes to be used in constraints*************
**
***Girder 1 left
*Nset, nset=girder_1l1, instance=Girder-1
374
*Nset, nset=girder_1l2, instance=Girder-1
30
*Nset, nset=girder_1l3, instance=Girder-1
347
***girder 2 right
*Nset, nset=girder_2r1, instance=Girder-2
388
*Nset, nset=girder_2r2, instance=Girder-2
47
*Nset, nset=girder_2r3, instance=Girder-2
361
***girder 2 left
*Nset, nset=girder_2l1, instance=Girder-2
374
*Nset, nset=girder_2l2, instance=Girder-2
30
*Nset, nset=girder_2l3, instance=Girder-2
347
***girder 3 right
*Nset, nset=girder_3r1, instance=Girder-3
388
*Nset, nset=girder_3r2, instance=Girder-3
47
```

```
*Nset, nset=girder_3r3, instance=Girder-3
361
**
*Nset, nset=diaphragm_1_left, instance=Diaphragm-1
1
*Nset, nset=diaphragm_1_right, instance=Diaphragm-1
19
*Nset, nset=diaphragm_2_left, instance=Diaphragm-2
1
*Nset, nset=diaphragm_2_right, instance=Diaphragm-2
19
*Nset, nset=diaphragm_3_left, instance=Diaphragm-3
1
*Nset, nset=diaphragm_3_right, instance=Diaphragm-3
19
*Nset, nset=diaphragm_4_left, instance=Diaphragm-4
1
*Nset, nset=diaphragm_4_right, instance=Diaphragm-4
19
*Nset, nset=diaphragm_5_left, instance=Diaphragm-5
1
*Nset, nset=diaphragm_5_right, instance=Diaphragm-5
19
*Nset, nset=diaphragm_6_left, instance=Diaphragm-6
1
*Nset, nset=diaphragm_6_right, instance=Diaphragm-6
19
**
*************************Define constraint equations*****************
**
***girder1-2
*MPC
TIE, diaphragm_1_left, girder_2r1
*MPC
TIE, diaphragm_1_right, girder_1l1
*MPC
TIE, diaphragm_2_left, girder_2r2
*MPC
TIE, diaphragm_2_right, girder_1l2
*MPC
TIE, diaphragm_3_left, girder_2r3
*MPC
TIE, diaphragm_3_right, girder_1l3
**
***girder2-3
*MPC
TIE, diaphragm_4_left, girder_3r1
*MPC
TIE, diaphragm_4_right, girder_2l1
*MPC
TIE, diaphragm_5_left, girder_3r2
*MPC
TIE, diaphragm_5_right, girder_2l2
*MPC
TIE, diaphragm_6_left, girder_3r3
*MPC
TIE, diaphragm_6_right, girder_2l3
**
**************************DEFINE SURFACES FOR CONTACT*********************
**
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
65, 344, 1
*Elset, elset=girder_1_topflange, instance=Girder-1, generate
1593, 1872, 1
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
65, 344, 1
*Elset, elset=girder_2_topflange, instance=Girder-2, generate
1593, 1872, 1
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
65, 344, 1
*Elset, elset=girder_3_topflange, instance=Girder-3, generate
```

```
1593, 1872, 1
**
*Elset, elset=Deck_Elements_1, instance=Deck-1
Deck_Elements
**
*SURFACE, name=top_surface_girder_1, type=element
girder_1_topflange, SNEG
*SURFACE, name=top_surface_girder_2, type=element
girder_2_topflange, SNEG
*SURFACE, name=top_surface_girder_3, type=element
girder_3_topflange, SNEG
**
*SURFACE, name=bottom_surface_deck, type=element
Deck_Elements_1, SNEG
**
******************DEFINE NODES FOR BOUNDARY CONDITIONS******************
**
*Nset, nset=BC_Left, instance=Girder-1
14, 340, 341, 342, 11, 228, 229, 230, 12, 339, 1213, 1212, 1211, 227, 873, 872
871, 231, 338, 1210, 1209, 1208, 226, 870, 869, 868, 232, 337, 1207, 1206,
1205, 225
867, 866, 865, 233
*Nset, nset=BC_Right, instance=Girder-1
17, 363, 364, 365, 13, 343, 344, 345, 15, 390, 1397, 1398, 1399, 298, 1499,
1500
1501, 457, 391, 1394, 1395, 1396, 297, 1496, 1497, 1498, 456, 392, 1391, 1392,
1393, 296
1493, 1494, 1495, 455
*Nset, nset=BC_Left, instance=Girder-2
14, 340, 341, 342, 11, 228, 229, 230, 12, 339, 1213, 1212, 1211, 227, 873, 872
871, 231, 338, 1210, 1209, 1208, 226, 870, 869, 868, 232, 337, 1207, 1206,
1205, 225
867, 866, 865, 233
*Nset, nset=BC_Right, instance=Girder-2
17, 363, 364, 365, 13, 343, 344, 345, 15, 390, 1397, 1398, 1399, 298, 1499,
1500
1501, 457, 391, 1394, 1395, 1396, 297, 1496, 1497, 1498, 456, 392, 1391, 1392,
1393, 296
1493, 1494, 1495, 455
*Nset, nset=BC_Left, instance=Girder-3
14, 340, 341, 342, 11, 228, 229, 230, 12, 339, 1213, 1212, 1211, 227, 873, 872
871, 231, 338, 1210, 1209, 1208, 226, 870, 869, 868, 232, 337, 1207, 1206,
1205, 225
867, 866, 865, 233
*Nset, nset=BC_Right, instance=Girder-3
17, 363, 364, 365, 13, 343, 344, 345, 15, 390, 1397, 1398, 1399, 298, 1499,
1500
1501, 457, 391, 1394, 1395, 1396, 297, 1496, 1497, 1498, 456, 392, 1391, 1392,
1393, 296
1493, 1494, 1495, 455
**
*****************DEFINE ELEMENT SETS FOR SELF WEIGHT LOADS*************
**
*Elset, elset=Girder_1_Elements, instance=Girder-1
Girder_Elements
*Elset, elset=Girder_2_Elements, instance=Girder-2
Girder_Elements
*Elset, elset=Girder_3_Elements, instance=Girder-3
Girder_Elements
**
*****************DEFINE SURFACE FOR M1600 WHEEL LOAD*************
**
*Elset, elset=EL_Moving_Traffic, instance=Deck-1
1750, 1820, 2380, 2450, 1745, 1815, 2375, 2445, 1740, 1810, 2370, 2440, 1726,
1796, 2356, 2426
1721, 1791, 2351, 2421, 1716, 1786, 2346, 2416, 1691, 1761, 2321, 2391, 1686,
1756, 2316, 2386
1681, 1751, 2311, 2381
**
*SURFACE, name=M1600, type=element
EL_Moving_Traffic, SPOS
```

```
**
*Elset, elset=EL_Moving_Traffic_2, instance=Deck-1
560, 630, 1190, 1260, 555, 625, 1185, 1255, 550, 620, 1180, 1250, 536, 606,
1166, 1236
531, 601, 1161, 1231, 526, 596, 1156, 1226, 501, 571, 1131, 1201, 496, 566,
1126, 1196
491, 561, 1121, 1191
**
*SURFACE, name=M1600_2, type=element
EL_Moving_Traffic_2, SPOS
**
*************DEFINE SURFACE FOR DISTRIBUTED M1600 WHEEL LOAD*********
**
*Elset, elset=EL_Moving_Traffic_Dist, instance=Deck-1, generate
1611, 2520, 1
**
*SURFACE, name=M1600_Dist, type=element
EL_Moving_Traffic_Dist, SPOS
**
*Elset, elset=EL_Moving_Traffic_Dist_2, instance=Deck-1, generate
421, 1330, 1
**
*SURFACE, name=M1600_Dist_2, type=element
EL_Moving_Traffic_Dist_2, SPOS
**
*****************DEFINE SURFACE FOR DEAD LOAD FROM BARRIERS***********
**
*Elset, elset=Barrier_Load, instance=Deck-1, generate
71, 140, 1
*Elset, elset=Barrier_Load, instance=Deck-1, generate
2801, 2870, 1
**
*SURFACE, name=Barrier, type=element
Barrier_Load, SPOS
**
*****************DEFINE DIAPHRAGMS FOR OUTPUT DATA FILE***************
**
*Elset, elset=diaphragms, instance=Diaphragm-1, generate
1, 19, 1
*Elset, elset=diaphragms, instance=Diaphragm-2, generate
1, 19, 1
*Elset, elset=diaphragms, instance=Diaphragm-3, generate
1, 19, 1
*Elset, elset=diaphragms, instance=Diaphragm-4, generate
1, 19, 1
*Elset, elset=diaphragms, instance=Diaphragm-5, generate
1, 19, 1
*Elset, elset=diaphragms, instance=Diaphragm-6, generate
1, 19, 1
**
*TIE, NAME = TIE_1,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_1, bottom_surface_deck
*TIE, NAME = TIE_2,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_2, bottom_surface_deck
*TIE, NAME = TIE_3,TYPE=SURFACE TO SURFACE, adjust=yes
top_surface_girder_3, bottom_surface_deck
**
*Nset, nset=displacement_node, instance=Deck-1
1246
**
*End Assembly
**
*********************MATERIALS*****************
**
*Material, name=Concrete
*Density
 2400
*Elastic
 2.86e+10, 0.15
*Concrete
2.068e+07,     0.
```

```
3.792e+07, 0.0015
*Tension Stiffening
1.,    0.
0., 0.01
**
*Material, name=Steel_280
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 2.8e+8
**
*Material, name=Steel_300
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 3e+8
**
*Material, name=Steel_500
*Density
 7850
*Elastic
 2e+11, 0.3
*PLASTIC
 5e+8
 ********************BOUNDARY CONDITIONS***************************
**
** Name: Pinned Type: Displacement/Rotation
*Boundary
BC_Left, 1, 3
BC_Right, 2, 3
*****************APPLY STEPS****************************
**
** STEP: Contact
**
*Step, name=Contact
**Establish Contact
*Static
1., 1., 1e-05, 1.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
*End Step
** ----------------------------------------------------------------
**
** STEP: Load
**
*Step, name=Load
**Apply Loads
*Static
1., 1., 1e-05, 1.
**
** LOADS
**
** Name: Dead_Load_Girder_1   Type: Gravity
*Dload
Girder_1_Elements, GRAV, 9.81, 0, 0 , -1
** Name: Dead_Load_Girder_2   Type: Gravity
*Dload
```

```
Girder_2_Elements, GRAV, 9.81, 0, 0, -1
** Name: Dead_Load_Girder_3   Type: Gravity
*Dload
Girder_3_Elements, GRAV, 9.81, 0, 0, -1
**
** Name: Dead_Load_Deck         Type: Gravity
**
*Dload
Deck_Elements_1, GRAV, 9.81, 0, 0, -1
**
*****LOAD FIRST LANE****
**
*DSLOAD
M1600, P, 1246455
*DSLOAD
M1600_Dist, P, 4725
**
*****LOAD SECOND LANE****
**
*DSLOAD
M1600_2, P, 1246455
*DSLOAD
M1600_Dist_2, P, 4725
**
*****BARRIER LOADS*****
**
*DSLOAD
Barrier, P, 31440
**
** OUTPUT REQUESTS
**
***Restart, write, frequency=0
**
** OUPUT TO THE DATA FILE
**
*EL Print, elset=diaphragms
S
*El Print, elset=Deck_Elements_1
crack
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
*El Print, REBAR
S, RBFOR
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history
*Node output, nset=displacement_node
U3
*End Step
```