Analysis and Design of Concrete Bridge Using STAAD. Pro

Vikash Kumar Dubey [1], Parmeshwar Sahu[2], Akhand Pratap Singh[3], Shiva Verma

[1]M. Tech Scholar,[2]Assistant Professor ,[3]Assistant Professor,[4] Assistant Professor
Department of Civil Engineering

[1,2,3,4]Shri Rawatpura Sarkar University, Raipur (C.G.), India

ABSTRACT: - The rapid growth of transportation infrastructure has increased the demand for efficient, safe, and economical bridge systems. Concrete bridges, due to their durability, strength, and cost-effectiveness, are widely adopted in modern civil engineering projects. This study focuses on the structural analysis and design of a reinforced concrete bridge using STAAD.Pro, a widely used structural analysis and design software. The primary objective is to evaluate the behaviour of the bridge under various loading conditions and ensure that the design complies with relevant standards and safety requirements.

The bridge model is developed in STAAD.Pro by defining its geometry, material properties, support conditions, and load parameters. The structure typically consists of components such as the deck slab, longitudinal girders, cross girders, piers, and abutments. Material properties for reinforced concrete are assigned based on standard design codes. The modeling process includes discretizing the structure into finite elements, allowing accurate simulation of real-world behaviour.

Loading plays a crucial role in bridge design, and this study incorporates different types of loads including dead load, live load, impact load, wind load, and seismic load where applicable. Dead loads include the self-weight of structural elements, while live loads are based on vehicular traffic as per standard loading codes. Load combinations are generated to analyse the most critical conditions the structure may experience during its service life.

The analysis is performed using STAAD.Pro's advanced finite element capabilities. The software computes internal forces such as bending moments, shear forces, and axial forces in various structural components. The results are evaluated to identify critical sections that require careful design. The structural response is assessed for stability, strength, and serviceability criteria.

Following the analysis, the design phase is carried out to ensure that all structural elements can safely resist the applied loads. Reinforcement detailing is done for beams, slabs, and columns based on calculated stresses and moments. The design adheres to relevant codes such as IRC or IS standards, ensuring compliance with national guidelines. Parameters such as deflection limits, crack width, and load-carrying capacity are checked to maintain serviceability and durability.

One of the key advantages of using STAAD.Pro is its ability to optimize the design. By iteratively modifying member sizes and reinforcement details, the most economical and efficient design can be achieved without compromising safety. The software also provides detailed reports and graphical outputs, which help engineers interpret results effectively and make informed decisions.

Keywords: Concrete Bridge, STAAD.Pro, Structural Analysis, Reinforced Concrete, Bridge Design, Load Analysis, Finite Element Method, IRC Codes, Structural Modeling, Civil Engineering Infrastructure.

---***---

1. Introduction

Bridges are vital components of transportation infrastructure, serving as critical links that facilitate the movement of people, goods, and services across physical obstacles such as rivers, valleys, railways, and roads. The design and construction of bridges have evolved significantly over time, driven by advancements in materials, structural engineering principles, and computational tools. Among various types of bridges, reinforced concrete bridges are widely used due to their durability, cost-effectiveness, and adaptability to different site conditions. The analysis and design of such structures require precision, adherence to safety standards, and the ability to predict structural behavior under various loading conditions.

In modern civil engineering practice, software tools have become indispensable for efficient and accurate structural analysis. One such powerful tool is STAAD.Pro, a widely used structural analysis and design software developed by Bentley Systems. STAAD.Pro enables engineers to model complex structures, apply various loads, analyze structural responses, and design structural elements in accordance with international design codes. Its versatility and user-friendly interface make it particularly suitable for bridge engineering applications.

The analysis and design of a concrete bridge involve multiple stages, including conceptual planning, structural modeling, load determination, analysis, design of components, and validation of results. Initially, the type of bridge is selected based on site conditions, span length, traffic requirements, and economic considerations. Reinforced concrete bridges, such as slab bridges, girder bridges, and box culverts, are commonly adopted for short to medium spans due to their simplicity and structural efficiency.

Once the bridge type is selected, a detailed structural model is created in STAAD.Pro. This model represents the geometry of the bridge, including spans, supports, deck slabs, girders, piers, and abutments. Accurate modeling is essential, as it directly influences the reliability of analysis results. The software allows the definition of material properties such as concrete grade, modulus of elasticity, Poisson’s ratio, and density, which are crucial for realistic simulation of structural behavior.

Load calculation is a fundamental aspect of bridge design. Bridges are subjected to various types of loads, including dead loads (self-weight of the structure), live loads (vehicular traffic), impact loads, wind loads, seismic loads, and temperature effects. In India, these loads are typically considered according to standards such as IRC (Indian Roads Congress) codes. STAAD.Pro provides the capability to define and apply these loads systematically, ensuring that all possible loading scenarios are accounted for during analysis.

After defining the structure and loads, the analysis phase is carried out. STAAD.Pro uses advanced numerical methods, primarily the finite element method (FEM), to compute internal forces such as bending moments, shear forces, and axial forces in structural members. The results obtained from the analysis help engineers understand how the bridge will behave under different load combinations. Critical sections of the bridge are identified based on maximum stress and deformation values.

Following the analysis, the design phase begins. In this stage, structural elements such as slabs, beams, girders, and columns are designed to safely resist the applied loads. Reinforcement details are calculated to ensure that the concrete structure can withstand tensile and compressive stresses. STAAD.Pro assists in this process by providing design modules that comply with relevant design codes, thereby reducing manual calculations and minimizing the risk of errors.

Objectives:

The main objectives of this project are as follows:

1. To understand the role and importance of bridge bearings in structural systems.
2. To study different types of bridge bearings and their applications.
3. To model a bridge structure using STAAD.Pro for analysis purposes.
4. To analyze the forces and reactions acting on bridge bearings under various load conditions.
5. To design suitable bridge bearings based on analysis results.
6. To ensure that the designed bearings satisfy safety, serviceability, and durability requirements.
7. To compare manual calculations with software-generated results (if applicable).
8. To gain practical knowledge of using STAAD.Pro in bridge engineering design.

Scope of the Study:

This project primarily focuses on the structural analysis and design of bridge bearings using STAAD.Pro software. The scope includes modeling a simplified bridge structure, applying relevant loads, analyzing the structural response, and designing bearings based on the obtained reactions.

The study is limited to standard loading conditions such as dead load, live load, wind load, and seismic load. Advanced factors such as fatigue analysis, long-term material degradation, and complex dynamic interactions may not be covered in detail.

2. Literature Review (National & International Studies)

1. Springer Nature (2022) presented a study on elastomeric bearings emphasizing their role in reducing vibrations from traffic, wind, and earthquakes, highlighting the importance of stability analysis.
2. Researchers developed **nonlinear analytical models** to study interaction between vertical and horizontal loads in bridge bearings, improving prediction accuracy of real behavior.
3. Studies on **lead rubber bearings (LRB)** showed enhanced energy dissipation capacity compared to conventional elastomeric bearings.
4. Experimental validation confirmed that **finite element models** can effectively simulate seismic isolator behavior.
5. Research by Elsevier (2019) investigated elastomeric bearings in steel girder bridges and found performance issues under high loads, emphasizing the need for proper design.
6. Field studies indicated that **large-sized elastomeric bearings** may not perform well under complex loading conditions without proper analysis.

7. Fatigue analysis studies highlighted that **bearing deterioration** is a critical factor affecting long-term bridge safety.
8. Advanced modeling using ABAQUS showed that **material modeling (Ogden model)** improves accuracy in predicting elastomer behavior.
9. Researchers emphasized the importance of **horizontal and vertical stiffness parameters** in bearing design.
10. Studies confirmed that **buckling and instability** are major failure modes in elastomeric bearings.
11. Research on **polyurethane elastomer bearings** showed improved shear strength and load capacity compared to conventional rubber bearings.
12. Investigations on **temperature effects in LRB** demonstrated reduction in energy dissipation capacity due to heat generation.
13. Finite element studies proved that **rotation affects vertical stiffness**, which must be considered in design.
14. Comparative studies of **bearing types (pot, spherical, elastomeric)** showed elastomeric bearings are economical but limited in heavy-load applications.
15. International guidelines stress the use of **advanced simulation tools** like STAAD.Pro, ABAQUS, and OpenSees for accurate analysis.
16. A review by Quantum University researchers analyzed RCC and steel bridges using STAAD.Pro and concluded that the software provides efficient load analysis and design optimization.
17. Studies in India emphasize the use of **STAAD.Pro for bridge modeling**, allowing accurate determination of reactions at supports (bearings).
18. Research shows that **STAAD.Pro improves code compliance** with Indian standards (IRC codes) and reduces manual errors.
19. Indian researchers have highlighted the importance of **load combinations (dead, live, seismic)** in bearing design using software tools.
20. Studies on elastomeric bearings using **ANSYS Workbench** demonstrated improved stress distribution through optimized design.
21. Research indicates that **optimized bearing geometry reduces stress concentration** and enhances durability.
22. Indian bridge projects commonly use **laminated elastomeric bearings** due to low maintenance and cost-effectiveness.
23. Several studies highlight the need for **seismic-resistant bearing design** in earthquake-prone regions of India.
24. Research suggests that **software-based modeling (STAAD.Pro)** is more reliable than traditional manual calculations for complex bridges.
25. Studies show that **bearing failures in Indian bridges** are often due to improper design, installation errors, or lack of maintenance.
26. Researchers recommend **periodic inspection and monitoring** of bridge bearings to ensure long-term performance.
27. Indian case studies demonstrate that **temperature variation significantly affects bearing movement and stress**.
28. Studies emphasize the importance of **proper selection of bearing type** based on span length and load conditions.
29. Research indicates that **integration of structural health monitoring systems** with software analysis is a future direction.

3. Methodology:

1 Structural analysis of concrete bridges

The purpose of a structural analysis is to evaluate the response of a structure. This can be of interest for many reasons, where the most common are design of new structures or assessment of existing ones. In design one aims towards providing a distribution of sectional forces, see Figure 2.1, while assessment aims towards accurately describing the response during loading and/or failure.

Four examples of methods for structural analysis are presented. These are:

- Linear elastic analysis
- Linear elastic analysis with limited redistribution
- Plastic analysis
- Non-linear analysis

Out of these methods it is only the non-linear which is capable of accurate prediction of the response during loading and describe the complex force redistribution taking place when cracking of concrete and yielding of reinforcement occurs. This means that it is only the non-linear analysis which accurately predicts the behavior of the structure in service state, and the mode of ultimate failure. However, the non-linear analysis requires substantial effort in establishment and post- processing of the model, as well as a large computational effort. It also requires the knowledge of the complete layout of the structure beforehand, making it a method suitable for accurate assessment of existing structures but not suitable for design purposes since this knowledge is not available at a design stage. Another major drawback for non-linear modelling in a design stage of reinforced concrete bridges is that non-linear analysis does not allow for load superposition. For bridge design applications with many different loads and load combinations it is essential from a practical point of view that load superposition is possible.

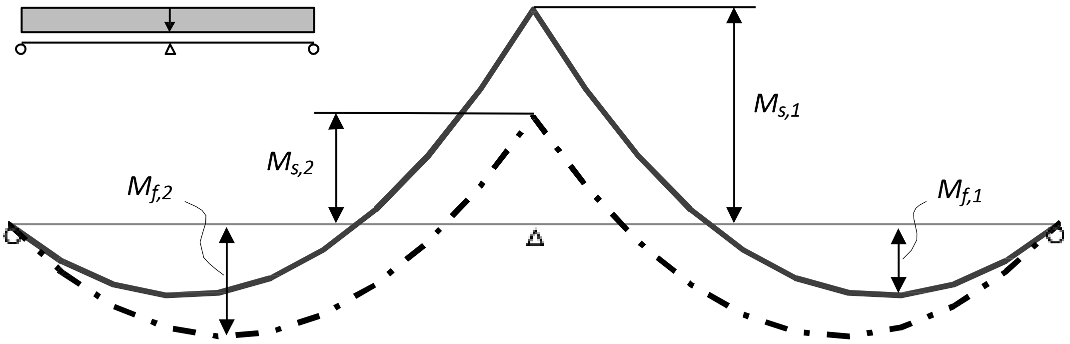


Figure 2.1. Possible moment distributions for design in ultimate limit state of a uniformly loaded continuous beam.

When performing a structural analysis at a design stage the aim is to provide a realistic and suitable distribution of sectional forces which fulfils equilibrium and can be used for design of the cross-sections in ULS. Since most bridge structures are statically indeterminate structures, there are many distributions (in fact an infinite number) that fulfils equilibrium, though of course not all are suitable for design about plastic redistribution and serviceability.

1.1 Linear elastic modelling

A linear elastic model assumes that the behavior of the structure is linearly dependent of the applied load. This is a simplification of the behavior of reinforced concrete. The linear elastic model will only describe the "true" distribution of forces under certain conditions, such as uncracked sections. Those conditions are generally never achieved since concrete structures often crack even for relatively low service loads. Concrete will crack and force redistribution will take place in the structure (providing that the structure is statically capable of redistribution). Consequently, the designer can choose a distribution in ultimate limit state. The structure will then, providing that it has sufficient plastic rotational capacity, adapt to the provided capacities in each section. Hence, the distribution in ULS will approach the distribution provided by a linear elastic analysis if reinforcement is designed accordingly.

1.2 The finite element method and element formulations

The finite element method (FEM) is a general tool for solving differential equations suitable for structural engineering applications. FEM is capable of handling large structural mechanics problems by discretizing the problem into a finite number of elements, which in turn is governed by equations. Since the element equations govern the model and the results, it is important that designers have a fair understanding of the underlying assumptions of these elements.

The elements used in FEM can roughly be categorized in three different categories: continuum elements, structural elements, and special purpose elements. Continuum elements describe the structure as a continuum and give the stress-state in the structure. This includes 3D solid elements and 2D plane stress/strain elements. Since these elements work with the stresses of the structure, they describe the real behavior of the structure and lack some of the limitations of the structural elements. However, the output from an analysis based on such elements is often massive and difficult to apply on reinforced concrete design since the design of such structures are more easily done based on sectional forces. To design concrete structures according to current praxis on basis of 3D-volume elements, integration of cross-sections must be performed to get the sectional reactions.

Structural elements are based on the equations of for example beam and plate theory. This makes structural elements suitable for design since they provide sectional forces directly for each cross-section. Structural elements also allow for a simpler and more intuitive modelling process, see Figure 2.2.

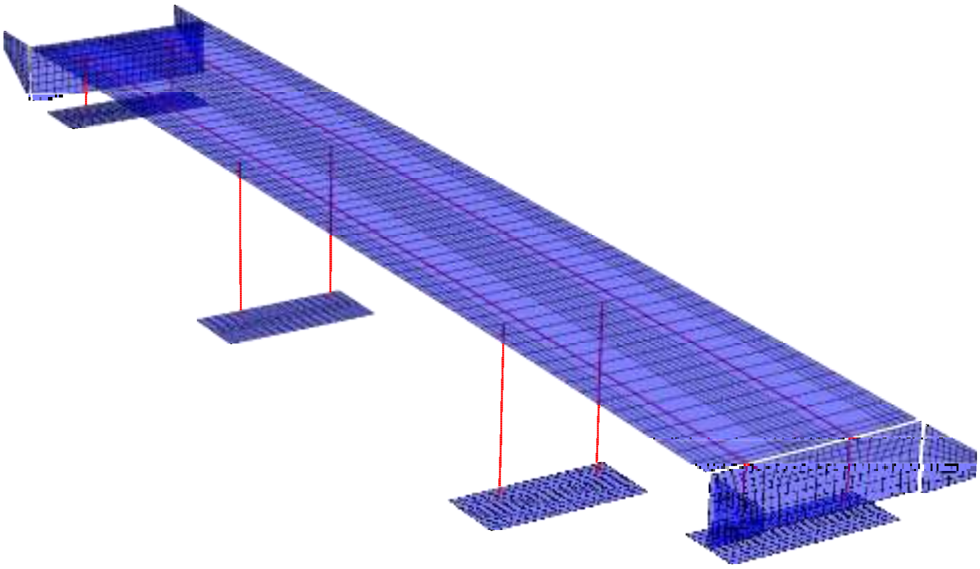


Figure 2.2. Example of finite element structural analysis model of a double beam bridge. Beam elements shown as lines (red) represent main beams and columns, shell elements shown as surfaces (blue) represent bridge deck, end shields, wing walls and foundation slabs

The basic assumption of beam- and shell elements is that they rely on linear strain distribution over the cross-section (e.g., plane sections remain plane during loading, Bernoulli-theorem). This assumption has large impact on certain areas of a structure, such as frame corners, deep beams, or holes, where the strain distribution diverges from the assumed linear distribution. To cope with this a designer might use different techniques, discussed further in chapter 4.

Special purpose elements are used to describe certain conditions in the model, such as interaction between structural parts or foundations.

1. Design of reinforced concrete bridges based on linear elasticity

The response of reinforced concrete is highly non-linear, Plos (1996), where effects such as cracking and yielding of reinforcement have large impact on the stiffness of the structure. This causes redistribution of sectional forces when stiffer sections become more stressed, which is why a linear elastic model is unable to describe the behaviour of a reinforced concrete structure. However, thanks to the structures ability to redistribute forces linear elastic models can be used for design. If the structure has sufficient capacity for plastic rotation, which governs the ability to redistribute forces, the structure will be able to utilize its capacity in all critical sections before a mechanism is formed and failure occurs.

By this reasoning, any distribution of sectional forces fulfilling equilibrium is valid for design in ultimate limit state, if the structure can fully develop the provided capacity in critical sections by yielding of weaker sections. If sufficient ductility is provided in yielded sections, redistribution will continue until all critical sections yield and a mechanism is formed causing structural failure.

Design according to linear elastic analysis is generally considered as a good design approach. It is normally assumed that a design according to linear elastic analysis will require only small amounts of plastic rotation for the sections to utilize their maximum capacities,

1.1 Critical section

The critical cross sections for design depend on the expected modes of failure. For a cast connection, such as a frame corner or monolithic column, the critical crack for bending failure will form along the column or wall surface, Sustainable Bridges (2007); hence this is where reinforcement can be expected to yield first. When designing a structure according to a linear elastic moment distribution, it is therefore sufficient to design for the moment at the column or wall face.

The critical shear crack for such cast connections can be assumed to have an inclination less than 45 degrees. The critical section for shear will therefore be situated at a distance equal to the internal lever-arm from the face of the column or wall, Sustainable Bridges (2007), or approximated to the thickness of the member, Rombach (2004). The shear force inside of that section will be carried to the column/wall and not be critical for shear failure.

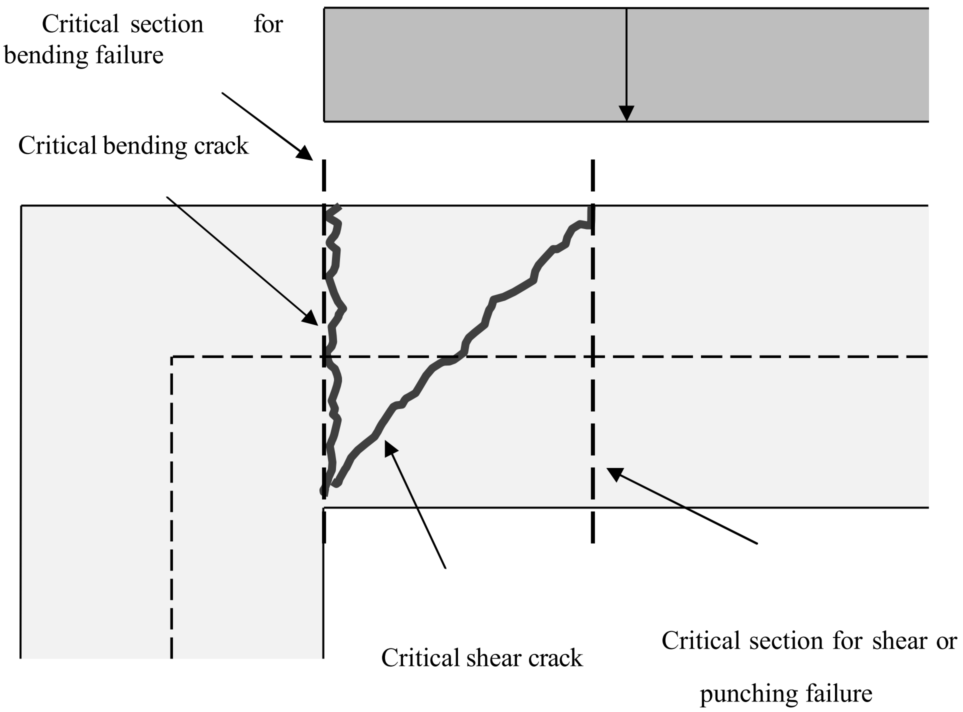


Figure 3.1. Illustration of critical sections at a support region of a frame corner loaded on the top surface.

1.2 Redistribution of moments

Due to the non-linear behaviour and the capacity of a structure to redistribute forces and moments, a redistribution of moments compared to the linear elastic solution is possible. Redistribution will take place when concrete cracks and when reinforcement yields. Continuous redistribution will also take place in the concrete due to its non-linear behaviour after cracking (stage II). When a concrete slab cracks, torsional moments will be taken as bending moments in the sections. Redistribution will also take place between sections, e.g., from support to field sections, and within sections, e.g., from primary to secondary reinforcement directions in a slab.

In Eurocode, the allowable redistribution of sectional moments is governed by the need for plastic rotation and ductility of the cross sections, SS-EN 1992-1 (2008). When a design has been made of a cross-section the ductility can be checked with relative ease by using the ratio between the compressive zone of concrete, x_u, and the effective height of the cross-section, d, see equation 3.1.

$$\underline{x_u} \leq \underline{0.80\delta - 0.35} \qquad d \qquad \beta_c$$

$$\frac{x_u}{d} \le \frac{0.80\delta - 0.45}{\beta_c}$$ for concrete grades C12/15 to C50/60

for concrete grades C55/67 and

higher (3.1)

In addition, the ductility of the reinforcement steel should allow for sufficient redistribution, SS-EN 1992-2 (2005), according to equation 3.2.

$\delta \ge 0.7$ for reinforcement steel of ductility class B and C for reinforcement steel of ductility class A (3.2

$\delta \ge 0.8$

1.3 Design of beams

Design of beam elements can be made based on sectional forces acquired from a linear elastic structural analysis. For this a non-linear cross-sectional analysis can be used, Sustainable Bridges (2007).

The forces applied to a cross-section are to be resisted by the reactions in the concrete compressive zone and the reactions in tensile reinforcement. Compressed concrete will together with the tensile reinforcement form a force couple to balance the moment acting on the section.

For preliminary simplified design, the required reinforcement area can be estimated using simplified expressions, Al-Emrani, et al. (2008). By assuming an internal lever arm and yielding of tensile reinforcement the resisting moment can be calculated and required amount of reinforcement estimated.

Estimation of internal lever arm of a ductile section subjected to pure bending, see equation 3.3.

$$z \approx 0.9 \times d \qquad (3.3)$$

Estimation of moment capacity with yielding of reinforcement can then expressed according to equation 3.4.

$$M_{Rd} \approx f_{yd} \times A_s \times 0.9d \qquad (3.4)$$

Which gives an estimation of required reinforcement area according to equation 3.5.

$$A_s \approx \frac{M_{Ed}}{f_{yd} \times 0.9d} \qquad (3.5)$$

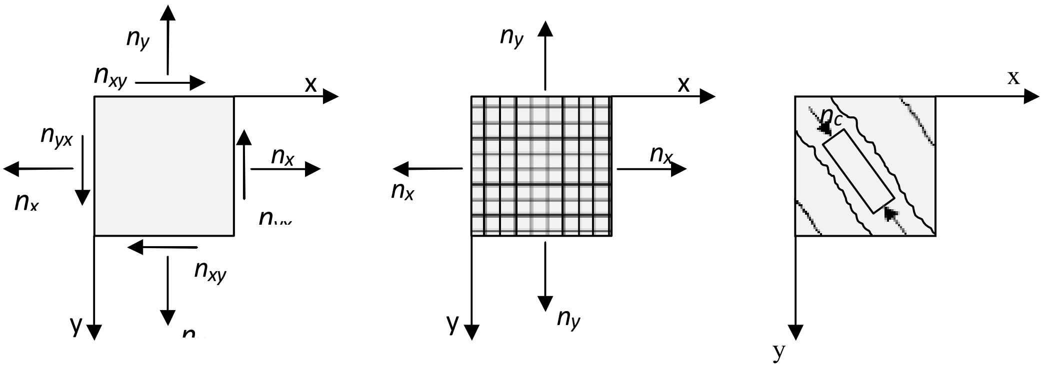


Figure 3.3. Stress resultants in a 3D-membrane element, resulting reinforcement forces and compressive strut in concrete.

A general shell element is subjected to a combination of membrane and bending actions. The design of a combined membrane/bending-element can be done by combining the plate-bending and membrane action models into a sandwich model, see Figure 3.4, Fib Bulletin 45 (2008), SS-EN 1992-2 (2005). In the sandwich model, the cross section is divided into three layers where the outer layers are subjected to membrane action and the middle layer resists the out-of-plane shear force and provides a lever arm for the outer membranes to resist the bending moment.

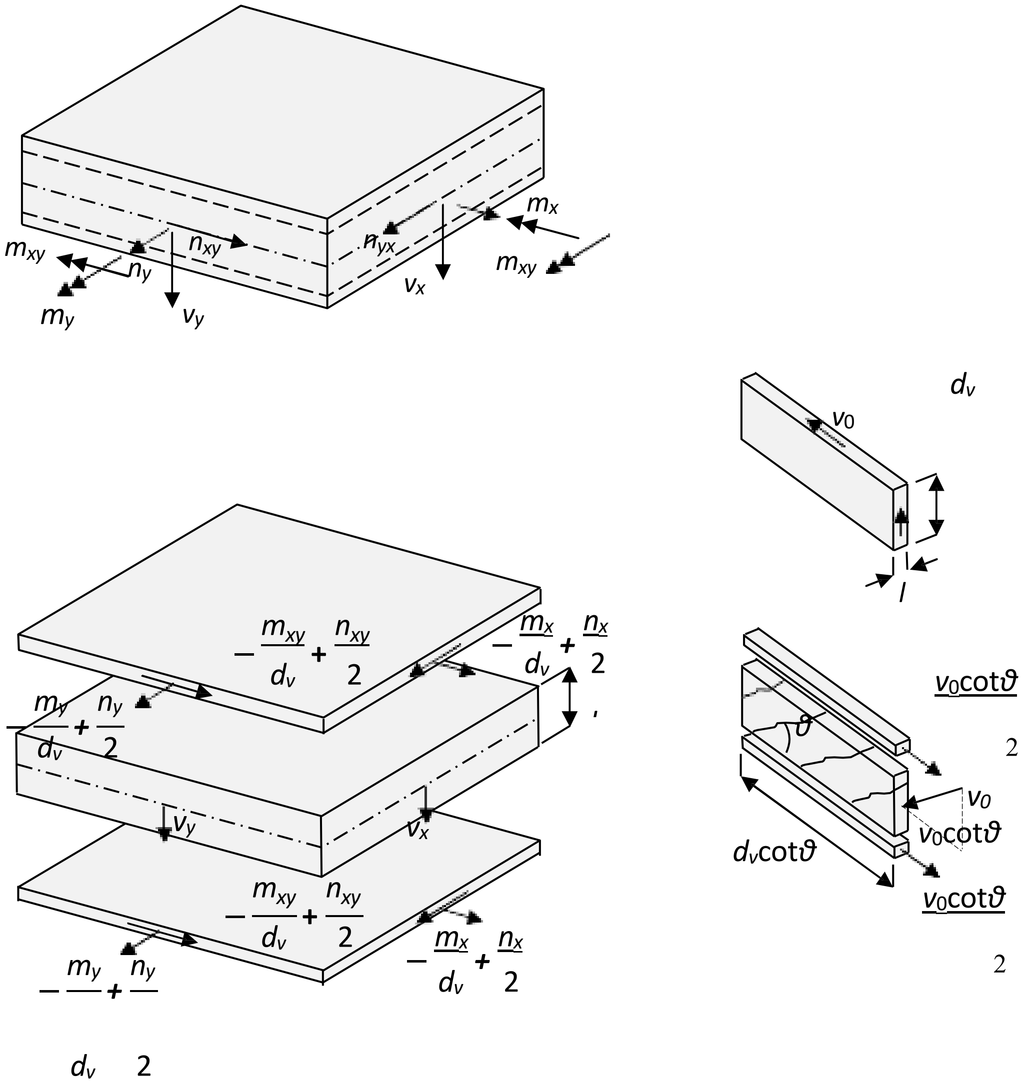


Figure 3.4. Stress resultants in a shell element (top left), layers and force distribution in the sandwich model (bottom left). To the right, contribution to outer layer membrane forces due to inclined shear cracking in the mid-layer. Adopted from Fib Bulletin 45 (2008).

By combining the models above for plate- and membrane elements using the middle layers thickness, d_v, as internal level arm the expressions in equations 3.8 are obtained.

$$\begin{aligned} a_{sx} f_y &\geq \frac{m_x}{d_v} + \frac{n_x}{2} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} \right| \\ a_{sy} f_y &\geq \frac{m_y}{d_v} + \frac{n_y}{2} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} \right| \\ a'_{sx} f_y &\geq -\frac{m_x}{d_v} + \frac{n_x}{2} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} \right| \\ a'_{sy} f_y &\geq -\frac{m_y}{d_v} + \frac{n_y}{2} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} \right| \end{aligned} \tag{3.8}$$

whereas. is the required reinforcement area and f_y is the yield limit of the reinforcement steel. If the middle core layer contains transverse shear cracks the expressions are modified to equations 3.9.

$$\begin{aligned} a_{sx} f_y &\geq \frac{m_x}{d_v} + \frac{n_x}{2} + \frac{v_x^2}{2v_0} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} + \frac{v_x v_y}{2v_0} \right| \\ a_{sy} f_y &\geq \frac{m_y}{d_v} + \frac{n_y}{2} + \frac{v_y^2}{2v_0} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} + \frac{v_x v_y}{2v_0} \right| \\ a'_{sx} f_y &\geq -\frac{m_x}{d_v} + \frac{n_x}{2} + \frac{v_x^2}{2v_0} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} + \frac{v_x v_y}{2v_0} \right| \\ a'_{sy} f_y &\geq -\frac{m_y}{d_v} + \frac{n_y}{2} + \frac{v_y^2}{2v_0} + \left| \frac{m_{xy}}{d_v} + \frac{n_{xy}}{2} + \frac{v_x v_y}{2v_0} \right| \end{aligned} \tag{3.9}$$

where v_0 is the shear resultant. In the above expression the common choice of crack inclination $\theta=45^\circ$ has been chosen.

This model is somewhat simplified since it is assumed that the core layer does not contribute to the transfer of membrane forces. The internal level arm is assumed to be the same for both reinforcement directions. This can be handled by modifying the model, which results in a more complicated iterative model. However, for the scope of this thesis the simplified model presented above is sufficient.

1.4 Traffic loads on bridges

To simulate the loading situation due to traffic Eurocode uses a combination of concentrated and distributed loads. The carriageway of the structure is divided into discrete traffic lanes where the loads are applied, see Figure 3.5.

Loads are moved along and between the traffic lanes to capture the response for vehicles being situated at different positions. The results are then combined into an envelope which gives the maximum reaction on each element.

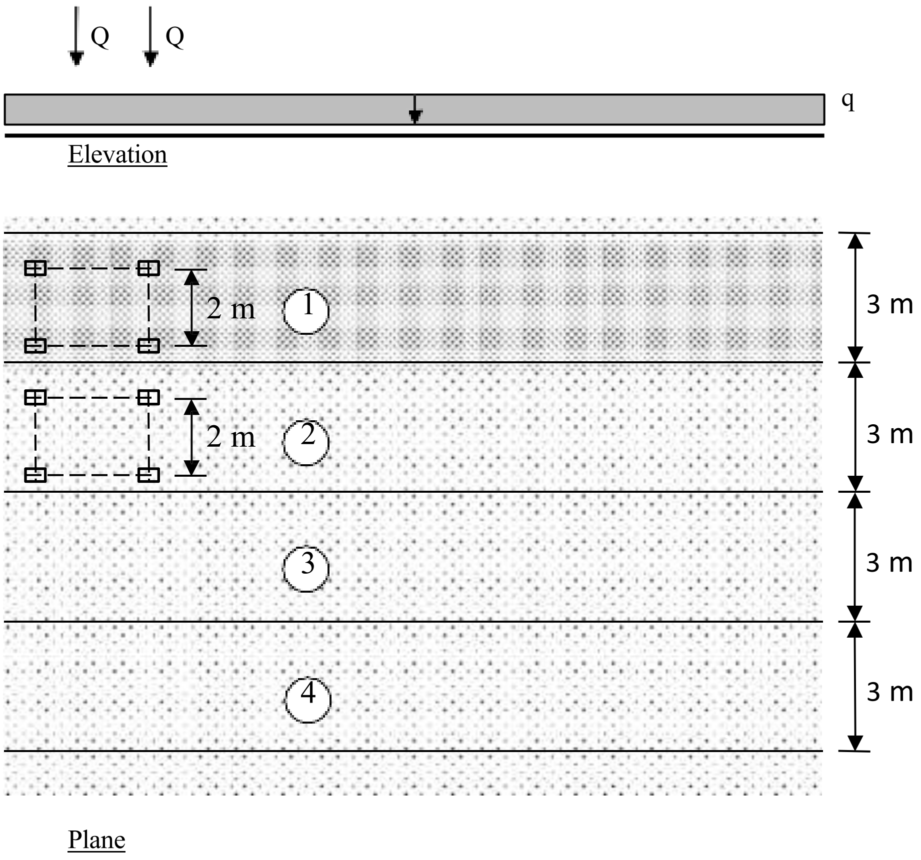


Figure 3.5: Illustration of lane division of a bridge carriageway. All lanes are loaded with a distributed load, while some lanes are loaded with point loads simulating vehicle positions according to certain rules.

Concentrated loads are applied in groups simulating wheel pairs (axle loads) and bogie loads, see Figure 3.5. Each axle load consists of two concentrated loads, and two axle loads make up one bogie load. Distributed loads are applied onto the whole traffic lane. Values for axle and distributed loads according to Eurocode and Swedish annex are presented in Table 3.1.

Table 3.1. Values for axle and distributed loads for traffic load model

Traffic lane	**Distributed load q [kN/m^2]**	**Axle loads Q [kN]**
1	6.3	270
2	2.5	180
3	2.5	0
Remaining lanes	2.5	0
Remaining surface	2.5	0

Horizontal loads on bridges (loads in the bridge plane) comes from vehicles braking, accelerating, or turning on the bridge. Braking loads are calculated as a part of the total vertical loads acting on a traffic lane. Acceleration loads are defined as the braking loads acting in opposite direction, which practically means that the load can have both positive and negative sign.

2 Practical modelling

Performing 3D structural analysis of concrete bridges with FEM allows the user to control several aspects of the modelling. These aspects range from choice of geometric and element representation to choosing to what extent and how closely the model needs to resemble the structural system.

2.1 Element mesh

The finite element mesh needs to be sufficiently dense to capture the proper response of the structure. A general rule of thumb says that the element size for shell elements should be equal to or smaller than the thickness of the elements.

Near critical sections there must be a sufficient number of elements between singular peak values (such as pinned connections) and the critical section. Mesh dependency studies have shown that for different meshes the difference in results is small only one element away from the peak value and two elements away it is negligible, Davidson, (2003) and Sustainable Bridges (2007).

2.2 Structural element types

As stated before, structural elements are suitable for structural analysis in a design stage of bridges and structures since the output (sectional forces) allows for simple and intuitive design of the structure. Structural elements are also rather effective at describing the actions of the structure, which is important for analysis of bridge structures where design codes require analysis of several load cases, positions, and load combinations.

2.3 Discontinuity regions and frame corners

A structural system can be divided into B- and D-regions. This is done in order to distinguish between areas in a structure where the state of strain diverge from the plane strain assumption (Bernoulli hypothesis) under which beam, and plate theory are valid. Hence, D-regions (or discontinuity regions) are areas in a structure where the strains no longer remain linear over the cross-section, see Figure 4.1.

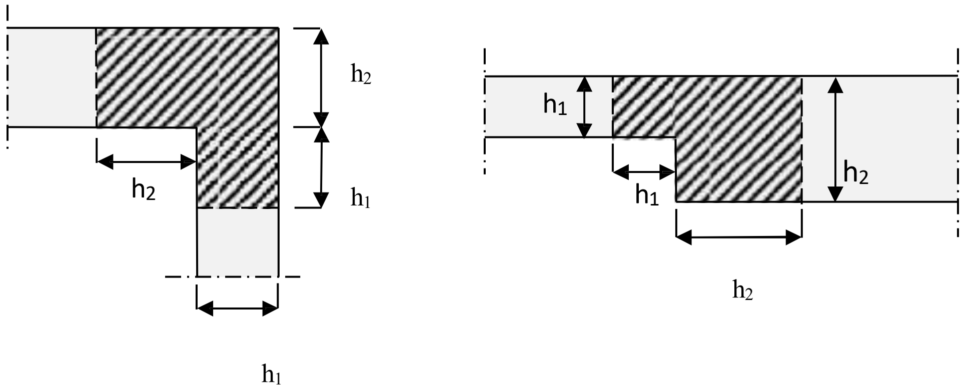
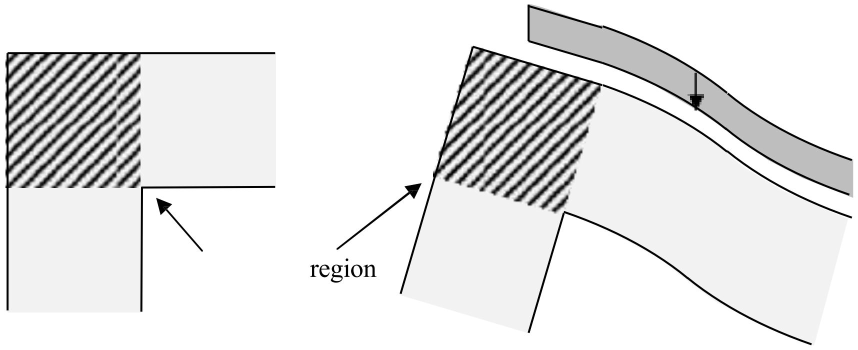
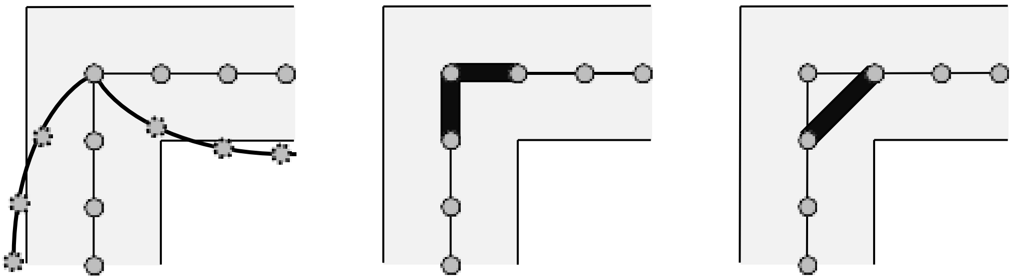


Figure 4.1. Examples of discontinuity regions, D-regions, where the strain distribution will differ from the linear strain distribution predicted by beam theory. The extent of the discontinuity is often assumed to be equal to the width of the element. Adopted from Engström (2011b)

These regions may not only affect the response locally, but the modelling of them is also important to correctly assess the response globally. In for example a frame corner, see Figure 4.2, the strain distribution no longer remains linear and, when modelled in detail, the elements within the corner cannot move independently of each other. Often the Centre line is modelled with beam or shell elements and, since the corner behaves like a diaphragm, its stiffness becomes underestimated. Therefore, the elements within the corner region should be coupled to simulate this effect.



Deformation around corner during loading



Deformation neglecting corner rigidity

Modelling methods to account for corner rigidity – stiff coupling of the corner

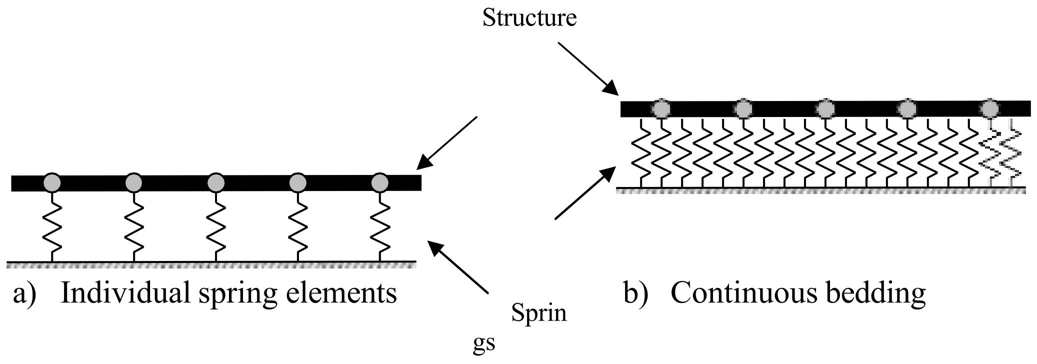
Figure 4.2. Response of a frame corner during loading and the response of a model neglecting frame corner rigidity. Alternative models for accounting frame corner rigidity are also shown,

The stiffness increase can be modelled using stiff connections, either assigning the corner elements with rigid properties (very large/infinite stiffness) or introducing an additional, inclined, stiff truss element into the corner region,

When assigning stiff material certain care must be taken, since if the stiffness is too great in relation to the ordinary stiffness, numerical problems when performing the analysis might occur, Also, when assigning stiff properties for shell elements it is important to model the stiffness increase in the correct direction. In for example a slab frame bridge modelled with shell elements; the stiffness increase is only relevant in the normal direction of the frame. Increasing the stiffness in the transverse direction will cause the stiffness of the wall to increase, causing errors in load cases were bending of the walls occur. However, these effects might be difficult to spot; since the structures principal direction of action is in the longitudinal direction the increased transverse stiffness will only influence certain load cases where transverse bending of the walls will occur. Therefore, the effect might be difficult to detect if the response is not evaluated for these individual load cases. Equation 4.2 shows the constitutive relationship for orthotropic material in plane stress elements, such as shells subjected to in-plane loading.

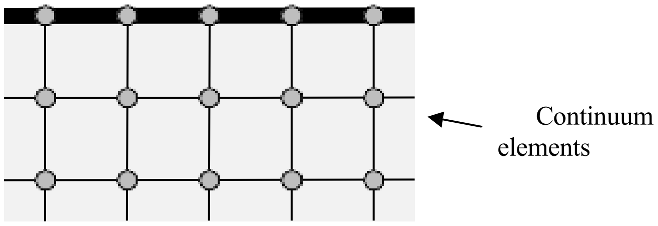
2.4 Soil-Structure Interaction

In many cases the supports will deform to some extent when loaded, for example in shallow foundations. Here the soil will deform under the foundation slab when loaded, which will influence the structural response. Consequently, it should be included in the structural model. The soil material and its influence on the structural system can be included in different ways; some examples are presented in Figure 4.3.



a) Individual spring elements

b) Continuous bedding



c) Continuum elements

Figure 4.3. Various models for elastic support. The simplest and most convenient methods to include the foundation materials deformation are by springs (models a – b). A more advanced model would be to represent the surrounding soil material by a continuum.

The deformation of the soil material can be included by representing it as springs, either with discrete springs situated at element nodes, see Figure 4.3a, or with an element formulation including continuous bedding, see Figure 4.3b. The stiffness of the springs is characterized by a bedding modulus, representing the normal stiffness of the soil. When using individual spring elements at element nodes, see Figure 4.3a, the spring stiffness will also be dependent on the spacing between the nodes, i.e., it will be element mesh dependent. For the continuous bedding model, the spring stiffness is included in the element stiffness matrix and therefore not mesh dependent. However, with such a model it is generally only possible to define springs normal to the elements. It should be noted that the spring methods will give the same results, provided that the element mesh is sufficiently dense.

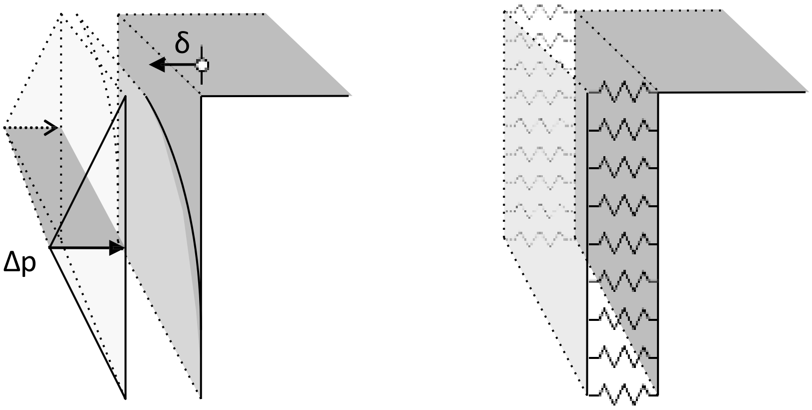
A simple way to assess the bedding modulus is to perform a simplified settlement calculation with an arbitrary load. The bedding modulus can then be estimated as a linear relation between settlement and applied load, equation 4.2.

$$k = F / d \qquad (4.2)$$

Methods based on spring foundations (or Winkler foundations) will not account for shear interaction between springs, leaving surrounding soil stress-less. This can often be neglected, though it might be important to account for in some situations. Examples are adjacent plates where one foundation does not settle independently of the other or shallow uniformly loaded foundations where an ordinary spring model will not result in any member forces. There are methods for including the shear stiffness of the soil, for example with a so-called Pasternak foundation, Blaauwendraad (2010) and Caselunghe & Eriksson (2012), though it is not covered further in this thesis.

Since linear material parameters are generally used, a spring model is unable to describe uplift of the foundation correctly since such a case will result in tension in the springs, not present in the real case where the slab will instead lift from the soil. By using non-linear springs this could be handled though, according to the reasoning in chapter 2, such a model is incompatible with design of bridges.

Interaction with surrounding soil material is relevant also for backfilled vertical members, such as frame walls or end shields on a bridge. The backfill material will provide resistance against horizontal deflection of the structural member, affecting the structural response and should therefore be included in a structural analysis. Backfill can be modelled in a similar manner as foundation interaction, where springs describe the soil in a very simplified manner. Another way to include the backfill is by adding an external load simulating the increased earth pressure due to deflection, Figure 4.4.



a) b)

Figure 4.4. Earth pressures increase models; (a) model where an additional load, Ap, is added against the vertical member dependent on the members deflection at the upper edge, δ. (b) Model like spring foundation models described in 4.4.

In the model presented by TR Bro the increased earth pressure is given a maximum value at the mid height of the vertical member and the value *0* at top and bottom edge of the member. The load *Ap* is defined as expressed in equation 4.3.

$$\Delta p = c \cdot \gamma \cdot z \cdot \beta \qquad (4.3)$$

where:

$$c = \begin{cases} 300 & \text{if earth pressure is favorable} \\ 600 & \text{if earth pressure is unfavorable} \end{cases}$$

γ density of soil material

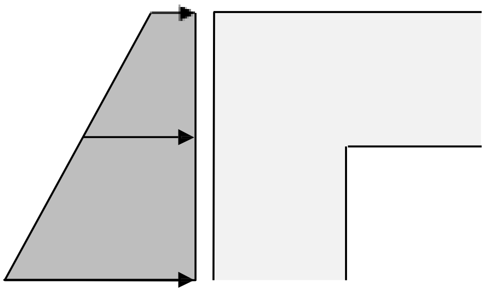
z depth from surface

$\beta = \frac{\delta}{h}$ ratio between top deflection, δ, and height, h, of the member

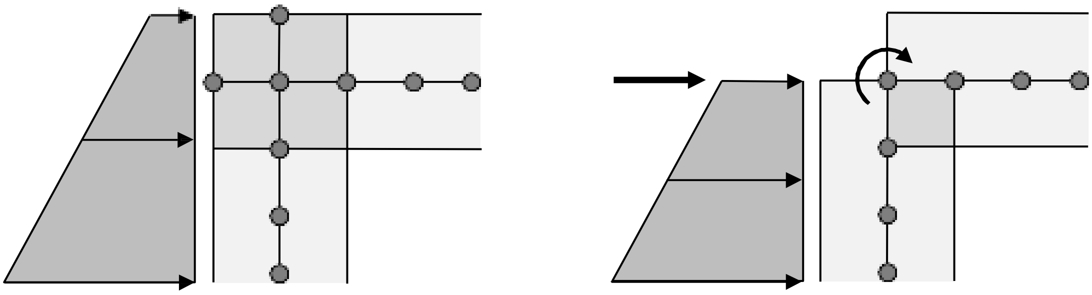
2.5 Load application

When loaded against a corner region, a structural model represented by member centerlines will not account for loading outside of the centerlines, for example frame corners. This load can be accounted for in different ways presented in Figure 4.5.

- Extending elements over the corner
- Adding point load and moment to account for load and eccentricity of load,



Real load case



Model with elements extended over the corners

Model including load and moment due to loading outside of centerline

Figure 4.5. Various models for accounting loading outside of centerlines.

4. Results:

1.4.1 Methodology of comparison

Comparison of the models was made in a simplified manner focusing on the required reinforcement in the main beams. This was calculated for the beam model according to the simplified method presented in section 3.3, again presented as equation 6.1.

1.4.2 Resulting primary reinforcement in bridge main beams

To verify the models, the total support reactions are compared in Table 6.1. Since the differences are less than 1 %, it is practically negligible.

Table 6.1. Comparison of support reactions for support 1 (end support) and support 2 (mid support). Reactions evaluated at the bottom of the foundation slabs. In general, the differences are small, the largest difference is noticed in the moment around an axis perpendicular to the supports.

Support reaction		**Beam**	**Shell**	**Diff. [%]**
Self-weight				
Vertical support 1	kN]	-2387	-2381	0%
Vertical support 2	kN]	-4752	-4752	0%
parallel support 2	kN]	70	71	1%
Moment perpendicular support 2	kNm]	118	120	2%
Traffic characteristic				
Vertical support 1	kN]	-1239	-1240	0%
Vertical support 2	kN]	-1715	-1715	0%
Parallel support 2	kN]	-109	-110	1%
Moment parallel support 2	kNm]	-1928	-1902	-1%
Perpendicular support 2	kN	-170	-158	-7%

	]			
Moment perpendicular support 2	kNm]	-290	-293	1%
ULS				
Vertical support 1	kN]	-5731	-5937	4%
Vertical support 2	kN]	-9401	-9399	0%
parallel support 2	kN]	-105	-106	0%
Moment parallel support 2	kNm]	-3044	-3010	-1%
Perpendicular support 2	kN]	-268	-250	-7%
Moment perpendicular support 2	kNm]	-370	-372	1%

Similarities between the models are also visible when studying the sectional moment in the bridge deck as well as the deflections, shown in Figure 6.7 for the self-weight load case.

Sectional moment in primary, x, direction m_x [kNm/m]

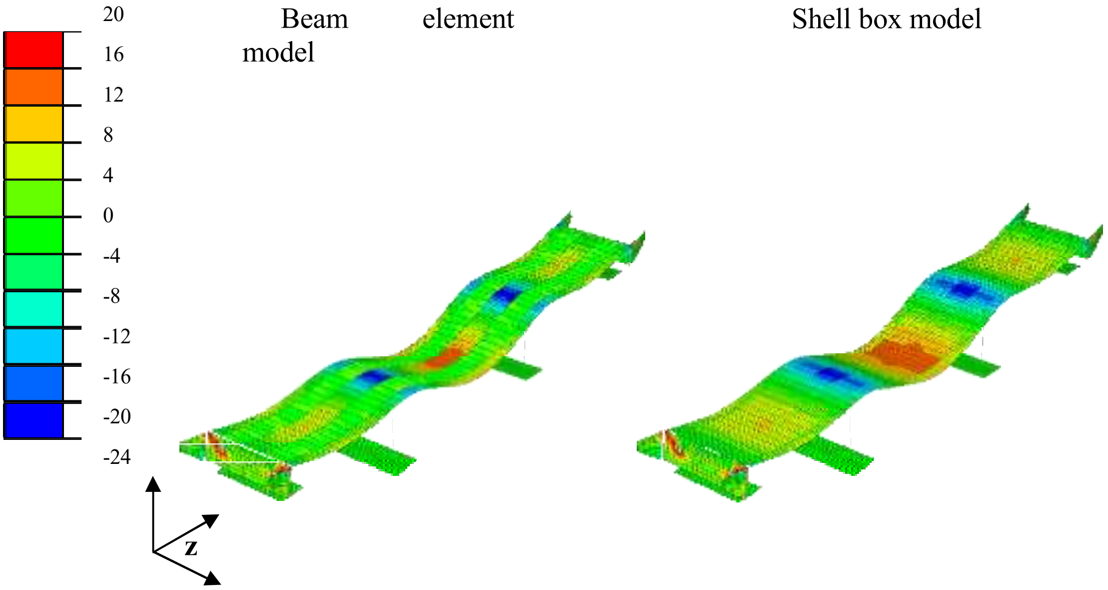


Figure 6.7. Simple comparison of primary sectional moment and deformations in self weight load case. The models show similar behavior in both parameters. The area above the beams in the beam element model shows no stress since the load in that area is carried by the beam elements below the deck. Deflections scaled in both models with a factor 500.

Also, when studying the main reinforcement requirement, it becomes evident that the difference between the modelling methods is small, see Figure 6.8 and Figure 6.9. One small difference which can be observed is that the shell model shows a requirement for more bottom reinforcement, and less top reinforcement indicating a different distribution between span and support sections.

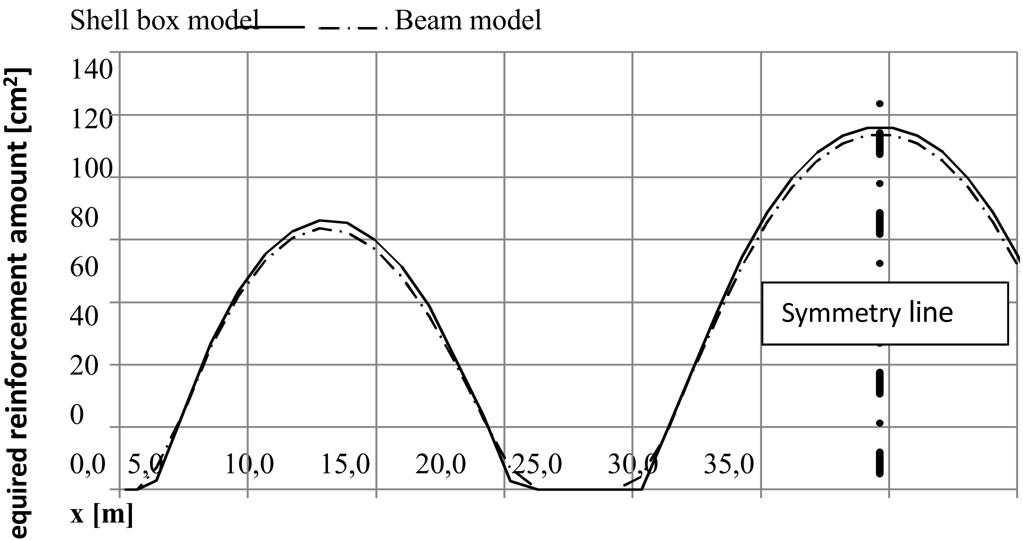


Figure 6.8. Required main bottom reinforcement in one of the beams. Slightly higher requirement for the shell model than the beam element model.

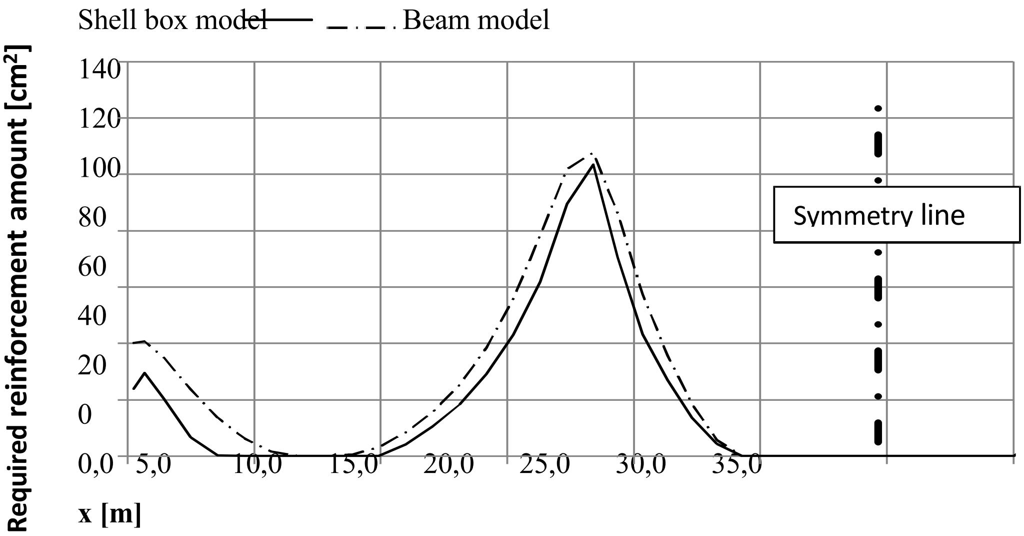


Figure 6.9. Required main top reinforcement in one of the beams. The beam element model shows a slightly higher reinforcement requirement than the shell element model.

Discussion on modelling with shell elements

The main objective of the study performed in Chapter 6 was to illustrate an alternative modelling method, and it was shown that it is possible to model beam structures with shell elements. However, to draw more accurate conclusions a more thorough analysis of the results is needed. This could be done for example by analyzing and comparing different sections in both main and transverse directions.

In general, only very small differences was noticed between the models, and since the total support reactions of both models are practically equal it is safe to assume that both models are loaded in the same way. Most distinct observed differences between the models are differences in sectional force distribution, both between and within cross-sections. Since it has been shown that both models describe the same loading situation both models should be valid for design, providing that the checks for the cross-sections capability to develop their full capacity are made.

One main advantage of the shell model is that it probably simplifies the design of the intermediate areas between main beams and bridge deck. Since the transition between these areas for the shell model is smoother, and doesn't involve any coupling of different element types, the shell model provides more accurate information about the response and requirement of reinforcement in these areas. This indicates that these areas can be designed in a more intuitive way which better reflects the needs in the structure.

Suggestions for further work within the subject:

- Development of guidelines for modelling in 3D, together with easy-to-use methods for model verification.
- Adapt current design codes to 3D-modelling, for example load models in TK- bro.
- Further studies on the impact of structural analysis on the design, for example by verifying design based on linear elastic structural analysis with non-linear FE-models.
- Study the influence of design based on linear elastic structural analysis on verification of response in SLS.

5. Conclusion

In this thesis different modelling procedures for structural analysis in design of concrete bridges are studied. The results show that in most cases there are only small differences between the models and procedures for structural analysis. However, it was identified that when modelling according to some principles inadvertent restraint might be introduced to the model if certain care was not taken. This might alter the response drastically, while still be difficult to detect when studying envelopes in ULS.

Introduction of inadvertent restraint was mainly an issue when modelling with stiff shell elements in frame corners. It was identified that lack of verification and difficulties in interpreting results could easily lead to mistakes and undesired results.

It was also noticed that some load models give differences in results. Though it was not studied in detail, it could be seen that different methods of modelling frame wall and soil interaction gave different moment distributions over the frame walls. The load model for earth pressure increase presented in Trafikverkets recommendations document, was more difficult and less intuitive to use in 3D analyses.

In general, it could be said that verification and interpretation of 3D models can be difficult. It is also difficult to assess the behavior of a 3D model under certain types of loading beforehand, for example temperature load effects. Therefore, it is always important to study the results carefully and critically to assess their credibility. Since modelling in 3D is in principle a requirement for structural analysis today, there is a need for guidelines and easy to use verification methods.

The response of concrete structures is non-linear, while the design of such a structure is made based on a linear structural analysis. This is possible due to the structures ability to adapt, provided capacities by redistribution of sectional forces to stiffer regions in the structure. As the purpose of a structural analysis for design is not to accurately describe the response of the structure, other parameters for the structural analysis model should be prioritized, such as usability. Important features regarding usability include model construction, verification, and interpretation.

Since the impact of choices in model construction is relatively small, if errors are avoided, a structural engineer could be relatively free in constructing structural analysis models. As the difference between different models is small it is more important to focus on that a model does not introduce errors than it is entirely accurate. In short, one could say that it is more important to avoid errors than it is to model accurately. Therefore, it is important that an analysis model is easily verified with simpler models since errors otherwise easily arise in 3D modelling.

It is important to point out that these suggestions are only valid for design of structures, when evaluating the response of existing structures different approaches are necessary.

References

1. Al-Emrani, M., Engström, B., Johansson, M. & Johansson, P. (2008): *Bärande konstruktioner Del 1* (Load bearing structures part 1. In Swedish). Department of Civil and Environmental Engineering, Chalmers University of Technology, Göteborg.
2. Blaauwendraad, J. (2010) *Plates and FEM - Surprises and Pitfalls.* Springer, Dordrecht.
3. Broo, H., Lundgren, K. & Plos, M. (2008): *A guide to non-linear finite element modelling of shear and torsion in concrete bridges.* Department of Civil and Environmental Engineering, Chalmers University of Technology, Göteborg.
4. Caselunghe, A. & Eriksson, J. (2012): *Structural Element Approaches for Soil- Structure Interaction.* MSc. Thesis. Department of Civil and Environmental Engineering, Chalmers University of Technology, Göteborg.
5. Dassault Systèmes (2008) *Abaqus User's Manual.* Dassault Systèmes, United States of America.
6. Davidson, M. (2003): *Strukturanalys av brokonstruktioner med finita elementmetoden*
    - *Fördelning av krafter och moment* (Structural analysis of bridge structures with the finite element method - Distribution of forces and moments. In Swedish). Brosamverkan Väst, Göteborg.
7. Engström, B. (2011a): *Design and analysis of continuous beams and columns,* Department of Civil and Environmental Engineering, Chalmers University of Technology, Göteborg.
8. Engström, B., (2011b). *Design and analysis of deep beams, plates and other discontinuity regions.* Department of Civil and Environmental Engineering, Chalmers University of Technology, Göteborg.
9. Fib Bulletin 45 (2008): *Practitioner's Guide to Finite Element Modelling of Reinforced Concrete Structures.* CEB-FIB, Lausanne.
10. O'Brien, J. E. & Keogh, L. D. (1999): *Bridge deck analysis.* E & FN Spon, Cornwall.
11. Plos, M. (1996): *Finite element analyses of reinforced concrete structures.* Department of Civil and Environmental Engineering, Chalmers University of Technology, Göteborg.
12. Rombach, G. A. (2004): *Finite element design of concrete structures.* Thomas Telford, Cornwall.
13. Scanscot Technology AB (2010): *Brigade/Plus User's Manual.* Scanscot Technology AB, Sweden.
14. SS-EN 1991-2 (2007). *Eurokod 1: Laster på bärverk - Del 2: Trafiklast på broar*

(Eurocode 1: Actions on structures - Part 2: Traffic loads on bridges. In Swedish). SIS Förlag AB, Stockholm.

15. SS-EN 1992-1-1, 2008. *Eurokod 2: Dimensionering av betongkonstruktioner - Del 1- 1: Allmänna regler och regler för byggnader* (Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings. In Swedish).
16. SS-EN 1992-2 (2005): *Eurokod 2: Dimensionering av betongkonstruktioner - Del 2: Broar* (Eurocode 2: Design of concrete structures - Part 2: Concrete bridges - Design and detailing rules. In Swedish). SIS Förlag AB, Stockholm.
17. Sustainable Bridges (2007): *Non-Linear Analysis and Remaining Fatigue Life of Reinforced Concrete Bridges.* Sustainable Bridges - Assessment for Future Traffic Demands and Longer Lives.
18. Trafikverket (2009a): *TK Bro* (Technical Requirements for Bridges. In Swedish).
19. Trafikverket, Borlänge.
20. Trafikverket (2009b): *TR Bro* (Technical Recommendations for Bridges. In Swedish).Trafikverket, Borlänge.
21. Wang, C. K., Salmon, C. G., & Pincheira, J. A. *Reinforced Concrete Design*. John Wiley & Sons.
22. Nilson, A. H., Darwin, D., & Dolan, C. W. *Design of Concrete Structures*. McGraw-Hill.
23. Lin, T. Y., & Burns, N. H. *Design of Prestressed Concrete Structures*. Wiley.
24. Leonhardt, F. *Prestressed Concrete: Design and Construction*. Wilhelm Ernst & Sohn.
25. Menn, C. *Prestressed Concrete Bridges*. Birkhäuser.
26. Hambly, E. C. *Bridge Deck Behaviour*. Chapman & Hall.
27. Barker, R. M., & Puckett, J. A. *Design of Highway Bridges*. Wiley.
28. Podolny, W., & Muller, J. M. *Construction and Design of Prestressed Concrete Segmental Bridges*. Wiley.
29. Cusens, A. R., & Pama, R. P. *Bridge Deck Analysis*. Wiley.
30. Johnson, R. P. *Composite Structures of Steel and Concrete*. Blackwell Publishing.
31. Chen, W. F., & Duan, L. *Bridge Engineering Handbook*. CRC Press.
32. Timoshenko, S. P., & Young, D. H. *Theory of Structures*. McGraw-Hill.
33. Zienkiewicz, O. C., & Taylor, R. L. *The Finite Element Method*. Butterworth-Heinemann.
34. Ghali, A., Favre, R., & Elbadry, M. *Concrete Structures: Stresses and Deformations*. CRC Press.
35. MacGregor, J. G., & Wight, J. K. *Reinforced Concrete: Mechanics and Design*. Pearson.
36. Nowak, A. S., & Collins, K. R. *Reliability of Structures*. McGraw-Hill.
37. EN 1992. *Eurocode 2: Design of Concrete Structures*.
38. EN 1998. *Eurocode 8: Design of Structures for Earthquake Resistance*.
39. Indian Roads Congress (IRC:112). *Code of Practice for Concrete Road Bridges*.
40. IRC:6. *Standard Specifications and Code of Practice for Road Bridges – Loads and Stresses*.
41. IS 456:2000. *Plain and Reinforced Concrete – Code of Practice*.
42. IS 1343:2012. *Prestressed Concrete – Code of Practice*.