Finite element-based elasto-plastic optimum reinforcement dimensioning of spatial concrete panel structures

Author(s):
Tabatabai, Seyed Mohammad Reza

Publication Date:
1996

Permanent Link:
https://doi.org/10.3929/ethz-a-001734882

Rights / License:
In Copyright - Non-Commercial Use Permitted
Finite Element-based Elasto-Plastic Optimum Reinforcement Dimensioning of Spatial Concrete Panel Structures

Seyed Mohammad Reza Tabatabai

Institute of Structural Engineering
Swiss Federal Institute of Technology (ETH Zurich)

1996
Preface

Reinforcement dimensioning of concrete structures is an everyday task for many practicing civil engineers. For this purpose, simplified hand methods are often applied, many of which, based on the lower bound theorem of plasticity theory, aim at finding an equilibrium system capable of somehow carrying the applied loads. On the other hand, with the spread of computers, much attention has been paid to precise stress analysis based on elasticity theory. Despite obvious achievements in specific areas, problems still exist in integrating finite element analysis with design rules towards a rational reinforcement dimensioning and in finding an appropriate balance between the complexity and effectiveness of the model with the ease of comprehension and use for practicing engineers.

These considerations were the motivation for starting a research project in 1992 for the development of a new approach for dimensioning two and three-dimensional reinforced concrete structures, i.e. two-dimensional walls and flat slabs and three-dimensional panel structures. Dr. Tabatabai dealt with the general case of spatial structures in which each flat panel arbitrarily oriented in space can be specified as acting in either pure membrane or combined membrane-bending action. While simpler structures may adequately be designed with simplified methods, for such complicated structures the application of sophisticated computer-based approaches is inevitable.

The approach proposed here for the ultimate load dimensioning of spatial concrete panel structures has two objectives. By regarding each finite element as a dimensioning unit loaded by its nodal forces, the concept of a finite element is extended from analysis to design. The second aim is the optimum reinforcement dimensioning. Rationalization is accomplished by taking advantage of the engineer's experience. Optimization, i.e. reinforcement minimization is achieved by taking into account stress redistribution based on the lower bound theorem of plasticity theory.

As a major part of his dissertation, Dr. Tabatabai has developed the program ORCHID integrating the finite element method, plasticity theory and linear programming. In addition to the structural engineering aspects, much emphasis has also been put on the software engineering side in order to make the program an efficient and useful tool for reinforced concrete designers.

Zurich, November 1996

Prof. Dr. E. Anderheggen
Contents

Abstract ................................................................................................................. 1

Zusammenfassung ................................................................................................. 2

1  Problem definition and scope formulation ...................................................... 3
   1.1  Reinforcement dimensioning today ........................................................... 3
      1.1.1  Dimensioning based on linear elastic stress analysis ............................ 3
      1.1.2  Truss model, stress field and strip methods .......................................... 4
      1.1.3  Elasto-plastic nonlinear analysis ............................................................ 5
   1.2  Scope of this work ....................................................................................... 6
      1.2.1  Extension of finite element concept from analysis to design ................. 7
      1.2.2  Reinforcement rationalization and optimization ..................................... 8
      1.2.3  Development of the corresponding computer program .......................... 9
   1.3  Assumptions and limitations .................................................................... 9
   1.4  Overview ..................................................................................................... 10

2  Basics of the finite element method, plasticity theory and linear programming .. 11
   2.1  The finite element method ........................................................................ 11
      2.1.1  Modelling membrane panels ............................................................... 11
      2.1.2  Modelling bending panels .................................................................... 14
      2.1.3  Finite element discretization ............................................................... 17
      2.1.4  Requirements of an element formulation ............................................. 21
   2.2  Plasticity theory ......................................................................................... 22
      2.2.1  Elastic and perfectly plastic material behavior ...................................... 22
      2.2.2  Limit theorems of plasticity theory ....................................................... 24
      2.2.3  Generalized variables and the theory of the plastic potential ................ 25
      2.2.4  Shakedown theorem ............................................................................ 27
   2.3  Linear programming .................................................................................. 28
      2.3.1  Formulation of a linear program ........................................................... 29
      2.3.2  Simplex algorithm .............................................................................. 31

3  Yield conditions for an infinitesimal element ................................................... 33
   3.1  Yield conditions for a membrane infinitesimal element ............................... 33
      3.1.1  Nonlinear yield conditions ................................................................. 33
      3.1.2  Linearized yield conditions ................................................................. 36
      3.1.3  Skew reinforcement ............................................................................. 38
   3.2  Yield conditions for a membrane-bending infinitesimal element ................. 39
      3.2.1  Pure bending ....................................................................................... 39
      3.2.2  Sandwich model for combined membrane-bending action .................... 42
Contents

4  First concept:
Element-based dimensioning and yield conditions for a finite element  ....... 47
  4.1 Deficiencies of the usual approach ........................................ 47
  4.2 Finite elements as dimensioning units ................................... 49
  4.2.1 Yield conditions formulation for generalized variables ........... 49
  4.2.2 Yield conditions formulation based on element nodal forces ...... 51
  4.3 Generalized strains and stresses for a finite element ................. 55
  4.3.1 Strain modes for membrane ........................................... 56
  4.3.2 Strain modes for bending .............................................. 58
  4.4 Yield conditions for a quadrilateral finite element ................... 60
    4.4.1 Yield conditions for a membrane finite element................ 61
    4.4.2 Yield conditions for a bending finite element .................. 65
    4.4.3 Yield conditions for a membrane-bending finite element ....... 69
  4.5 Yield conditions for a triangular finite element ..................... 70
  4.6 Transverse shear in bending finite elements .......................... 71
  4.7 Mesh parameter for error estimation ................................... 73

5  Second concept:
Optimization by superposition of self-equilibrating stress states .......... 79
  5.1 Basic idea in a simple example ........................................... 79
  5.2 Reinforcement evaluation for the linear elastic solution ............. 83
    5.2.1 Formulation of the minimum weight objective function ........... 83
    5.2.2 Constraint formulation of the linearized yield conditions ....... 86
    5.2.3 Building the linear program for the linear elastic solution .... 87
  5.3 Force redistribution by introducing fictitious plastic strains ....... 90
    5.3.1 Progressive optimization approach ................................ 91
    5.3.2 Initial strains for membrane ....................................... 97
    5.3.3 Initial curvatures for bending ..................................... 97
    5.3.4 Modifications of the tableau ...................................... 99
  5.4 Ultimate load analysis ................................................ 101

6  ORCHID: A program for
“Optimum Reinforced Concrete Highly Interactive Dimensioning” .......... 105
  6.1 The structure of ORCHID .................................................. 105
  6.2 The graphical user interface of ORCHID ................................ 108
  6.3 Free formulation element model ........................................ 111
    6.3.1 Stiffness matrix calculation ....................................... 111
    6.3.2 Membrane and bending triangular elements ....................... 115
    6.3.3 Membrane and bending quadrilateral elements .................... 116
  6.4 Specific features of the finite element model ........................ 117
    6.4.1 Combination of membrane and bending panels .................... 117
    6.4.2 Modelling incompatible meshes and eccentric panel connections .. 119
  6.5 External loads and boundary conditions ................................ 124
  6.6 Equation solver ........................................................ 125
## Contents

7  **Examples** ............................................................. 127
   7.1 T-beam modelled by two panels ................................ 127
   7.2 Three-panel plate with stiffening walls ....................... 133
   7.3 Effects of model assumptions in optimization ............... 137

8  **Closure** ............................................................. 141
   8.1 On the proposed method .......................................... 141
   8.2 Suggestions for future work .................................... 143

**Symbols** ............................................................. 145

**List of figures** ....................................................... 148

**Literature** ............................................................ 152
Abstract

A new method is proposed for the ultimate load reinforcement dimensioning of spatial concrete panel structures. It aims at providing civil engineers with a system which exploits the advantages of today’s established approaches while alleviating some of their deficiencies (chapter 1). The finite element method, plasticity theory and linear programming are integrated in a unified manner so as to obtain a rational reinforcement distribution (chapter 2). The approach can be used for both the dimensioning and the evaluation of the load-carrying capacity of a structure with given reinforcement. The method is based on two basic concepts. The first new idea is finite element-based reinforcement dimensioning. In the conventional approach, yield conditions are applied to the stresses at some points of the structure as obtained e.g. from a linear elastic finite element analysis (chapter 3). The method proposed here, unlike the conventional one, regards each finite element as a dimensioning unit which must withstand its element nodal forces. These are statically equivalent to the stresses within the element and are in perfect equilibrium both with one another and with the external loads. The idea is therefore not to investigate the stress distribution but to find an equilibrium system of nodal forces which can carry the external loads. Thus, the proposed method can also be viewed as a generalization of the truss model approach. The concepts of generalized strains and stresses are employed for deriving yield conditions for quadrilateral and triangular finite elements in membrane, bending and combined membrane-bending actions. The idea of finite element discretization is extended from analysis to design in a natural way (chapter 4).

The second new idea concerns stress optimization and rationalization. Based on the shakedown and lower bound theorems of plasticity theory, self-equilibrating homogeneous stress states are superimposed on the linear elastic solution. These fictitious redistribution cases are not determined by a nonlinear incremental analysis but are chosen at will. Their distributions and intensities are found by linear programming techniques and various optimization criteria. For the task of dimensioning, these superimposed self-equilibrating stress states result in a more rational steel distribution with a lower weight (chapter 5).

Program ORCHID (Optimum Reinforced Concrete Highly Interactive Dimensioning) was developed for the dimensioning of spatial concrete panel structures. The structures handled by ORCHID are general three-dimensional structures consisting of flat panels that can be arbitrarily oriented in space. Each panel can act in pure membrane or in combined membrane-bending action. ORCHID is designed to provide practicing engineers with an effective tool for reinforcement dimensioning. The interactive program performs all necessary calculations and provides the engineer with all required information, while the major decisions are left to him (chapter 6). As the complexity of the structural system grows, the method becomes more effective and useful. For greater clarity, however, simple examples are presented (chapter 7).
Zusammenfassung

Eine neue Methode für die Bemessung der Bewehrung von räumlichen, aus ebenen Teilen bestehenden Stahlbetontragwerken aufgrund von Tragsicherheitsanforderungen wird vorgeschlagen. Ziel der Arbeit ist die Schaffung eines Werkzeuges, das die Vorteile der heutigen Methoden berücksichtigt und ihre Nachteile möglichst vermeidet (Kapitel 1). Die Methode der Finiten Elemente, Plastizitätstheorie und lineare Programmierung werden dafür eingesetzt (Kapitel 2). Das vorgeschlagene Verfahren, das sowohl für die Bemessung als auch für die Traglastermittlung anwendbar ist, basiert auf zwei Grundkonzepten.


Das zweite neue Konzept betrifft die Bewehrungsoptimierung. Basierend auf dem Einspielsatz und dem unteren Grenzwert der Plastizitätstheorie werden homogene Eigenspannungszzustände mit der linearen elastischen Lösung einer Finiten Element Berechnung überlagert. Diese fiktiven Umlagerungsfallen werden nicht etwa aus einer nichtlinearen inkrementellen Analyse bestimmt sondern frei, d.h. aufgrund von Optimierungskriterien gewählt. Ihre Verteilung und Intensität werden mittels linearer Programmierung ermittelt, was zu einer rationelleren Bewehrungsverteilung mit weniger Gewicht führt (Kapitel 5).

1 Problem Definition and Scope Formulation

This chapter aims at defining the problem and explaining the solution proposed to tackle it. In the first section, a short review of the status of reinforcement dimensioning today is made. Three well-known approaches are discussed and their advantages and shortcomings mentioned. In the second part, the scope of the present work is explained and its main goals formulated. These try to exploit the advantages of today’s reinforcement design as much as possible and alleviate some of their deficiencies. The chapter ends with a classification of the most important assumptions and limitations of the suggested approach after which the topics to be discussed in the subsequent chapters are summarized.

1.1 Reinforcement dimensioning today

This section classifies the main approaches taken today for dimensioning reinforced concrete structures. After a brief explanation of the methods, the major advantages and disadvantages of each of them are discussed. The section, however, does not review the methods based on the upper bound theorem, notably Johansen’s yield line theory, and only addresses those primarily based on equilibrium principles.

1.1.1 Dimensioning based on linear elastic stress analysis

In this approach, the classical linear elastic theory is applied to uncracked unreinforced homogeneous concrete to determine the stresses at different points within the structure. Today, this is usually done by the application of standard linear elastic finite element analysis programs. This elastic stress distribution forms the basis of steel dimensioning. Some reinforcement fields are chosen and the amount of steel in each field is found such that, on the assumption of the usual utilization of the bars up to the design values, the total reinforcement provided at each point can resist the elastic stresses.

The advantages of the elastic method can be classified as follows:

Generality and ease of use: The generality of the well-known finite element approach and the availability of corresponding computer programs, as well as the ease in applying this method are the reasons for it becoming widespread among reinforced concrete designers.

Easy handling of multiple load cases: The method can easily accommodate multiple load cases which is of great practical importance, that would otherwise be extremely cumbersome for non-computerized methods to handle.
Elastic deformations available: The application of the finite element method provides the elastic deformation of the structure, which is very valuable since it furnishes the necessary information for fulfilling the serviceability requirements.

Suitable for crack propagation control: This method results in the provision of more reinforcement in the areas where the formation of cracks first occurs. It can, therefore, be useful for accommodating the crack formation and propagation.

The drawbacks of the elastic method are:

No evaluation of the ultimate load: An elastic analysis gives no indication of the ultimate load-carrying capacity of the structure.

Stress peaks at singularity points: A purely elastic solution results in high stress values at points of singularity which are not representative of the physical structural behavior.

Stress analysis as a goal: Here, a precise stress analysis is the goal, but for a structural concrete designer, dimensioning is the main concern. His main interest is in the evaluation of a reinforcement distribution that optimally carries the applied loads.

1.1.2 Truss model, stress field and strip methods

In the truss model approach, the continuum is modelled by a truss system with the force distribution in the truss elements being expected to represent the force distribution in the continuum accurately enough for design purposes. In the discontinuous stress field approach, the continuum is discretized by a set of stress fields, such that in each field, the stresses are assumed to be constant. The distribution of these constant stresses must show how the applied forces are transferred by the structure to its supports. In the strip method, the structure is simplified into a set of strips (beams) carrying the load in their respective directions. The reinforcement design then follows.

The strengths of these methods are:

Dimensioning is the main concern: The idea here is not to investigate the actual structural behavior under certain loads, but rather to create an equilibrium system which is capable of carrying the external loads.

Safe design based on lower bound principles: These approaches are based on the lower bound theorem of plasticity theory which is known to provide a lower bound to the collapse load and therefore result in a safe design.

But they show the following disadvantages:

Too crude a model: With the advance of numerical methods and powerful computers, these methods present a somewhat crude model to handle the design of steel for complex concrete structures.
1.1 Reinforcement dimensioning today

Not a general method: These methods were originally conceived as hand methods which should enable the engineer to make a first evaluation of structural behavior in the early stages of his design. Attempts have been made to extend and implement the truss model approach [Schlaich 89] and the stress field method [Hajdin 90] into computer programs. The basic problem involved is the realization of a suitable system which can model the structure adequately while requiring little help from the engineer to accomplish this task. Also, they are not general and only applicable to a specific type of simple problem.

Elastic deformations not available: These methods do not provide any information on deformations which are of importance for fulfilling serviceability requirements.

1.1.3 Elasto-plastic nonlinear analysis

In a step-by-step nonlinear analysis, the variation of the applied loads on the structure is specified by the load time history and the structural behavior is investigated in a series of short steps so that the stress distribution and redistribution can be followed continuously. Indeed, the development and improvement of the methods related to the elasto-plastic nonlinear analysis has long been and continues to be a favorite topic for research among academics.

The most important feature of this method can therefore be stated as enabling a

Precise investigation of structural behavior: Clever and rather complex material models for concrete behavior have been developed and implemented into computer programs.

In reinforced concrete dimensioning, there has been very little interest in nonlinear analysis and at most, it is restricted to the analysis of structural frames to study the formation of plastic hinges especially for dynamic loading as in the case of earthquakes. The reasons are:

Reinforcement must be known in advance: The amount of reinforcement must be known in advance. The method is therefore more suitable for a thorough investigation of an existing design rather than for a new design.

Too complex for practicing engineers: For two-dimensional reinforced concrete continua, a complete nonlinear analysis is almost never performed in practice. The fact is that its complexity and the lack of know-how to correctly model the structure and properly interpret the results has discouraged most practice-oriented civil engineers from making use of these research advances. Furthermore, assuming that initial stresses are zero (or at least, known) may be incorrect for large structures.

Not necessary for conventional applications: In everyday designs, the effort spent on a nonlinear analysis outweighs the amount of useful information the engineer expects to get and indeed regards as necessary for the task of reinforcement design.
1.2 Scope of this work

In view of the deficiencies outlined above, the lack of a suitable design approach becomes more apparent as the complexity of the structural system grows from frames to wall elements, to slabs and finally to general three-dimensional structures. At one end of the spectrum, reinforcement of frame structures is well-known, is mainly governed by code specifications and there is little room for altering the steel distribution. For general 3D structures which behave as a continuum with significant interaction between different parts, great advantages can be gained if the design engineer is able to easily examine a large number of various steel layouts. For the complicated structures consisting of flat panels which are handled here, the practice before the spread of the finite element method was to somehow extract a panel from the system and model it with a series of line loads at the edges where the considered panel was connected to the rest of the structure. The designer would then proceed with the dimensioning of this panel as when a single wall or slab element is to be reinforced. He would then take care of the reinforcement at the edges of panel connections. With the development of general purpose computer programs, the problem of the analysis of such structures was overcome. However, the gap between the tasks of analysis and design was further widened.

![Figure 1.1 Examples of spatial panel structures](image)

Considering all these problems for an integrated analysis and reinforcement design, a project was initiated in 1992 for the development of a practice-oriented approach and the corresponding computer programs for the reinforcement dimensioning of concrete structures [Anderheggen, et al. 94]. It has resulted in the development of three separate computer programs for the reinforcement design of walls [Despot 95], flat slabs [Steffen 96] and the general three-dimensional structures reported here. This project tries to introduce a system which should inherit the advantages of the various existing methods while trying to alleviate some of their deficiencies. The main characteristics of such a system are:

- The finite element method is applied to make the approach general, furnish the elastic structural response and to easily handle multiple load cases.
1.2 Scope of this work

- Dimensioning is the main concern and not a precise stress analysis.
- The elastic solution is modified taking advantage of the concepts of plasticity theory.
- It represents a compromise between the sophistication and complexity of the model and the ease of use of the corresponding program.

1.2.1 Extension of finite element concept from analysis to design

Analysis and design have been, so far, mostly perceived as two distinct activities with very little in common. The function of analysis is to determine displacements and stresses within the structure. Based on these values, design is concerned with the provision of adequate strength to resist these stresses. Unlike the mechanical engineer, the civil engineer is basically less interested in knowing the exact stress distributions within his structure. This is mainly due to the big uncertainties involved in the construction process, material behavior and future loadings which makes the task of a thorough investigation both unnecessary and indeed impossible. His primary objective, therefore, is to design a system that is capable of carrying in one way or another the external loads. This fact together with the attractiveness of the finite element method resulted in the idea of moving away from the usual viewpoint and extending the concepts of finite element discretization to reinforced concrete design.

In the context of the finite element model, we know that the resulting stresses calculated at the common node of adjacent elements are discontinuous and thus not in equilibrium with one another despite the fact that they belong to the very same node. To overcome this situation, which only does not arise in the special case of beam elements, programmers apply arbitrary averaging techniques to end up with representative stress values at each node of the finite element model. Having obtained these values, the reinforced concrete designer applies the well-known yield conditions formulated for infinitesimal $dx$-$dy$ element to check whether the provided concrete can resist the compressive force resultants or determine the reinforcement needed to carry the tensile forces.

In the new paradigm of finite element-based dimensioning, a finite element is regarded as a dimensioning unit which must be capable of carrying the loads applied to it. These loads are the element nodal forces which are transferred from each element to its neighboring elements. The nodal forces are statical equivalents of the forces transmitted across the boundaries between the elements and are in exact equilibrium with each other and are easily obtainable from the nodal displacements. In this way, not only the conventional stress calculations are avoided but also the idea of the finite element method is extended from analysis to design in a natural way. Of course, as a result of this approach, the well-established yield conditions formulated for an infinitesimal element must now be derived for the finite element. This represents a major task in this approach.
1. Problem Definition and Scope Formulation

1.2.2 Reinforcement rationalization and optimization

In section 1.1.1, it was shown that the most wide-spread approach currently used for reinforcement dimensioning is to perform a linear elastic analysis and then design the required reinforcement assuming that the reinforcement is to be fully utilized up to its tensile yield strength. In most of the programs used in practice, the computer provides a set of so-called isoline plots of stresses within the structure. On the basis of this information and other practical considerations, the engineer decides what kind of reinforcement fields he wants to introduce. The steel content provided in the fields must be such that the resistance provided at a fully plastic condition is everywhere equal to or greater than the stress resultant considered. From experience with the finite element analysis, however, we know that the linear elastic solution can, in the so-called singular points, result in very high stress concentrations which are a consequence of the assumed linear elastic behavior. These singular points are usually the points of application of heavy single loads, column supports and on the structure’s boundaries. On the basis of his experience, the design engineer usually tries to flatten out these stress peaks by applying some non-standard methods which result in providing some additional reinforcement in the neighboring areas of such critical regions. In doing so, he resorts to a well-understood concept of reinforced concrete behavior beyond its elastic limit as the formation of cracks in the concrete and the yielding of steel results in a stress redistribution within the structure.

Considering these facts and based on the lower bound (or static) theorem of plasticity, a new approach is presented which fully utilizes this stress redistribution within the structure. According to the lower bound theorem, the superposition of self-equilibrating states of stress on any structure in equilibrium under the external loads does not alter the load-carrying capacity of the structure. This paves the way to optimization. Exploiting this principle, one can introduce appropriate fictitious self-equilibrating states of stress, which superimposed on the linear elastic stress distribution result in a more favourable reinforcement arrangement.

The goal here is both rationalization as well as optimization of the reinforcement distribution. Rationalization requires a direct and active involvement of the engineer in the design process. Since the choice of steel fields can be influenced by many factors and represents a major decision, it is one which should be directly made by the designer. Optimization means that under the specific considerations, the structure must be able to safely carry its loads while minimizing a specific cost function. Although the costs of the construction material often represent a rather small fraction of the total costs of the construction, the reinforcement can well be a good portion of the construction material costs. Also since it is a good quantifiable parameter, the optimization problem will be formulated to minimize the weight of the total steel necessary. The task of optimization, therefore, is one which can be well formulated in a computer program.
1.2.3 Development of the corresponding computer program

As a part of this dissertation, program ORCHID (standing for: Optimum Reinforced Concrete Highly Interactive Dimensioning) was developed for the optimum reinforcement design of 3D concrete panel structures. The structure can consist of several concrete panels arbitrarily oriented in space where each panel can be specified to behave in a pure membrane state or in a combination of membrane and bending action. The panels can be eccentrically connected to one another accommodating any thickness differences or other constructional details. There are two major issues considered in the development of ORCHID. The first goal is of course to develop a tool enabling us to examine the merit, applicability, problems and possible solutions for the two basic new ideas of this work. As one can see later, the present optimization approach can lead to results which are all safe but quite different. In other words, as the final decision-maker, the design engineer is capable of influencing the stress redistribution and direct the program the way to go. We regard this as one of the advantages of the present approach and indeed what design is all about. To take full advantage of this opportunity, it is indispensable to have a very user-friendly environment with a suitable graphical user interface, with which the designer can easily and effectively interact with the computer. Beyond this, the second aim is to realize a system for computer-aided reinforcement dimensioning in which the program performs all necessary calculations and provides appropriate data for helping the engineer to make the major decisions. It is believed that a rational final design can only be accomplished by such an interaction between the engineer and the computer. Program ORCHID represents an attempt in this direction.

1.3 Assumptions and limitations

The scope of this work was formulated in section 1.2. Based on the previously mentioned requirement that a compromise should be made between the complexity of the model and the ease of practical application for the design engineer, a series of simplifying assumptions should be made, the most fundamental of which will be summarized here.

- **Small displacements and strains:** This enables the formulation of equilibrium equations on the undeformed structure. It also allows the use of principle of virtual work, which is the key to proving the limit theorems.

- **No time-dependent effects:** The loads are assumed to act statically and no dynamic effects are included. Also, time-dependent aspects of concrete behavior like creep and shrinkage are not considered.

- **Basic assumptions of simple plasticity theory apply:** Limit theorems of plasticity theory are applied here for the task of optimum design. Consequently, all underlying assumptions which
enable one to apply this method, e.g. perfectly plastic material behavior without strain hardening or softening and the unimportance of the time history of applied forces are assumed.

- **Underreinforced structures:** Since the most important assumption for the applicability of plasticity theory is the capacity of the structure for plastic deformation and stress redistribution, the method is applicable only for those types of structures for which failure under short term loading is primarily governed by the ductile nature of the mild steel and not the brittle behavior of concrete. As a result, only underreinforced structures are dealt with here.

- **Ultimate load design:** According to code specifications, a structure must be designed for the two requirements of serviceability and strength. The objective of the work presented here is to address solely the issue of design for strength requirements. The serviceability conditions, which, depending on the type of structure and applied loading, may govern the design, are not dealt with here. However, useful data like structural displacements for the linear elastic solution that can be of great value for checking serviceability are provided by ORCHID.

- **No treatment of structural detailing:** Specific issues like detailing at supports, reinforcement configuration at the edges of neighboring panels, anchorage lengths, etc. are not dealt with. These vary in different codes and are best considered on a case by case basis and with the judgement of the engineer and are unsuitable for automation within a computer program.

### 1.4 Overview

This project attempts to integrate the concepts of the finite element method, plasticity theory and linear programming, in a unified manner for the task of reinforcement design. Chapter 2 provides a brief review of the basic principles underlying these three topics. In chapter 3, the linearized yield conditions are derived for an infinitesimal element in pure membrane, in bending and in combined membrane-bending action by making use of a sandwich model. Chapter 4 deals with the derivation of the linearized yield conditions of a finite element in membrane, bending and combined membrane-bending based on its generalized nodal forces. In chapter 5, the optimization of steel distribution by the introduction of fictitious plastic strains is dealt with. The formation of the linear program for reinforcement evaluation for the first linear elastic solution is discussed. The self-equilibrating stress states under membrane and bending action are then presented and the methods for the expansion of the tableau explained. The extension of the method from reinforcement minimization towards ultimate load analysis is then discussed. Chapter 6 is devoted to the explanation of program ORCHID, where the general aspects of program functionality and interface together with the free formulation element model used and special features like the numerical treatment of eccentric panel connections are presented. Simple examples to demonstrate various aspects of the method using ORCHID are shown in chapter 7. Concluding remarks are given in chapter 8.
2 Basics of the Finite Element Method, Plasticity Theory and Linear Programming

Therefore, O students, study mathematics
and do not build without foundations.

Leonardo da Vinci

The new approach presented here tries to integrate in a unified manner the areas of finite element analysis, plasticity theory and linear programming for the task of optimum reinforcement design. In this chapter, the fundamental concepts of these three topics, as related to this work, are presented.

2.1 The finite element method

The structures to be dealt with in this work are assumed to be made up of panels in membrane and bending states. This section reviews the basic concepts which lead to the derivation of the differential equations modelling the behavior of such panels. It then briefly discusses the underlying concepts of the finite element method which are used to discretize the continuum and numerically approximate structural behavior under the applied loads. Linear elasticity is assumed.

2.1.1 Modelling membrane panels

An ideal membrane element is one, in which the applied forces act in the middle plane of the element. In our application, panels in a pure membrane state represent a plane stress state in an \(x-y\) plane with the stresses \(\sigma_z\), \(\tau_{yz}\) and \(\tau_{zx}\) equal to zero. Structural behavior under the applied loads is described by the components of the displacement vector \(u_m\) in the panel’s local \(x\) and \(y\) Cartesian directions and of the membrane strain and stress vectors \(\epsilon_m\) and \(\sigma_m\) defined by

\[
\begin{align*}
\epsilon_m &= \begin{cases}
\epsilon_{xx} \\
\epsilon_{yy} \\
2\epsilon_{xy}
\end{cases} = \begin{cases}
\epsilon_x \\
\epsilon_y \\
\gamma
\end{cases} \\
\sigma_m &= \begin{cases}
n_{xx} \\
n_{yy} \\
n_{xy}
\end{cases} = \begin{cases}
n_x \\
n_y \\
n_{xy}
\end{cases}
\end{align*}
\]

(2.1)

where, with the assumption of constant stress distribution through the thickness \(h\), the sectional forces \(n_x\), \(n_y\) and \(n_{xy}\) are taken as generalized stresses and are defined by

\[
\begin{align*}
n_x &= \int_{-h/2}^{h/2} \sigma_x \, dz = \sigma_x \, h \\
n_y &= \int_{-h/2}^{h/2} \sigma_y \, dz = \sigma_y \, h \\
n_{xy} &= \int_{-h/2}^{h/2} \tau_{xy} \, dz = \tau_{xy} \, h
\end{align*}
\]

(2.2)
2. Basics of the Finite Element Method, Plasticity Theory and Linear Programming

With the assumption of small displacements and small strains, Fig. 2.1 establishes the relationship between the components of the displacement and strain vectors as

\[
\begin{align*}
\epsilon_m &= \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix} = A_m \mathbf{u}_m
\end{align*}
\]

(2.3)

with \(A\) being the operator matrix. In the isotropic plane stress case, the generalized stress and strain vectors specified in (2.1) are related through the elasticity matrix \(D_m\) where \(E > 0\) and \(0 \leq \nu \leq 0.5\) are Young’s modulus and Poisson’s ratio, respectively. \(h\) is the element thickness and \(\mathbf{e}_{m0}\) is the given vector of initial strains.

\[
\sigma_m = D_m (\epsilon_m - \mathbf{e}_{m0}) h \quad \text{with} \quad D_m = \frac{E}{(1 - \nu^2)} \begin{bmatrix} 1 & \nu & . \\ \nu & 1 & . \\ . & . & \frac{1 - \nu}{2} \end{bmatrix}
\]

(2.4)

The forces acting on an infinitesimal membrane element are shown in Fig. 2.2. The formulation of a moment equilibrium condition on this rectangular \(dx-dy\) membrane element gives

\[
n_{xy} = n_{yx}
\]

(2.5)

With the body forces denoted as \(f_B\), force equilibrium in the \(x\) and \(y\) directions results in the following equilibrium differential equations relating the normal and shear forces to the distributed external load components:

\[
\begin{align*}
\frac{\partial n_x}{\partial x} + \frac{\partial n_{xy}}{\partial y} + f_{B_x} &= 0 \\
\frac{\partial n_y}{\partial y} + \frac{\partial n_{xy}}{\partial x} + f_{B_y} &= 0
\end{align*}
\]

or:

\[
A_m^T \sigma_m + f_B = 0
\]

(2.6)
2.1 The finite element method

For a membrane element subjected to the three stress components as in Fig. 2.3, a section can be found in which the resultant shear force vanishes \( n_{nt} = n_{nt} = 0 \). These normal stresses are then referred to as the principal stresses.

The corresponding angle is found from

\[
\psi_0 = \frac{1}{2} \tan^{-1} \left( \frac{2n_{xy}}{n_x - n_y} \right) \tag{2.7}
\]

while the values of the principal stresses are given by

\[
n_{nt} = \frac{1}{2} \left( n_x + n_y \pm \sqrt{(n_x - n_y)^2 + 4(n_{xy})^2} \right) \tag{2.8}
\]
2.1.2 Modelling bending panels

The major distinction between membrane and bending elements is that in pure membrane action, the external forces act in the element middle plane, whereas in pure bending, they are perpendicular to this plane. The basic proposition in plate bending analyses is that the structure is thin in one dimension so that the following assumptions can be made:

- The stress through the thickness (i.e. perpendicular to the mid-surface) of the plate is zero.
- Material particles that are originally on a straight line perpendicular to the mid-surface of the plate remain on a straight line during deformations.

![Displacement components in plate bending](image)

**Figure 2.4 Displacement components in plate bending**

The displacement of a node on the plate’s mid-surface under the external forces can be described by its translation $w$ along the $z$ axis perpendicular to the mid-surface and the two rotations about the middle plane’s coordinate axes $x$ and $y$ (Fig. 2.4). The three components of the displacement of any point are then given in terms of these parameters by

$$u_b(x,y,z) = \begin{bmatrix} u \\ v \\ w \end{bmatrix} = \begin{bmatrix} z \theta_y(x,y) \\ -z \theta_x(x,y) \\ w(x,y) \end{bmatrix}$$

(2.9)

The components of the bending displacement vector $u_b$ in the $x$ and $y$ directions and the strain and stress vectors $\epsilon_b$ and $\sigma_b$ are

$$u_b = \begin{bmatrix} \theta_x \\ \theta_y \\ w \end{bmatrix} \quad \epsilon_b = \begin{bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{bmatrix} \quad \sigma_b = \begin{bmatrix} m_x \\ m_y \\ m_{xy} \\ q_x \\ q_y \end{bmatrix}$$

(2.10)

The two bending moments $m_x$ and $m_y$ and twisting moment $m_{xy}$ for a unit length leading to the generalized stress vector $\sigma_b$ are defined by the integrals
2.1 The finite element method

\[ m_x = \int_{-h/2}^{h/2} \sigma_x z \, dz \quad m_y = \int_{-h/2}^{h/2} \sigma_y z \, dz \quad m_{xy} = \int_{-h/2}^{h/2} \tau_{xy} z \, dz \]  

(2.11)

whereas the shear forces \( q_x \) and \( q_y \) are given by

\[ q_x = \int_{-h/2}^{h/2} \tau_{xz} \, dz \quad q_y = \int_{-h/2}^{h/2} \tau_{yz} \, dz \]  

(2.12)

Using the operator matrix \( A_b \) for bending, the relationship between the strain vector \( \varepsilon_b \) and the displacement vector \( u_b \) is given by

\[
\varepsilon_b = \begin{pmatrix}
\gamma_x \\
\gamma_y \\
\gamma_{xy}
\end{pmatrix} = \begin{pmatrix}
-\frac{\partial}{\partial x} & \cdot & \cdot \\
\frac{\partial}{\partial y} & -\frac{\partial}{\partial y} & \cdot \\
\frac{\partial}{\partial x} & \frac{\partial}{\partial y} & -\frac{\partial}{\partial y}
\end{pmatrix} \begin{pmatrix}
\theta_x \\
\theta_y \\
w
\end{pmatrix} = A_b u_b
\]  

(2.13)

The forces acting on an infinitesimal bending element are shown in Fig. 2.5. The relationship between the shear forces \( q_x \) and \( q_y \) and the bending and twisting moments \( m_x, m_y \) and \( m_{xy} \) is given by the equilibrium equations

\[ q_x = \frac{\partial m_x}{\partial x} + \frac{\partial m_{xy}}{\partial y} \quad q_y = \frac{\partial m_y}{\partial y} + \frac{\partial m_{xy}}{\partial x} \]  

(2.14)

\textbf{Figure 2.5 Stress resultants on an infinitesimal plate bending element}
Formulating force equilibrium in z direction gives
\[
\frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} = \frac{\partial^2 m_x}{\partial x^2} + 2 \frac{\partial^2 m_{xy}}{\partial x \partial y} + \frac{\partial^2 m_y}{\partial y^2} = f_{B_z}\tag{2.15}
\]

In the Kirchhoff plate theory, the additional assumption is made that a straight line originally perpendicular to the mid-surface remains perpendicular to it during deformation, i.e. shear deformations are neglected. The rotations \(\theta_x\) and \(\theta_y\), therefore, do not represent independent variables but are functions of the displacement component \(w\)
\[
\theta_x = \frac{\partial w}{\partial y} \quad \theta_y = -\frac{\partial w}{\partial x}\tag{2.16}
\]
The displacement vector \(\mathbf{u}_b\), strain vector \(\mathbf{\varepsilon}_b\) and stress vector \(\mathbf{\sigma}_b\) of (2.10) are then simplified to
\[
\mathbf{u}_b = \{w\} \quad \mathbf{\varepsilon}_b = \begin{bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{bmatrix} \quad \mathbf{\sigma}_b = \begin{bmatrix} m_x \\ m_y \\ m_{xy} \end{bmatrix}\tag{2.17}
\]
Similarly, (2.13) reduces to
\[
\mathbf{\varepsilon}_b = \begin{bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{bmatrix} = \begin{bmatrix} -\frac{\partial \theta_y}{\partial x} \\ \frac{\partial \theta_x}{\partial y} \\ \frac{\partial \theta_x}{\partial x} - \frac{\partial \theta_y}{\partial y} \end{bmatrix} = \begin{bmatrix} \frac{\partial^2}{\partial x^2} \\ \frac{\partial^2}{\partial y^2} \\ 2 \frac{\partial^2}{\partial x \partial y} \end{bmatrix}\{w\} = \mathbf{A}_b \mathbf{u}_b\tag{2.18}
\]
In the isotropic case, the stress and strain vectors specified in (2.17) are related through the elasticity matrix \(\mathbf{D}_b\)
\[
\mathbf{\sigma}_b = \mathbf{D}_b (\mathbf{\varepsilon}_b - \mathbf{\varepsilon}_{b0}) \quad \text{with} \quad \mathbf{D}_b = \frac{E h^3}{12(1-\nu^2)} \begin{bmatrix} 1 & \nu & \cdot \\ \nu & 1 & \cdot \\ \cdot & \cdot & \frac{1-\nu}{2} \end{bmatrix}\tag{2.19}
\]
While \(m_x\), \(m_y\) and \(m_{xy}\) are viewed as generalized stresses for the classical Kirchhoff plate theory, \(q_x\) and \(q_y\) are regarded as the generalized reactions.
Substituting (2.18) and (2.19) in (2.15) and assuming a constant stiffness, i.e. constant \(E\), \(h\) and \(\nu\), gives the well-known differential equation for isotropic plates of constant thickness as
\[
\frac{\partial^4 w}{\partial x^4} + 2 \frac{\partial^4 w}{\partial x^2 \partial y^2} + \frac{\partial^4 w}{\partial y^4} = \frac{12 (1-\nu^2)}{E h^3} f_{B_z}\tag{2.20}
\]
Similar to the membrane element in which principal stresses were defined, principal moments can also be defined for a bending element. In a section with the principal moments, the twisting moment $m_w$ is null. The determination of the angle of the section and the values of principal moments follows the same description given in Fig. 2.3 and leads to the same results given by (2.7) and (2.8), if the normal stress $n$ is everywhere substituted by the moment $m$.

In bending elements, the examination of shear trajectories is a good description of the way the applied forces are transferred from their points of application to the supporting boundaries. Equivalent shear stresses for a coordinate system rotated by an angle $\varphi$ are shown in Fig. 2.6.

![Figure 2.6 Shear stress transformation in a bending element](image)

From Fig. 2.6, it can be found that the principal shear stress $q_0$, in which $q_x = 0$, corresponds to an angle

$$\varphi_0 = \tan^{-1} \left( \frac{q_y}{q_x} \right)$$

(2.21)

and equals to

$$q_0 = \sqrt{q_x^2 + q_y^2}$$

(2.22)

### 2.1.3 Finite element discretization

The basic concept in the finite element method is that the continuum is idealized as an assemblage of discrete structural elements which are interconnected at nodal points on the element boundaries. The element stiffness matrices corresponding to the global degrees of freedom of the structural idealization are calculated and the total stiffness matrix is formed by assembling the element stiffness matrices. The solution of the equilibrium equations of the assemblage of elements yields the nodal displacements which are then used to calculate element stresses.

In solving the differential equations of equilibrium, an exact solution is obtained if all three requirements of stress equilibrium, kinematic compatibility and material constitutive law are fulfilled. In beam analysis, it is possible to derive the exact element matrices (exact for beam theory) but for two and three dimensional continua, finding an exact solution is impossible.
except for some very simple cases. As a result, approximate numerical approaches such as the finite element method are applied. In this approach, interpolation functions usually in the form of polynomials are used which approximate the actual displacements. Although global equilibrium is always satisfied, these approximate interpolation functions result in the violation of the differential equations of equilibrium both within each element and between the elements. The error reduces as the finite element idealization of the continuum is refined.

The finite element discretization is briefly presented here for a general panel in space. For the specific type of structural element in membrane and bending which are dealt with here, the corresponding parameters should of course be applied.

The equilibrium of the three-dimensional body in space is considered. The external forces acting on the body are surface tractions $f_S$, body forces $f_B$ and concentrated forces $F_i$. These forces have in general three components corresponding to the three coordinate axes as shown in Fig. 2.7. The displacements from the unloaded configuration for all nodes are stored in vector $\mathbf{A}$. The strain components corresponding to $\mathbf{A}$ are given by the vector $\mathbf{e}$, the stresses corresponding to $\mathbf{e}$ by the vector $\mathbf{\sigma}$.

![Figure 2.7 General three-dimensional body in equilibrium](image)

In order to calculate the response of the body to the applied forces, one method is to solve the governing differential equations of equilibrium for appropriate boundary and compatibility conditions. An equivalent approach, which is employed in the finite element method, is to express the equilibrium of the body by means of the virtual work principle. This principle states that the equilibrium of the body requires that for any compatible, small virtual displacements which satisfy the kinematic continuity and boundary conditions, the total internal virtual work is equal to the total external virtual work, i.e.
\[
\int_V \delta \epsilon^T \sigma \, dV = \int_V \delta A^T f_B \, dV + \int_S \delta A^T_S f_S \, dS + \sum_{i=1}^{NN} \delta A^T_i F_i \tag{2.23}
\]

with \( NN \) being the total number of nodes, the integrals being evaluated over the volumes and surfaces of the elements. The internal virtual work given on the left hand side of (2.23) is equal to the actual stresses \( \sigma \) undergoing the virtual strains \( \delta \epsilon \) corresponding to the virtual displacements \( \delta A \). The external work given on the right hand side of (2.23) is equal to the actual forces \( f_B, f_S \) and \( F_i \) undergoing the virtual displacements \( \delta A \). In (2.23), any compatible set \( \delta A, \delta \epsilon \) may be substituted.

The displacements \( \mathbf{u}^e \) within an element \( e \) measured in a local coordinate system \( (x, y, z) \), to be chosen conveniently, are assumed to be a function of the displacements \( \mathbf{a}^e \) of the element’s nodal points, the two vectors being related through the displacement interpolation matrix \( \mathbf{H}^e \)

\[
\mathbf{u}^e(x, y, z) = \mathbf{H}^e(x, y, z) \mathbf{a}^e \tag{2.24}
\]

Formally, the displacements of the element’s nodes \( \mathbf{a}^e \) can be extracted from the global displacement vector \( \mathbf{A} \) using (2.25) where the topology matrix \( \mathbf{J}^e \) is defined by \( J^e_{ij} = 1 \) if the element’s degree of freedom \( a^e_i \) corresponds to the global degree of freedom \( A_j \) and \( J^e_{ij} = 0 \) if otherwise.

\[
\mathbf{a}^e = \mathbf{J}^e \mathbf{A} \tag{2.25}
\]

It should be noted that the introduction of the matrix \( \mathbf{J}^e \) is solely for the sake of consistence in the presentation of formulae here. The tasks of assemblage and extraction of element matrices into and from the global matrices is, of course, accomplished by other computationally more efficient means.

The element strains are calculated from the element displacements by

\[
\epsilon^e = \mathbf{A} \mathbf{u}^e = \mathbf{A} \mathbf{H}^e \mathbf{a}^e = \mathbf{B}^e \mathbf{a}^e \tag{2.26}
\]

with the operator matrix \( \mathbf{A} \) and the strain-displacement matrix \( \mathbf{B}^e \). The stresses in a finite element are related to the element strains and the element initial strains, using

\[
\sigma^e = \mathbf{D}^e (\epsilon^e - \epsilon^e_0) = \mathbf{D}^e \epsilon^e - \sigma^e_{i0} \quad \text{with} \quad \sigma^e_{i0} = \mathbf{D}^e \epsilon^e_{i0} \tag{2.27}
\]

with \( \mathbf{D}^e \) and \( \sigma^e_{i0} \) being the element elasticity matrix and initial stresses, respectively. By rewriting (2.23) as a sum of integrations over the volume and areas of all finite elements and substituting (2.24) to (2.27) in (2.23), we find the equilibrium equations that correspond to the nodal point displacements of the assemblage of finite elements as
\[
\delta A^T \left( \sum_{e=1}^{NE} f^e \left( \int_{V^e} B^e^T D^e B^e dV^e \right) J^e \right) A = \\
\delta A^T \left( \sum_{e=1}^{NE} f^e \int_{V^e} H^e f_B^e dV^e + \sum_{e=1}^{NE} \int_{S^e} H^e f_S^e dS^e - \sum_{e=1}^{NE} \int_{V^e} B^e^T \sigma^e_{e0} dV^e + F_C \right) \tag{2.28}
\]

The surface displacement interpolation matrices \(H^e_S\) are obtained from the volume displacement interpolation matrices \(H^e\) in (2.24) by substituting the element surface coordinates. Equation (2.28) can therefore be summarized as an equation system

\[
KA = F \quad \text{with} \quad F = F_B + F_S - F_{e0} + F_C \tag{2.29}
\]

where the matrix \(K\) is the stiffness matrix of the element assemblage

\[
K = \sum_{e=1}^{NE} f^e K^e J^e \quad \text{with} \quad K^e = \int_{V^e} B^e^T D^e B^e dV^e \tag{2.30}
\]

and the load vector \(F\) includes the effect of the element body forces,

\[
F_B = \sum_{e=1}^{NE} f^e f_B^e \quad \text{with} \quad F_B^e = \int_{V^e} H^e f_B^e dV^e \tag{2.31}
\]

the effect of the element surface forces,

\[
F_S = \sum_{e=1}^{NE} f^e f_S^e \quad \text{with} \quad F_S^e = \int_{S^e} H^e f_S^e dS^e \tag{2.32}
\]

the effect of the element initial stresses

\[
F_{e0} = \sum_{e=1}^{NE} f^e f_{e0} \quad \text{with} \quad F_{e0}^e = \int_{V^e} B^e^T \sigma^e_{e0} dV^e \tag{2.33}
\]

and the concentrated loads \(F_C\).

The element local stiffness matrix \(K^e\) is singular since it contains the rigid body modes. For membrane elements, 3 rigid body modes exist, namely the two translations along the two perpendicular element axes and a rigid body rotation about the axis perpendicular to the element plane. Similarly, 3 rigid body modes characterize bending elements, two rotations about the two axes and a translation in the transverse direction. The rank of a square matrix is defined as the number of linearly independent rows and columns of the matrix. As a result, the rank of the element stiffness matrix is the number of degrees of freedom minus the number of rigid body modes, since for these modes, a vector \(x\) exists where
2.1 The finite element method

\[ K^e x = 0 \quad \text{with} \quad x \neq 0 \]  \hspace{1cm} (2.34)

The global stiffness matrix \( K \) in (2.29) is initially positive semi-definite when it is constructed by the summations specified in (2.30) since the unconstrained structure can undergo rigid body displacements and rotations. This will become positive-definite after the boundary conditions are applied and \( K \) is modified accordingly. The solution of the equation system given by (2.29) gives the displacements of all the nodes. The nodal displacements of each element are then extracted from the displacement vector \( A \) and inserted into (2.26) and (2.27), which furnish the element strains and stresses.

2.1.4 Requirements of an element formulation

As the number of elements is increased, any proper finite element solution should converge to the exact solution of the differential equations that govern the response of the mechanical idealization of the actual physical problem. If the analytical solution of the exact response of the mechanical idealization cannot be obtained, the convergence of the finite element solutions can only be measured by the fact, that all basic kinematic, static and constitutive conditions contained in the mechanical idealization must ultimately (at convergence) be satisfied.

Depending on the specific displacement-based finite element formulation, convergence may be \textit{monotonic} or \textit{non-monotonic}. For monotonic convergence, the elements must be \textit{complete} and \textit{kinematically compatible (or conform)}. The requirement of completeness means that the displacement functions of the element must be able to represent the \textit{rigid body modes} and the \textit{constant strain states}. A rigid body mode is a type of displacement where no strains are developed in the element. The necessity for constant strain states comes from the physical understanding that in the limit of continuous refinement where the element size approaches zero, the strain in each element approaches a constant value and any complex variation of strain within the structure can be approximated. The requirement for compatibility means that the displacements within the elements and across the element boundaries must be continuous. Physically, compatibility assures that no gaps or overlappings occur between elements under the applied loads. Satisfaction of compatibility is not always easy, especially for continuum bending elements. Nevertheless, it has been observed that good solutions can still be obtained even if this condition is violated. Consequently, much research has been directed towards the development of incompatible element formulations such as mixed and hybrid finite element models.

By relaxing the compatibility condition, the problem may arise that although an individual element may be able to represent all constant strain states (i.e. be complete), when the element is used in an assemblage, the incompatibilities between elements may prohibit constant strain states from being represented. The completeness condition is therefore introduced on
an element assemblage to guarantee at least a non-monotonic convergence. As a test to investigate whether an assemblage of incompatible (non-conforming) elements guarantees convergence, the *patch test* has been suggested. In this test, a patch of elements is subjected to the boundary nodal point forces that in an exact analysis correspond to constant strain conditions. If the element strains do actually represent the constant strain conditions, the patch test is passed, i.e. the completeness condition is satisfied for the element assemblage.

Another non-conforming finite element formulation is the so-called *free formulation model* which has been shown to give excellent results and was chosen to be implemented in program ORCHID. More on this element model is presented in chapter 6.

### 2.2 Plasticity theory

The theory of plasticity is concerned with structural behavior beyond the elastic range of certain materials. A complete investigation of elasto-plastic structural behavior can be achieved by a nonlinear analysis where the distribution of forces and the introduction and propagation of plastic deformations can be followed in the form of incremental steps. Clever, but complicated material models are developed and implemented in some general purpose finite element programs which make the task of nonlinear step-by-step analysis possible. Plasticity theory, however, can also be used for *limit analysis*. In this approach, the history of loading and its effects on the real structural behavior over time are not taken into account, the goal being to investigate the structure at the point of collapse.

In this section, some simple models for material behavior are first reviewed and the limit theorems of plasticity theory are formulated. This is followed by a brief discussion of generalized variables and the theory of plastic potential after which the application of the shakedown theorem for handling multiple load cases is presented.

#### 2.2.1 Elastic and perfectly plastic material behavior

In order to investigate the behavior of different materials, engineers use simplified models which describe the mechanical properties in a simple, yet sufficiently accurate, way for engineering purposes. An *ideal material* is therefore assumed. The most familiar material models which we are concerned with are briefly explained here.

The best-known ideal material is the *perfectly elastic material*. This model is characterized by a one-to-one correspondence between stress $\sigma_{ij}$ and strain $\epsilon_{ij}$. The stress depends only on the strain and not on the history of that strain. If the relation is linear

$$\sigma_{ij} = D_{ijkl} \epsilon_{kl}$$

(2.35)
with \( D_{ijkl} \) being material constants, then the material is called linear elastic. Since both \( \sigma_{ij} \) and \( \epsilon_{ij} \) are second order tensors, it follows that \( D_{ijkl} \) is a fourth order tensor with, in general, 81 constants. For a material, whose elastic properties are not a function of direction, 2 independent elastic constants are sufficient to describe the behavior completely. For isotropic materials, these two are the Young’s modulus \( E \) and Poisson’s ratio \( \nu \). In terms of \( E \) and \( \nu \), the stress-strain relationship is given by

\[
\epsilon_{ij} = \frac{1 + \nu}{E} \sigma_{ij} - \frac{\nu}{E} \sigma_{kk} \delta_{ij}
\]

\[
\sigma_{ij} = \frac{E}{1 + \nu} \epsilon_{ij} + \frac{\nu E}{(1 + \nu)(1 - 2\nu)} \epsilon_{kk} \delta_{ij}
\]

in which \( \delta_{ij} \) is a component of the unit matrix \( I_{3\times3} \) with 1 for the diagonal and 0 for the off-diagonal terms. It should be noted that in the above formulae, the summation convention holds. No residual strains remain after the unloading process with zero energy losses.

The linear elastic model accommodates the mechanical behavior only under sufficiently small stresses, i.e. in the elastic range. It does not represent the mechanical properties correctly beyond this limit, e.g. the theory of elasticity predicts infinite stresses at the bottom of a sharp notch. Therefore, the elastic analysis of stresses in the neighborhood of such points of singularity does not furnish results with physical significance.

![Figure 2.8 Perfectly plastic materials in a uniaxial stress state](image)

\( E = \tan \alpha \)

\( \sigma_y \)

\( \epsilon \)

\( \epsilon_e \)

\( \epsilon_p \)

\( \sigma \)

\( \epsilon \)

\( E \rightarrow \infty \)

**Figure 2.8** Perfectly plastic materials in a uniaxial stress state

The second category of materials is called elastic-plastic material. In contrast to the perfectly elastic materials, the mechanical energy needed for deform ing the material is only partly recovered on unloading. The rest of the energy is dissipated into heat. An elastic, perfectly plastic material is characterized by its elasticity modulus \( E \) in the elastic range and the yield limit \( \sigma_y \) beyond which plastic flow under constant stress takes place (Fig. 2.8 a).

The rigid, perfectly plastic material is obtained by increasing the elasticity modulus in an elastic, perfectly plastic material towards infinity \( E \rightarrow \infty \). As a result, the elastic strains vanish \( \epsilon_e \rightarrow 0 \) (Fig. 2.8 b).
2.2.2 Limit theorems of plasticity theory

The limit theorems of plasticity theory provide efficient tools with the help of which the load-carrying capacity of structures can be estimated. The term *load-carrying capacity* was introduced by [Drucker, et al. 51] for an elastic, perfectly plastic continuum as the load intensity at which unlimited (unbounded) deformation could occur if all changes in the geometry of the continuum were neglected. Alternatively, [Hill 51] defined the *yield limit* of a rigid, perfectly plastic continuum as the load intensity at which deformation first becomes possible as the load is increased. From the practical point of view, the two definitions are basically equivalent and one can conclude that if the elastic deformations are small

\[
\text{the yield limit of a rigid, perfectly plastic continuum coincides with the load-carrying capacity of the corresponding elastic, perfectly plastic continuum.}
\]

This conclusion can also be made from the following preliminary theorem, which is used in proving the limit theorems.

*When the limit load is reached and the deformation proceeds under constant load, all stresses remain constant; only plastic (not elastic) increments of strain occur.*

This theorem states that elastic characteristics play no part in the collapse state at the limit load. Thus, the application of the elastic-perfectly plastic stress-strain rate relation becomes the same as the use of the rigid-perfectly plastic stress-strain rate relation for the material.

A *stress field* is said to be *statically admissible* if it satisfies the equilibrium and statical boundary conditions for the actual load. Such a stress field is called *stable* if its stresses nowhere exceed the yield limit.

A *velocity field* is said to be *kinematically admissible* if it satisfies the kinematical constraints of the system. Such a velocity field is called *unstable* for given loads if the work of the external loads is greater than or equal to the internal dissipation work.

The *lower bound (or static) theorem* of limit analysis can be stated as follows:

*In a rigid, perfectly plastic continuum, plastic flow cannot occur under loads for which a stable, statically admissible stress field can be found.*

In physical terms, the lower-bound theorem expresses the ability of the ideal body to adjust itself to carry the applied loads if at all possible.

The *upper bound (or kinematic) theorem* can be formulated as this:

*In a rigid, perfectly plastic continuum, plastic flow must occur under loads for which an unstable, kinematically admissible velocity field can be found.*

In physical terms, the upper bound theorem states that if a possibility of failure exists, the ideal body will collapse.
2.2.3 Generalized variables and the theory of the plastic potential

In order to investigate the behavior of a system, one can consider appropriate generalized variables which accommodate the deformations accurately enough within engineering approximations. A set of linearly independent strain modes $e_i$ are chosen which together should appropriately model the real strain mode $e$ of the continuum. The amplitude of each strain mode $e_i$ is denoted by the scalar parameter $q_i$, so that the investigation of a continuum will be reduced to the consideration of these $n$ generalized strain parameters $q_i$. A linear combination of the adopted $n$ linearly independent strain modes $e_i$ multiplied by their corresponding amplitudes $q_i$ should, in principle, be capable of producing the approximate value of the deformation at any point within the continuum accurately enough within the confines of the simplifying assumptions made.

$$e = \sum_{i=1}^{n} e_i q_i = e q$$  \hspace{1cm} (2.38)

The term generalized stress was introduced to indicate that the variables used to specify the state of stress need not have the dimension of a stress [Prager 55]. When a set of generalized strains $q_1, ..., q_n$ has been chosen as the specification of the state of strain in a continuum, the corresponding generalized stresses $S_1, ..., S_n$ are defined by the condition that

$$dW = S_1 dq_1 + ... + S_n dq_n$$  \hspace{1cm} (2.39)

is the work that stresses $S$ do on infinitesimal increments of strain $dq$.

The concept of generalized variables can be illustrated by a simple example of a beam section under bending and normal forces. A sandwich model is considered where the core carries only shear forces while the thin layers on the two sides withstand axial forces (Fig. 2.9).

![Diagram of a beam section](image)

Figure 2.9 Generalized strains and stresses for a beam section

The state of strain at a cross section of the beam could then be specified by the strains $\varepsilon_b$ and $\varepsilon_t$ at the two flanges. The corresponding stress components would be $\sigma_b$ and $\sigma_t$. Alternatively, the unit extension $\varepsilon$ of the central fibre and the curvature $\chi$ could be taken as the generalized strains. In terms of $\varepsilon_b$ and $\varepsilon_t$, these are defined by
\[ \varepsilon = \frac{1}{2}(\varepsilon_b + \varepsilon_t) \quad \chi = \frac{1}{h}(\varepsilon_b - \varepsilon_t) \]

(2.40)

The corresponding generalized stresses would then be the axial force \( N \) and bending moment \( M \) which, in terms of \( \sigma_b \) and \( \sigma_t \), are given by

\[ N = A(\sigma_b + \sigma_t) \quad M = \frac{Ah}{2}(\sigma_b - \sigma_t) \]

(2.41)

Let the yield limit of an elastic, perfectly plastic continuum be given by a continuously differentiable relation between the generalized stresses \( S_i \)

\[ \phi(S_1, ..., S_n) = 0 \]

(2.42)

and let the sign of the function \( \phi \) be such that states of stress below the yield limit furnish negative values of \( \phi \). If the generalized stresses are interpreted as the coordinates of the stress point in an \( n \)-dimensional stress space, the yield condition (2.42) represents a surface called the yield surface which is postulated to be convex (Fig. 2.10 a). Since the stress-free state with vanishing \( S_i \) components is below the yield limit, the convex yield surface encloses the origin of coordinates. For a point \( S = (S_1, ..., S_n) \) on the yield surface and another neighboring point to \( S \) which is reached by an infinitesimal transition \( dS \) along the yield surface, we have

\[ d\phi = \frac{\partial \phi}{\partial S_1} dS_1 + ... + \frac{\partial \phi}{\partial S_n} dS_n = 0 \]

(2.43)

since both points on the yield surface satisfy (2.42). Since (2.43) holds for all vectors of stress increment that are tangential to the yield surface, the vector with the components \( \left( \frac{\partial \phi}{\partial S_1}, ..., \frac{\partial \phi}{\partial S_n} \right) \) is normal to the yield surface at the considered stress point. Moreover, since \( \phi \) changes from negative to positive values as one crosses the yield surface coming from the interior, this vector has the direction of the exterior normal.

\[ \text{Figure 2.10 Yield surface in the generalized stress space} \]
By assuming the convexity of the yield surface and denoting a non-plastic stress state by $S^\ast$, for any non-plastic stress increment $S^\ast - S$

$$(S^\ast - S) \, dq_p \leq 0$$ (2.44)

The vectors that represent the increments of stress and plastic strain are orthogonal, i.e. the stress increment does not produce any work on the increment of plastic strain.

$$dS_1 \, dq_{p_1} + \ldots + dS_n \, dq_{p_n} = 0$$ (2.45)

A comparison between (2.43) and (2.45) leads to

$$dq_{p_i} = \lambda \, \frac{\partial \Phi}{\partial S_i} \quad i = 1..n \quad \lambda > 0$$ (2.46)

The relationship between the yield condition (2.42) and the flow rule (2.46) was first introduced by [von Mises 28] and is known as the theory of the plastic potential. [Koiter 53] generalized this theory by allowing the yield limit to be specified by a number $m$ of yield functions

$$\Phi_i(S_1, \ldots, S_n) = 0 \quad \ldots \quad \Phi_m(S_1, \ldots, S_n) = 0$$ (2.47)

A state of stress $S_1, \ldots, S_n$ is below the yield limit if all these yield functions have negative values. For a state of stress at the yield limit, at least one yield function vanishes while none has a positive value.

As a special case, Koiter’s generalization can be used to approximate the nonlinear yield surface with a set of linear functions (Fig. 2.10 b). The plastic strain increments can be denoted by a matrix $N$ whose column $j$ gives the components in the stress space of the vector $N_j$ normal to the yield surface $\Phi_j$. If the vector $N_j$ is scaled to unity, then the product $N_j S$ represents the distance of the yield plane $\Phi_j$ from the coordinate origin. The yield condition (2.42) is given by

$$NS - r = 0$$ (2.48)

in which $r$ is the vector of provided resistances for all yield surfaces.

### 2.2.4 Shakedown theorem

If the loads on a structure are not monotonically increased or if it is subjected to a set of loads with varying intensities, the structure may collapse at load factors below the load-carrying capacity factor of any single load case. This may be caused by alternating plastic flow in tension and compression causing an early fatigue failure or by progressive plastic flow which produces plastic deformation increments at each load cycle leading to unacceptably large deformations.
Also, as there is no one-to-one stress-strain correspondence in the presence of plastic strains, the structural response can depend on the load history. Once one accounts for plastic deformation in the structural design process, it seems natural to ensure that for any possible history of loading, its plastic deformation will stabilize, i.e. the structure will shake down.

In principle, the shakedown question can be answered by examining the structural behavior by means of a step-by-step procedure. With no necessity to evaluate the future stress or strain state, a criterion first established by Bleich and later generalized by Melan facilitates the investigation whether the structure can shake down to a state of stress where it can carry further cycles of loading in a purely elastic manner. The shakedown or adaptation criterion may be formulated as follows:

*An elastic, perfectly plastic structure will shake down for given extreme values of the load factors of any number of independent load cases if and only if there exists a self-equilibrating state of stress that nowhere leads to stresses beyond the yield limit when this is superimposed on the elastic responses of the structure to the extreme states of loading.*

It is important to emphasize that the real self-equilibrating stress state is not necessarily sought, rather any (fictitious) self-equilibrating stress state can be used which fulfills the requirements.

The above theorem has a very desirable consequence on the way optimization problems for multiple load cases are handled. Instead of including every individual load case in the optimization process, one can establish the extreme cases for all the load cases and superimpose the self-equilibrating stress state on these extreme states. Although the optimum thus evaluated is likely to differ to some extent from the optimum if all load cases were included, the savings with regard to computational effort, i.e. time and size, will undoubtedly justify this simplification. It is also clear that by doing this, the design will be on the safer side than if each load case were included in the optimization process. This approach is also the one adopted in the present work as discussed in chapter 5.

### 2.3 Linear programming

In this section, the general formulation of a linear program is first presented and the simplex algorithm for the solution of such optimization problems is outlined afterwards.
### 2.3.1 Formulation of a linear program

Linear programming in general consists of the task of minimizing a linear cost function or maximizing a linear profit function under a set of linear constraints. In the following, we will concentrate on the minimization problem. The maximization problem is quite analogous.

In mathematical terms, a set of \( n \) optimization variables \( x_j \) are to be found which minimize the value of a linear objective function \( Z \) defined by

\[
Z(x_j) = c_0 + \sum_{j=1}^{n} c_j x_j
\]  

(2.49)

and satisfy the \( m \) linear constraint inequalities

\[
b_i \leq \sum_{j=1}^{n} a_{ij} x_j \quad i = 1..m
\]  

(2.50)

and possibly the \( n \) non-negativity constraints

\[
x_j \geq 0 \quad j = 1..n
\]  

(2.51)

The cost coefficients \( c_j \) and the constraint coefficients \( b_i \) and \( a_{ij} \) are given constants. The constant coefficient \( c_0 \) represents the initial costs. Any solution satisfying all constraint inequalities (2.50) and (2.51) is called a feasible solution. Among all feasible solutions, the one minimizing the objective function is called an optimum solution.

Each of the linear inequality constraints (2.50) can be changed into an equality by the introduction of a slack variable \( s_i \) for the \( i \)-th constraint. The optimization formulation will then be:

\[
Z(x_j) = c_0 + \sum_{j=1}^{n} c_j x_j
\]  

(2.52)

\[
s_i = -b_i + \sum_{i=1}^{n} a_{ij} x_j \quad i = 1..m
\]  

(2.53)

\[
s_i \geq 0 \quad x_j \geq 0 \quad j = 1..n
\]  

(2.54)

A positive slack variable \( s_i \) shows to what extent one is on the safe side of constraint \( i \), whereas a non-acceptable negative slack variable indicates the degree of violation of the corresponding constraint. A vanishing \( s_i \) says that one is right at the limit of constraint \( i \). One can view (2.53) as an equation system with \( m \) equations and \( m + n \) unknowns \( x_j \) and \( s_i \). In matrix form and in the framework of a tableau, the above problem can be shown in the following figure:
For the initial tableau at the starting stage as shown in Fig. 2.11, the \( m \) dependent variables \( s_i \), which through (2.53) are functions of \( x_j \) are called basic variables. The independent \( x_j \) variables are referred to as non-basic variables. A basic solution is a solution obtained by setting \( n \) non-basic variables equal to zero and finding the corresponding \( m \) basic variables. If the basic solution thereby calculated satisfies all the constraints then we have a basic feasible solution.

The \( x_j \) variables of a linear program with \( n \) unknowns build an \( n \)-dimensional space. As a linear function of the \( x_j \) variables, each constraint represents a hyperplane which all together, form a convex polyhedron encompassing the solution space. Each basic solution is a point on the intersection of \( n \) hyperplanes. The transition from one basic solution to another consists of a set of elementary row operations which are referred to as pivot transformation or, briefly, pivoting. If \( a_{ij} \) is chosen as pivot, then pivoting results in an exchange of the non-basic variable \( x_j \) with the basic variable \( s_i \) so that \( x_j \) becomes a basic variable while \( s_i \) becomes non-basic.

Within this step, the tableau of Fig. 2.12 (a) will be transformed to that of Fig. 2.12 (b).

**Figure 2.11** Formulation of tableau of a general linear program

**Figure 2.12** Tableau modification for one pivoting exchange step
2.3.2 Simplex algorithm

The Simplex algorithm is a well-established procedure for tackling linear programming problems. It is based on the fundamental theorem of linear programming which states that,

*If an optimum feasible solution exists, then there exists a feasible basic solution which is optimum.*

The optimization procedure therefore consists of a series of pivot transformations as described before, until an optimum feasible solution is found.

For the minimization problem, two cases are distinguished here which are of interest to us with regard to the reinforcement optimization discussed in chapter 5. The first case is an exchange step from an unfeasible minimum towards a feasible optimum solution. A natural choice for the starting basic solution for the non-basic variables $x_j$ is the null vector. The selection of the pivot row and column is conducted through a set a rules which is shown in the following. The flow diagram of the pivoting steps which will result in an optimal feasible solution is given in Fig. 2.13.

![Flow diagram](image)

*Figure 2.13 Exchange step from an unfeasible minimum towards a feasible optimum solution*

The algorithm tries to locate the pivot $a_{pq}$ in such a way that an exchange step for this pivot results in the most favourable transition towards a feasible solution. Due to the convexity of the linear program, the first feasible solution found using this algorithm represents also the optimum solution.
The second case is one where a feasible solution exists but it can further be optimized. The solution algorithm is shown in the flow diagram of Fig. 2.14. The selection of the pivot at each step is such that the most effective transition towards the optimum solution is achieved.

**Figure 2.14** Exchange step from a feasible non-optimum towards a feasible optimum solution

In the optimization formulation defined by (2.49) to (2.51), if the optimization variables $x_j$ are not strictly defined to be non-negative, then the simplex algorithm is slightly different. In this case, the columns with these variables are selected as the first pivot columns for which exchange steps take place. Later, when selecting the pivot rows, the rows corresponding to these variables, which have been exchanged, are excluded from the consideration of the potential pivot rows, which can be exchanged.
3 Yield Conditions for an Infinitesimal Element

This chapter deals with the derivation of yield conditions for an infinitesimal element. It serves as the basis for the derivation of yield conditions for a finite element which will be presented in the next chapter. It also provides a basis of comparison for the two approaches. The yield conditions for an infinitesimal element in a pure membrane state are first derived after which the cases of pure bending and combined membrane-bending action are handled.

3.1 Yield conditions for a membrane infinitesimal element

In this section, yield conditions for a membrane infinitesimal element are derived. Nonlinear yield conditions for an orthogonally reinforced element are first found by combining the yield conditions of concrete and steel. A linear approximation of these nonlinear yield conditions is then suggested. The problem of skew reinforcement is subsequently discussed which enables the application of previously derived yield conditions for an orthogonal reinforcement in the general case of a series of arbitrarily oriented reinforcement bars. This section is intended to serve for membrane only panels and also as a basis for the derivation of yield conditions for the general case of combined membrane-bending panels.

3.1.1 Nonlinear yield conditions

An infinitesimal $dx- dy$ reinforced concrete membrane element with an orthogonal reinforcement net parallel to the two coordinate axes $(x,y)$ is considered. The composite element can be viewed as the combination of its two concrete and reinforcement components as shown in Fig. 3.1. The concrete is assumed to be homogeneous while the actual discrete reinforcing bars are assumed to be of small diameter relative to the other panel dimensions and closely spaced so that each steel layer can be considered to be replaced by an equivalent continuous uniform sheet of which an infinitesimal element is considered here. The yield conditions for concrete and steel are derived separately and then appropriately superimposed to give the yield conditions for the composite material [Müller 78].
3. Yield Conditions for an Infinitesimal Element

The generalized stresses $n_x = \sigma_x h$, $n_y = \sigma_y h$ and $n_{xy} = \tau_{xy} h$ are simply referred to as stresses for the sake of simplicity. The contributions of concrete and steel in resisting these stresses are distinguished by the superscripts $c$ and $s$, respectively.

![Figure 3.1 Reinforced concrete infinitesimal element in membrane state](image)

The tensile strength of concrete is neglected so that it resists only compressive forces, while the reinforcement is assumed to act in a planar uniaxial stress state in the direction of the reinforcement bars capable of withstanding tensile forces. Due to the usually low reinforcement content and the significant variations of concrete compressive strength, the action of reinforcement bars under compressive forces is also neglected.

![Figure 3.2 Concrete material model](image)

Concrete is assumed to be an elastic ideally plastic homogeneous isotropic material. The idealized stress-strain curve for the concrete model adopted for the task of reinforcement design is shown in Fig. 3.2 (a). The material is assumed to obey the modified Coulomb material model which is characterized by the two types of sliding and separation failures. These are functions of the coefficient of friction $\mu = \tan \varphi$, cohesion $c$ and the tensile strength $f_t$. The tensile strength of concrete is neglected, i.e. $f_t = 0$ (Fig. 3.2 b). By plotting Mohr’s circle in the case of pure compression, the relationship between concrete compressive strength $f_c$ and the material coefficients $\varphi$ and $c$ can be found as

$$f_c = \frac{2c \cos \varphi}{1 - \sin \varphi} = 2c \tan \left( \frac{\pi}{4} + \frac{\varphi}{2} \right)$$  \hspace{1cm} (3.1)
For a simpler mathematical description, a square yield locus is adopted and the increase of the biaxial concrete compressive strength in comparison with its uniaxial strength is therefore neglected (Fig. 3.3 a). The yield conditions for concrete in terms of the principal stresses \( n_1^c, n_2^c \) and normal and shear stresses \( n_{xy}^c, n_{xy}^c \) is then given by

\[
-r_m^c \leq n_{1,2}^c = \frac{1}{2} \left( n_x^c + n_y^c \pm \sqrt{(n_x^c - n_y^c)^2 + 4(n_{xy}^c)^2} \right) \leq 0
\]  

(3.2)

which is equivalent to

\[
(n_{xy}^c)^2 \leq n_x^c n_y^c \hspace{1cm} n_x^c \leq 0 \hspace{1cm} n_y^c \leq 0
\]  

(3.3)

\[
(n_{xy}^c)^2 \leq (r_m^c + n_x^c) (r_m^c + n_y^c) \hspace{1cm} r_m^c + n_x^c \geq 0 \hspace{1cm} r_m^c + n_y^c \geq 0
\]  

(3.4)

**Figure 3.3** Yield conditions for an infinitesimal concrete element in a membrane state

The concrete resistance \( r_m^c \) is defined by

\[
r_m^c = \kappa f_c h
\]  

(3.5)

with \( f_c \) and \( h \) being the concrete compressive strength and element thickness, respectively. \( \kappa \leq 1 \) is a reduction factor taking into account the fact that the effective panel thickness and concrete strength at failure may be smaller than their corresponding nominal values.

As shown in Fig. 3.3 (a), in the space of principal stresses, the yield conditions represent a square with its side lengths equal to the concrete resistance compressive strength \( r_m^c \). In the space of the normal and shear stresses, the envelope consists of two cones (Fig. 3.3 b). As expected, the intersection of the non-plastic region with the zero shear stress plane \( n_{xy}^c = 0 \) gives the square yield surface in the space of the principal stresses.

The yield conditions for the orthogonal steel net parallel to the coordinate axes \( x-y \) as shown in Fig. 3.4 are
3. Yield Conditions for an Infinitesimal Element

\[0 \leq n_x^i \leq A_s^i f_y = r_{mx}^i \quad 0 \leq n_y^i \leq A_y^i f_y = r_{my}^i \quad n_{xy}^i = 0 \quad (3.6)\]

in which \(f_y\) is the steel tensile yield strength and \(A_s^i, A_y^i, r_{mx}^i\) and \(r_{my}^i\) denote the cross sectional area and yield resistance of reinforcement per unit width and in the \(x\) and \(y\) directions, respectively.

![Figure 3.4 Yield conditions for an orthogonal reinforcement net](image)

For given reinforcement resistances \(r_{mx}^i\) and \(r_{my}^i\) within the grey-shaded rectangle of Fig. 3.4, the admissible stress points \(n_x, n_y\) and \(n_{xy}\) lie on or within the cone-shaped concrete yield surface of Fig. 3.3 (b) if the cone is translated in such a way that its apex \(O\) is moved to the point \((r_{mx}^i, r_{my}^i)\) of Fig. 3.4. Consequently, the nonlinear yield condition surface is found by the envelope of all cone surfaces with their apex within the grey-shaded rectangle of Fig. 3.4. This will result in an envelope which is shown with its contour lines in Fig. 3.5.

![Figure 3.5 Nonlinear yield conditions for an infinitesimal reinforced concrete membrane element](image)

### 3.1.2 Linearized yield conditions

The simplest form of the linearization of nonlinear concrete yield conditions is to approximate the cone by a pyramid as shown in Fig. 3.3 (b). The linearized yield conditions for the combined action of concrete and steel of the infinitesimal orthogonally reinforced concrete element will then be found analogous to the approach presented for the nonlinear case,
namely by sliding the pyramid’s apex in the steel’s square yield surface. The shape of the
envelope thus found and the corresponding linear equations for the yield surface are given in
Fig. 3.6.

In Fig. 3.6, the yield surfaces with an odd subscript correspond to the positive shear values
while those with an even subscript refer to the surfaces in the negative $n_{xy}$ half space. The
equation number for each yield surface is written at the center of the corresponding surface.

![Diagram showing yield conditions](image)

**Figure 3.6** Linearized yield conditions of an infinitesimal reinforced concrete membrane element

The inequalities presented in Fig. 3.6 can alternatively be formulated in a matrix notation as

$$N_m\sigma_m - r_m \leq 0$$

with the stress vector $\sigma_m$ given by (2.1), the normal matrix $N_m$ summarized in

$$N_m^T = \begin{bmatrix} N_m^{sT} & N_m^{cT} \end{bmatrix}$$

$$N_m^{sT} = \begin{bmatrix} 1 & 1 & 0 & 0 \\ 0 & 0 & 1 & 1 \\ 1 & -1 & 1 & -1 \end{bmatrix}, \quad N_m^{cT} = \begin{bmatrix} -1 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & -1 & 0 & 0 \\ 1 & -1 & 1 & -1 & 1 & -1 \end{bmatrix}$$

and the resistance vector $r_m$ defined as

$$r_m^T = \begin{bmatrix} r_m^{sT} \\ r_m^{cT} \end{bmatrix}$$

$$r_m^{sT} = \begin{bmatrix} r_{mx}^s & r_{my}^s \end{bmatrix}, \quad r_m^{cT} = \begin{bmatrix} r_m^c \\ r_m^c \\ r_m^c/2 \\ r_m^c/2 \end{bmatrix}$$

The values in column $i$ of matrix $N_m$ represent the components of a vector perpendicular to
yield plane $\phi_i$ onto $n_x$, $n_y$ and $n_{xy}$ axes.
3.1.3 Skew reinforcement

In the previous section, an orthogonal reinforcement was assumed parallel to the coordinate axes \(x\)-\(y\) with reference to which the yield conditions were derived. In general, a series of arbitrarily-oriented uniaxial reinforcement bars overlapping one another provide the reinforcement at that point (Fig. 3.7).

![Superposition of reinforcement layers for an infinitesimal element](image)

**Figure 3.7 Superposition of reinforcement layers for an infinitesimal element**

The stress in all the bars is assumed to reach the tensile yield strength \(f_y\). For one layer of skew reinforcement with an angle \(\alpha\) to the \(x\) axis, the equivalent \(\sigma\) stresses on the infinitesimal element in the \(x\)-\(y\) directions are

\[
\sigma^s_m = \begin{pmatrix} n_x^s \\ n_y^s \\ n_{xy}^s \end{pmatrix} = \begin{pmatrix} \cos^2 \alpha & \cos \alpha \sin \alpha \\ \sin^2 \alpha & \cos \alpha \sin \alpha \end{pmatrix} A^s f_y
\]  

(3.12)

For a skew reinforcement with \(NRL\) reinforcement layers, the total equivalent stress is equal to the sum of the contributions of each of these reinforcement layers. The equivalent total stress resultant provided by steel layers calculated in this way is multiplied by the \(N\alpha\) matrix defined in (3.9) which leads to the following formula for calculating the steel resistance sub-vector \(r_m^s\) in (3.11)

\[
r_m^s = \begin{pmatrix} r_{x+}^s \\ r_{x-}^s \\ r_{y+}^s \\ r_{y-}^s \end{pmatrix} = \begin{pmatrix} \sum \left( \cos^2 \alpha + \cos \alpha_j \sin \alpha_j \right) A_j^s f_{y_j} \\ \sum \left( \cos^2 \alpha - \cos \alpha_j \sin \alpha_j \right) A_j^s f_{y_j} \\ \sum \left( \sin^2 \alpha + \cos \alpha_j \sin \alpha_j \right) A_j^s f_{y_j} \\ \sum \left( \sin^2 \alpha - \cos \alpha_j \sin \alpha_j \right) A_j^s f_{y_j} \end{pmatrix} = \begin{pmatrix} \sum a_{1j} A_j^s \\ \sum a_{2j} A_j^s \\ \sum a_{3j} A_j^s \\ \sum a_{4j} A_j^s \end{pmatrix}
\]  

(3.13)

in which \(A_j^s\) is the steel area per unit width for the \(j\)-th reinforcement layer. \(\alpha_j\) and \(f_{y_j}\) refer to the inclination angle of reinforcement layer \(j\) to the \(x\) axis and steel yield strength, respectively. The summation is done over all reinforcement layers. If the value of any of the terms in parentheses projecting the resistance onto an axis, i.e. any \(a_{ij}\), becomes negative, then that term is taken as zero in the summation.
3.2 Yield conditions for a membrane-bending infinitesimal element

In this section, the yield conditions for bending infinitesimal elements with and without membrane action are derived. The elements considered are assumed to be reinforced in two layers, at the top and bottom, by straight bars. An effective depth is considered at each side to approximate the reinforcement layers lying on each other in different directions. The general assumptions are similar to those specified in the previous sections. The contribution of the compressive reinforcement to the increase of the load-carrying capacity is neglected since this is on the safe side, it significantly simplifies the calculations and also the difference is very slight for low degrees of reinforcement.

The assumption of a ductile material behavior has been verified by a variety of experiments for cases where the reinforcement yields and compressive forces in the concrete are not dominant. Surprisingly, this assumption has also resulted in qualitatively correct results for cases with dominant compressive forces. As a result, the yield conditions are only formulated for the reinforcement, i.e. concrete crushing is assumed not to govern the design so that the corresponding yield conditions need not to be incorporated.

3.2.1 Pure bending

As the simplest case, a section with unit width is considered in which only bottom reinforcement is provided along the $X$ axis (Fig. 3.8). All tensile forces are carried by steel whereas compressive forces are resisted by concrete [Nielsen 84].

![Figure 3.8 Reinforced element subjected to pure bending](image)

According to the assumptions made so far, force equilibrium at failure for the cross section shown in Fig. 3.8 gives

$$A^t f_y = a f_c$$  \hspace{1cm} (3.14)

The yield moment per unit length in pure bending is given by

$$M_p = A^t f_y (d - \frac{1}{2}a)$$  \hspace{1cm} (3.15)
3. Yield Conditions for an Infinitesimal Element

For a bending element orthogonally reinforced along the $X$ and $Y$ axes, yield conditions can be derived [Wolfensberger 64]. The idea is that at any cross section at an angle $\varphi$ with the $X$ axis and with a normal direction $n$ perpendicular to this section, we must have

$$-M_{pn}^l \leq m_n \leq M_{pn}^b$$  \hspace{1cm} (3.16)

$$m_{nq} = -M_{pnq}^l \quad \text{if} \quad m_n = -M_{pn}^l$$  \hspace{1cm} (3.17)

$$m_{nq} = M_{pnq}^b \quad \text{if} \quad m_n = M_{pn}^b$$  \hspace{1cm} (3.18)

The plastic moments $M_{pn}^b$, $M_{pnq}^b$, $M_{pn}^l$ and $M_{pnq}^l$ are calculated by transforming $M_{ps}^b$, $M_{ps}^l$, $M_{px}^b$ and $M_{py}$ respectively. Similarly, $m_n$ and $m_{nq}$ are found by transforming $m_x$, $m_y$, and $m_{xy}$. These will be substituted into (3.16) and the extreme cases will be studied in terms of the angle $\varphi$ such that the dependency on $\varphi$ is removed. This will result in the nonlinear plasticity conditions for an orthogonally reinforced element formulated in Fig. 3.9 (a). The linear inequalities given in Fig. 3.9 (b) are approximations for the nonlinear inequalities of Fig. 3.9 (a). They nowhere violate the nonlinear yield conditions. In graphical terms, the convex surface represented by Fig. 3.9 (b) is fully encompassed by the surface given by Fig. 3.9 (a). The formulation given by Fig. 3.9 (c) is fully equivalent to that of Fig. 3.9 (b) and is gained by substituting each inequality with $|m_{xy}|$ by two inequalities with $\pm m_{xy}$.

$$m_s \leq M_{ps}^b \qquad m_s + |m_{xy}| \leq M_{ps}^b \qquad m_s + m_{xy} \leq M_{ps}^b$$

$$m_y \leq M_{py}^b \qquad m_y + |m_{xy}| \leq M_{py}^b \qquad m_y - m_{xy} \leq M_{py}^b$$

$$(M_{ps}^l - m_s) (M_{py}^l - m_y) \geq m_{xy}^2 \qquad -m_s + |m_{xy}| \leq M_{ps}^l \qquad m_y + m_{xy} \leq M_{py}^l$$

$$-m_s \leq M_{ps}^l \qquad -m_s + |m_{xy}| \leq M_{ps}^l \qquad m_y - m_{xy} \leq M_{py}^l$$

$$-m_y \leq M_{py}^l \qquad -m_y + |m_{xy}| \leq M_{py}^l \qquad -m_s + m_{xy} \leq M_{ps}^l$$

$$(M_{ps}^l + m_s) (M_{py}^l + m_y) \geq m_{xy}^2 \qquad -m_s - m_{xy} \leq M_{ps}^l \qquad -m_y \leq M_{py}^l$$

$$-m_y - m_{xy} \leq M_{py}^l$$

\textbf{Figure 3.9 Yield conditions for an orthogonally reinforced plate bending element}

(a) Nonlinear (b),(c) Linearized

Similar to the case of membrane elements, the inequalities presented in Fig. 3.9 can alternatively be formulated in a matrix notation as
\[ N_b \sigma_b - r_b \leq 0 \] (3.19)

with the stress vector \( \sigma_b \) given by (2.17), the normal matrix \( N_b \) summarized in

\[
N_b^T = \begin{bmatrix}
1 & 1 & 0 & 0 & -1 & -1 & 0 & 0 \\
0 & 0 & 1 & 1 & 0 & 0 & -1 & -1 \\
1 & -1 & 1 & -1 & 1 & -1 & 1 & -1
\end{bmatrix}
\] (3.20)

and the resistance vector \( r_b \) defined as

\[
r_b^T = \{ M_{px}^b \ M_{py}^b \ M_{px}' \ M_{py}' \ M_{px}^t \ M_{py}^t \ M_{px}'^t \ M_{py}'^t \} \] (3.21)

As before, the values in column \( i \) of matrix \( N_b \) represent the components of a vector perpendicular to yield plane \( \phi_i \) onto \( m_x \), \( m_y \) and \( m_{xy} \) axes.

The nonlinear and linearized reinforcement yield conditions for an infinitesimal bending element with equal reinforcement at the top and bottom layers along the \( x \) and \( y \) axes is shown in Fig. 3.10.

**Figure 3.10** Yield conditions for an infinitesimal concrete element in the bending state with

\[ M_{px}^b = M_{py}^b = M_{px}^t = M_{py}^t \]

For the case of reinforcement fields in arbitrary directions, it can be shown that a similar approach to the one presented for an orthogonally reinforced bending element will lead to similar nonlinear yield conditions as in Fig. 3.9 (a) with \( m_{xy} \) replaced by \( (m_{xy} - M_{pzy}) \) for the corresponding bottom and top reinforcements [Wolfensberger 64]. The equivalent plastic resistances \( M_{pzn}^b, M_{pzn}'^b, M_{pzn}^t \) and \( M_{pzn}'^t \) are found by projecting the resistances of the reinforcements in arbitrary directions onto the \( x \) and \( y \) axes using the transformations given in section 3.1.3. The linear inequalities presented in Fig. 3.11 (b) are the approximation for the nonlinear yield surface specified in Fig. 3.11 (a).
3. Yield Conditions for an Infinitesimal Element

\[
m_x \leq M_{px}^b \\
m_y \leq M_{py}^b \\
(M_{px}^b - m_x) (M_{py}^b - m_y) \geq (m_{xy} - M_{pxy}^b)^2 \\
-m_x \leq M_{px}^t \\
-m_y \leq M_{py}^t \\
(M_{px}^t + m_x) (M_{py}^t + m_y) \geq (m_{xy} - M_{pxy}^t)^2
\]

(a) Nonlinear  (b) Linearized

\[m_x + m_y \leq M_{px}^b + M_{pxy}^b \]
\[m_x - m_y \leq M_{px}^b - M_{pxy}^b \]
\[m_x + m_y \leq M_{py}^b + M_{pxy}^b \]
\[m_x - m_y \leq M_{py}^b - M_{pxy}^b \]
\[m_x + m_y \leq M_{px}^t + M_{pxy}^t \]
\[m_x - m_y \leq M_{px}^t - M_{pxy}^t \]
\[m_x + m_y \leq M_{py}^t + M_{pxy}^t \]
\[m_x - m_y \leq M_{py}^t - M_{pxy}^t \]

**Figure 3.11 Yield conditions for an arbitrarily reinforced plate bending element**

It should be noted that \(M_{px}^b, M_{py}^b, M_{px}^t\) and \(M_{py}^t\) always have positive values whereas \(m_x, m_y, m_{xy}, M_{pxy}^b\) and \(M_{pxy}^t\) can be positive or negative. In matrix notation, (3.19) and (3.20) remain unchanged while the new resistance vector will be

\[
\mathbf{r}_b^T = \begin{bmatrix}
M_{px}^b & M_{pxy}^b & M_{py}^b & M_{pxy}^t & M_{px}^t & M_{pxy}^t & M_{py}^t & M_{pxy}^t
\end{bmatrix}
\]

(3.22)

### 3.2.2 Sandwich model for combined membrane-bending action

The six generalized stress components for a combined membrane-bending element are shown in Fig. 3.12.

**Figure 3.12 Stress components in a combined membrane-bending element**
It has not yet been possible to derive a simple yield condition in this case in a closed analytical form. Rajendran and Morley developed a numerical procedure for the determination of points on the exact yield surface [Rajendran, et al. 74].

A common approach to tackle this problem is to utilize the yield conditions for membrane elements derived previously. In this method, a sandwich model is assumed, which consists of two reinforcement cover layers at the top and bottom and a concrete core withstanding transverse loads [Nielsen 84]. The bending moments are viewed as a couple of normal forces applying on the two layers and the twisting moment similarly results in shear forces at the top and bottom (Fig. 3.13). \(d_q\) is the distance between the center of the reinforcement layers at top and bottom. With the assumption of equal cover thickness \(c\) on two sides, we will have \(d_q = h - c\) where \(h\) is the element thickness.

![Figure 3.13 Sandwich model for a membrane-bending element](image)

Using this model, the formulation of yield conditions for the reinforcement of an underreinforced element will be accomplished by imposing the membrane yield conditions on each of these two cover layers for the forces specified. This will lead to the following yield conditions for the top and bottom layers

\[
\frac{1}{2} N_m^t \sigma_m - \frac{1}{d_q} N_m^t \sigma_b - r_m^t \leq 0
\]  

(3.23)

\[
\frac{1}{2} N_m^b \sigma_m + \frac{1}{d_q} N_m^b \sigma_b - r_m^b \leq 0
\]  

(3.24)
with the normal matrix $N_{m}^{p}$ and resistance vector $r_{m}^{s}$ defined by (3.9) and (3.11), respectively.

The concrete compressive strength of the top and bottom layers is controlled by the following additional constraints, respectively

$$\pm 2\left(\frac{n_{xy}}{2} - \frac{m_{xy}}{d_{q}}\right) \leq c f_{c} \quad (3.25)$$

$$\pm 2\left(\frac{n_{xy}}{2} + \frac{m_{xy}}{d_{q}}\right) \leq c f_{c} \quad (3.26)$$

If the compressive strength of the cover layers is exceeded, then the thickness $c$ of the concrete cover has to be increased and the calculations have to be repeated with a correspondingly reduced value of $d_{q}$.

The resistance vector $r_{m}^{s}$ in the general case of arbitrary reinforcement layers, must be evaluated for the top and bottom layers separately by using (3.13). The sandwich model of Fig. 3.13 can also be extended to include a contribution of transverse shear in reinforcement design [Marti 90].

![Figure 3.14 Design for transverse shear](image-url)
In Fig. 3.14, \( \varphi_0 \) and \( q_0 \) are defined by (2.21) and (2.22), respectively. The angle of inclination of the diagonal compressive stress field \( \theta \) can be freely selected. Usually, values between 25 and 45 degrees are used.

If the nominal shear stress due to the principal shear force, i.e. \( q_0/d_q \) does not exceed a shear strength limit \( \tau_c \) \((= 0.17 \sqrt{f'_c} \) according to ACI) it can be assumed that there are no diagonal cracks in the concrete. A state of pure shear develops within the concrete core (Fig. 3.14 a), and no transverse reinforcement has to be provided.

\[
\frac{q_0}{d_q} < \tau_c \tag{3.27}
\]

If (3.27) is not satisfied and diagonal cracking of the core takes place, a model can be assumed as shown in Fig. 3.14 (b). The resulting transverse reinforcement ratio required normal to the plane of the slab would, according to this model, be

\[
\rho_q = \frac{q_0 \tan \theta}{d_q f_y} \tag{3.28}
\]

Formulating equilibrium conditions as depicted in Fig. 3.14, the stress components as a result of the shear force \( q_0 \) in the two cover layers would be

\[
\sigma_x = \frac{q_0^2}{2q_0 \tan \theta} \quad \sigma_y = \frac{q_0^2}{2q_0 \tan \theta} \quad \sigma_{xy} = \frac{q_0 q_y}{2q_0 \tan \theta} \tag{3.29}
\]

Their contribution to the stresses in the two cover layers is shown in Fig. 3.15.

Figure 3.15 Sandwich model for a membrane-bending element including shear reinforcement

It should be noted before closing this chapter that some approaches have been suggested for the optimum reinforcement design for stresses at an infinitesimal point for the cases of membrane, bending and combined membrane-bending actions. A summary of and references to
these works can be found in [Nielsen 84]. In this chapter, reinforcement minimization in this regard was not discussed. [Morley 70] and [Morley, et al. 77] proposed methods to tackle the optimum design of the reinforcement in a concrete slab element required to resist given combinations of membrane forces and bending moments. In the latter paper, the authors suggested a general filled sandwich method and the aspects of automation and a simplified method for hand calculation were also explained.
4 First Concept: Element-based Dimensioning and Yield Conditions for a Finite Element

In this chapter, the first idea of the new dimensioning method is explained. The deficiencies of the usual approach based on an infinitesimal element are discussed. Then, the idea of handling each finite element as a dimensioning unit is proposed. Generalized strains and stresses are established for a quadrilateral finite element after which yield conditions are derived for a finite element in three cases of membrane, bending and combined membrane-bending action. Based on the yield conditions for quadrilateral elements, those for a triangular finite element are suggested. Finally, the mesh parameter is introduced as a means for error estimation for the proposed yield conditions.

4.1 Deficiencies of the usual approach

In the previous chapter, the yield conditions for an infinitesimal element were presented. Having obtained the stress distribution which is usually found assuming linear elasticity and graphically described by isoline plots or by color scale, the design engineer uses the formulae given in the previous chapter to calculate the necessary reinforcement. It is interesting to have a closer look at the procedure used for calculating the stresses, on which the engineer bases his design, within the framework of a displacement-based finite element program.

The first results of a finite element calculation based on the displacement method are the displacements of the nodes. Assuming linear elasticity, these are calculated by solving the linear system of equations formulated in (2.29) which gives the displacements of all nodes. The nodal displacements and, for bending elements, rotations $a$ of each element can then be extracted from this displacement vector. The stresses $\sigma$ within the element can now be evaluated using the relation

$$\sigma = D B a - \sigma_{e0}$$

(4.1)

where the elasticity matrix $D$, the strain-displacement matrix $B$ and the stress vector $\sigma_{e0}$ of the initial strains were formulated in section 2.1. This relation gives the stresses at any point of the element in terms of the element nodal displacements and the $B$ matrix which is a function
4. First Concept: Element-based Dimensioning and Yield Conditions for a Finite Element

of the coordinates of the stress point considered. We know that in the displacement-based finite element approach, the displacements from element to element are continuous but the stress components are not continuous unless a stress field is analysed which is contained in the element formulation. This is always the case for beam elements but it is practically never the case in continuum elements except for some very simple structural and loading configurations. These stress discontinuities, of course, will decrease as the element mesh is refined.

Another observation in stress calculation is that stresses in some points in an element can be significantly more accurate when compared with the exact solution than at other points. In particular, it has been observed that stresses can be considerably more accurate at the Gauss integration points than at the nodal points of an element. Because of this and also due to some technical programming aspects, the usual practice in finite element programs is to calculate the stresses at the integration points within each element and then employ interpolation procedures to predict the stresses at the element nodes. The stresses calculated at the nodal points in this way are then simply averaged for all adjoining elements at that node to obtain one value for the stress component at a nodal point (Fig. 4.1 a). Although this nodal averaging technique usually leads to acceptable results from an engineering point of view, it does involve problems especially in the case of three-dimensional panel structures, where for the nodes along the edges of adjoining panels, the contributing stresses of an edge node from the elements in different panels are evaluated in different panel coordinate systems so that their simple averaging is meaningless (Fig. 4.1 b).

As expected and, for the same reason, the element stresses are also discontinuous along the boundaries of adjoining elements.

This problem and other considerations discussed in the first chapter resulted in a new paradigm in the derivation of the yield conditions for a finite element with non-infinitesimal dimensions instead of the common approach of working with an infinitesimal element. The new method is explained in the coming sections.

Figure 4.1 Evaluation of nodal average stresses in the finite element approach
4.2 Finite elements as dimensioning units

In this section, the concept of handling each finite element as a dimensioning unit, which forms the first concept of the new approach, is presented. First, the mathematical derivation, which enables the formulation of yield conditions for generalized variables, is discussed. Afterwards, the element nodal forces are proposed as generalized variables for which dimensioning should take place.

4.2.1 Yield conditions formulation for generalized variables

The theory of plastic potential, proposed by von Mises connects the yield condition and the flow rule of a plastic solid. This is a hypothesis and several researchers have tried to provide a basis for it which in turn is also based on some other postulates. The version formulated by von Mises was also restricted to an infinitesimal volume element with strains $\varepsilon_{ij}$ and stresses $\sigma_{ij}$. The generalization of the theory of the plastic potential as formulated in section 2.2.3 is due to Prager [Prager 55]. Ziegler provided a purely mathematical proof for the theory in Prager’s generalized form based on the single assumption that it holds in von Mises’ sense for an infinitesimal element of volume [Ziegler 61 b]. The proof can be given for rigid perfectly plastic [Ziegler 60] as well as elastic perfectly plastic materials [Ziegler 61 a].

For a rigid plastic infinitesimal element, the states of strain and stress are given by the strain vector $\varepsilon$ and stress vector $\sigma$ in the 9 dimensional Euclidean space $R_9$. If the scalar product in $R_9$ is defined by

$$\sigma^T \varepsilon = \sigma_{ij} \varepsilon_{ij}$$  \hspace{1cm} (4.2)

where the summation notation holds, then the work per unit volume done on an infinitesimal increment of strain is

$$dW = \sigma_{ij} d\varepsilon_{ij} = \sigma^T d\varepsilon$$  \hspace{1cm} (4.3)

Due to the symmetry of the strain and stress tensors, the vectors $\varepsilon$ and $\sigma$ actually lie in a linear subspace $R_6$ of $R_9$. With the assumption that the theory of plastic potential as formulated by von Mises is valid, it follows from (2.44) that for any non-plastic stress increment $\sigma^* - \sigma$

$$(\sigma^* - \sigma)^T d\varepsilon = (\sigma^*_{ij} - \sigma_{ij}) d\varepsilon_{ij} \leq 0$$  \hspace{1cm} (4.4)

Also, we have

$$d\sigma^T d\varepsilon = d\sigma_{ij} d\varepsilon_{ij} \geq 0$$  \hspace{1cm} (4.5)
For a rigid plastic finite element with non-infinitesimal dimensions, the states of strain and stress are given in a function space $F$ by the strain and stress vectors $e$ and $s$, their components $\varepsilon_{ij}(x_k)$ and $\sigma_{ij}(x_k)$ being functions of the coordinates $x_k$. With the definition of the scalar product in $F$ by the volume integral

$$s^T e = \int \sigma_{ij} \varepsilon_{ij} \, dV$$

(4.6)

extended over the finite element volume, the work done in an infinitesimal increment of strain would be

$$dW = \int \sigma_{ij} \, d\varepsilon_{ij} \, dV = s^T \, de$$

(4.7)

The above representation is neither restricted to kinematically admissible states of strain nor to statically admissible states of stress. It holds for any states for which the integrals (4.6) and (4.7) exist. In general, certain infinitesimal elements of the finite element reach their local yield limit for states of stress $s$ still within the yield limit of the finite element. Plastic flow sets in when a sufficiently large domain in the element has become plastic. At this stage, the state of stress in any infinitesimal element of the finite element either lies below or at its local yield limit. Since none of the infinitesimal elements undergoes plastic flow in a non-plastic stress increment over the entire finite element, it follows from (4.4) that

$$(s^* - s)^T \, de = \int (\sigma_{ij}^* - \sigma_{ij}) \, d\varepsilon_{ij} \, dV \leq 0$$

(4.8)

for any non-plastic stress increment $s^* - s$. Also from (4.5)

$$ds^T \, de = \int d\sigma_{ij} \, d\varepsilon_{ij} \, dV \geq 0$$

(4.9)

It can therefore be concluded that the theory of plastic potential, if valid for the infinitesimal element, likewise applies for the finite element as a whole.

In section 2.2.3, the concept of generalized strains and stresses was outlined in which the treatment of a rigid plastic body was simplified to the investigation of a finite set of $n$ parameters. Therefore, we consider only states of strain $\varepsilon_{1...e_{k}}...e_{n}$ which belong to a certain subspace $F_n$ of $F$. There is no similar restriction with respect to the states of stress. Obviously, the simplification from $F$ towards $F_n$ is only justified as long as it does not appreciably modify the actual state of strain. If $e_k$ denotes the state of strain in function space $F$ corresponding to the generalized strain $q$ in subspace $F_n$ with $q_k = 1$, $q_i = 0 \ (i \neq k)$, then the state of strain $e$ in subspace $F_n$ is defined by
\[ e = \sum_{k=1}^{n} q_k e_k \]  

(4.10)

Considering (4.7), (4.10) and (2.39), the work done by the stress \( s \) on an infinitesimal strain increment \( de \) belonging to \( F_n \) is

\[ dW = s^T de = s^T \sum_{k=1}^{n} e_k dq_k = \sum_{k=1}^{n} s^T e_k dq_k = \sum_{k=1}^{n} S_k dq_k = S^T dq \]  

(4.11)

Hence, the generalized stresses are given by the scalar products

\[ S_k = s^T e_k \]  

(4.12)

Consequently, any state of stress \( s \) in \( F \) is represented in \( F_n \) by a unique vector \( S \) which, conversely, is the image of an infinity of vectors \( s \).

From (4.11) and (4.8), we obtain

\[ (S^* - S)^T dq = (s^* - s)^T de \leq 0 \]  

(4.13)

with \( S^* - S \) being an arbitrary non-plastic stress increment. Similarly, from (4.11) and (4.9) we obtain

\[ dS^T dq = ds^T de \geq 0 \]  

(4.14)

This shows that the theory of plastic potential, if valid for an infinitesimal element, likewise applies to the treatment of the finite element in generalized strains and stresses.

### 4.2.2 Yield conditions formulation based on element nodal forces

When a stable structure discretized into a finite element mesh is subjected to external loads, each finite element exerts forces on the neighboring nodes of adjoining elements. These element nodal forces \( p \) have several advantages when compared with the conventional stresses at infinitesimal points. Firstly, they are much easier to evaluate than stresses. While the evaluation of stresses involves a range of calculations as specified in section 4.1, the element nodal forces can be gained by a simple multiplication of the element stiffness matrix \( k \) with the vector of element nodal displacements \( a \)

\[ p = \int B^T \sigma \, dV = k \, a \]  

(4.15)

Secondly, for any \( a \), the nodal forces \( p \) are always in exact equilibrium with one another within an element. This is a direct consequence of the singularity of the element stiffness matrix.
and the fact that any finite element stiffness formulation must accommodate rigid body motion, i.e. we must have \( k a_{rb} = 0 \). In other words, for any rigid body displacement field \( a_{rb} \) with vanishing strains \( \mathbf{e} = B a_{rb} = 0 \), the work of the nodal forces \( \mathbf{p} \) on the displacements \( a_{rb} \) vanishes.

\[
p^T a_{rb} = a^T k a_{rb} = 0
\]  

(4.16)

Thirdly, unlike the stresses, the force components transferred at the nodes from each element to neighboring ones exactly equal those forces with which the considered element is loaded by the rest of the structure at that node. In fact, the global system of equations is solved by fulfilling these nodal equilibrium conditions.

Finally, the nodal forces \( \mathbf{p} \) are statically equivalent to the tractions acting on the element boundaries and the stress resultants within the element volume through the application of the virtual work principle which is applied in the derivation of the element stiffness matrix.

In view of all the points considered so far, the nodal forces \( \mathbf{p} \) represent a suitable choice on the basis of which yield conditions are to be formulated. Each finite element can be regarded as being loaded with the nodal forces \( \mathbf{p} \) in the same way that an infinitesimal element with the side lengths \( dx_i \) is subjected to stresses \( \sigma_{ij} \). A basic idea of the new dimensioning method, therefore, is to treat each finite element as an independent dimensioning unit which must be capable of carrying its loads \( \mathbf{p} \) without plastic flow taking place.

![Figure 4.2](image)

**Figure 4.2 Transition from infinitesimal to finite element-based dimensioning**

The element-based design approach proposed here can be justified by the lower bound theorem of plasticity theory. The nodal forces \( \mathbf{p} \) satisfy both internal and external equilibrium exactly and are therefore statically admissible. Provided that they also satisfy the yield condition constraints, strength requirements are then fulfilled. The discretization here is attributed to the fact, that the nodal forces \( \mathbf{p} \) satisfy the microscopic equilibrium conditions only in an integral form. The differential equations derived previously for the infinitesimal element with the
side lengths $dx_i$ are only weakly satisfied. This, however, is the basis of all finite element models and of the finite element concept in general.

The vector of the nodal forces $p$ has a dimension equal to the total number $N^e$ of the degrees of freedom of the finite element. If the number of rigid body modes of the finite element is $M^r$, then the number of independent strain modes of the element equals $m = N^e - M^r$. This is a direct consequence of the fact that the components of the nodal force vector $p$ cannot be chosen arbitrarily, but must be in equilibrium within themselves. As a result, for each finite element, $m$ linearly independent nodal force vectors can be found whereby, any vector of nodal forces producing strains in the element, i.e. not resembling a rigid body mode, can be reproduced as a linear combination of these $m$ vectors.

This issue can also be viewed from the point of view of the mathematical group theory and particularly the representation theory. These theories provide the mathematical tools for simplifying physical problems which exhibit different forms of symmetry. For a finite element with $N^e$ degrees of freedom, a canonical set of $N^e$ basis vectors $e_i$ can be constructed. Each basis vector $e_i$ has a dimension equal to $N^e$ where the $i$-th component of vector $e_i$ is equal to 1 while all other components vanish.

$$G_{\text{canonical}} = \{e_1, ..., e_{N^e}\} \quad e_i^T = [0, ..., 0, 1, 0, ..., 0]$$

The set of canonical basis vectors obviously represents a linearly independent system and any nodal load combination on the finite element can be constructed as a linear combination of these basis vectors. Representation theory enables one to represent the vector space $G_{\text{canonical}}$ as the combination of orthogonal subspaces. Each of these subspaces is given by new sets of basis vectors, each of which are linear combinations of the old ones.

$$g_i = \sum_{j} e_j T_{ji} \quad G_{\text{new}} = [g_1, ..., g_{N^e}]$$

The newly derived vectors exhibit symmetry properties of the element under consideration. For a four noded isoparametric membrane element with 8 degrees of freedom, a series of transformations are performed on the canonical basis vector. The transformations consisted of rotations and projections [Stiefel 79]. The set of load configurations thus found is shown in Fig. 4.3.

This set of load combinations found by applying representation theory can be classified into the three categories representing rigid body modes $g_1$ to $g_3$, constant strain modes $g_4$ to $g_6$ and higher order (linear) strain modes $g_7$ and $g_8$ as in the finite element terminology. For example, the rotational rigid body mode $g_3$ is found from the canonical basis vectors $e_1$ to $e_8$ by
\[ g_3 = e_1 - e_2 + e_3 + e_4 - e_5 + e_6 - e_7 - e_8 \] (4.19)

for the degree of freedom order shown in Fig. 4.3. For the sake of conciseness, the transformation matrices of (4.18) are substituted by a set of \((\pm 1)\) scalar factors in (4.19).

Since rigid body modes produce no strains in the element, the state of strain \(\mathbf{\epsilon}\) within the element can be formulated as a linear combination in terms of the constant strain modes \(E_c\) and higher order strain modes \(E_h\).

\[ \mathbf{\epsilon} = E_c \mathbf{q}_c + E_h \mathbf{q}_h = \begin{bmatrix} E_c & E_h \end{bmatrix} \begin{bmatrix} \mathbf{q}_c \\ \mathbf{q}_h \end{bmatrix} = \mathbf{E} \mathbf{q} \] (4.20)

with \(\mathbf{q}\) being a vector of strain multipliers for the strain modes \(E\).

![Figure 4.3 Linearly independent nodal forces for an isoparametric membrane finite element](attachment:figure43.png)
The values of the real stresses $\sigma$ at each infinitesimal point of the finite element, for which dimensioning should take place, are not known. The principle of virtual work can, however, be applied in order to find statically equivalent generalized stresses $S_j$ which represent appropriate integrals of the real stresses $\sigma$. The same principle was used to find the nodal forces $p$ which again are statically equivalent to the real stresses $\sigma$. The vectors of virtual strains $\delta e$ are grouped in the strain matrix $E$ with $E_j = \delta e$. The corresponding virtual displacement vector $\delta a$ is summarized in the displacement matrix $\hat{U}$ with $\hat{U}_j = \delta a$. The following work equation can then be written

$$\delta q_j S_j = \int E_j^T \sigma \, dV = \hat{U}_j^T \int B^T \sigma \, dV = \hat{U}_j^T p \quad (4.21)$$

In (4.21), $\delta q_j$ is the virtual generalized strain associated with $S_j$. $E_j$ and $\hat{U}_j$ are the virtual element strains and nodal displacements that must be kinematically compatible with $\delta q_j$ and also satisfy the requirement that

$$E_j = B \hat{U}_j \quad (4.22)$$

For the $S_j$ components to be independent of one another, an additional condition to be imposed on the choice of the $E_j$ vectors is that they must be linearly independent, i.e. orthogonal to each other

$$\int E_i^T E_j \, dV = 0 \quad \text{if } i \neq j$$

$$\neq 0 \quad \text{if } i = j \quad \quad i,j = 1..m \quad (4.23)$$

Choosing virtual strain fields with a unit amplitude $\delta q_j = 1$, (4.21) reduces to

$$S_j = \int E_j^T \sigma \, dV = \hat{U}_j^T p \quad (4.24)$$

from which generalized stress components $S_j$ are found separately for pure membrane and bending elements.

In the following derivations, the thickness of the finite element is assumed to be constant and the reinforcement distribution is also assumed to remain unchanged within the element.

### 4.3 Generalized strains and stresses for a finite element

In this section, the generalized strains and stresses are derived for a quadrilateral finite element in membrane and bending.
4.3.1 Strain modes for membrane

Quadrilateral free formulation membrane elements are considered. A total of five independent strain modes are chosen, the first three of which correspond to constant strain modes whereas the additional two represent higher order (linear) modes.

In each of the three constant strain modes, one of the three strain components $\epsilon_x$, $\epsilon_y$ or $\gamma_{xy}$ is equal to 1 while the other two are zero. The two higher order modes are characterized by a linear variation of strain components $\epsilon_x$ and $\epsilon_y$ along the element’s $\xi$ and $\eta$ axes whose origin is at the element centroid. The strain vectors $\delta \epsilon$ thus chosen can be grouped in these matrices:

$$E_{mc} = \begin{bmatrix} 1 & . & . \\ . & 1 & . \\ . & . & 1 \end{bmatrix} \quad E_{mh} = \begin{bmatrix} \eta & . \\ . & \xi \\ . & . \end{bmatrix} \quad E_m = \begin{bmatrix} E_{mc} & E_{mh} \end{bmatrix} \quad (4.25)$$

Subscript $m$ stands for membrane while subscripts $c$ and $h$ refer to constant and higher order (linear) modes. $E_{mj}$ gives the strain component $i$ for the strain mode $j$. Consequently, each column of the $E_m$ matrix represents a strain mode where one strain component is non-zero while the others vanish.

The nodal displacements of a finite element which produce the above strains can be found using the operator matrix $A_m$ defined by (2.3) if the coordinate axes notation is changed from $x - y$ to $\xi - \eta$. This establishes the relationship between the strain matrix $E_m$ and the displacement matrix $U_m^d$ as

$$E_m = A_m U_m^d \quad (4.26)$$

The nodal displacement matrices will then be

$$U_m^d = \begin{bmatrix} \xi & . & \eta^2/2 \\ . & \eta & \xi^2/2 \\ . & . & . \end{bmatrix} \quad U_m^h = \begin{bmatrix} \xi \eta & -\eta^2/2 \\ -\xi^2/2 & \xi \eta \\ . & . & . \end{bmatrix} \quad (4.27)$$

$$U_{mc} = \begin{bmatrix} U_m^d \\ U_m^r \end{bmatrix} \quad U_{mh} = \begin{bmatrix} U_m^d \\ U_m^r \end{bmatrix} \quad U_m = \begin{bmatrix} U_{mc} & U_{mh} \end{bmatrix} \quad (4.29)$$

$U_{mij}$ gives the displacement component $i$, namely $u$ and $v$ for a unit strain $j$. $U_{mij}$ gives the rotational component which, for a free formulation membrane element model, is defined by (6.3). For an isoparametric membrane element model, $U_{mc}$ and $U_{mh}$ do not exist.
The strain modes $E_m$, i.e. the sub-vectors $E_{mj}$ are shown in Fig. 4.4 for a rectangular element. For better visualization, instead of taking a unit strain, the constant strain modes are plotted for a strain of 0.5 while the linear strain modes have a factor of 0.2.

![Figure 4.4 Strain modes for a rectangular membrane free formulation finite element](image)

The generalized stresses $S_m$ corresponding to the five strain modes $E_m$ and derived from (4.24) are given by

$$
S_{m1} = \int_A n_\zeta \, dA \quad S_{m2} = \int_A n_\eta \, dA \quad S_{m3} = \int_A n_\zeta\eta \, dA
$$

$$
S_{m4} = \int_A n_\zeta \, dA \quad S_{m5} = \int_A n_\eta \, dA
$$

(4.30)

(4.31)

It should be noted that due to the choice of the generalized stresses $n_\zeta$, $n_\eta$ and $n_\zeta\eta$ for the stress vector $\sigma_m$ which represents the total force over the thickness, the integrals over the element volume reduce to integrals over the element area. The three components in (4.30) represent stress integrals over the element volume while the two generalized stresses of (4.31) act like a moment about the element centroid. Indeed, it resembles the bending moment of a rigid ideally plastic beam cross section about its neutral axis.

The numerical evaluation of $S_m$ defined by (4.30) and (4.31) is based on the formula specified by (4.24). The 3x5 $U_m$ matrix given by (4.29) is found for each element’s nodal points by substituting the coordinates of the node in $U_m$. Summarizing the matrices thus found in a $\hat{U}_m$ matrix
\[ \hat{U}_m = \begin{bmatrix} U_{m1}^1 \\ \vdots \\ U_{m4}^4 \end{bmatrix} \] (4.32)

we are now able to find \( S_m \) by multiplying \( \hat{U}_m \) by the vector of membrane nodal forces \( p_m \).

\[ S_m = U_m^T p_m \] (4.33)

### 4.3.2 Strain modes for bending

Quadrilateral free formulation bending elements are considered in this section. A total of five independent strain modes are chosen, the first three of which correspond to constant strain modes whereas the additional two represent higher order (linear) modes.

In each of the three constant strain modes, one of the three curvature components \( \chi_{\xi}, \chi_{\eta} \) and \( \chi_{\xi\eta} \) is equal to 1 while the other two are 0. The two higher order modes are characterized by a linear variation of the curvature components \( \chi_{\xi} \) and \( \chi_{\eta} \) along the element’s \( \xi \) and \( \eta \) axes whose origin is at the element centroid. The strain vectors \( \delta \epsilon \) so chosen are grouped in these matrices:

\[ E_{bc} = \begin{bmatrix} 1 & . & . \\ . & 1 & . \\ . & . & 1 \end{bmatrix} \quad E_{fh} = \begin{bmatrix} \xi \\ . \\ . \end{bmatrix} \quad E_b = \begin{bmatrix} E_{bc} & E_{fh} \end{bmatrix} \] (4.34)

\( E_{bij} \) gives the strain component \( i \) for the strain mode \( j \). The two strain components here are the two curvatures \( \chi_{\xi} \) and \( \chi_{\eta} \) and the twist \( \chi_{\xi\eta} \). Consequently, each column of the \( E_b \) matrix represents a strain mode where one strain component is equal to 1 while the others vanish. Subscript \( m \) stands for bending while subscripts \( c \) and \( h \), as before, refer to constant and higher (linear) order modes.

Using the operator matrix \( A_b \) defined by (2.18) and having

\[ E_b = A_b U_b^d \] (4.35)

the nodal displacements and rotations of a finite element which produce the above curvatures can be found from
4.3 Generalized strains and stresses for a finite element

\[
U_{bc}^r = \begin{bmatrix}
\xi
& \eta
d\xi
& d\eta
\end{bmatrix}
\quad U_{bh}^r = \begin{bmatrix}
\frac{\xi^2}{2}
d\xi
& d\eta
\frac{\eta^2}{2}
\end{bmatrix}
\]

(4.36)

\[
U_{bc}^d = \begin{bmatrix}
\frac{\xi^2}{2} & \frac{\eta^2}{2} & \frac{\xi\eta}{2}
\end{bmatrix}
\quad U_{bh}^d = \begin{bmatrix}
\frac{\xi^3}{6}
& \frac{\xi\eta^2}{6}
\end{bmatrix}
\]

(4.37)

\[
U_{bc} = \begin{bmatrix}
U_{bc}^r
U_{bc}^d
\end{bmatrix}
\quad U_{bh} = \begin{bmatrix}
U_{bh}^r
U_{bh}^d
\end{bmatrix}
\quad U_b = \begin{bmatrix}
U_{bc}
U_{bh}
\end{bmatrix}
\]

(4.38)

\(U_{bc}^d\) gives the displacement component \(w\) for a unit curvature (or twist) \(j\). \(U_{bh}^r\) gives the rotational components \(\xi\) and \(\eta\) which are functions of \(w\) and were defined by (2.16).

The strain modes \(E_b\), i.e. the sub-vectors \(E_{bj}\) are shown in Fig. 4.5 for a rectangular element. No precise scale holds among the different modes.

\[\text{Figure 4.5 Strain modes for a rectangular bending finite element}\]

The generalized stresses \(S_b\) corresponding to the five strain modes \(E_b\) and derived from (4.24) are given by

\[
S_{b1} = \int_A m_\xi dA \quad S_{b2} = \int_A m_\eta dA \quad S_{b3} = \int_A m_{\xi\eta} dA
\]

(4.39)

\[
S_{b4} = \int_A \xi m_\xi dA \quad S_{b5} = \int_A \eta m_\eta dA
\]

(4.40)
The numerical evaluation of $S_b$ defined by (4.39) and (4.40) is again based on the formula specified by (4.24). The $3 \times 5$ $U_b$ matrix given by (4.38) is calculated for each element’s nodal points by substituting the coordinates of the node in $U_b$. Summarizing the matrices thus found in a $\hat{U}_b$ matrix

$$\hat{U}_b = \begin{bmatrix} U_b^T_{1} \\ \vdots \\ U_b^T_{4} \end{bmatrix}$$

(4.41)

$S_b$ can then be found by multiplying $\hat{U}_b$ by the vector of bending nodal forces $p_b$.

$$S_b = \hat{U}_b^T p_b$$

(4.42)

### 4.4 Yield conditions for a quadrilateral finite element

In this section, the yield conditions for a quadrilateral finite element for the three cases of membrane, bending and membrane-bending actions are derived. In each case, they are first found for an orthogonally reinforced rectangular finite element. These yield conditions are then extended to an arbitrarily-shaped quadrilateral element with several steel layers.

The aim is to formulate linearized yield conditions in terms of the generalized stresses $S$. For membrane and bending elements each with 5 generalized stress components as derived in section 4.3, the yield conditions define a polyhedral yield surface in the five dimensional $S$ space. It can be found on the basis of the lower bound theorem of the plasticity theory stating that if any stable, statically admissible stress field is found, then structural safety is guaranteed.

Among all possible statically admissible stress fields, one suitable stress field is therefore chosen and the yield conditions formulated for this admissible stress state $\sigma^a$. For this purpose, every $S_j$ is associated with a statically equivalent stress field $\sigma_j^a$ whose work on the virtual strain $E_j$ for an amplitude $\delta q_j = 1$ is equal to the work done by the real but unknown stresses $\sigma$ within the element on $E_j$ and, therefore, equal to the work done by the known element nodal forces $p$ on the virtual nodal displacements $\hat{U}_j$.

$$S_j = \int E_j^T \sigma dV = \int E_j^T \sigma^a dV = \hat{U}_j^T p$$

(4.43)

In the context of the virtual work principle, each $S_j$ is in equilibrium with its associated $\sigma_j^a$, or in other words, the generalized stresses $S$ and $\sigma^a$ are in equilibrium with the real element stresses $\sigma$ and nodal forces $p$. 

60
4.4 Yield conditions for a quadrilateral finite element

For the sake of clarity, the approach leading to the derivation of the linearized yield conditions for a membrane finite element is first explained for a rectangular element with orthogonal reinforcement (Fig. 4.6 a). The modifications necessary for an extension to an arbitrarily-shaped quadrilateral element with different reinforcement layers are discussed afterwards.

The admissible stress field $\sigma^a$ is chosen in the following manner: The element is subdivided by its natural coordinate system $\xi - \eta$ with its origin at the element centroid into four quadrants. The stress state within each quadrant is assumed to be constant while it normally varies from one quadrant to the other (Fig. 4.6 b).

![Figure 4.6 Rectangular membrane finite element](image)

The stresses $\sigma_{ni}^a$ corresponding to $S_{ni}$ for the first three components $i = 1, 2, 3$ are chosen the same in all four quadrants with a magnitude of $S_{ni}/A$. The stress distribution in each quadrant for $S_{m4}$ and $S_{m5}$ is shown in Fig. 4.7.

![Figure 4.7 Admissible stress state for linear modes of membrane quadrilateral finite elements](image)

The magnitudes of $\sigma_{m4}^a$ and $\sigma_{m5}^a$ are given by

$$\sigma_{m4}^a = \frac{S_{m4}}{L_\xi}$$

$$\sigma_{m5}^a = \frac{S_{m5}}{L_\eta}$$

(4.44)
in which the first moments of area are defined by

\[ I_{\xi} = \int_A |\eta| \, dA = \bar{\eta} A \quad I_{\eta} = \int_A |\xi| \, dA = \bar{\xi} A \]  \hspace{1cm} (4.45)

For the rectangular element of Fig. 4.6 (a) we have

\[ I_{\xi} = 2b^2a \quad I_{\eta} = 2a^2b \quad A = 4ab \quad \bar{\eta} = \frac{b}{2} \quad \bar{\xi} = \frac{a}{2} \]  \hspace{1cm} (4.46)

The relationship between the admissible stress field \( \sigma_m^a \) and the generalized stress vector \( S_m \) can therefore be summarized as

\[
\begin{bmatrix}
  n_{\xi}^a \\
  n_{\eta}^a \\
  n_{\xi\eta}^a
\end{bmatrix}
= \frac{1}{A}
\begin{bmatrix}
  1 & \ldots & \pm \frac{1}{\bar{\eta}} \\
  \ldots & \ldots & \ldots \\
  \ldots & \ldots & \ldots
\end{bmatrix}
\begin{bmatrix}
  S_{m1} \\
  S_{m2} \\
  S_{m3} \\
  S_{m4} \\
  S_{m5}
\end{bmatrix}
\hspace{1cm} \sigma_m^a = \frac{1}{A} I_m^1 S_m \hspace{1cm} (4.47)
\]

For each of the four quadrants, the \( I_m^1 \) matrix with the appropriate sign for the last two columns as shown in Fig. 4.6 (b) will be chosen and the corresponding \( \sigma_m^a \) will then be found.

The linearized yield conditions derived for an infinitesimal membrane element and formulated in Fig. 3.6 are restated here. The 10 equations comprise 4 conditions (4.48) for reinforcement and 6 conditions (4.49) and (4.50) for concrete.

\[
\begin{align*}
  n_{\xi} \pm n_{\xi\eta} & \leq r_{\xi}^i \\
n_{\eta} \pm n_{\xi\eta} & \leq r_{\eta}^i \\
n_{\xi} \pm n_{\xi\eta} & \leq r^c \\
n_{\eta} \pm n_{\xi\eta} & \leq r^c \\
\pm 2n_{\xi\eta} & \leq r^c
\end{align*}
\hspace{1cm} (4.48) \hspace{1cm} (4.49) \hspace{1cm} (4.50)
\]

The constant stress state in each of the four quadrants must satisfy the yield conditions (4.48) to (4.50). In the general case, this will result in a total of 40 inequalities. For the special case of the rectangular element of Fig. 4.6, these 40 constraints reduce to the following 18 inequalities because some of the constraints in the four quadrants are identical.

\[
\begin{align*}
  \frac{S_{m1}}{A} \pm \frac{S_{m3}}{A} \pm \frac{S_{m4}}{A\bar{\eta}} & \leq r_{\xi}^i \\
  \frac{S_{m2}}{A} \pm \frac{S_{m3}}{A} \pm \frac{S_{m5}}{A\bar{\xi}} & \leq r_{\eta}^i \\
  -\frac{S_{m1}}{A} \pm \frac{S_{m3}}{A} \pm \frac{S_{m4}}{A\bar{\eta}} & \leq r^c \\
  -\frac{S_{m2}}{A} \pm \frac{S_{m3}}{A} \pm \frac{S_{m5}}{A\bar{\xi}} & \leq r^c \\
  \pm \frac{2S_{m3}}{A} & \leq r^c
\end{align*}
\hspace{1cm} (4.51) \hspace{1cm} (4.52) \hspace{1cm} (4.53)
\]
4.4 Yield conditions for a quadrilateral finite element

A comparison between these two sets of yield conditions is interesting. \( S_{m1}/A \), \( S_{m2}/A \) and \( S_{m3}/A \) represent averages of the real (generalized) stresses \( n_\xi \), \( n_\eta \) and \( n_\xi \eta \) over the element area. Apart from the \( S_{m4} \) and \( S_{m5} \) terms, the yield conditions for the non-infinitesimal finite element therefore resemble those of an infinitesimal element if the stresses \( n_\xi \), \( n_\eta \) and \( n_\xi \eta \) were substituted by their corresponding averaged values. The additional \( S_{m4} \) and \( S_{m5} \) terms take into account the transition from an infinitesimal to a finite element.

![Figure 4.8 Reinforcement dimensioning direction for membrane finite element](image)

A general quadrilateral element with non-orthogonal reinforcement as shown in Fig. 4.8 will now be handled. The reinforcement over the element area is constant and consists of a set of steel layers in different directions. An orthogonal \( r - s \) system making an angle \( \beta \) with the \( x \) axis is assumed. Its value can be specified interactively by the program user. Indeed, the \( r - s \) coordinate system is the one in which the linearized yield conditions have to be formulated and the reinforcement dimensioning will take place. Using (3.13) the equivalent resistances in the \( r - s \) system can be evaluated. The generalized stresses \( S_m \) are formulated with respect to the element’s natural coordinate axes \( \xi - \eta \) whose origin is at the centroid of the element.

![Figure 4.9 Stress transformation to a rotated coordinate system](image)
The three components of the stress vector $\sigma$ in the membrane state can be transformed from an $x - y$ system into a rotated $r - s$ system (Fig. 4.9) using the following transformation matrix

$$\sigma_{rs} = T \sigma_{xy} \begin{bmatrix} n_r \\ n_s \\ n_{rs} \end{bmatrix} = \begin{bmatrix} \cos^2\psi & \sin^2\psi & \sin 2\psi \\ \sin^2\psi & \cos^2\psi & -\sin 2\psi \\ -\sin \psi \cos \psi & \sin \psi \cos \psi & \cos 2\psi \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_{xy} \end{bmatrix}$$

(4.54)

The stresses in each of the four quadrants are calculated in the dimensioning $r - s$ system using

$$\sigma_{m}^{n} = \frac{1}{A} T I_m^l S_m$$

(4.55)

The resistance vector for all the quadrants is the same and is calculated from (3.11) where $r^c$ is defined by (3.5) while the components of $r^s$ are found from (3.13). These are substituted in

$$N_m \sigma_{m}^{n} - r_m \leq 0$$

(4.56)

with the normal matrix $N_m$ defined by (3.8). This finally gives

$$N_m T I_m^l S_m - A r_m \leq 0$$

(4.57)

In (4.57), since the configuration of the reinforcement in a finite element is assumed to be constant, the transformation matrix $T$ and the resistance vector $r_m$ will be the same for all four regions $A$, $B$, $C$ and $D$. The form of the constraints which (4.57) leads to is demonstrated in Fig. 4.10. In each region, the first four constraints correspond to tension in steel, the second four constraints are related to compression in concrete and the last two accommodate shear in concrete. A comparison of Fig. 4.10 with the formulae given by (4.51) to (4.53) for a rectangular element shows that each yield condition for reinforcement in tension and concrete in compression appears twice, e.g. the inequalities formulated for reinforcement in $\xi$ direction using (4.51) for region $A$ is the same as those of region $B$. As a result, checking only one of the two regions is sufficient for $r^c_\xi$. The total 32 inequalities, therefore, reduce to 16. The additional 8 constraints corresponding to shear in concrete reduce to 2 for the rectangular case considered before. With this consideration and having in mind that for the general case of non-rectangular elements, the difference between these pairs of similar inequalities is rather marginal, it was possible to reduce the number of inequalities from 40 to 24 in which the considered constraints in each region are marked with a grey box in Fig. 4.10. Except for shear in concrete, which is controlled everywhere, the directions in each region where the rest of the constraints are checked are also shown in Fig. 4.10.
The inequalities of (4.57) can therefore be given in the condensed form of

\[ \widetilde{N}_m S_m - A \tilde{r}_m \leq 0 \]  

(4.58)

where the 24x5 matrix \( \widetilde{N}_m \) is constructed by evaluating the matrix multiplication \( N_m T^T_m \) for each of the components with grey-shaded background in Fig. 4.10. The 24x1 resistance vector \( \tilde{r}_m \) is given by

\[ \tilde{r}_m^T = \begin{bmatrix} \tilde{r}_m^s \cr \tilde{r}_m^c \end{bmatrix} \]  

(4.59)

\[ \tilde{r}_m^s = \begin{bmatrix} r_1^s & r_2^s & r_3^s & r_4^s & r_5^s & r_6^s \end{bmatrix} \]  

(4.60)

\[ \tilde{r}_m^c = \begin{bmatrix} r^c & \ldots & r^c \cr \frac{r^c}{2} & \ldots & \frac{r^c}{2} \end{bmatrix} \]  

(4.61)

with \( r_j^c \) given by (3.13).

### 4.4.2 Yield conditions for a bending finite element

The derivation of the yield conditions for the bending finite element follows the same principles outlined before where an admissible stress field \( \sigma^a \) was chosen.

Here again, the linearized yield conditions for a bending finite element are first derived for a rectangular element with orthogonal reinforcement in the top and bottom layers (Fig. 4.11 a). The extension to an arbitrarily-shaped quadrilateral element with a series of reinforcement fields at the two layers is discussed afterwards.
4. First Concept: Element-based Dimensioning and Yield Conditions for a Finite Element

The admissible stress field $\sigma^a_b$ is chosen by subdividing the element into four quadrants with a constant stress distribution and the origin of the natural coordinate system $\xi - \eta$ at the element centroid (Fig. 4.11 b). The stresses $\sigma^a_{bi}$ corresponding to $S_{bi}$ for the first three components $i = 1, 2, 3$ are chosen to be the same in all four quadrants with a magnitude of $S_{bi}/A$. The stress distribution in each quadrant for $S_{b4}$ and $S_{b5}$ is shown in Fig. 4.12.

\begin{equation}
\begin{pmatrix}
m^a_\xi \\
m^a_\eta \\
m^a_{\xi\eta}
\end{pmatrix} = \begin{bmatrix}
1 & \pm \frac{1}{\xi} & . \\
. & 1 & \pm \frac{1}{\eta} \\
. & . & 1
\end{bmatrix} \begin{pmatrix}
S_{b1} \\
S_{b2} \\
S_{b3} \\
S_{b4} \\
S_{b5}
\end{pmatrix}
\end{equation}

The linearized yield conditions derived for the reinforcement of an infinitesimal bending element and formulated in Fig. 3.9 are restated here in the element’s $\xi - \eta$ system.

\begin{align}
m_\xi \pm m_{\xi\eta} & \leq M_{p\xi}^b \\
m_\eta \pm m_{\xi\eta} & \leq M_{p\eta}^b \\
-m_\xi \pm m_{\xi\eta} & \leq M_{p\xi}^I \\
-m_\eta \pm m_{\xi\eta} & \leq M_{p\eta}^I
\end{align}
By substituting the values for moments from (4.62) into (4.63) and (4.64) we will get a total of 32 inequalities. For the special case of the rectangular element of Fig. 4.11, these 32 constraints reduce to the following 16 inequalities because the other 16 would be identical

\[
\frac{S_{b1}}{A} \pm \frac{S_{b3}}{A} \pm \frac{S_{b4}}{A\xi} \leq M_{p\xi}^b
\]
\[
\frac{S_{b2}}{A} \pm \frac{S_{b3}}{A} \pm \frac{S_{b5}}{A\eta} \leq M_{p\eta}^b
\]
\[
\frac{-S_{b1}}{A} \pm \frac{S_{b3}}{A} \pm \frac{S_{b4}}{A\xi} \leq M_{p\xi}^t
\]
\[
\frac{-S_{b2}}{A} \pm \frac{S_{b3}}{A} \pm \frac{S_{b5}}{A\eta} \leq M_{p\eta}^t
\]

The transition from infinitesimal yield conditions (4.63) and (4.64) to those for a finite element given by (4.65) and (4.66) in bending is analogous to the case of the membrane state discussed before.

In the general case of an arbitrary element shape and steel layers, a sandwich model is adopted, in which several reinforcement fields in arbitrary directions are located at the top and bottom layers (Fig. 4.13). Dimensioning directions can be specified interactively by the user for each of the two layers. An equivalent orthogonal reinforcement layout is then found with respect to these directions.

![Figure 4.13 Reinforcement dimensioning direction for bending finite element](image)

The stresses in each of the four quadrants are calculated in the \( r - s \) dimensioning system using

\[
\sigma_m^{ab} = \frac{1}{A\delta_{r}} T^b I^b_s S_b
\]
\[
\sigma_m^{st} = \frac{1}{A\delta_{s}} T^t I^t_s S_h
\]

with the transformation matrices \( T^b \) and \( T^t \) found by substituting the corresponding dimensioning angles of the two layers in (4.54). \( \delta_{r} \) is the distance between the bottom and top steel layers (Fig. 3.13). The resistance vector for all quadrants in each layer is the same and is
found from (3.11) where the components of \( r^b_{m} \) and \( r^s_{m} \) are found from (3.13) for the bottom and top layers, respectively. They will then be substituted in

\[
N^s_m a^b_m - r^b_m \leq 0 \quad \quad \quad N^s_m a^s_m - r^s_m \leq 0
\]

with the normal matrix \( N^s_m \) defined by (3.9). This finally gives

\[
N^s_m T^b I^b_b S_b - A d_q r^b_m \leq 0 \quad \quad \quad N^s_m T^s I^s_b S_b - A d_q r^s_m \leq 0
\]

The form of the constraints which (4.69) leads to is shown in Fig. 4.14. Comparing Fig. 4.14 with the formulae given by (4.65) and (4.66) for a rectangular element shows that each yield condition arises twice. The corresponding 32 inequalities reduce therefore to 16. As in the membrane case, this fact was exploited so that 16 constraints were chosen to be controlled for the reinforcement. The selected constraints in each region are marked with a grey box in Fig. 4.14. The directions in each region where the constraints are formulated are also shown in the same figure.

\[
A \quad B \quad C \quad D
\]

\[
\begin{array}{c|c|c|c|c}
 r^s_m & \xi + \xi \eta & \eta + \xi \eta & \xi + \xi \eta & \eta + \xi \eta \\
 r^s_m & \eta + \xi \eta & \eta + \xi \eta & \eta + \xi \eta & \eta + \xi \eta \\
 r^s_m & \xi + \xi \eta & \xi + \xi \eta & \xi + \xi \eta & \xi + \xi \eta \\
 r^s_m & \eta + \xi \eta & \eta + \xi \eta & \eta + \xi \eta & \eta + \xi \eta \\
\end{array}
\]

\textbf{Figure 4.14 Selected constraints in the four zones of a bending finite element}

The inequalities of (4.69) can be given in the condensed form of

\[
\tilde{N}_{mb} S_b - A d_q \tilde{r}^s_m \leq 0 \quad \quad \quad \tilde{N}_{mb} S_b - A d_q \tilde{r}^s_m \leq 0
\]

where the 16x5 matrices \( \tilde{N}_{mb} \) and \( \tilde{N}_{mb} \) are constructed by evaluating the matrix multiplications \( N^s_m T^s I^s_b \) and \( N^s_m T^b I^b_b \) for each of the grey-colored components in Fig. 4.14. The 16x1 resistance vectors \( \tilde{r}^s_m \) and \( \tilde{r}^s_m \) are found by determining the following vector for the two layers.

\[
\tilde{r}^s_m = \{ r^s_1 \quad r^s_2 \quad r^s_3 \quad r^s_4 \quad r^s_5 \quad r^s_6 \quad r^s_7 \quad r^s_8 \quad r^s_9 \quad r^s_{10} \quad r^s_{11} \quad r^s_{12} \quad r^s_{13} \quad r^s_{14} \quad r^s_{15} \quad r^s_{16} \}
\]

with \( r^s_j \) given by (3.13).
4.4.3 Yield conditions for a membrane-bending finite element

In section 3.2.2, a sandwich model was adopted to derive the constraints for an infinitesimal element subjected to both membrane forces and bending moments. There, the membrane forces were split equally between the two layers and the corresponding yield conditions were superimposed on the one for the bending moments. This gives the following inequalities for reinforcement in the two directions of the top and bottom layers.

\[
\frac{n_\xi}{2} + \frac{m_\xi}{d_q} \pm \left( \frac{n_\eta}{2} + \frac{m_\eta}{d_q} \right) \leq r_b^b \\
\frac{n_\eta}{2} + \frac{m_\eta}{d_q} \pm \left( \frac{n_\xi}{2} + \frac{m_\xi}{d_q} \right) \leq r_b^b 
\]  
(4.72)

\[
\frac{n_\xi}{2} - \frac{m_\xi}{d_q} \pm \left( \frac{n_\eta}{2} - \frac{m_\eta}{d_q} \right) \leq r_r^b \\
\frac{n_\eta}{2} - \frac{m_\eta}{d_q} \pm \left( \frac{n_\xi}{2} - \frac{m_\xi}{d_q} \right) \leq r_r^b 
\]  
(4.73)

A similar approach is taken here by combining the two cases presented in the new approach of element-based design using the formulae given in sections 4.4 and 3.2.2. If in (4.72) and (4.73), \( n_\xi, n_\eta, m_\xi, m_\eta \) are replaced by their equivalent values for the finite element-based dimensioning model from (4.47) and (4.62) we get

\[
\frac{S_m}{2A} \pm \frac{S_{m4}}{2A^2} + \left( \frac{S_{b1}}{A_{d,q}} \pm \frac{S_{b4}}{A_{d,q}} \right) \pm \left( \frac{S_{m3}}{2A} + \frac{S_{b3}}{A_{d,q}} \right) \leq r_b^b 
\]  
(4.74)

\[
\frac{S_m}{2A} \pm \frac{S_{m5}}{2A^2} + \left( \frac{S_{b2}}{A_{d,q}} + \frac{S_{b5}}{A_{d,q}} \right) + \frac{S_{m3}}{2A} + \frac{S_{b3}}{A_{d,q}} \leq r_r^b 
\]  
(4.75)

\[
\frac{S_m}{2A} \pm \frac{S_{m4}}{2A^2} + \left( \frac{S_{b1}}{A_{d,q}} \pm \frac{S_{b4}}{A_{d,q}} \right) \pm \frac{S_{m3}}{2A} - \frac{S_{b3}}{A_{d,q}} \leq r_r^b 
\]  
(4.76)

\[
\frac{S_m}{2A} \pm \frac{S_{m5}}{2A^2} + \left( \frac{S_{b2}}{A_{d,q}} \pm \frac{S_{b5}}{A_{d,q}} \right) \pm \frac{S_{m3}}{2A} - \frac{S_{b3}}{A_{d,q}} \leq r_r^b 
\]  
(4.77)

Constraints (4.74) to (4.77) lead to 32 inequalities in which both constant modes for membrane and bending as well as linear modes for the two are included.

For the general case of arbitrary reinforcement in the two layers, the yield conditions for an infinitesimal element given by (3.23) and (3.24) are given with the stress states \( \sigma_m \) and \( \sigma_b \) being substituted by statically admissible stress states \( \sigma_m^a \) and \( \sigma_b^a \)

\[
\frac{1}{2} N_m \sigma_m - \frac{1}{d_q} N_m' \sigma_b^a - r_m^{ab} \leq 0 
\]  
(4.78)

\[
\frac{1}{2} N_m \sigma_m + \frac{1}{d_q} N_m' \sigma_b^a - r_m^{ab} \leq 0 
\]  
(4.79)
Substituting (4.55) and (4.67) for \( \sigma_{m}^{x} \) and \( \sigma_{b}^{y} \) gives

\[
\frac{1}{2} N_{m}^{s} T^{I} I_{m} S_{m} - \frac{1}{d_{q}} N_{m}^{s} T^{I} I_{b} S_{b} - A r_{m}^{st} \leq 0 \tag{4.80}
\]

\[
\frac{1}{2} N_{m}^{s} T^{I} I_{m} S_{m} + \frac{1}{d_{q}} N_{m}^{s} T^{I} I_{b} S_{b} - A r_{m}^{sb} \leq 0 \tag{4.81}
\]

in which \( N_{m}^{s} \) is given by (3.9), \( T^{I} \) and \( T^{b} \) by (4.54) calculated separately for the top and bottom layers, \( I_{m} \) by (4.47), \( I_{b}^{I} \) by (4.62), \( S_{m} \) by (4.33), \( S_{b} \) by (4.42) and finally \( r_{m}^{st} \) and \( r_{m}^{sb} \) evaluated from (3.11) for the top and bottom layers, respectively.

The same selection of regions as shown in Fig. 4.14 is applied for the bending contributions. For the membrane contributions, the first two rows of the selections shown in Fig. 4.10 are taken for both the top and bottom layers.

The inequalities of (4.80) and (4.81) can be given in the condensed form of

\[
\frac{1}{2} \tilde{N}_{m}^{st} S_{m} - \frac{1}{d_{q}} \tilde{N}_{mb}^{st} S_{b} - A \tilde{r}_{m}^{st} \leq 0
\tag{4.82}
\]

\[
\frac{1}{2} \tilde{N}_{m}^{sb} S_{m} + \frac{1}{d_{q}} \tilde{N}_{mb}^{sb} S_{b} - A \tilde{r}_{m}^{sb} \leq 0
\tag{4.83}
\]

The 16x5 matrices \( \tilde{N}_{m}^{st} \) and \( \tilde{N}_{m}^{sb} \) are constructed by evaluating the matrix multiplications \( N_{m}^{s} T^{I} I_{m} \) and \( N_{m}^{s} T^{I} I_{b}^{I} \) for the two layers based on the grey-shaded components for steel in Fig. 4.10. The 16x5 matrices \( \tilde{N}_{mb}^{st} \) and \( \tilde{N}_{mb}^{sb} \) are similarly constructed by calculating the matrix multiplications \( N_{m}^{s} T^{I} I_{b} \) and \( N_{m}^{s} T^{I} I_{b}^{I} \) for each of the grey-shaded components in Fig. 4.14.

The 16x1 resistance vectors \( \tilde{r}_{m}^{st} \) and \( \tilde{r}_{m}^{sb} \) are found using (4.71).

### 4.5 Yield conditions for a triangular finite element

For the triangular elements, only the constant strain modes are taken into consideration and the higher order modes are left out. The \( I_{m}^{I} \) matrix in (4.47) is changed to a 3x3 unit matrix. The generalized stress vectors for membrane and bending will be

\[
S_{m} = \begin{cases} S_{m1} \\ S_{m2} \\ S_{m3} \end{cases} \quad S_{b} = \begin{cases} S_{b1} \\ S_{b2} \\ S_{b3} \end{cases}
\tag{4.84}
\]
The $E_{m h}$, $U_{mh}$, $E_{bh}$ and $U_{bh}$ matrices corresponding to the higher order modes do not exist and the only contribution to the $E$ and $U$ matrices is from the constant modes. The $\hat{U}_m$ and $\hat{U}_b$ matrices are still found by the same form as described in (4.32) and (4.41) taking only three nodes. Subsequently, $S_m$ and $S_b$ are calculated using (4.33) and (4.42) with the corresponding element nodal force vectors $p_m$ and $p_b$. The yield conditions (4.57) for membrane, (4.69) for bending and (4.80) and (4.81) for the combined membrane-bending actions still hold with the new definitions for $I_m^1$, $S_m$, $I_b^1$ and $S_b$. It can be seen that for triangular elements, this is equivalent to using the conventional yield conditions formulated for an infinitesimal element by replacing the stress components $a_m$ and $a_b$ by their corresponding averaged values over the element area, namely $S_m/A$ and $S_b/A$, respectively.

### 4.6 Transverse shear in bending finite elements

In the sandwich model of section 3.2.2, it was shown how the transverse shear is dealt with for an infinitesimal element under bending action. A similar procedure is used here to handle this problem in the finite element-based dimensioning.

As explained in section 2.1.2, the shear stresses in the classical Kirchhoff theory are viewed as generalized reactions. Based on the principle of virtual work and similar to the membrane and bending cases, a displacement field can still be defined which furnishes average transverse shear stress values for a finite element. This is given by

$$
U^{r \ q}_{bc} = \begin{bmatrix}
\cdot & \cdot \\
\cdot & \cdot 
\end{bmatrix} \quad U^{d \ q}_{bc} = \begin{bmatrix}
-\xi & -\eta 
\end{bmatrix} \quad U^q_{bc} = \begin{bmatrix}
U^{r \ q}_{bc} \\
U^{d \ q}_{bc}
\end{bmatrix}
$$

(4.85)

The corresponding strain modes for transverse shear in bending are shown in Fig. 4.15.

![Figure 4.15 Strain modes for transverse shear (with no rotations) in bending](image)

The stresses $S^q_{b1}$ for transverse shear action in bending elements are given by

$$
S^q_{b1} = \int_A q_x \, dA \quad S^q_{b2} = \int_A q_y \, dA
$$

(4.86)
The numerical evaluation of $S_b^q$ defined by (4.86) is accomplished by multiplying $\tilde{U}_b^q$ by the vector of bending nodal forces $p_b$.

\[
\tilde{U}_b^q = \begin{bmatrix}
U_b^{q_1} \\
\vdots \\
U_b^{q_4}
\end{bmatrix}
\]

(4.87)

\[
S_b^q = \tilde{U}_b^q p_b
\]

(4.88)

The admissible stress field $\sigma_{b1}^{qa}$ corresponding to $S_b^q$ is chosen by assuming a constant distribution over the entire element area.

\[
\sigma_{b1}^{qa} = \frac{S_b^q}{A} \quad \sigma_{b2}^{qa} = \frac{S_b^q}{A}
\]

(4.89)

The admissible transverse shear stresses $S_{b1}^q$ and $S_{b2}^q$ are substituted into the equations given by (3.29) for $q_x$ and $q_y$ which give

\[
\sigma_x^q = \frac{(S_{b1}^q)^2}{2(S_{b0}^q)\tan \theta} \quad \sigma_y^q = \frac{(S_{b2}^q)^2}{2(S_{b0}^q)\tan \theta} \quad \tau_{xy}^q = \frac{(S_{b1}^q)(S_{b2}^q)}{2(S_{b0}^q)\tan \theta}
\]

(4.90)

with $S_{b0}^q$ being the principal generalized transverse shear stress and defined as

\[
S_{b0}^q = \sqrt{(S_{b1}^q)^2 + (S_{b2}^q)^2}
\]

(4.91)

If the effect of the transverse shear in the amount of the layer reinforcement of the two layers is to be incorporated according to the model explained in [Marti 90], then the stress values $\sigma_x^q \pm \tau_{xy}^q$ and $\sigma_y^q \pm \tau_{xy}^q$ from (4.90) must be added to the left hand sides of the appropriate equations of (4.74) to (4.77).

Condition (3.27) which determined whether transverse reinforcement is required becomes

\[
\frac{S_{b0}^q}{d_q} < \tau_c
\]

(4.92)

If (4.92) is not satisfied then the amount of the required transverse reinforcement ratio will be

\[
\rho_q = \frac{(S_{b0}^q)\tan \theta}{d_q f_y}
\]

(4.93)
A few remarks should be made here concerning reinforcement dimensioning for transverse shear:

- Due to brittle concrete behavior, shear collapses are normally without prior warning and therefore are dangerous. Special care should be taken when dealing with high transverse shear forces, in particular at the points of application of concentrated forces, columns and along boundaries of panel connections.

- A uniform assumption of shear distribution over the entire finite element area is only justified if the element area is small enough or, alternatively, if the variation of shear forces within the element is not significant.

- In the finite element analysis, the values calculated for transverse shear are in general less accurate than those of the moments, since shear forces represent derivatives of bending moments.

### 4.7 Mesh parameter for error estimation

The underlying concepts for handling each finite element as a dimensioning unit is independent of the element model but the yield conditions do depend to some extent on the type of element model adopted. Since in the new approach, the yield conditions are directly derived for the element nodal forces, a general statement which could be made here is that as the number of degrees of freedom of an element model increases, the complexities with regard to both the derivation and number of yield condition constraints grows.

It was shown in Fig. 4.3 that for a quadrilateral isoparametric membrane finite element, the 5 generalized stresses can describe all possible nodal load combinations to which the finite element can be subjected. For a quadrilateral free formulation membrane finite element with 3 degrees of freedom for each node (similar to a bending finite element), 3 load combinations represent the rigid body modes and produce no generalized stresses, so that 9 linearly independent nodal load combinations exist. In the derivation of the yield conditions for a membrane element as described before (and also a bending finite element), 5 generalized stresses were chosen which means that in the 12 dimensional space of nodal forces, or alternatively the 9 dimensional space of generalized stresses, a subspace of dimension 4 remains uncontrolled. The stress space can therefore be split into a primary subspace \( p' \) or \( \sigma' \) and a secondary subspace \( p'' \) or \( \sigma'' \).

\[
\begin{align*}
  p &= p' + p'' \\
  \sigma &= \sigma' + \sigma''
\end{align*}
\]  \hspace{1cm} (4.94)

In the following, the subscripts \( m \) and \( b \) for membrane and bending are left out and the corresponding matrices must be substituted for the respective vectors and matrices. With \( \hat{U} \) being
the adopted displacement modes of the primary subspace as defined before, the element nodal forces or stresses of the primary subspace lead to the generalized stresses $S$

$$S = \hat{U}^T \mathbf{p}' = \int E^T \sigma' \, dV$$

(4.95)

whereas the generalized stresses of the secondary subspace vanish for the displacement modes of the primary subspace $\hat{U}$.

$$0 = \hat{U}^T \mathbf{p}'' = \int E^T \sigma'' \, dV$$

(4.96)

The element stresses of the primary subspace are derived from the selected strain modes of the primary subspace using a $C$ matrix to be determined.

$$\sigma' = E \, C$$

(4.97)

Inserting (4.97) in (4.95) gives

$$S = \left( \int E^T E \, dV \right) C$$

(4.98)

It was shown in (4.23) that the strain modes are orthogonal to each other and the integral in (4.98) gives a diagonal matrix. Inverting this matrix and substituting for $C$ from (4.98) into (4.97) gives

$$\sigma_{m} = \begin{bmatrix} \frac{1}{A} & \cdots & \frac{\eta}{I_\xi} \\ \cdots & \frac{1}{A} & \cdots \\ \cdots & \frac{1}{A} & \cdots \\ \frac{1}{A} & \cdots & \frac{\xi}{I_\eta} \end{bmatrix} \begin{bmatrix} S_{m1} \\ S_{m2} \\ S_{m3} \\ S_{m4} \\ S_{m5} \end{bmatrix}$$

(4.99)

and

$$\sigma_{b} = \begin{bmatrix} \frac{1}{A} & \cdots & \frac{\xi}{I_\eta} \\ \cdots & \frac{1}{A} & \cdots \\ \cdots & \frac{1}{A} & \cdots \\ \frac{1}{A} & \cdots & \frac{\eta}{I_\xi} \end{bmatrix} \begin{bmatrix} S_{b1} \\ S_{b2} \\ S_{b3} \\ S_{b4} \\ S_{b5} \end{bmatrix}$$

(4.100)

for membrane and bending, respectively.
The second moments of inertia $I_{\xi}^2$ and $I_{\eta}^2$ about the two axes as used in (4.99) and (4.100) are defined by

$$I_{\xi}^2 = \int \eta^2 dA \quad I_{\eta}^2 = \int \xi^2 dA \tag{4.101}$$

The strain energy $U'$ of the considered deformation modes in the primary subspace can be found from

$$U' = \frac{1}{2} \int \sigma'^T \epsilon' dV = \frac{1}{2} \int \sigma'^T D^{-1} \sigma' dV \tag{4.102}$$

in which the relationship $\epsilon' = D^{-1} \sigma'$ is used. The total strain energy $U$ from the real stresses $\sigma$ can be directly found from the element nodal forces and displacements

$$U = \frac{1}{2} \int \sigma^T \epsilon = \frac{1}{2} \int \sigma^T D^{-1} \sigma dV = \frac{1}{2} p^T a \tag{4.103}$$

A so-called mesh parameter $\kappa$ is now introduced which compares the strain energy $U'$ of the considered deformation modes in the primary subspace with the total strain energy $U$

$$\kappa = \frac{U'}{U} \tag{4.104}$$

The mesh parameter has a unit value if the element is subjected to nodal forces which can be fully constructed from the modes in the primary subspace. It will be zero only if the secondary modes build the actual nodal force combinations. The mesh parameter, therefore, is an indicator to what extent ignoring the secondary modes is justified.

The value of the mesh parameter is important for those elements which are loaded to their limit, i.e. if the difference between the provided resistance for a particular constraint and the corresponding force resultant is small, then the mesh parameter must be close enough to unity to ensure that no significant contribution of the secondary subspace remains unchecked. The control of the mesh parameter is particularly important during the optimization phase to ensure that the reinforcement minimization process does not go too far by exploiting these unchecked nodal force combinations.

The nodal force combinations of the unchecked subspace $p''$ are shown in Fig. 4.16 for a square membrane finite element, while it can be seen for a square bending finite element in Fig. 4.17.
4. First Concept: Element-based Dimensioning and Yield Conditions for a Finite Element

![Figure 4.16 Membrane nodal force combinations in the unchecked subspace of a finite element](image)

It is interesting to note in both figures 4.16 and 4.17 that, since the chosen element for which the uncontrolled force combinations are shown is a square finite element and hence has a point of symmetry about its center, each of the four load combinations shown in membrane and bending can be found from another load combination set in the group by rotating it 90 degrees about the $\zeta$ axis at the element center. This indeed conforms with the representation theory briefly mentioned in section 4.2.2.

![Figure 4.17 Bending nodal load combinations in the unchecked subspace of a finite element](image)

A qualitative geometrical description of the projection of the yield conditions into a subspace is shown in Fig. 4.18.
A comparison between the isoparametric elements and the adopted free formulation model is given in the table of Fig. 4.19 for triangular and quadrilateral membrane and bending elements.

With regard to Fig. 4.19, a few points should be made here:

- In general, element models with few degrees of freedom are more favourable. On the other hand, we know from the finite element point of view that as the number of degrees of freedom of a finite element model increases, the results of the finite element analysis become more accurate. A compromise is therefore necessary. Since the emphasis here is on dimensioning, it is advisable that in reaching this compromise, the ease of use and consistency of the element model with regard to dimensioning aspects, in particular with respect to the optimized dimensioning approach proposed in the next chapter, is given a higher priority than the accuracy of the finite element analysis model. Consequently, element models with, e.g. side nodes are avoided here despite their higher accuracy in the analysis sense.

- It is more desirable to adopt element models which accommodate element strain modes with as little effort as possible. Simplifications are allowed as long as appropriate tools such as the mesh parameter described here are available which examine the extent of inaccuracies introduced in the system by such simplifying assumptions.
4. First Concept: Element-based Dimensioning and Yield Conditions for a Finite Element

![Table showing comparison between different element models in finite element-based dimensioning](image)

Figure 4.19 Comparison between different element models in finite element-based dimensioning

- In the finite element analysis, every element model must include the rigid body and constant strain modes as explained in section 2.1.4. Higher order modes can be introduced to increase the accuracy of the results. In the derivation of the yield condition constraints throughout this chapter, it was observed that constant strain modes have always been taken into account while some higher (linear) order strain modes were introduced for quadrilateral elements to better accommodate the modes of deformation and failure of the finite element. Rigid body modes were left out since they produce displacements but no strains. It can be seen that this approach for the finite element at the level of dimensioning is very much in line with that for analysis. This clearly demonstrates how the concepts of the finite element method are expanded from analysis to design in a consistent and natural way.
5 Second Concept: Optimization by Superposition of Self-Equilibrating Stress States

In this chapter, the second concept of the new dimensioning method is described. The basic idea is first explained in the context of a simple example of a two bar element system under axial load. The procedures leading to the evaluation of the reinforcement distribution corresponding to the linear elastic solution are then explained. The subject of force redistribution by the introduction of fictitious plastic strains is addressed afterwards. Finally, it is explained how the proposed procedure can also be applied for the ultimate load analysis of a structure with given reinforcement amounts.

5.1 Basic idea in a simple example

Using the simplest conceivable example, this section explains in detail how the concept of superposition of self-equilibrating stress states can be employed in order to optimize the reinforcement distribution within the structure. The simple structure consists of two bar elements of the same length $l$, cross section $A$ and Young’s modulus $E$ which have to carry an external longitudinal load $F$ along their common node (Fig. 5.1). At both boundaries, the two bar structure is fixed. The middle node represents only a translational degree of freedom along the bar axis and the only generalized stress component in each element is the normal force which remains constant along the element length. The generalized strain and stress components are $\varepsilon$ and $p$, respectively. Finally, the concrete compressive strength is $f_c$ and the yield strength of the reinforcement is denoted by $f_y$.

It is assumed that the necessary reinforcement amounts for fulfilling serviceability requirements are already provided in the two members so that the additional reinforcement corresponding to $F$ should be optimized, i.e. minimized for ultimate load considerations only.

According to the lower bound theorem of the plasticity theory, the superposition of any self-equilibrating stress states does not alter the load-carrying capacity of the structure. This means that as long as equilibrium conditions are satisfied, self-equilibrating stress states can be introduced which result in the redistribution of the forces within the structure. As an example for these self-equilibrating stress states, one can think of an increased temperature in a
part of the structure. If the structure is in equilibrium under the external loads, a rising temperature in one region does produce stresses in the system and change the overall stress distribution but it does not change the load-carrying capacity according to plasticity theory.

On the basis of the classical theory of reinforced concrete design, concrete with its brittle material behavior is not supposed to yield in compression before the ductile reinforcement has become plastic. Underreinforced structures where the yielding of reinforcement and the formation of cracks can act as a warning before the structure collapses are therefore preferred. As a result, no plastic distribution stress states are allowed which exploit concrete crushing.

In our simple example, therefore, we can only introduce a longitudinal strain $\varepsilon_1 = \mu \geq 0$ which can be regarded as heating element 1 such that its elongation equals $\mu l$. One should solve the global equation system for the additional right hand side vector again and then superimpose it on the original linear elastic solution. The final distribution of nodal forces in the elements is calculated from

$$p = p_e + p^l$$

(5.1)

in which $p_e$ is the nodal force corresponding to the linear elastic solution and the initial distribution nodal force $p^l$ is determined from

$$p^l = p_{\text{dist}} - p_{e0}$$

(5.2)

Here, $p_{\text{dist}}$ is the force due to the additional redistribution stress state and $p_{e0}$ is the force due to the initial strain $\varepsilon_1 = \mu$ introduced for element 1.

![Figure 5.1 Force redistribution in a two element bar system](image)

The steel contents $A^s_1$ and $A^s_2$ are assumed to be constant along the element length. The optimization problem consists of four yield conditions, namely for steel and concrete in the two
elements, three non-negativity conditions and the objective function which is the minimization of the total steel weight.

Minimize \((A_1^i + A_2^i) l\) with:

\[
\frac{F}{2} \leq A_1^i f_y + \frac{AE}{2} \mu \quad : 1^i
\]

\[-f_c' A - \frac{F}{2} \leq A_3^i f_y + \frac{AE}{2} \mu \quad : 1^c
\]

\[-f_c' A + \frac{F}{2} \leq -\frac{AE}{2} \mu \quad : 2^c
\]

\[A_1^i, A_2^i \geq 0 \quad \mu \geq 0 \quad (5.3)
\]

In Fig. 5.2, the graphical presentation of the optimization problem in the three-dimensional space of the 3 optimization variables is given. The non-negativity conditions allow only a solution in the positive subspace depicted. Yield conditions \(2^i\) and \(1^c\) are not shown since the former is always satisfied and the latter is always less critical than \(2^c\). The other 2 conditions represent planes cutting (or limiting) the feasible solution space. Any point in the three-dimensional space of \((A_1^i, A_2^i, \mu)\) that lies in the feasible region, which is limited by, and on the admissible side of, the planes shown in Fig. 5.2, represents a feasible solution. The vector of the objective function can translate (but not rotate) in space to find the solution of the optimization problem.

Three cases can be distinguished here:

if \(0 \leq F \leq f_c' A\)

optimum: \(AE \mu = F\)

\(A_1 = A_2 = 0\) \quad (5.4)

if \(f_c' A \leq F \leq 2f_c' A\)

optimum: \(AE \mu = 2f_c' A - F\)

\(A_1^i f_y = F - f_c' A\)

\(A_2 = 0\) \quad (5.5)

if \(2f_c' A \leq F\)

not allowed (increase \(A\) or \(f_c'\)) \quad (2^c) \quad (5.6)

The points of the optimum solution for the two cases are marked with a circle in Fig. 5.2. These results should be compared with the dimensioning according to the linear elastic solution (\(\mu = 0\)) which is marked with a square:

\[A_1^i f_y = \frac{F}{2} \quad A_2 = 0\]

and \(F \leq 2f_c' A\) \quad (5.7)

Fig. 5.2 clearly shows the reinforcement reduction by adopting this method.

As shown in Figs. 5.1 and 5.2, as far as the ultimate load design is concerned, the design engineer is able to choose any of the solutions within the feasible region. For various reasons,
most notably fulfilling code requirements for serviceability, he may also decide not to go as low as the minimum solution found and control his design by imposing additional conditions like a minimum value of reinforcement in element 1 for this example. This will then correspond to the solution represented by the dotted line in Fig. 5.1.

![Graphical presentation of the optimization problem for the two element beam](image)

**Figure 5.2** Graphical presentation of the optimization problem for the two element beam

It should be noted that due to the simplicity of this example, the optimization lead to a reduction of the reinforcement weight by just an increased compression in concrete. For normal cases, the other consequence of optimization which is even more common is to relocate some reinforcement from one region to another region where the provided steel with a smaller weight is more effective than where it was put at the first stage.

Moment redistribution in frame structures through the formation of plastic hinges has long been investigated by researchers. [Anderheggen 66] is such a work, in which linear programming techniques were employed in a similar fashion to optimize reinforcement in statically indeterminate structural frames.

The idea explained in the above example will be extended for tackling problems in a continuum. Although the basic principle remains unchanged, the following complexities arise:

- The yield condition in generalized stresses is formulated for the whole finite element as opposed to the conventional infinitesimal volume element. This was addressed in chapter 4.

- The structure consists of a large number of elements. Examining all possible forms of plastic redistribution forces is out of the question. Some heuristic, yet efficient, approach must be implemented for this task.
Continuum structures are highly statically indeterminate. As a result, the capacity for a force redistribution is very high. This enhances opportunities for optimization as well as increases the complexities involved.

The linear program constructed for the optimization formulation cannot include all the element constraints and plastic distributions. This would result in too many constraints, i.e. in an excessively large linear program which is not practical and sometimes impossible to solve.

5.2 Reinforcement evaluation for the linear elastic solution

In this section, the steps towards the evaluation of the reinforcement distribution for the linear elastic solution are discussed. Having explained the basic concepts of linear programming in section 2.3, we now see how this method is applied to the optimum reinforcement dimensioning.

5.2.1 Formulation of the minimum weight objective function

In the practice of reinforced concrete design, the geometry of the structure and dimensions of different parts are usually parameters which the design engineer is less capable of influencing. The amount and the layout of the reinforcement nets and bars, on the other hand, are two parameters which he is indeed responsible to figure out.

Based on experience from everyday designs, we know that the selection of the layout of the reinforcement fields must be solely made by the engineer himself. This choice is a major decision that can be affected by many restrictions or considerations and can be only formulated on a case by case basis. An automation of this task will most probably result in unpractical, inefficient and expensive layouts. For given broad steel configurations, however, it is very appropriate to minimize the cost of the total steel required which, on the one hand, is a reasonable portion of the total construction material cost and, on the other hand, is very suitable for automation within a computer program.

After solving the structure for a set of external load cases, the engineer defines for each panel a series of reinforcement fields. His choice of these fields can be based on experience, engineering judgement, possible restrictions and any other consideration he may find appropriate. In making his decision, the engineer is assisted by the user-friendly program which, through numerical and graphical outputs, furnishes him the necessary data regarding the linear elastic behavior of the structure, in particular the principal stress distributions and structural deformations (chapter 6).

Each reinforcement field, which can belong to only one panel, covers a series of finite elements and each finite element can belong to a number of reinforcement fields. A finite element cannot be only partly covered by any reinforcement field.
Two reinforcement field types can be distinguished here. A *uniaxial field* (bars) is one in which reinforcement bars of the same size are located parallel to each other (Fig. 5.3 a). A *biaxial field* (net) is defined as one in which two uniaxial fields are formed perpendicular to one another. The total area of the reinforcement per unit width in the secondary direction of a biaxial field is $k$ times that of the primary direction (Fig. 5.3 b). Each reinforcement field is characterized by the following parameters:

- *angle* $\alpha$ which is the angle between the (primary) reinforcement bars and corresponding panel’s local $x_p$ axis
- *steel yield strength* $f_y$ of the reinforcement bars to be used
- *ratio factor* $k$ in the case of biaxial (net) reinforcement
- *cost factor* $c$
- *minimum reinforcement area per unit width* $A^{\min}$ which is greater than or equal to zero.

The cost factor $c$ is introduced to enable the engineer to accommodate parameters other than steel weight which influence the total cost of the reinforcement. This can include various cost factors affected by the supply of particular reinforcement fields or their positioning on the site. The default value for $c$ for all reinforcement layers is 1 so that, initially, the total steel weight is to be minimized. In the following representations of the tableau, it is assumed that the $c_j$ factor corresponding to the area of zone $A_{ij}$ is 1. If $c_j$ is chosen different from 1 for any of the reinforcement fields, then $A_{ij}$ must be substituted everywhere by $c_j A_{ij}$.

The steel content of each reinforcement field, uniaxial or biaxial, represents at this stage an optimization variable.

A bending panel has two different *layers* of reinforcement which we refer to as the *top* and *bottom* layers with respect to the panel’s local $t$ axis. The reinforcement layout at the top and bottom layers of a bending panel are independent. Although in the case of a membrane panel of moderate thickness, the usual practice is to put reinforcement of the same size and layout on the two sides of the panel’s thickness to have a better crack distribution, however, for the sake of simplicity, both in this text and within the computer presentation, membrane panels

---

**Figure 5.3** Uniaxial (bar) and biaxial (net) reinforcement fields

---

84
are always regarded as being reinforced by a user-defined layout in a \textit{central} layer (Fig. 5.4) meaning that bending action is ignored. The two layer construction will simply proceed in this case for the same configurations at both sides and with half the calculated steel contents.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{figure5_4}
\caption{Definition of \textit{a) user-defined reinforcement fields b) program-generated reinforcement zones in membrane and bending panels}}
\end{figure}

Having introduced the concept of reinforcement fields, a \textit{reinforcement zone} is now defined as a group of elements which are all covered by the same set of reinforcement fields. In addition, reinforcement zones are chosen in such a way that they cover a closed region without a hole.
Each finite element can belong to several reinforcement fields but only one reinforcement zone. In bending panels, each element belongs to two reinforcement zones covering its top and bottom steel layers. The assignment of each element to its respective reinforcement zone is done by the program on the basis of the user-defined reinforcement fields (Fig. 5.4).

### 5.2.2 Constraint formulation of the linearized yield conditions

As pointed out in section 5.2.1, the reinforcement in the secondary direction is perpendicular to the primary reinforcement and is \( k \) times its area. Since the value for \( k \) is predefined by the user (often \( k = 1 \)), the biaxial reinforcement field represents only one variable. The resistances provided by the steel in the perpendicular direction (subscript \( \perp \)) can be found from

\[
\{ r^s_\perp \} = \left\{ \begin{array}{c} r^s_{\perp -} \\ r^s_{\perp +} \\ r^s_{\perp -} \\ r^s_{\perp +} \end{array} \right\} = \left\{ \begin{array}{c} \sum (\sin^2 \alpha_j - \cos \alpha_j \sin \alpha_j) k_j A^s_{j} f_y \\ \sum (\sin^2 \alpha_j + \cos \alpha_j \sin \alpha_j) k_j A^s_{j} f_y \\ \sum (\cos^2 \alpha_j - \cos \alpha_j \sin \alpha_j) k_j A^s_{j} f_y \\ \sum (\cos^2 \alpha_j + \cos \alpha_j \sin \alpha_j) k_j A^s_{j} f_y \end{array} \right\} = \left\{ \begin{array}{c} a_{i\perp j} A^s_{j} \\ a_{2\perp j} A^s_{j} \\ a_{3\perp j} A^s_{j} \\ a_{4\perp j} A^s_{j} \end{array} \right\} (5.8)
\]

The summation is performed over all steel fields by which the element is covered. For membrane elements, the resistance for the primary reinforcement was given by (4.60) where the components \( r^s_j \) were given by (3.13). The contribution of the secondary reinforcement to the total resistance vector \( \tilde{r}_m \) can still be determined from (4.60) if the components \( r^s_j \) are substituted by \( r^s_{\perp j} \) and the corresponding values are found from (5.8). The angle \( \alpha_j \) to be put in (3.13) and (5.8) equals \( \alpha'_j - \beta \) where \( \alpha'_j \) is the angle between the positive \( x \) axis and the \( j \)-th steel layer (Figs. 4.8 & 4.13) while \( \beta \) represents the element dimensioning direction (Fig. 4.8).

For membrane-bending elements, the vector \( \tilde{r}_m \) is evaluated for both layers using (4.71) and (3.13) which furnishes the contribution of the primary reinforcement. Similar to the case of membrane elements, this is added to the contribution of the secondary reinforcement found by substituting \( r^s_{\perp j} \) into (4.71) and evaluating this vector using (5.8). Obviously, the \( \alpha'_j \) and \( \beta \) values are taken from their respective top and bottom layers as shown in Fig. 4.13.

The resistances are therefore known linear functions of the unknown reinforcement content variables \( A^s_{j} \). Each inequality of the yield condition (4.58) for membrane can be re-formulated in the following form depending on whether it represents a condition for steel or concrete

\[
f_i - A \sum (a_{ij}^s + a_{i\perp j}^s) A^s_{j} \leq 0 \quad (5.9)
\]

\[
f_i - r^c \leq 0 \quad (5.10)
\]
Similarly, the steel conditions for membrane-bending elements formulated in (4.82) and (4.83) can be given by

$$f_i = A \sum \left( a_{i}^{a} + a_{i}^{a+} \right) A_j^a \leq 0$$  \hspace{1cm} (5.11)

$$f_i = A \sum \left( a_{i}^{b} + a_{i}^{b+} \right) A_j^b \leq 0$$  \hspace{1cm} (5.12)

In (5.9) to (5.12), $f_i$ represents the resultant force in the considered direction while the sum is the total resistance as a function of the unknown reinforcement contents $A_j^x$. The element area is denoted by $A$.

Another constraint which the user can impose is an explicit input of a minimum reinforcement content for a particular field. This option was implemented primarily to take into account the specifications by design codes which aim to ensure a uniform crack distribution. The constraint is formulated by

$$A_j^x \geq A_j^{x \text{min}}$$  \hspace{1cm} (5.13)

For steel constraints (5.9), (5.11) and (5.12), the resistance provided by the minimum reinforcement $r_i^{x \text{min}}$ is subtracted from the applied force resultant $f_i$. This resistance is given by

$$r_i^{x \text{min}} = A \sum \left( a_{i}^{x} + a_{i}^{x+} \right) A_j^{x \text{min}}$$  \hspace{1cm} (5.14)

with the minimum value of reinforcement area for the $j$-th layer given by $A_j^{x \text{min}}$. As a result, the new definition of the unknown variable $A_j^x$ would be the amount of reinforcement necessary in addition to the already provided minimum reinforcement.

### 5.2.3 Building the linear program for the linear elastic solution

So far, the user has generated the finite element model of the structure, specified a set of external load cases $l_1...l_{NL}$ where $NL$ is the number of load cases and solved the structure for all these load cases. The generalized stresses are subsequently calculated. At this stage, the user specifies a series of load combinations $L_1...L_{NLC}$ where $NLC$ is the number of load combinations and $L_j$ is defined as

$$L_j = \sum_{i=1}^{NL} c_{ij} l_i$$  \hspace{1cm} (5.15)

$c_{ij}$ is the factor for load case $l_i$ in load combination $L_j$. He then defines interactively the reinforcement fields for each panel as described in section 5.2.1.
The flow diagram for evaluating the reinforcement contents corresponding to the linear elastic solution is shown in Fig. 5.5.

Figure 5.5 Flow diagram of reinforcement design for the linear elastic stress distribution

Among all $L_j$ load combinations, the program then finds the extreme state generalized stresses for each constraint of each element. The corresponding algorithm is shown in Fig. 5.6.

Figure 5.6 Evaluation of the extreme state forces for the linear elastic solution
The tableau for the linear program will now be established as formulated in Fig. 5.7. Concrete constraints corresponding to compressive forces do not enter the tableau at this stage since their fulfilment is assumed to be independent of the amount of steel provided.

![Diagram](image)

**Figure 5.7 Building the linear program for the linear elastic stress distribution**

It can be seen that for each reinforcement zone, only the two worst cases in the two reinforcement dimensioning directions are introduced in the tableau. For membrane-bending panels, this is done for each of the two layers. The tableau of the linear program will then have the setup shown in Fig. 5.8.

![Setup of the tableau for reinforcement of the linear elastic solution](image)

**Figure 5.8 Setup of the tableau for reinforcement of the linear elastic solution**

The $A^j_i$ variables are shown as $X^j_i$ to signify that they are the unknowns of the linear program. $NF$ is the number of reinforcement fields. $A_{ij}$ is the area of the zone which is equal to the total area of all elements that include reinforcement field $j$, i.e. $X^j_i$ and are all covered by the same reinforcement fields. If a minimum amount of reinforcement area is prescribed for a specific field, then this condition can be imposed on the tableau by introducing an additional constraint row as shown in Fig. 5.8.
In our application of a reinforcement cost minimization problem, the starting basic solution \( x = 0 \), namely zero reinforcement or zero reinforcement in addition to the predefined minimum values for all reinforcement fields, is indeed a minimum but not a feasible solution since it violates those constraint rows in the tableau where \( s_i = -b_i < 0 \). This is not the case only when the provided reinforcement at all elements can resist the applied loads. The solution of the linear program now proceeds based on the principles elaborated in section 2.3. The no feasible solution case shown in Fig. 2.13 indicates that one or more constraints cannot be satisfied for this status of the tableau. It means that the provided steel configuration for that element is not capable of carrying the tensile force resultants in that particular direction and therefore a new reinforcement field in the appropriate direction must be introduced.

After solving the linear program, all constraints must be checked. The procedure for this task can in general be formulated as in Fig. 5.9 although for the tableau at this stage, only a check on all concrete constraints is required. If any of the concrete yield conditions are violated, it implies that the concrete thickness is too small to resist the applied compressive forces. Increasing panel thickness or using concrete with a higher strength will then be the remedy.

**Figure 5.9 Checking all constraints**

### 5.3 Force redistribution by introducing fictitious plastic strains

The reinforcement distribution corresponding to the linear elastic solution found at this stage is normally uneconomical, particularly because of the stress peaks at points of singularity, which is a characteristic of the elasticity theory in general. In this section, the progressive optimization approach towards a rational reinforcement distribution is first outlined. The fictitious initial strains for membrane action and curvatures for bending are then presented and the subsequent modifications of the optimization tableau are explained. It should be emphasized that the introduced plastic strain distributions are fictitious, meaning that they do not necessarily represent the actual stress redistribution under the applied loads.
5.3.1 Progressive optimization approach

The basic concept of the optimization approach was outlined in section 5.1 for a bar element subjected to axial forces. In a series of steps, homogeneous self-equilibrating states of stress are superimposed on the existing equilibrium system with the aim of redistributing the forces in such a way that the total steel weight diminishes. A few points must be elaborated at this stage about the optimization process: As pointed out before, based on the lower bound theorem as long as equilibrium is satisfied, the choice of the initial strains is free. This means that the criteria which determine the distribution and intensity of the initial strain cases are primarily determined with regard to efficiency of the linear optimization program.

It is also important to note, that in general the time spent by the linear program very rapidly grows as the size of the tableau increases. In this regard, it is crucial to perform the relevant tasks of the optimization process as much as possible outside of the tableau, and only to introduce those parameters and conditions into the optimization tableau which are significant. It is in this context that when establishing the tableau for the first linear elastic solution, only the most unfavorable constraints were introduced. These considerations also explain why the extreme state values among all load combinations were chosen instead of separately introducing each load combination in the tableau.

A numerical investigation of the size of the tableau without any attempt to reduce it may prove interesting. Using the table of Fig. 5.10, one can see the importance of a specially-tailored approach for handling the tableau. The 16/32 constraints per bending element depend whether membrane effects are included or not. Each initial strain redistribution produces 4 new columns in the tableau and the full size is when redistribution stress states are generated for each one of the elements. It can be seen how different parameters affect the overall size.

- If each external load case were to be included in the tableau individually, the size of the tableau would have grown proportionally to the number of these load cases. This was avoided by resorting to the shakedown theorem and taking the extreme state values of all load cases.

- For a full size tableau for optimization, a mesh with twice as many elements results in four times as big a tableau, i.e. the tableau size varies as the square of the number of elements.

Considering that typical PCs and workstations today have 16 / 32 MB RAM respectively, one can see that for a real problem, working with the full tableau is out of the question. In addition to this major restriction, the fact is that on the one hand, a great majority of the constraints do not control the design and also initial strain distributions in many of the elements cannot reduce the total steel weight at all. Furthermore, the bigger the size of the tableau, the more the linear program is prone to numerical instabilities, and the time spent also grows very rapidly with the increased tableau size.
5. Second Concept: Optimization by Superposition of Self-Equilibrating Stress States

<table>
<thead>
<tr>
<th></th>
<th>fields / panel e.g.</th>
<th>elements e.g.</th>
<th>constraints / element</th>
<th>full tableau linear elastic</th>
<th>full tableau elasto-plastic</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 membrane panel</td>
<td>7</td>
<td>500</td>
<td>24</td>
<td>0.77 MB</td>
<td>192 MB</td>
</tr>
<tr>
<td>1 bending panel</td>
<td>2x7</td>
<td>500</td>
<td>16/32</td>
<td>1.192 MB</td>
<td>128/256 MB</td>
</tr>
</tbody>
</table>

**Figure 5.10** Example of a full tableau size without progressive optimization

Having in mind these points, a procedure was developed which greatly enhances the numerical efficiency of the optimization process. In the *progressive optimization* method (the word “incremental”, which is very much associated with nonlinear analysis is deliberately avoided), the optimization tableau is progressively expanded in the two directions and updated at each stage. This is depicted in Fig. 5.11 and can be summarized in the following:

Starting from the tableau of the linear elastic solution, a few redistribution stress states are added and the linear program is solved for a new minimum. All constraints are then checked outside of the tableau, the violated constraints are introduced in the tableau and a new optimum is found. This procedure is repeated until the appropriate solution is obtained.

**Figure 5.11** Progressive optimization and expansion of the tableau

Furthermore, generating redistribution stress states for each finite element and introducing all these in the tableau in one run is computationally impossible. As a result, either some elements have to be grouped together or the redistribution should be performed on some selected finite elements. In general, since the choice of the redistribution stress states is free, any heuristic model can be adopted which is computationally efficient. One suitable approach
is to group several neighboring finite elements together for which the same initial strains are introduced for all the elements within the selected group. At the first stage, an appropriate choice for these groups of elements can be the reinforcement zones as defined in section 5.2.1 and shown in Fig. 5.4. As the optimization proceeds, these zones can be subdivided continuously into smaller zones and the same procedure repeated. Ultimately, each sub-zone will correspond to one single finite element in the model. For normal applications, the optimization process is likely to be interrupted either manually by the design engineer or automatically by the program due to various reasons, in particular the instability of the tableau, before the optimization goes that far. The flow diagram of the optimization process is given in Fig. 5.12.

**Figure 5.12** Flow diagram of the optimization stage
The task of optimization at each stage is performed on a panel selected by the user. However, the whole structure is included in the tableau to take into account the interaction among different panels. The user can switch between panels at different stages of the optimization process. For a chosen panel, initial strains or curvatures are introduced as described in sections 5.3.2 and 5.3.3. Having evaluated the equivalent nodal forces producing the initial strains or curvatures in all the selected elements, the program then assembles these nodal forces into load vectors which we refer to as redistribution load cases. Since the global stiffness matrix has already been triangularized for the solution of the linear elastic load cases, the solution of the equation system for these additional redistribution load cases consists only of a forward elimination and backward substitution which can be accomplished very quickly. The generalized stresses in each element corresponding to these new load cases are then determined in all the elements throughout the structure.

The next step as shown in the flow diagram of Fig. 5.12 is to include the redistribution load cases in the existing tableau which is shown in Fig. 5.13, where \( m \) represents the number of redistribution load cases introduced in one stage. The new columns must first be modified according to the rules outlined in section 5.3.4 and shown in Fig. 5.19. The form shown in Fig. 5.13 is the one before the modification is made. Among all redistribution load cases, only those columns of the new redistribution matrix will be introduced in the tableau whose objective function coefficient is negative, i.e. which may be able to reduce the total steel weight. This means that the introduction of these self-equilibrating states of stress can result in a reduction of steel weight. The existing tableau, whose current solution is feasible but not optimum, is now solved using the algorithm previously described in Fig. 2.14.

\[
Z = \begin{bmatrix}
X'_1 & \ldots & X'_j & \ldots & X'_{NF} & \mu_1 & \ldots & \mu_m \\
\ast & A_{i1} & \ldots & A_{ij} & \ldots & A_{im} & 0. & \ldots & 0. \\
\ast & \ast & \ldots & \ast & \ldots & \ast & \ast & \ldots & \ast \\
\vdots & \vdots & \ldots & \vdots & \ldots & \vdots & \vdots & \ldots & \vdots \\
-s_i & -f_i^{\text{rmin}} & 0. & \text{or} & (a_{ij}^{\nu} + a_{ij}^{\mu}) & -f_{ij} \\
-s_i & -\mu_i^{\text{min}} & 0. & \ldots & 0. & 1. & 0. & \ldots & 0. \\
-s_j & \mu_i^{\text{max}} & 0. & \ldots & 0. & -1. & 0. & \ldots & 0. \\
\end{bmatrix}
\]

**Figure 5.13** Expansion of the tableau for new initial strains
In Fig. 5.13, $f_{ij}^I$ is the resultant force $f_i$ corresponding to the $i$-th constraint for the redistribution load case $j$. The vector $f_{ij}^I$ is found for membrane elements from

$$f_{ij}^I = \tilde{N}_{mc} S_{mcj}^I$$  \hspace{1cm}  \text{(5.16)}$$

in which $\tilde{N}_{mc}$ is a 24x3 matrix including the first 3 columns of the 24x5 matrix $\tilde{N}_m$ introduced in (4.58). Similarly, $f_{ij}^I$ is found for membrane-bending elements from

$$f_{ij}^{Ih} = \tilde{N}_{mbc} S_{bcj}^I \hspace{1cm} f_{ij}^{Ih} = \tilde{N}_{mbc} S_{bcj}^I$$  \hspace{1cm}  \text{(5.17)}$$

The 16x3 matrices $\tilde{N}_{mbc}$ and $\tilde{N}_{mbc}$ for the two layers contain the first 3 columns, i.e. the constant modes of the 16x5 matrices $\tilde{N}_{mb}$ and $\tilde{N}_{mb}$ previously defined in (4.70).

In sections 5.3.2 and 5.3.3, it will be shown how strain modes are introduced for an element in membrane or bending and the corresponding element nodal displacements $a_{ij}^e$ and the equivalent nodal forces $p_{ij}^e$ are calculated. Each of the elements subjected to these initial strains has a contribution to the redistribution nodal force vector $F_{ij}^I$. This is found by summing up the contribution of the nodal force vector $p_{ij}^e$ for each redistributed element as

$$F_{ij}^I = \sum F_{ij}^e p_{ij}^e$$  \hspace{1cm}  \text{(5.18)}$$

Having solved the equation system and thus obtained the nodal displacements of all nodes for the redistribution load case $F_{ij}^I$, the nodal forces are found by

$$p_{ij}^e = K^e (J^e A_{ij}^e - a_{ij}^e)$$  \hspace{1cm}  \text{(5.19)}$$

In (5.19), $J^e A_{ij}^e$ is the extracted element nodal displacement after solving the global system of equations. These displacements resulted from loading the structure with the chosen self-equilibrating states of stress. The element nodal displacement vector $a_{ij}^e$ due to the introduced plastic strains should only be included in (5.19) for those elements which are directly subjected to the plastic strains. For non-distributed elements, $a_{ij}^e$ represents a null vector. Having obtained the element nodal forces $p_{ij}^e$, the corresponding element generalized stresses $S_{ij}^e$ for the constant modes can be found by

$$S_{ij}^e = U_e p_{ij}^e$$  \hspace{1cm}  \text{(5.20)}$$
The corresponding \( \tilde{U}_c \) must be used for membrane and bending using (4.32) and (4.41) while only the components corresponding to the constant modes are taken.

In Fig. 5.13, the factors for the redistribution load vectors in the row of the objective function are initially zero since the redistribution load cases represent no cost. Numerically, it is possible to impose lower and upper limits on the size of the redistribution coefficients. This is basically similar to imposing a prescribed minimum value for a reinforcement field which was explained in section 5.2.3.

Having solved the linear program, all constraints will be checked for all elements outside of the tableau to ensure that the introduction of the new redistribution load cases has not resulted in the violation of constraints not included in the tableau. If this is the case, these violated constraints will be introduced as new rows at the bottom of the tableau which is shown in Fig. 5.14. The new rows are first modified according to the rules outlined in section 5.3.4 and shown in Fig. 5.20. For membrane panels, some of these violated constraints possibly correspond to concrete under compression. The coefficients for the reinforcement in this row will then be zero and the concrete resistance \( r_i^c \) will be subtracted from the resultant force in the first column as shown in Fig. 5.14.

\[
Z = \begin{bmatrix}
X_i' & \ldots & X_j' & \ldots & X_{NF}' & \mu_1 & \ldots & \mu_m \\
0 & A_{z1} & \ldots & A_{zj} & \ldots & A_{zm} & * & \ldots & * \\
* & * & \ldots & * & \ldots & * & * & \ldots & * \\
\vdots & \vdots & \ldots & \vdots & \ldots & \vdots & \vdots & \ldots & \vdots \\
-s_i - (f_i - r_i^{c \text{min}}) & 0 & \text{or} & (a_{iy}^r + a_{iy}^{r \perp}) & -f_i^r & 0 & \ldots & 0 & 0 \\
\end{bmatrix}
\]

\[
s_i = \begin{bmatrix}
-s_i - (f_i - r_i^{c \text{min}}) & 0 & \text{or} & (a_{iy}^r + a_{iy}^{r \perp}) & -f_i^r & 0 & \ldots & 0 & 0 \\
-s_i - (f_i - r_i') & 0 & 0 & 0 & -f_i^r \\
\end{bmatrix}
\]

**Figure 5.14** Expansion of the tableau for new violated constraints

The new tableau is solved again to find a feasible optimum solution using the algorithm given in Fig. 2.13.
5.3.2 Initial strains for membrane

For membrane finite elements, four possible initial strain modes $\epsilon_{1m}'$ to $\epsilon_{4m}'$ are chosen as a combination of the constant strain modes. These correspond to the membrane normal matrix $N_{nm}$ as specified in (3.9). The shape of these four modes are shown in Fig. 5.15 for a square element. It can be clearly seen that no concrete crushing takes place.

\[
\epsilon_{1m}' = \begin{pmatrix}
1 \\
0 \\
1
\end{pmatrix} \quad \epsilon_{2m}' = \begin{pmatrix}
1 \\
0 \\
-1
\end{pmatrix} \quad \epsilon_{3m}' = \begin{pmatrix}
0 \\
1 \\
1
\end{pmatrix} \quad \epsilon_{4m}' = \begin{pmatrix}
0 \\
1 \\
-1
\end{pmatrix}
\]

**Figure 5.15** Initial strain combinations shown on a square membrane finite element

The membrane nodal forces producing $\epsilon_{1m}'$ to $\epsilon_{4m}'$ can be found from (6.4) and (6.9) using the lumping matrix (6.7) for the free formulation membrane element model used in the program. The corresponding element nodal displacements causing these unit strains are found from (4.32), evaluated only for the constant strain modes.

5.3.3 Initial curvatures for bending

A total of four curvature modes are adopted which resemble the first four column vectors building the normal matrix $N_b$ for bending finite elements as specified in (3.20). They represent combinations of the constant curvature modes. The shape of these modes are shown in Fig. 5.16 for a rectangular bending finite element.

\[
\epsilon_{1b}' = \begin{pmatrix}
1 \\
0 \\
1
\end{pmatrix} \quad \epsilon_{2b}' = \begin{pmatrix}
1 \\
0 \\
-1
\end{pmatrix} \quad \epsilon_{3b}' = \begin{pmatrix}
0 \\
1 \\
1
\end{pmatrix} \quad \epsilon_{4b}' = \begin{pmatrix}
0 \\
1 \\
-1
\end{pmatrix}
\]

**Figure 5.16** Initial curvature combinations shown on a square bending finite element
It can be seen in (3.20) that the second set of four column vectors is identical to the first four vectors with the opposite sign. This means that if the coefficient of the curvature modes adopted here is allowed to be positive as well as negative, then only half of the eight curvature combinations should be taken into account, which is why the first four modes are taken here. This will reduce the size of the linear program for optimization. Only these first four modes are shown here in Fig. 5.16.

The bending nodal forces producing \( \epsilon_{ib}' \) to \( \epsilon_{4b}' \) can be found from (6.5) and (6.9) using the lumping matrix (6.8) for the free formulation bending element model adopted here. The corresponding element nodal displacements causing these unit curvatures are found from (4.41) evaluated only for the constant curvature modes.

It can be seen that, unlike the membrane case where plastic strain redistributions which nowhere produce concrete crushing are introduced, the plastic curvatures proposed here do assume some concrete crushing to take place in one of the concrete layers (Fig. 5.17 a).

![Figure 5.17 Plastic curvature modes (a) with and (b) without concrete crushing](image)

In order to reduce the degree of concrete crushing or fully eliminate it, it is possible to combine plastic strain redistribution modes with the imposed curvature modes (Fig. 5.17 b). Consequently, four new plastic strain columns and two new constraint rows must then be introduced in the tableau of the linear program. These two new constraints control the relationship between the coefficients of the bending curvatures and membrane strain modes such that

\[
\begin{align*}
\epsilon_{ipm}' + c \epsilon_{ipb}' & \geq 0 \\
0 & \leq c \leq \frac{d_s}{2} \\
\epsilon_{ipm}' - c \epsilon_{ipb}' & \geq 0 \\
i & = 1..4
\end{align*}
\]

(5.21)

It should be noted that this will greatly increase the size of the linear program and also makes the tableau more susceptible to numerical problems and inaccuracies. The current version of program ORCHID does not include the additional constraints of (5.21).
5.3.4 Modifications of the tableau

As explained in the optimization procedure, the tableau of the linear program is constantly updated and expanded by introducing new columns for plastic redistribution load cases and new rows for including violated constraints as shown in Fig. 5.11. Each time the tableau is to be updated, the new vectors must be modified to consider the changes which have taken place before in the tableau. The idea is therefore to find out at stage \( k \) before introducing a new vector into the tableau what changes would have been performed on this vector if it had been included in the tableau from the beginning. These changes have to be performed on the new vector outside the tableau after which the modified vector can be added to the existing tableau.

The exchange step was described in section 2.3.1. After performing \( k \) such exchange steps, the components of the vectors of basic variables \( s \) and non-basic variables \( x \) can each be regrouped and split into two subvectors

\[
s \rightarrow (s_{m-k}, s_k) \quad x \rightarrow (x_{n-k}, x_k)
\]

in which vectors with the subscript \( m - k \) refer to the non-exchanged variables while those with the subscript \( k \) correspond to the exchanged ones. The tableau at stage \( k \) and the corresponding regrouped tableau in the beginning before starting to perform any exchange operation are shown in Fig. 5.18.

![Regrouped tableau](image)

**Figure 5.18** Regrouped tableau in the beginning \( O_0 \) and after \( k \) exchange steps \( O_k \)

If new columns as a set of redistribution load cases are to be added to the tableau, the modified matrix is shown in Fig. 5.19.

The modified tableau with the newly introduced redistribution load cases represents a feasible solution which can further be optimized. The solution algorithm was given in the flow diagram of Fig. 2.14.
5. Second Concept: Optimization by Superposition of Self-Equilibrating Stress States

Figure 5.19 Introduction of new columns (redistribution load cases) in the tableau at step \( k \) \((O_k)\)

Fig. 5.20 shows the necessary modifications on the violated constraints to be added as additional rows to the existing tableau.

Figure 5.20 Introduction of new rows (constraints) in the tableau at step \( k \) \((O_k)\)

The modified tableau with the newly introduced constraints represents an unfeasible minimum solution in which some constraints are violated. The corresponding simplex algorithm, which will result in an increase of the reinforcement weight in exchange for the fulfilment of the violated constraints, is the one previously discussed and demonstrated in Fig. 2.13.
5.4 Ultimate load analysis

The reinforcement dimensioning optimization problem described so far was a minimization. The constraints of the linear program are element yield conditions. The unknowns are the amount of reinforcement in each field and the magnitude of the redistribution load case factors. The objective function consisted of the total steel weight to be minimized.

Alternatively, an ultimate load analysis problem can be formulated. The distribution and amount of reinforcement is prescribed. The constraints are still the element yield conditions. The objective function, however, is the maximization of the load factor for a given load configuration which the provided (known) reinforcement is capable of carrying. Similar to the approach presented before, the idea here is also to introduce appropriate self-equilibrating states of stress which result in a stress redistribution within the structure so as to maximize the load factor. Therefore, the unknowns are the load factor and the magnitude of the redistribution load case factors.

It should be emphasized that similar to the optimization approach outlined before, where the fictitious redistribution stress states did not represent the real stress redistributions within the structure, in the ultimate load analysis approach suggested here, the introduced plastic strains are again fictitious. As a result, structural deformation after the ultimate load analysis does not necessarily correspond to the actual collapse mechanism of the structure at the limit load.

Fig. 5.21 shows a concise form of the tableaus for a better comparison between the reinforcement minimization and the ultimate load factor maximization problems. In the minimization problem (a), the unknowns are $X^e$ and $\mu$ while in the maximization problem (b), these are $\lambda$ and $\mu$. Note that $X^e$ is a known vector in the ultimate load analysis of case (b).

\[
\begin{array}{ccc}
1 & X^e & \mu^T \\
\hline
& A_e & 0 \\
& - f_e & r & - f^i \\
\end{array}
\]

(a) reinforcement minimization

\[
MIN \quad A_e X^e \\
with \quad - f_e + r X^e - f^i \mu \geq 0
\]

\[
\begin{array}{ccc}
1 & \lambda & \mu^T \\
\hline
& A_e X^e & 0 \\
& r X^e & - f_e & - f^i \\
\end{array}
\]

(b) ultimate load maximization

\[
MAX \quad \lambda(A_e X^e) \\
with \quad r X^e - \lambda f_e - f^i \mu \geq 0
\]

Figure 5.21 Comparison between reinforcement minimization and ultimate load maximization
The ultimate load maximization as formulated in Fig. 5.21 (b) can be changed to a minimization problem by one of the two approaches shown in Fig. 5.22.

With the data structure and algorithms for the reinforcement minimization problem available, the implementation of the form suggested in Fig. 5.22 (a) is easier and involves less modifications than the one given by Fig. 5.22 (b).

\[
\begin{array}{ccc}
1 & \frac{1}{\lambda} & \mu^T \\
A_iX^i & 0 & \\
-f_c & rX^i & -f^i \\
\end{array}
\]

\[
\begin{array}{ccc}
1 & \lambda & \mu^T \\
-A_iX^i & 0 & \\
rX^i & -f_c & -f^i \\
\end{array}
\]

(a) \( \min \frac{1}{\lambda} (A_iX^i) \) with \(-f_c + \frac{1}{\lambda} rX^i - f^i \mu \geq 0\)

(b) \( \min -\lambda (A_iX^i) \) with \(rX^i - \lambda f_c - f^i \mu \geq 0\)

**Figure 5.22** Alternatives for ultimate load maximization transformed into a minimization problem

The tableau corresponding to the linear program for the ultimate load analysis is shown in Fig. 5.23. At first, it consists of two columns of the resultant forces and resultant resistances. The first column is basically the same as that of Fig. 5.8 for reinforcement minimization. The terms of the second column correspond to the total reinforcement resistance provided at each field.

\[
Z = 0. \sum_{j} A_{ij} X_j^i \\
\]

\[
s_i = \frac{1}{\lambda} (a_{ij} + a_{ij}^*) X_j^i \\
s_i = \frac{1}{\lambda} (a_{ij} + a_{ij}^*) X_j^i \\
\]

**Figure 5.23** Adopted tableau for the ultimate load analysis
5.4 Ultimate load analysis

If any of the constraints are violated at this stage, i.e. if the provided reinforcement does not sustain the resulting stresses of the linear elastic solution, then one exchange step is necessary to bring the tableau into a feasible solution state.

The redistribution load cases can now be introduced in a similar way to the reinforcement optimization case.

Two remarks should be made at the end of this chapter:

- The optimization procedure described here did not take any detailing aspects into consideration. Despite the great importance of appropriate detailing, in the context of this work, the contribution of this reinforcement to the total steel amount is marginal. On the other hand, it was not the goal of this research project to program any specific design code or constructional recommendations or practices in a computer software. Therefore, it is expected that the design engineer pays enough attention to the detailing aspects of his specific problem. However, it should be mentioned that depending on the reinforcement layout and other structural specifications, anchorage length for reinforcement in different fields can, in some cases, affect the total steel weight quite substantially. This is normally more important if a large number of reinforcement fields are defined. In any case, the engineer can partially take care of the question of anchorage length while defining the reinforcement fields, i.e. he can specify the reinforcement fields in such a way that some room is provided at the borders of these fields for the reinforcement to be anchored.

- In this chapter, instead of using one single term for the self-equilibrating stress states introduced for the optimization, different terminologies were adopted in order to emphasize on specific aspects of these stress states. The reference as plastic was used to make the distinction from a design based on the linear elastic analysis results. It was also explained that the stress states introduced are fictitious and do not necessarily represent the real plastic stress redistributions taking place in the structure. Therefore, the term initial strain was also used instead of plastic strain in different occasions. On the other hand, the term redistribution load case was introduced in section 5.3.1 and used frequently afterwards. It is clear that these self-equilibrating stress states are very much different in their nature from the external loads. Still, they have been referred to as load cases, because from the finite element point of view, they represent a right hand side vector in the global equation system very much similar to the external load vectors. Although they are self-equilibrating, they do result in stress distributions within the structure which modify the linear elastic stress distribution and lead to a stress redistribution, and this is why the term redistribution was also used.

The approach and algorithms explained in this chapter for the optimum dimensioning of three-dimensional concrete panel structures are implemented in the interactive computer program ORCHID (Optimum Reinforced Concrete Highly Interactive Dimensioning). In the
next chapter, the general aspects of the graphical user interface of ORCHID together with some particular features of the finite element model are explained.
6 ORCHID: A Program for “Optimum Reinforced Concrete Highly Interactive Dimensioning”

Program ORCHID (Optimum Reinforced Concrete Highly Interactive Dimensioning) has been developed for the optimum reinforcement dimensioning of three-dimensional concrete structures consisting of flat panels. The issues concerning the reinforcement calculation and the elasto-plastic optimization have already been discussed. In this chapter, two objectives are pursued. The first is to give a very brief description of the design and structure of ORCHID. In the second part, some of the numerical algorithms concerned with the finite element calculations, which can be helpful for other programmers of such a system, are explained. The emphasis here is on the computer implementation of the numerical procedures. More on the theoretical background can be found in the literature which is referenced wherever possible.

6.1 The structure of ORCHID

The major steps towards finding the desired optimum reinforcement layout are depicted in the flow diagram of Fig. 6.1. The abbreviations FE, LP and PT stand for Finite Element, Linear Programming and Plasticity Theory, respectively.

![Diagram](image-url)
The steps leading to the creation and solution of the linear elastic finite element model are shown by the flow diagram of Fig. 6.2. The first stage in the program is to read an ASCII text file in which the general input data of the specific structure are specified. This input mainly consists of the geometry defining the boundaries of each panel, the type of the panel (membrane only or membrane-bending), the panels’ thicknesses and the concrete material properties. Based on the geometrical data description, a finite element mesh is automatically generated with quadrilateral and triangular elements. The user now interactively imposes the boundary conditions on the finite element model and defines the external load cases. The types of boundary conditions and the loading types implemented in ORCHID are given in section 6.5. At any time, the user can modify the input of boundary conditions, load case specifications, material properties and panel thicknesses interactively. Any changes in the overall geometry of the structure, however, must be made in the original ASCII input file after which the process as shown in Fig. 6.2 must be repeated.

At this stage, the program is able to solve the structure for all specified load cases, i.e. to determine all nodal displacements and generalized stresses within each finite element. With the aid of the user-friendly interface, the engineer can then examine structural deformations under combinations of the applied load cases with different load factors for the whole structure or for specific panels. Also, the distribution of different generalized stresses can be seen for each panel in various graphical representation forms.

![Figure 6.2 Flow diagram for “Creating and solving the linear elastic finite element model”](image)

The steps leading to the evaluation of the reinforcement content for the linear elastic stress distribution were shown in the flow diagram of Fig. 5.5. The effective support provided by the program in visualizing the stress distribution within the structure is of great value for the design engineer in choosing the reinforcement fields. In this task he can, of course, effectively
use his engineering judgement and include any additional constraints or requirements his design is subjected to.

Having the reinforcement values for all the fields throughout the structure for the linear elastic solution, the third step of Fig. 6.1 can follow, which is shown in more detail in the flow chart of Fig. 5.12.

The overall design of ORCHID is shown in Fig. 6.3. The program is built up of three separate parts which are connected to one another and exchange data through the data management system.

A: linear elastic FE
- FE meshing
- calculation of element stiffness types
- assemblage of stiffness matrix and load vectors
- solution of equation system
- calculation of generalized stresses

B: linear optimization program
- construction and expansion of tableau
- simplex algorithm and optimization
- ultimate load analysis

The function of the Input / Output (IO) routines is to perform the loading of the first input file into memory and to save the reinforcement results before exiting the program. It is possible to store the information regarding the finite element model, load cases and reinforcement field

\[ \text{Data management system} \]

\[ \text{Input – Output (IO) routines} \]
- reading geometrical input
- reading / writing FE model
- reading / writing load cases
- reading / writing reinforcement field input
- writing reinforcement output
  - eventual data transfer to
    - CAD programs for drafting

\[ \text{Graphical User Interface (GUI) routines} \]
- drawing of whole structure and single panels
- drawing of displacements and stresses
- drawing of reinforcement and redistribution fields
- user-machine interaction
  - interactive inputting, modifying and checking,
    - zooming, scaling, rotating, animating,
    - picking, selecting

**Figure 6.3 Modularity of ORCHID**
specifications in separate files. In this case, if the engineer desires to work on the same structure in a new session, the tasks of mesh generation and renewed specification of the above data can be dispensed with. All files are written in an ASCII format so that the user is able to look at or possibly modify them.

The numerical calculation routines comprise two groups: all relevant calculations for the linear elastic finite element solution and the linear program for the reinforcement optimization. The interaction of the two groups during the progressive optimization process occurs when the newly introduced fictitious plastic strain load cases are solved by the linear equation solver and the generalized stresses are calculated at each stage.

6.2 The graphical user interface of ORCHID

The Graphical User Interface (GUI) routines include all the procedures which enable an effective interaction between the user and the program in real time. This includes the presentation of a variety of input and output data like geometry and dimensions, structural deformations, generalized stresses, principal stresses, reinforcement fields, etc. in both numerical and graphical form. A library of graphical objects was used to develop the user interface. The Simple User Interface Toolkit (SUIT) is a collection of basic graphical objects which served as a library on the basis of which the interface was built up [SUIT 92]. The current version of the program ORCHID was developed in C (ANSI–C) programming language on a SUN–Sparc workstation running under the UNIX operating system. A view of the graphical user interface is shown in Fig. 6.4.

The main parts of the graphical user interface are explained at this stage:

The pull-down control menu controls the general user interaction with the program. Each menu entry consists of a set of sub-menus each of which either performs a specific task or activates another set of dialog boxes which in turn interact with the user. The basic menu palette consists of the following four menus:

File: includes the commands for loading new projects, saving, printing, exporting and providing general information on the active project and finally for exiting the program.

2D View: controls the presentation of an active panel in the 2D view of active panel window. It activates a series of dialogue boxes through which the user can define what parameters must be plotted and in which form they should be presented. These include nodes and elements and their numberings, various stress results, reinforcement, redistribution fields and a wide range of other parameters which are too lengthy to be mentioned here.
6.2 The graphical user interface of ORCHID

Figure 6.4 Graphical User Interface (GUI) of ORCHID

Window: defines in which form the 2D view of active panel window should be presented. This includes automatic or manual switching between one or two sub-windows so that e.g. for the reinforcement presentation of a membrane panel, only one window is active while for bending panels, two windows are activated to examine and compare the reinforcement of the top and bottom layers simultaneously.

Mode: activates the mode in which the structure is active. These include:

Mesh: to generate a finite element mesh for a structure whose geometrical description is stored in a data file.
6. ORCHID: A Program for "Optimum Reinforced Concrete Highly Interactive Dimensioning"

**3D View:** to transform the 2D view of active panel window to a 3D View window so that the spatial structure can be examined more easily and effectively.

**Geometry:** to interactively define, check and modify material properties and thicknesses, boundary conditions, panel types, etc.

**Loading:** to interactively define, check and modify external load cases.

**Solve:** to define, check and modify load combinations, limit load states, solve the finite element system for these load cases and show the displacement and stress values in graphical and numerical form.

**Design:** to define, check and modify the reinforcement fields, calculate the reinforcement evaluation for the linear elastic stress distribution and perform the elasto-plastic distribution as well as the ultimate load analysis for a prescribed reinforcement distribution.

As the Design mode is changed, the corresponding commands of the active mode will be activated. These commands enable the user to interact with the program in the selected mode.

The window for optimization chart is only active in the Design mode which, in a normal design process, is the option mostly active. In the phase of reinforcement dimensioning for the linear elastic solution, it shows the percentage of the required steel weight for each panel with respect to the total reinforcement weight in the form of column charts. The user can alternatively ask for a graphical description of the comparative reinforcement weights of different reinforcement fields within a reinforcement layer of the active panel. At each stage of the elasto-plastic optimization, the above information is also presented for the optimized reinforcement distribution next to their original linear elastic values. This enables the engineer to actively follow the optimization process and view the consequences of the selection of reinforcement fields and the introduction of each redistribution load case among different reinforcement fields and panels.

The window for 2D View of active panel shows the active panel in that panel’s local coordinate system. The user can switch among different panels at all stages in order to see or manipulate them in the 2D view of active panel window. The window can be split into two sub-windows which is particularly useful for presenting the reinforcement of the two layers for combined membrane-bending panels when the Design mode is active.

The window for 3D View of spatial structure is always active and shows the overall structural shape. The mesh and structural deformations under various load combinations can be viewed here. It also helps to facilitate working in the 2D view of active panel window, e.g. if an element is selected to examine its stress values in the active 2D panel, the element is marked to show where it is exactly located in the three-dimensional finite element system. For a better visualization in 3D, options like dynamic rotation and inactivating selected panels are implemented.
6.3 Free Formulation element model

The finite element-based dimensioning and stress redistribution concepts presented in chapters 4 and 5 are basically independent of the finite element model adopted. Some minor changes may be necessary from one element model to the other, e.g. the free formulation model for membrane elements has a rotational degree of freedom which does not exist in the conventional isoparametric element models. At any case, these changes are secondary and very straightforward. In this section, the finite element model adopted in ORCHID is discussed. Emphasis here is less on theoretical details, which are obtainable from the given literature, rather on the procedures leading to the numerical calculation of the element stiffness matrices for membrane and bending of triangular and quadrilateral elements. More on the free formulation element model can be found in [Bergan 80], [Bergan, et al. 84 a,b] and [Bergan, et al. 85 a,b].

6.3.1 Stiffness matrix calculation

The nodal degrees of freedom, nodal forces and the generalized stresses in membrane and bending according to the free formulation theory are defined as

\[
v_m = \begin{bmatrix} d_x \\ d_y \\ r_z \end{bmatrix}, \quad F_m = \begin{bmatrix} F_x \\ F_y \\ M_z \end{bmatrix}, \quad \sigma_m = \begin{bmatrix} n_{xx} \\ n_{xy} \end{bmatrix}
\]

(6.1)

\[
v_b = \begin{bmatrix} r_x \\ r_y \\ d_z \end{bmatrix}, \quad F_b = \begin{bmatrix} M_x \\ M_y \\ F_z \end{bmatrix}, \quad \sigma_b = \begin{bmatrix} m_{xx} \\ m_{xy} \end{bmatrix}
\]

(6.2)

where \(d, r, F\) and \(M\) denote displacement, rotation, force and moment, respectively. The subscripts \(m\) and \(b\) denote membrane and bending.

The specified degrees of freedom resemble those of other element models except for the membrane rotational degree of freedom \(r_z\). This is defined as

\[
r_z = \frac{1}{2} \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)
\]

(6.3)

The physical interpretation of \(r_z\) can be seen from Fig. 6.5 in which the deformations in the neighborhood of a node subjected to an in-plane displacement field \(u(x,y)\) are demonstrated. The angles \(\gamma_x\) and \(\gamma_y\) can be split into a rigid body rotation \(r_z\) and a shear strain \(\varepsilon_{xy} = \varepsilon_{yx}\) as depicted in Fig. 6.5. It can be seen that \(r_z\) corresponds to the rigid body rotation in the neighborhood of a node as a result of an arbitrary displacement field.
It should be noted that while $\varepsilon_{xy}$ depends on the choice of the coordinate axes $(x, y)$, the rotational degree of freedom $r_z$ is invariant with respect to this selection.

\[ \gamma_x = \frac{\partial v}{\partial x}, \quad \gamma_y = -\frac{\partial u}{\partial y} \]

\[ \varepsilon_{xy} = \frac{1}{2} (\gamma_x - \gamma_y) = \frac{1}{2} \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \]

\[ r_z = \frac{1}{2} (\gamma_x + \gamma_y) = \frac{1}{2} \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) \]

Figure 6.5 Definition of rotational degree of freedom in membrane free formulation elements

A lumping matrix $L$ is introduced which transforms a constant element stress field $\sigma_c$ or, alternatively, strain field $\varepsilon_c$ to its equivalent nodal forces $F_c$

\[ F_{mc}^j = L_{mc}^j \sigma_c = L_{mc}^j D_m \varepsilon_{mc} \]  \hspace{1cm} (6.4)  

\[ F_{bc}^j = L_{bc}^j \sigma_c = L_{bc}^j D_b \varepsilon_{bc} \]  \hspace{1cm} (6.5)  

with $j$ referring to $j$-th node and $c$ denoting a constant stress field. Adopting the abbreviations

\[ y_j - y_i = y_{ji}, \quad x_i - x_j = x_{ij} \]  \hspace{1cm} (6.6)  

the lumping matrix for membrane corresponding to node $j$ can be derived as

\[ L_m^j = \frac{1}{2} \begin{bmatrix} y_{ki} & \cdot & x_{ik} \\ \cdot & x_{ik} & y_{ki} \\ \frac{1}{4}(y_{ji}^2 - y_{kj}^2) & \frac{1}{4}(x_{ji}^2 - x_{jk}^2) & \frac{1}{2}(x_{ji} y_{ji} - x_{jk} y_{kj}) \end{bmatrix} \]  \hspace{1cm} (6.7)  

Similarly, the lumping matrix for bending is given by

\[ L_b^j = \frac{1}{2} \begin{bmatrix} y_{ik} & x_{ki} & y_{ki} \\ \cdot & x_{ki} & x_{ki} \\ \cdot & \cdot & \cdot \end{bmatrix} \]  \hspace{1cm} (6.8)  

The total lumping matrix for the whole triangular or quadrilateral element will then be

\[ L_m = \begin{bmatrix} L_m^1 \\ \cdot \\ \cdot \\ L_m^n \end{bmatrix}, \quad L_b = \begin{bmatrix} L_b^1 \\ \cdot \\ \cdot \\ L_b^n \end{bmatrix} \]  \hspace{1cm} \text{with} \quad n = 1, 3 \text{ or } 4 \]  \hspace{1cm} (6.9)  

112
The displacement field $u$ within an element is expressed by a complete rigid body and constant strain field expressions ($rc$ modes) and a set of higher order functions ($h$ modes)

$$u = N_{rc} q_{rc} + N_h q_h = [N_{rc} N_h] q = N q \quad (6.10)$$

$N_{rc}$ expresses a complete polynomial to a degree which corresponds to $rc$ modes for the given type of problem while $N_h$ are higher order functions. $q_{rc}$ and $q_h$ are the corresponding polynomial coefficients. The total number of $rc$ and $h$ modes should be equal to the number of degrees of freedom $n$ in the element nodal vector $a$. $N_{rc}$ and $N_h$ may be termed generalized functions. The strains $\epsilon$ are then given by

$$\epsilon = A u = \Delta N_{rc} q_{rc} + \Delta N_h q_h = B_{rc} q_{rc} + B_h q_h \quad (6.11)$$

$A$ is the appropriate strain-producing differential operator which was defined for membrane and bending by (2.3) and (2.18), respectively. $B_{rc}$ and $B_h$ contain gradients of the generalized displacement modes. The nodal displacements $a$ are found by inserting appropriate nodal coordinates in (6.10).

$$a = a_{rc} + a_h = G_{rc} q_{rc} + G_h q_h = [G_{rc} G_h] q = G q \quad (6.12)$$

It is necessary that all nodal values of the generalized patterns are linearly independent so that (6.12) may be inverted.

$$q = \begin{bmatrix} q_{rc} \\ q_h \end{bmatrix} = G^{-1} a = H a = \begin{bmatrix} H_{rc} \\ H_h \end{bmatrix} a \quad H = G^{-1} \quad (6.13)$$

The element stiffness matrix $k$ is constructed as the sum of a basic stiffness $k_b$ and a higher order stiffness $k_h$.

$$k = k_b + k_h \quad (6.14)$$

The basic stiffness is derived directly from the elasticity and lumping matrices

$$k_b = \frac{1}{V} L D L^T \quad (6.15)$$

in which $V$ is the volume of the element. The higher order stiffness $k_h$ is defined by (6.16) where $k_{qh}$ is the generalized stiffness matrix for the higher order modes

$$k_h = H_h^T k_{qh} H_h \quad k_{qh} = \int_{V} B_h^T D B_h \, dV \quad (6.16)$$
It can be shown that the above formulation is similar to the conventional stiffness matrix form provided a modified $B$ matrix is used.

$$ k = \int_B B^T D B \, dV \quad B = \frac{1}{V} L^T + B_h H_h $$

(6.17)

The evaluation of the integral in (6.17) is performed by a 2x2 Gaussian integration over the element area of quadrilateral elements. Triangular elements are numerically integrated with 3 sampling points (Fig. 6.6).

**Figure 6.6 Adopted integration points coordinates**

The higher order modes $B_h$ in the free formulation element model can be chosen with much more flexibility, i.e. rather freely. The only condition in their selection is the so-called energy orthogonality requiring that the work of the constant strain modes on the displacements of the higher order modes must vanish.

$$ \int_B B_h \, dV = 0 $$

(6.18)

This convergence criteria furnishes the following theorem:

*When a finite element is derived from a complete set of rigid body and constant strain modes plus a set of linearly independent higher order modes which is energy orthogonal to the first set, the element is convergent.*

It is much easier to find displacement patterns which satisfy the orthogonality condition (6.18) than functions that satisfy the inter-element compatibility requirements. Energy orthogonality can readily be obtained even in cases when conforming interpolation polynomials are impossible.
6.3.2 Membrane and bending triangular elements

a) Membrane action:

The generalized interpolation functions $N_m$ in the membrane action are defined in dimensionless Cartesian coordinates $(\xi, \eta)$ and $(\xi', \eta')$ as depicted in Fig. 6.7 (a). $(\xi, \eta)$ is defined by

$$
\xi = \frac{1}{\sqrt{A}} (x - x_C) \quad \eta = \frac{1}{\sqrt{A}} (y - y_C)
$$

(6.19)

in which $A$ is the element area and $(x_C, y_C)$ are the coordinates of the element centroid $C$. $(\xi', \eta')$ is found by an appropriate rotation of $(\xi, \eta)$ as shown in Fig. 6.7 (a). The interpolation functions for the rigid body, constant strain and higher order modes are defined as

$$
N_{mr} = \begin{bmatrix}
1 & -\eta \\
-\xi & 1
\end{bmatrix} \quad N_{me} = \begin{bmatrix}
\xi & \eta \\
\eta & \xi
\end{bmatrix}
$$

(6.20)

$$
N_{mh} = \begin{bmatrix}
\xi_1\eta_1 & \xi_2\eta_2 & \xi_3\eta_3 \\
-\frac{1}{2}\xi_1^2 & -\frac{1}{2}\xi_2^2 & -\frac{1}{2}\xi_3^2
\end{bmatrix}
$$

(6.21)

The coefficients corresponding to the rotational degree of freedom are found by applying (6.3) to the interpolation functions (6.20) and (6.21). The higher order $N_{mh}$ functions are rotated to the $(\xi, \eta)$ system after which the 9x9 $G_m$ matrix can be constructed by inserting the nodal coordinates followed by the determination of the 3x9 $B_m$ matrices at each integration point.

![Dimensionless coordinate systems for triangular elements for (a) membrane (b) bending](image)

**Figure 6.7** Dimensionless coordinate systems for triangular elements for (a) membrane (b) bending

b) Bending action:

The position of any point within a triangular element can uniquely be defined by 3 triangle coordinates $(\zeta_1, \zeta_2, \zeta_3)$ which are defined and related to the Cartesian coordinate system by
\[
\xi_i = \frac{A_i}{A} \quad \begin{bmatrix} 1 \\ x \\ y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix} \begin{bmatrix} \xi_1 \\ \xi_2 \\ \xi_3 \end{bmatrix}
\]

Fig. 6.7 (b) is a geometrical description of the triangle coordinate system. The interpolation functions for bending are then defined in this coordinate system as:

\[
N_{br} = \begin{bmatrix} \xi_1 & \xi_2 & \xi_3 \end{bmatrix} \quad N_{bc} = \begin{bmatrix} \xi_1 \xi_2 & \xi_2 \xi_3 & \xi_3 \xi_1 \end{bmatrix}
\]

\[
N_{bh} = \begin{bmatrix} \xi_1 (\xi_1 - \xi_2) & \xi_2 (\xi_2 - \xi_3) & \xi_3 (\xi_3 - \xi_1) \end{bmatrix}
\]

The functions corresponding to the degrees of freedom \( r_x \) and \( r_y \) are found by applying the definitions given in (2.16) to (6.23) and (6.24). The 9x9 \( G_b \) matrix and the 3x9 \( B_b \) matrices at each integration point can now be constructed.

### 6.3.3 Membrane and bending quadrilateral elements

**a) Membrane action:**

The non-dimensional coordinates \( \xi \) and \( \eta \) with origin at the centroid are defined by

\[
\xi = \frac{2}{\sqrt{A}} (x - x_c) \\
\eta = \frac{2}{\sqrt{A}} (y - y_c)
\]

quite similar to triangular elements. The \((\xi, \eta)\) system is chosen such that it follows the natural direction of the quadrilateral. Its coordinate directions are taken as the orthogonal directions of a best fit imaginary rectangle (Fig. 6.8).

![Local coordinates for a quadrilateral element](image)

**Figure 6.8 Local coordinates for a quadrilateral element**

The rigid body \( N_{mr} \) and constant strain \( N_{mc} \) interpolation functions for quadrilateral membrane elements are the same as those for triangular elements specified by (6.20). The interpolation functions for the higher order modes \( N_{mh} \) are defined by

\[
N_{mh} = \begin{bmatrix} \xi \eta & \eta^2 & 0 & 0 & \eta(\xi - \eta)^2 & -\eta(\xi + \eta)^2 \\ 0 & 0 & \xi \eta & \xi^2 & -\xi(\eta - \xi)^2 & \xi(\eta + \xi)^2 \end{bmatrix}
\]
The last two modes of \( N_{mh} \) with third order terms do not satisfy (6.18). As a result, the \( B_{mh} \) matrix must be corrected as

\[
B^*_h = B_h - \frac{1}{V} \int_V B_h \, dV
\]

(6.27)

\textit{b) Bending action:}

The interpolation functions for the \( rc \) and \( h \) modes are defined as

\[
N_{hr} = \begin{bmatrix} 1 & \xi & \eta \end{bmatrix} \quad N_{hc} = \begin{bmatrix} \xi^2 & \xi \eta & \eta^2 \end{bmatrix}
\]

(6.28)

\[
N_{hh} = \begin{bmatrix} \xi^3 & \xi^2 \eta & \xi \eta^2 & \eta^3 & \xi^3 \eta + c_2 \xi^2 - c_4 \xi \eta & \xi \eta^3 - c_5 \xi^2 \eta - c_5 \eta^3 \end{bmatrix}
\]

(6.29)

in which the coefficients of the second order term, added to the fourth order terms in order to achieve energy orthogonality requirements, are given by

\[
c_i = 3p_i \quad \text{with} \quad p_1 = \frac{1}{A} \int_A \xi^2 \, dA \quad p_2 = \frac{1}{A} \int_A \xi \eta \, dA \quad p_3 = \frac{1}{A} \int_A \eta^2 \, dA
\]

(6.30)

The construction of the 12x12 \( G_b \) matrix can follow similarly to the triangular elements.

6.4 Specific features of the finite element model

As noted earlier, the structure can in general consist of a few panels which are arbitrarily oriented and connected to one another in space. In this section, the specific features of the finite element model which could be helpful for other programmers of such a system are explained.

6.4.1 Combination of membrane and bending panels

Each of the panels of the structure can be defined as either a panel capable of carrying only membrane forces or as a bending panel subjected to both membrane forces and bending moments. In this section, an approach is presented which in numerical terms proved to be very efficient both with regard to the development of the code and to memory and computational effort.

The type of structure considered allowed the introduction of some special features in the finite element formulation. Advantages are obtained by the distinction between two types of nodes (Fig. 6.9): \textit{panel nodes} which solely belong to one panel and do not lie on the common edge of
two or more panels which are eccentrically connected to each other and *common nodes* which include all other nodes.

![Diagram showing distinction between panel and common nodes](image)

**Figure 6.9 Distinction between panel and common nodes**

For each panel, a *panel local coordinate system* \((x_p, y_p, z_p)\) is defined where the \(x_p\) and \(y_p\) axes lie in the panel plane and \(z_p\) is perpendicular to it considering the right hand rule. The set of degrees of freedom of each panel node refers to its respective panel local coordinate system. For common nodes, the reference coordinate system is the global \((X, Y, Z)\) axes. As a result, for the panel nodes which form the great majority of the nodes, no transformations of the element stiffness matrix are necessary from the local to the global coordinate system for assembling the global stiffness matrix. This is particularly useful in handling membrane panels with no bending action. It greatly reduces the computational time at this stage. For the few common nodes, of course, appropriate transformations are necessary before assembling the stiffness matrices of the surrounding elements these nodes belong to.

Another aspect is the choice of the order of the degrees of freedom for a node. In most general purpose finite element programs for analysing three-dimensional structures, the following order is chosen:

\[
DX \ DY \ DZ \quad RX \ RY \ RZ
\]  

(6.31)

where \(D\) and \(R\) refer to the translational (displacement) and rotational degrees of freedom, respectively. In our structure, the choice made was

\[
RX \ RY \ DZ \quad DX \ DY \ RZ
\]  

(6.32)

The difference here is that in (6.31) the membrane and bending degrees of freedom are mixed whereas in (6.32) they are separated. Also bearing in mind that the membrane and bending
stiffnesses of the element model used (see section 6.3.1) are decoupled and due to our specific type of problem, simplifications in various parts of the program were obtained. The most important one was related to the assemblage of the elements with their nodes in the panel’s coordinate system where no transformation was necessary except for the slave nodes to be discussed later. It should be noted, however, that choice (6.32) did in fact result in some complications with regard to the assemblage of the common nodes. All in all, the advantages outweigh the disadvantages in our model. More technical programming details are spared here.

![Image](image)

**Figure 6.10** Introduction of fictitious beams for adjoining membrane panels

As noted before, a panel can also be specified as being capable of withstanding only membrane forces. In this case, the bending stiffness matrices of this panel’s elements are not assembled in the global stiffness matrix. In the very special case, that several panels join each other and they are all specified as membrane panels, the common nodes at this intersection will have a rotational degree of freedom along the edge axis which will result in a zero diagonal term in the global stiffness matrix which is not desirable (Fig. 6.10 a). Another translational zero diagonal term will also be produced if two membrane panels which are in the same plane come together (Fig. 6.10 b). To overcome this problem, a fictitious beam is introduced along the connecting edge of these panels and its stiffness matrix is assembled. The beam constants should be non-zero small values and should not be so big as to influence the overall structural behavior. A square cross section with a side length equal to the minimum thickness of the adjoining panels was used for the fictitious beam.

### 6.4.2 Modelling incompatible meshes and eccentric panel connections

**a) Incompatible meshes:**

The term *incompatible meshes* is used for those cases in which a node of an element is located along the side of the adjacent element. This mostly arises when irregular boundaries are to be meshed or a portion of the mesh is refined (Fig. 6.11 a).
In the terminology used here, a slave node of one element is to be constrained to the free master nodes of the adjoining element. The slave can be constrained to one or at most two masters. In the latter case, the slave must lie between its two masters and on the line connecting them. For nodes along the edge of eccentric panel connections, the projection of the node on this line must satisfy this condition. The constraint here is that the slave sl has to remain on the side of the deformed shape of \( m1 - m2 \) when the mesh is deformed under the external loads. This does represent an incompatibility in deformations along \( m1 - sl \) and \( sl - m2 \) but it is consistent with the whole FE approach where these incompatibilities are tolerated since they diminish as the mesh is refined. The transformation matrix consists of the shape functions along one side of an element. Denoting by \( T_{m1-sl}^{m-i} \) the transformation matrix relating master node \( mi (i = 1,2) \) and slave node \( sl \) for the membrane action in non-rotated coordinate system (i.e. a system where the x axis is along \( m1 - m2 \) and points from \( m1 \) towards \( m2 \) as shown in Fig. 6.11 (b)), these matrices are:

\[
T_{m1-sl}^{m-i} = \begin{bmatrix}
1 - \xi & \cdot & \cdot \\
\cdot & 1 - 3\xi^2 + 2\xi^3 & (\xi - 2\xi^2 + \xi^3)l \\
\cdot & -6\xi + 6\xi^2 & 1 - 4\xi + 3\xi^2
\end{bmatrix}
\]  

(6.33)

\[
T_{m2-sl}^{m-i} = \begin{bmatrix}
\xi & \cdot & \cdot \\
\cdot & 3\xi^2 - 2\xi^3 & (-\xi^2 + \xi^3)l \\
\cdot & 6\xi - 6\xi^2 & -2\xi + 3\xi^2
\end{bmatrix}
\]  

(6.34)

where \( l \) is the length between \( m1 \) and \( m2 \) and \( \xi = l_{m1-sl} \). For the rotated coordinates, i.e. a system independent of the orientation of \( m1 - m2 \) and usually the panel's coordinate system (Fig. 6.11 c), the rotation matrix \( R \) is
6.4 Specific features of the finite element model

\[
R = \begin{bmatrix}
\cos \alpha & \sin \alpha \\
-\sin \alpha & \cos \alpha \\
. & . & 1
\end{bmatrix}
\]

(6.35)

The transformation matrix will then be

\[
T_{mi-sl}^{mn} = R^T T_{mi-sl}^{mn} R \quad (i = 1, 2)
\]

(6.36)

It should be noted here that so far, panel nodes in panel local coordinate systems have been considered. If the master nodes here are common nodes in the global system, then further transformations at the assemblage level will also be needed.

Handling the corresponding case for bending action is quite similar. The shape functions resemble the ones for membrane but the positions and some signs of the elements in matrices \(T_{mi-sl}^{mn} \) are different from \(T_{mi-sl}^{mn} \). Further elaboration is not given here.

b) Eccentric panel connections

A special approach must be adopted to accommodate eccentric panel connections. This case arises when panels with different thicknesses join each other (Fig. 6.12 a). Since the FE discretization assumes the panel plane to be located at the center of the panel height, the neighboring nodes are eccentrically connected to one another and special treatment is necessary along these panel boundaries.

One possible way of tackling the problem is the introduction of short very stiff beam elements between the two eccentric panels. This oversimplified way of by-passing the problem is not recommended since it will result in a badly-conditioned stiffness matrix. The big difference in the values of the stiffnesses of normal elements and these unrealistic beams can result in numerical problems during the solution of the equation system. This problem is handled directly at the assemblage level by introducing transformation matrices constraining the nodes at the eccentric boundaries. This method is described below.

![Figure 6.12 Eccentric panel connections](image)
Fig. 6.12 (b) shows a 6x6 (non-diagonal) block of the global stiffness matrix relating slave node sl with its master node ml. The pure bending bb and membrane mm interaction of the two nodes were treated in the previous section. This section deals with the cross interaction of the membrane and bending actions of the slave and its masters.

Physically, the mb action means that

- membrane forces on the slave sl result in bending forces in the master ml and
- bending displacements in the master ml result in membrane displacements in the slave sl.

As shown in Fig. 6.12 (b), no bm action exists.

The transformation matrices in the non-rotated coordinate system are

\[
T^{m1-sl}_{mb-n} = \begin{bmatrix}
1 & \Delta (1 - 4\xi + 3\xi^2) & -\Delta \left(\frac{-6\xi + 6\xi^2}{l}\right) \\
-\Delta (1 - \xi) & 1 & 0 \\
-\Delta \left(\frac{6\xi - 6\xi^2}{l}\right) & 0 & 1
\end{bmatrix}
\] (6.37)

\[
T^{m2-sl}_{mb-n} = \begin{bmatrix}
1 & \Delta (-2\xi + 3\xi^2) & -\Delta \left(\frac{6\xi - 6\xi^2}{l}\right) \\
-\Delta (\xi) & 1 & 0 \\
-\Delta \left(\frac{6\xi - 6\xi^2}{l}\right) & 0 & 1
\end{bmatrix}
\] (6.38)

in which \(\Delta = t_{sl} - t_{m}\) is the eccentricity between the two panels and \(\xi = \frac{t_{m1-sl}}{t_{m1-m2}}\).

In the rotated coordinate system, they are:

\[
T^{m1-sl}_{mb-r} = \begin{bmatrix}
-\sin \alpha \cos \alpha (T_{01} + T_{01}) & \cos^2 \alpha T_{01} - \sin^2 \alpha T_{10} & \cos \alpha T_{02} \\
\cos^2 \alpha T_{10} - \sin^2 \alpha T_{01} & \sin \alpha \cos \alpha (T_{01} + T_{10}) & \sin \alpha T_{02} \\
\cos \alpha T_{20} & \sin \alpha T_{20} & 1
\end{bmatrix}
\] (6.39)

where \(T_{ij} = T^{m1-sl}_{mb-n}[i][j]\) and \(i = 1, 2\).

Two additional comments should be made here for the case of only one master for a slave:

Firstly, the transformation matrices \(T^{m1-sl}_{mm-n}, T^{m1-sl}_{bb-n}\) and \(T^{m1-sl}_{mb-n}\) can be found by setting \(\xi = 0\) in \(T^{m1-sl}_{mm-n}, T^{m1-sl}_{bb-n}\) and \(T^{m1-sl}_{mb-n}\) or putting \(\xi = 1\) in \(T^{m2-sl}_{mm-n}, T^{m2-sl}_{bb-n}\) and \(T^{m2-sl}_{mb-n}\). Secondly, the matrices \(T^{m1-sl}_{mm-n}, T^{m1-sl}_{bb-n}\) will expectedly be both equal to the unit matrix \(I_{3x3}\).

**Assemble into the global stiffness matrix:**

So far, the required transformation matrices are established. At this stage, we consider the process of assembling the stiffness matrices into the global stiffness matrix. Here, we treat the
most general case where the nodes $P_{sl}$ and $Q_{sl}$ of one element are each a slave of 2 other nodes (Fig. 6.12 a). The $i$-th masters of the nodes $P$ and $Q$ are denoted by $P_{mi}$ and $Q_{mi}$ respectively where $i = 1, 2$. The 6x6 block matrix connecting nodes $P_{sl}$ and $Q_{sl}$ of an element is assembled at four different places in the global stiffness matrix according to Fig. 6.13 (b). $T^{P_{mi}}_{mb}$ is the transpose of the transformation matrix relating the $mb$ action of node $P_{sl}$ with its master $P_{mi}$.

![Figure 6.13](image-url)

**Figure 6.13 Modification of the stiffness matrix for slave nodes**

There are two further tasks to be performed. The first is the transfer of nodal loads on the slave to its masters while assembling the right hand side load vector. The second is the evaluation of the displacements of slaves from their masters after the equation system had been solved. The contributions of each of the following three cases in the load and displacement vector of the nodes is formulated below:

**pure bending action ($bb$):**

\[
p_b^{m1} : p_b^{m} = T_{bb}^{m1} p_b^{sl} \\
p_b^{m2} = T_{bb}^{m2} p_b^{sl}
\]  
(6.40)

\[
a_b^{m1}, a_b^{m2} : a_b^{sl} = T_{bb}^{m1} a_b^{m1} + T_{bb}^{m2} a_b^{m2}
\]  
(6.41)

**pure membrane action ($mm$):**

\[
p_{m}^{m1} : p_{m}^{m} = T_{ss}^{m1} p_{m}^{sl} \\
p_{m}^{m2} = T_{ss}^{m2} p_{m}^{sl}
\]  
(6.42)

\[
a_{m}^{m1}, a_{m}^{m2} : a_{m}^{sl} = T_{mm}^{m1} a_{m}^{m1} + T_{mm}^{m2} a_{m}^{m2}
\]  
(6.43)

**membrane-bending interaction ($mb$):**

\[
p_{b}^{m1} : p_{b}^{m} = T_{sb}^{m1} p_{m}^{sl} \\
p_{b}^{m2} = T_{sb}^{m2} p_{m}^{sl}
\]  
(6.44)

\[
a_{b}^{m1}, a_{b}^{m2} : a_{b}^{sl} = T_{mb}^{m1} a_{b}^{m1} + T_{mb}^{m2} a_{b}^{m2}
\]  
(6.45)
6. ORCHID: A Program for "Optimum Reinforced Concrete Highly Interactive Dimensioning"

6.5 External loads and boundary conditions

A variety of external load types have been implemented. These consist of concentrated nodal forces and moments in the global and the panel’s local coordinate system, prescribed nodal displacements and rotations, distributed normal, shear and transverse forces and distributed moments along each side of a finite element in the panel’s local coordinate system and distributed normal and shear forces on the entire element area in both the global and panel’s local coordinate system. Prescribed element strains and curvatures can also be dealt with as needed for the optimization phase described in chapter 5.

Structural self-weight can automatically be taken into account. For these different types of loading, the method in principle consists of finding the equivalent nodal forces which can then be included in the right hand side load vector. For distributed forces, the appropriate element interpolation functions are chosen so as to find the equivalent nodal forces. In the case of prescribed displacements and rotations, both the stiffness matrix as well as the right hand side load vector must be modified in such a way that the displacement found after solving the equation system furnishes the predefined displacement for that particular degree of freedom. The various loading types supported by ORCHID, except for prescribed element strains and curvatures, are shown in Fig. 6.14.

**Figure 6.14 Types of loads and boundary conditions implemented in ORCHID**

The types of boundary conditions implemented in ORCHID consist of fixed nodal degrees of freedom, elastically supported nodal degrees of freedom and elastically supported elements. These are also summarized in Fig. 6.14.

The accommodation of fixed and elastically supported nodal boundary conditions follows the standard rules specified in the finite element literature [Bathe 82]. If the whole element is elastically supported, the element bending stiffness matrix is modified to consider a uniformly distributed force perpendicular to the element surface whose (yet to be determined) magnitude is proportional to the average element displacement in the transverse direction.
6.6 Equation solver

The resulting linear system of equations to be solved is:

\[ KA = F \]  \hspace{1cm} (6.46)

in which \( K \) is the global stiffness matrix related to all non-slave nodes of the structure. \( F \) is the vector or matrix of external loads and \( A \) represents the nodal displacements which are to be calculated. The \( K \) matrix is positive definite if stable boundary conditions are imposed, and it is always sparse.

In general, the solution of such a system of equations can be performed using two different approaches. In an iterative method, an initial estimate for the displacements is made which, if no better value is known, could be a null vector. This estimate is successively corrected until the change in the last estimate of the displacement vector is small enough. The number of iterations mainly depends on the quality of the starting vector and on the conditioning of the matrix \( K \). This is the reason why the time of solution can only be estimated very approximately in an iterative method. The iteration, however, will always converge provided that \( K \) is positive definite.

Iterative methods are most suitable if a very large problem must be solved very few times. On the contrary, direct methods are best suited for cases in which a medium-size problem is to be solved many times. The difference is that in an iterative approach, the effort needed for solving an equation system with \( n \) load cases is \( n \) times the effort for a single load case, whereas in a direct method, the computational effort for the first load case may seem large because of the time-consuming triangularization process, but the effort for solving additional load cases is small since it only involves forward elimination and backward substitution.

In our application here, the size of the problem is moderate but a very large number of load cases have to be solved in the optimization stage. Therefore, it was decided to adopt a direct method. This is briefly discussed here.

All direct methods are based on variants of Gaussian elimination. In the case of a positive-definite coefficient matrix \( K \), Cholesky’s method can be used in which \( K \) is factorized into the product \( LL^T \), where \( L \) is a lower triangular matrix, and then the triangular systems \( Ly = F \) and \( L^TA = y \) are solved by forward elimination and backward substitution, respectively. When a sparse matrix is factored using Cholesky’s method, the matrix normally suffers fill, meaning that the triangular factor \( L \) will have non-zeros in some of the positions which are zero in \( K \).
For an NxN permutation matrix $P$, the matrix $PKP^T$ remains sparse and positive definite, so Cholesky’s method still applies

$$ (PKP^T)PA = PF $$

(6.47)

In general, a suitable choice of $P$ can often drastically reduce fill. If zeros are exploited, this can in turn imply a reduction in storage requirements and/or arithmetic requirements for the linear equation solver. The determination of an optimum $P$ is a problem of the broader topic in mathematics called graph theory. A heuristic algorithm which has been found to be very effective in finding efficient orderings with respect to fill is the so-called minimum degree algorithm. A modification of this algorithm, leading to improved performances for finite element matrix problems, was suggested by [George, et al. 78]. More on the evolution of this algorithm can be found in [George, et al. 89].
7 Examples

In this chapter, some examples are given to show how the elasto-plastic optimization based on the proposed approach can redistribute the forces in a rational way and simultaneously reduce total steel weight. The emphasis here is not on the quantitative values, rather the examples show in a qualitative manner the fundamental concepts and the possibilities and restrictions involved. As a result, only the geometrical dimensions, type of loading, boundary conditions and reinforcement fields are given and the precise values of stresses, reinforcement contents and arrangement, etc. are left out. The significance of the results is in their relative values and distribution which are graphically presented.

The examples provided point to the following aspects of optimization in particular:

- flattening out of stress peaks induced by the finite element model,
- concentrating reinforcement where it is most efficient for the ultimate design state,
- possibilities for the user to affect the load transfer mechanism in the elasto-plastic solution,
- effects of model assumptions on the resulting reinforcement layout.

7.1 T-beam modelled by two panels

In this example, a T-beam is modelled with two panels and continuum elements. The beam is fully constrained at one end while it is simply supported at the other end. It is subjected to a transverse load uniformly distributed along its axis. In the form of an ASCII file, the user first defines the general geometry of the structure and material properties. Based on this information, the program automatically generates a finite element mesh. The degree of refinement of the element mesh is controlled by the user through specifying in the input file the size of a typical element in a panel. The selected mesh here consists of 114 nodes and 90 elements. Although a very coarse mesh is deliberately adopted, it adequately serves the purpose of demonstrating some aspects of the proposed reinforcement dimensioning approach and the way this task is accomplished with program ORCHID. The user then specifies interactively the applied loads and boundary conditions. All displacement degrees of freedom for the nodes on the constrained side are blocked. Their rotational degrees of freedom are free. On the simply supported side, the vertical displacement degree of freedom of one node is blocked.
The panel representing the web of the girder is specified as a panel only in membrane action. The horizontal panel modelling the flange is defined as a combined membrane-bending panel. Since the flange panel is primarily under membrane action, it could also have been modelled as a membrane-only panel. However, one should note that if the flange panel were modelled as membrane-only, the vertical external loads had to be exclusively applied on the nodes in the middle which also belong to the web panel. If vertical forces were applied on the side nodes or the elements were subjected to transverse uniform forces, these membrane-only elements would have not been capable of transferring these forces to the web.

![Diagram](image)

**Figure 7.1** T-beam modelled with continuum elements and two panels

A view of the girder, its cross section and the undeformed and deformed mesh under the applied loads are shown in Fig. 7.1. After assembling the global stiffness matrix and load vectors, the equation system is solved and the nodal displacements are found for all nodes. From the element nodal displacements $a$, the element nodal forces $p$ are subsequently determined. Based on the element nodal forces $p$, the program then finds the element stresses corresponding to the three constant strain modes as discussed in chapter 4. For membrane action, these are $S_{m1}$, $S_{m2}$ and $S_{m3}$ which were defined by (4.30) and determined from (4.33). For bending action, they are $S_{b1}$, $S_{b2}$ and $S_{b3}$ defined by (4.39) and determined from (4.42). Also, transverse shear stresses $S_{b1}^q$ and $S_{b2}^q$ defined by (4.86) and calculated from (4.88) are found as well. When divided by the element area, these average stresses are then displayed in different
7.1 T-beam modelled by two panels

graphical and numerical forms on the screen. Investigating the direction and relative intensities of the principal stresses can generally demonstrate how the structure is behaving under the applied loads. This can be done for specific load cases and load combinations. The program then finds the worst case for each constraint among all load combinations. The tableau thus established by finding the hyperplanes encompassing all yield conditions for all load combinations represents the starting point for the optimization process.

In the present example, just one load case is applied and the limit state equals the structural response under this single load case. The elastic principal stresses in the T-beam without any stress redistribution are shown for the two panels in Fig. 7.2. The thickness of the lines is proportional to the stress intensities (within each plot of a panel but not between different panels) and the line directions correspond to those of the principal axes.

![flange panel](top view)

![web panel](side view)

**Figure 7.2 Distribution of principal membrane stresses of the T-beam**

Based on these results, the user can define the reinforcement fields for each panel. If the structure becomes too complicated, it may be difficult to define suitable reinforcement fields just by examining the stress distribution. A view of the required steel amounts, on the other hand, can be more helpful for this task. Since the reinforcement evaluation by the linear program prior to the optimization stage is extremely quick, it is advisable to evaluate the reinforcement corresponding to the linear elastic stress distribution in two steps. Firstly, one defines for each panel just two reinforcement fields which are perpendicular to one another, each covering the whole panel completely. If the panel includes bending action, this is done for both layers. These reinforcement fields are specified preferably along the dimensioning directions as discussed in sections 4.4.1 and 4.4.2. The linear program is then established and the reinforcement is evaluated for this configuration. For the T-beam in consideration, this is shown in Fig. 7.3 in which the steel amount is proportional to the line thicknesses. Although the flange panel is modelled as combined membrane-bending and therefore in two layers, for a better visualization and compactness in the following figures, the steel amount shown in one layer here equals the sum of the reinforcement amounts of the two layers.
Figure 7.3 Required reinforcement for the linear elastic stress distribution

Now, the user can directly see the amount and distribution of reinforcement from which it is even easier to get a picture about structural behavior than was the case using principal stresses. Based on this reinforcement distribution, the user subsequently defines all the additional reinforcement fields he regards as suitable. From the programming point of view, having the two perpendicular reinforcement fields covering the whole panel is also helpful since some elements, which from a practical point of view do not need any reinforcement, may strictly need some very small amount of reinforcement in a particular direction in order for all inequalities in the linear program to be satisfied. The user-defined reinforcement fields are shown in Fig. 7.4.

Figure 7.4 User-defined reinforcement fields for the T-beam

The reinforcement corresponding to the linear elastic stress distribution for the steel layout of Fig. 7.4 is shown in Fig. 7.5.

In Fig. 7.3, the required reinforcement is constant within each finite element. In Fig. 7.5, the reinforcement provided is constant over each user-defined reinforcement field, and so the required reinforcement only reaches the amount provided at a few points.
7.1  T-beam modelled by two panels

Figure 7.5 Required and provided reinforcement for the linear elastic stress distribution

The elasto-plastic optimization phase can follow now in which self-equilibrating stress states are introduced in order to alter the reinforcement distribution with the aim of minimizing the total steel weight. In this example, membrane self-equilibrating states of stress are found for the web panel and progressively introduced in the linear program. The determination of these stress states is always very fast, and so is the solution of the equation system for these additional redistribution load cases which involves only the forward elimination and backward substitution. The generally time-consuming solution of the linear program is, at each stage, also very fast so that the whole optimization process here is performed in a matter of seconds on a typical computer of today.

Figure 7.6 Reinforcement for the optimized elasto-plastic stress distribution

The most important results in terms of reinforcement weight percentages are given in Fig. 7.7. It should be noted that in these comparisons, other factors like the weight of the minimum reinforcement and additional reinforcement for detailing are not considered. Of course, taking into account these factors will change the final percentages of the T-beam.
### Figure 7.7 Reinforcement comparisons for the T-beam

Comparing the reinforcement distribution for the linear elastic (Fig. 7.5) and elasto-plastic (Fig. 7.6) solutions, a few points can be concluded:

- For the linear elastic solution of Fig. 7.5, the high tensile stress values and consequently large amount of required reinforcement in the finite element of the web panel’s upper left corner governs the amount of steel for the whole reinforcement zone which includes all elements covered by the same field. Such stress concentrations can not only arise from the physical behavior of the structure but can also be, in many cases, a result of the finite element model. In general, one remedy is the introduction of an additional reinforcement field in the regions of stress concentration, which may not always be practical. The elasto-plastic stress optimization has flattened out this stress peak and thus resulted in a smaller total steel weight.

- Longitudinal reinforcement along the girder axis is redistributed and reduced. In exchange, the reinforcement is mostly concentrated at the bottom of the girder where the lever arm for withstanding the applied loads is maximum. In this way, a more rational and effective use of reinforcement is accomplished.

- The amount of reinforcement in the flange panel at the fixed support for the elasto-plastic solution is slightly less than that of the linear elastic solution. This is in exchange of more reinforcement at the lowest reinforcement field (nr. 3) of the web panel. For different girder dimensions and reinforcement layouts, the optimization process may find it helpful for steel weight minimization to redistribute some or possibly all flange reinforcement from the support and in exchange increase the steel area in the span. This resembles a beam with a plastic hinge formed at the fixed support so that the beam carries the load as simply supported on both sides. In order to control the size and distribution of cracks, the engineer can prevent this from happening by explicitly specifying a specific minimum amount of reinforcement at the support of the flange panel. This further underlines the importance of the role of the engineer in interacting with the program.
7.2 Three-panel plate with stiffening walls

This example concerns with the elasto-plastic structural behavior of a plate panel under primarily bending action. The plate is simply supported along two of its edges, while on the other two edges, two parallel walls act as stiffeners mostly under membrane action. The horizontal plate is subjected to a uniform load in the transverse direction. All three panels are modelled as combined membrane-bending. The structural geometry, dimensions and the deformed mesh under the applied load are shown in Fig. 7.8.

Fig. 7.8 Structural geometry and deformed mesh

Fig. 7.9 shows the principal stresses as explained in the previous example. The membrane forces produced in the plate are due to the action of the two vertical side panels. If these two panels did not exist or if they were modelled instead by simple supports, no membrane forces would have been produced in the plate since the membrane and bending finite element formulations are decoupled.

The direction and intensities of the principal transverse shear forces in the plate clearly show how the distributed forces are transferred either directly or through the neighboring panels to the supports. The load transfer here (Fig. 7.9 c) resembles that of a square plate simply supported at the four edges and subjected to a uniform load.

Although in the optimization process the whole structure is modelled, only those results of the plate panel which are of primary interest are shown here. As explained in section 7.1, for each panel the minimum required number of reinforcement fields are introduced, and the required reinforcement amounts are found for all panels.
7. Examples

Figure 7.9 Stress distribution for the linear elastic solution

With two independent reinforcement fields for each layer of the plate panel and with a dimensioning direction parallel to the plate edges, the required and provided amount of reinforcement is shown in Fig. 7.10.

Figure 7.10 Required reinforcement
As shown in Fig. 7.10 for the top layer, some reinforcement must be provided in the corner areas. The reinforcement in these areas should not be provided just due to the finite element analysis results, which in many other cases are exaggerated because of the numerical model, but because of the provisions normally required by design codes for various reasons. Indeed, some codes may require some degree of reinforcement at midspan at the top layer in order to accommodate the stresses produced by concrete creep and shrinkage. As pointed out before, these issues are not of concern in this work. The assumption is that reinforcement requirements for serviceability criteria are considered separately and only the additional reinforcement needed for the ultimate load is minimized.

Different scenarios for the elasto-plastic solution are investigated. Starting with the solution of Fig. 7.10, the reinforcement minimization procedure is applied. The result is shown in Fig. 7.11. The plate is transferring the loads like a one way slab to the simple supports. The reduced amount of reinforcement is 53% of the total weight needed for the elastic solution of Fig. 7.10.

\[\text{Figure 7.11 Elasto-plastic load transfer no. 1}\]

In the second case, an additional uniaxial reinforcement field is introduced in the central area in a direction parallel to the vertical panels. A new reinforcement distribution corresponding to the linear elastic solution is then found. The corresponding reinforcement weight before optimization is almost the same as in the case of Fig. 7.10. Having performed the optimization process, both the reinforcement distribution and the directions of the principal transverse shear stresses in Fig. 7.12 clearly show that three regions are formed. The central region covered by the newly introduced reinforcement field transfers the load like a one way slab to the simple supports. Some smaller portion of the load is transferred to the two vertical panels which in turn transfer the load with a compressive arch action to the simple supports on the other two sides. The total amount of reinforcement is 50% of that of the elastic solution for this new configuration.
In the third scenario, a uniaxial reinforcement field in the perpendicular direction to the previous case is introduced. Like the previous case, the corresponding reinforcement weight before optimization is almost the same as in the case of Fig. 7.10. The subsequent optimization does not lead to the same behavior in the other direction. Most of the load is still transferred by a one way slab action directly to the simple supports. The rest of the load is primarily transferred through this additional field to the vertical panels and from there to the supports. This can be explained by considering the interaction between the vertical panels with the plate. If a situation similar to the case of Fig. 7.12 were supposed to take place in the perpendicular direction, then the compressive arch action in the vertical panels would produce a tensile force in the plate between the two simple supports. The reinforcement would then be redistributed in the other direction which is similar to what has happened now. The reinforcement amount in this case is 53% of its elastic amount.

The slight unsymmetries in the results after the optimization are due to the fact that the groups of the plastic strain and curvature distributions at the optimization level are not introduced symmetrically.
Comparing cases 2 and 3, it can be seen that in case 2, the load is transferred primarily by the bending action of the plate whereas in case 3, the plate partly acts as a tensile flange for the vertical panels which are in compressive arch action.

### 7.3 Effects of model assumptions in optimization

A simple one way slab supported by two vertical panels is considered. Different models are compared here to demonstrate the influence of the assumptions in the analysis and the subsequent reinforcement dimensioning of spatial structures and the optimization process. Important conclusions based on the results of this very simple example are drawn afterwards.

The horizontal slab is subjected to a uniform vertical load. The overall dimensions and panel thicknesses are shown in Fig. 7.14. The vertical panels are fully constrained at the bottom. Each of these two panels is covered by two independent reinforcement fields in the horizontal and vertical directions. Although in practice, some minimum amounts of steel are normally
provided, no minimum values are specified here in order to have a better comparison of the results. In the next figures, the line thicknesses are proportional to the amount of required reinforcement in the direction shown.

**Figure 7.14** Simple one way slab with two vertical panels

The different models investigated are the following:

1) All 3 panels are modeled as combined membrane-bending panels. The required reinforcement corresponding to the linear elastic solution is shown in Fig. 7.15 (a). The required amount is the basis for comparison and is equal to 100%.

2) If the top layer of the slab is covered with only two reinforcement fields in the two perpendicular directions along the slab edges, an elasto-plastic optimization of the solution of model 1 using ORCHID will result in a force redistribution and therefore new reinforcement amounts as shown in Fig. 7.15 (b). The negative moment on the slab edges are removed and compensated by an increase of the positive moment in the slab middle. This corresponds to the creation of a plastic hinge line along the two supporting edges. The total reinforcement amount is 62% of case 1.

3) Starting from the reinforcement configuration of case 1, two reinforcement fields are now added to the top layer as shown in Fig. 7.15 (c). The reinforcement amount for this new layout is 78% of case 1. The optimization process will lead to the reinforcement distribution as shown in Fig. 7.15 (c). Some of the top reinforcement is redistributed from the edges to the middle but the minimization does not go as far as in case 2. The total steel weight is 59% of case 1.

The two extreme case models approximating the above three-dimensional structure with a two-dimensional model would be

4) A simply supported beam on the two sides carrying the load by a positive field moment. This is also equivalent to the 3-panel model with the two vertical panels being defined as membrane-only.
5) A beam fully constrained along its edges where a positive moment with equal negative moments on the two sides are produced.

![Diagram of required reinforcement for linear elastic and elasto-plastic stress distribution.](Image)

**Figure 7.15** Required reinforcements for linear elastic and redistributed cases of a one way slab

The following conclusions can be drawn from this very simple but demonstrative example:

- In a spatial structure with interaction among different panels, isolating a panel and considering it separately is extremely difficult and bound to lead to gross simplifications which may not always be justified. This factor is more important for more complicated structural geometries and different load combinations.

- Engineering judgement can greatly influence the dimensioning process and show the program the way to go. The optimization process can be controlled by the introduction of reinforcement fields with a specified minimum value of reinforcement. They help in fulfilling the necessary ductility and serviceability requirements while the resistances they provide are also taken into account for investigating overall strength requirements.
A suitable dimensioning can be accomplished by exploiting the engineer’s experience and the program’s computational capabilities. In this example, minimum reinforcement according to code specifications must be provided everywhere and an additional reinforcement field in the top layer can control the crack propagation if the slab is not supposed to simply rest on the two vertical panels. Some of the reinforcement for these fields at the top can be redistributed to the bottom layer by allowing small cracks to appear along the edges. The degree of force redistribution depends on the amount of the applied load, the structural dimensions and the detailing of the edges. The bottom line is that the detailing must be compatible with the model adopted for analysis and design.
8 Closure

In this chapter, the most important aspects of the present work are summarized and some suggestions on the possibilities for future research in the continuation of the proposed method are discussed.

8.1 On the proposed method

In the context of this work, a new method for the ultimate load dimensioning of general reinforced concrete spatial structures consisting of flat panels was presented. The following aspects were considered:

- *Generality*: The finite element method was the underlying basis for the approach. As a result, the method is general and easily applicable to all sorts of structures consisting of flat panels.

- *Practice-oriented approach*: Only the linear elastic finite element method was used, thus avoiding nonlinear incremental analysis with complex material models. This makes the method easier to understand for a typical engineer and attractive for his everyday design.

- *Design for ultimate load*: The design is based on strength requirements for ultimate load. However, the linear elastic solution is available, which helps to examine the serviceability conditions.

- *Design-oriented approach*: The emphasis of the method was based on design, not on a precise stress analysis. The yield conditions were formulated for the nodal forces of the finite element rather than the conventional stress values computed for an infinitesimal point somewhere in each element. The idea of finite element discretisation was extended from analysis to design in a natural way.

- *Elasto-plastic design and stress redistribution*: Based on the lower bound theorem of plasticity, the introduction of self-equilibrating states of stress was suggested in order to alter the linear elastic stress distribution with the aim of reinforcement weight minimization and optimization. Unlike nonlinear step-by-step analysis, these states of stress are fictitious, i.e. can be
chosen at will. Their selection was governed by optimization criteria within the framework of linear programming.

In the context of this dissertation, the program ORCHID was developed to implement and investigate the proposed ideas and also to serve as a tool for optimum reinforcement dimensioning of general spatial structures consisting of flat panels. The program should help the structural designer in the task of reinforced concrete dimensioning in two different ways:

Inexperienced designers, in particular, can take advantage of the possibilities in easily investigating different reinforcement layouts corresponding to the linear elastic solution. Since the program is conceived to directly address the problem of design, the engineer can immediately see the consequences of his choice of reinforcement fields. Without having to handle a large amount of stress data, he is therefore able to conveniently examine different scenarios and consequently choose the most suitable option. Engineers with little design experience are urged to make use of the elasto-plastic optimization stage rather cautiously. It is helpful to introduce the redistribution load cases in order to flatten out the stress peaks and to obtain a smoother stress and reinforcement distribution in specific regions in the vicinity of singularity points. One should, however, be very cautious with significantly deviating from the linear elastic response.

An experienced designer, on the contrary, knows in advance by examining the linear elastic solution furnished by the program, how he would like to define his reinforcement fields. The ease in examining reinforcement configurations corresponding to the linear elastic solution is, as a result, not as vital to him as was the case for the inexperienced designer. The elasto-plastic optimization stage, on the other hand, can be promising since his engineering judgement together with the computational capabilities of the program can lead to a favourable elasto-plastic reinforcement solution. He can therefore show the program the way to go as well as control how far to proceed.

It can be concluded that the present approach and the developed computer program based on it is not intended to be a push-button piece of computer software to accomplish the task of reinforced concrete dimensioning. It is conceived to be a powerful tool and not a substitute for the design engineer.

In using the proposed approach, the engineer should take the following points into account:

- The application of the elasto-plastic reinforcement dimensioning approach is only valid if a certain degree of ductility of the reinforced concrete composite material is available. Although concrete is a brittle material, numerous experimental investigations in the past thirty years have shown that the application of plasticity theory for underreinforced concrete structures is justified. Already in 1961, Drucker wrote [Drucker 61]
“Although concrete is a material of very limited deformability, indications are that the load-carrying capacity of reinforced and prestressed structures will, in time, be computed on the basis of the limit theorems of plastic analysis and design”.

The design engineer should, on the one hand, try to use steel bars with acceptable deformation capacity and also try to ensure a suitable reinforcement arrangement within the structure so that the necessary ductility requirements are fulfilled. The provision of minimum reinforcement and a smooth distribution of steel bars with small diameters at short intervals can be helpful for this task.

- Experience has shown that reinforced concrete structures almost never collapse under normal circumstances due to a lack of overall strength simply because they are generally overdimensioned. Except for abnormal loading cases, e.g. earthquakes, unsatisfactory detailing has been mainly responsible when structural collapse has occurred. Therefore, extra attention should be paid to critical regions and possible failure modes like punching shear. In addition, the programming of specific national standards for reinforcement dimensioning was not of importance in this research project and should therefore be taken into account by the design engineer separately.

### 8.2 Suggestions for future work

The following improvements could be made:

- The same approach can be applied with minor modifications to other element models. It was explained in section 4.7 that elements with less accuracy from the analysis point of view, e.g. isoparametric elements, may be more suitable for the optimum design through stress redistribution suggested here. A study of other element models in comparison with the adopted free formulation element model may prove to be useful.

- Today, many structures are designed using prestressing techniques. Expanding the program to include prestressing would widen the range of structures that can be handled.

- With the rapid improvements both in hardware and software, ORCHID like every other program can be redesigned and improved in many ways. New ideas from object-oriented programming can be applied which make the current version of the program more suitable for expansion and integration.

- The current method used for the selection and the introduction of plastic strain redistribution load cases is quite efficient but heuristic. There is surely much room for improvement in this regard.

- Performing a nonlinear analysis for a reinforcement distribution found by the proposed method is useful both from an academic point of view and possibly for some real-world designs.
as well. Since the plastic strain redistribution load cases introduced here are fictitious and do not necessarily have much to do with the real structural failure mode, performing a subsequent nonlinear analysis may provide additional information on how the structure really behaves. The nonlinear calculation, however, is inevitably prone to the drawbacks outlined in the first chapter.
Symbols

**bold** characters for vectors and matrices

**normal** characters for scalars

**Abbreviations:**

- ANSI: American National Standards Institute
- ASCII: American Standard Code for Information Interchange
- FEM: Finite Element Method
- GUI: Graphical User Interface
- IO: Input / Output
- LP: Linear Programming
- MAX/MIN: Maximize/Minimize, Maximum/Minimum
- MB: Megabyte
- ORCHID: Optimum Reinforced Concrete Highly Interactive Dimensioning (program name)
- PT: Plasticity Theory
- RAM: Random Access Memory

**Greek symbols (vectors and matrices):**

- $\boldsymbol{\varepsilon}$: strain vector
- $\sigma$: stress vector
- $\Delta$: operator matrix

**Greek symbols (scalars):**

- $\alpha, \beta, \theta$: angle
- $\psi$: angle
- $\epsilon$: strain
- $\chi$: curvature
- $\xi$: position of slave wrt. its masters
- $\xi, \eta, \zeta$: element coordinate system
- $\mu$: factors for redistribution stress states
- $\kappa$: mesh parameter
- $\nu$: Poisson's ratio
- $\rho$: reinforcement ratio
- $\sigma$: normal stress
- $\tau$: shear stress

**Greek symbols (functions):**

- $\phi, \Phi$: yield condition functions
- $\partial$: partial derivative
### Symbols

<table>
<thead>
<tr>
<th>Roman symbols (vectors and matrices):</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>a</strong></td>
</tr>
<tr>
<td><strong>B</strong></td>
</tr>
<tr>
<td><strong>D</strong></td>
</tr>
<tr>
<td><strong>F</strong></td>
</tr>
<tr>
<td><strong>J</strong></td>
</tr>
<tr>
<td><strong>K</strong></td>
</tr>
<tr>
<td><strong>O</strong></td>
</tr>
<tr>
<td><strong>q</strong></td>
</tr>
<tr>
<td><strong>R</strong></td>
</tr>
<tr>
<td><strong>S</strong></td>
</tr>
<tr>
<td><strong>u</strong></td>
</tr>
<tr>
<td><strong>A</strong></td>
</tr>
<tr>
<td><strong>d</strong></td>
</tr>
<tr>
<td><strong>E</strong></td>
</tr>
<tr>
<td><strong>H</strong></td>
</tr>
<tr>
<td><strong>k</strong></td>
</tr>
<tr>
<td><strong>N</strong></td>
</tr>
<tr>
<td><strong>p</strong></td>
</tr>
<tr>
<td><strong>r</strong></td>
</tr>
<tr>
<td><strong>s</strong></td>
</tr>
<tr>
<td><strong>T</strong></td>
</tr>
<tr>
<td><strong>U</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Roman symbols (scalars):</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>A</strong></td>
</tr>
<tr>
<td><strong>d</strong></td>
</tr>
<tr>
<td><strong>dx, dy</strong></td>
</tr>
<tr>
<td><strong>D</strong></td>
</tr>
<tr>
<td><strong>c</strong></td>
</tr>
<tr>
<td><strong>d_q</strong></td>
</tr>
<tr>
<td><strong>( \delta )</strong></td>
</tr>
<tr>
<td><strong>E</strong></td>
</tr>
<tr>
<td><strong>f_c</strong></td>
</tr>
<tr>
<td><strong>f_c'</strong></td>
</tr>
<tr>
<td><strong>f_t</strong></td>
</tr>
<tr>
<td><strong>f_y</strong></td>
</tr>
<tr>
<td><strong>F</strong></td>
</tr>
<tr>
<td><strong>h</strong></td>
</tr>
<tr>
<td><strong>L</strong></td>
</tr>
<tr>
<td><strong>l_c</strong></td>
</tr>
<tr>
<td><strong>m, M</strong></td>
</tr>
<tr>
<td><strong>m_i</strong></td>
</tr>
<tr>
<td><strong>n, N</strong></td>
</tr>
<tr>
<td><strong>NE</strong></td>
</tr>
<tr>
<td><strong>NF</strong></td>
</tr>
<tr>
<td><strong>NL</strong></td>
</tr>
<tr>
<td><strong>NLC</strong></td>
</tr>
<tr>
<td><strong>NN</strong></td>
</tr>
<tr>
<td><strong>NP</strong></td>
</tr>
<tr>
<td><strong>NR</strong></td>
</tr>
<tr>
<td><strong>q</strong></td>
</tr>
<tr>
<td><strong>r</strong></td>
</tr>
<tr>
<td><strong>R</strong></td>
</tr>
<tr>
<td><strong>s</strong></td>
</tr>
<tr>
<td>Symbol</td>
</tr>
<tr>
<td>--------</td>
</tr>
<tr>
<td>$sl$</td>
</tr>
<tr>
<td>$u,v,w$</td>
</tr>
<tr>
<td>$U$</td>
</tr>
<tr>
<td>$V$</td>
</tr>
<tr>
<td>$W$</td>
</tr>
<tr>
<td>$x,y,z$</td>
</tr>
<tr>
<td>$X,Y,Z$</td>
</tr>
</tbody>
</table>

**Superscripts:**
- $a$: admissible
- $b$: bottom layer
- $c$: concrete
- $d$: displacement
- $e$: element number
- $I$: initial strains/curvatures
- $p$: panel number
- $q$: transverse shear
- $r$: rotation
- $s$: steel
- $t$: top layer

**Subscripts:**
- $b$: bending
- $c$: constant strain modes
- $e$: elastic
- $h$: higher order (first degree) strain modes
- $i,j$: auxiliary variables referring to $i,j$th term
- $l$: load case number
- $m$: membrane
- $n,t$: normal/tangential directions
- $p$: plastic
- $r$: rigid body modes
- $z$: zone
List of Figures

chapter 1
p. 6  fig. 1.1  Examples of spatial panel structures

chapter 2
p. 12  fig. 2.1  Linear strain-displacement relationship in a membrane panel
p. 13  fig. 2.2  Stress resultants in an infinitesimal membrane element
p. 13  fig. 2.3  Normal stress transformation in a membrane element
p. 14  fig. 2.4  Displacement components in plate bending
p. 15  fig. 2.5  Stress resultants on an infinitesimal plate bending element
p. 17  fig. 2.6  Shear stress transformation in a bending element
p. 18  fig. 2.7  General three-dimensional body in equilibrium
p. 23  fig. 2.8  Perfectly plastic materials in a uniaxial stress state
p. 25  fig. 2.9  Generalized strains and stresses for a beam section
p. 26  fig. 2.10  Yield surface in the generalized stress space
p. 30  fig. 2.11  Formulation of tableau of a general linear program
p. 30  fig. 2.12  Tableau modification for one pivoting exchange step
p. 31  fig. 2.13  Exchange step from an unfeasible minimum towards a feasible optimum solution
p. 32  fig. 2.14  Exchange step from a feasible non-optimum towards a feasible optimum solution

chapter 3
p. 34  fig. 3.1  Reinforced concrete infinitesimal element in membrane state
p. 34  fig. 3.2  Concrete material model
p. 35  fig. 3.3  Yield conditions for an infinitesimal concrete element in a membrane state
p. 36  fig. 3.4  Yield conditions for an orthogonal reinforcement net
p. 36  fig. 3.5  Nonlinear yield conditions for an infinitesimal reinforced concrete membrane element
p. 37  fig. 3.6  Linearized yield conditions of an infinitesimal reinforced concrete membrane element
p. 38  fig. 3.7  Superposition of reinforcement layers for an infinitesimal element
p. 39  fig. 3.8  Reinforced element subjected to pure bending
chapter 4

p. 48    fig. 4.1  Evaluation of nodal average stresses in the finite element approach
p. 52    fig. 4.2  Transition from infinitesimal to finite element-based dimensioning
p. 54    fig. 4.3  Linearly independent nodal forces for an isoparametric membrane finite element
p. 57    fig. 4.4  Strain modes for a rectangular membrane free formulation finite element
p. 59    fig. 4.5  Strain modes for a rectangular bending finite element
p. 61    fig. 4.6  Rectangular membrane finite element
p. 61    fig. 4.7  Admissible stress state for linear modes of membrane quadrilateral finite elements
p. 63    fig. 4.8  Reinforcement dimensioning direction for membrane finite element
p. 63    fig. 4.9  Stress transformation to a rotated coordinate system
p. 65    fig. 4.10 Selected constraints in the four zones of a membrane finite element
p. 66    fig. 4.11 Rectangular bending finite element
p. 66    fig. 4.12 Admissible stress state for linear modes of bending quadrilateral finite elements
p. 67    fig. 4.13 Reinforcement dimensioning direction for bending finite element
p. 68    fig. 4.14 Selected constraints in the four zones of a bending finite element
p. 71    fig. 4.15 Strain modes for transverse shear (with no rotations) in bending
p. 76    fig. 4.16 Membrane nodal force combinations in the unchecked subspace of a finite element
p. 76    fig. 4.17 Bending nodal force combinations in the unchecked subspace of a finite element
p. 77    fig. 4.18 Projection of yield conditions in a subspace
p. 78    fig. 4.19 Comparison between different element models in finite element-based dimensioning


chapter 5

p. 80  fig. 5.1 Force redistribution in a two element bar system
p. 82  fig. 5.2 Graphical presentation of the optimization problem for the two element beam
p. 84  fig. 5.3 Uniaxial (bar) and biaxial (net) reinforcement fields
p. 85  fig. 5.4 Definition of a) user-defined reinforcement fields b) program-generated reinforcement zones in membrane and bending panels
p. 88  fig. 5.5 Flow diagram of reinforcement design for the linear elastic stress distribution
p. 88  fig. 5.6 Evaluation of the extreme state forces for the linear elastic solution
p. 89  fig. 5.7 Building the linear program for the linear elastic stress distribution
p. 89  fig. 5.8 Setup of the tableau for reinforcement of the linear elastic solution
p. 90  fig. 5.9 Checking all constraints
p. 92  fig. 5.10 Typical full tableau size without progressive optimization
p. 92  fig. 5.11 Progressive optimization and expansion of the tableau
p. 93  fig. 5.12 Flow diagram of the optimization stage
p. 94  fig. 5.13 Expansion of the tableau for new initial strains
p. 96  fig. 5.14 Expansion of the tableau for new violated constraints
p. 97  fig. 5.15 Initial strain combinations shown on a square membrane finite element
p. 97  fig. 5.16 Initial curvature combinations shown on a square bending finite element
p. 98  fig. 5.17 Plastic curvature modes (a) with and (b) without concrete crunching
p. 99  fig. 5.18 Regrouped tableau in the beginning O₀ and after k exchange steps Oₖ
p. 100 fig. 5.19 Introduction of new columns (redistribution load cases) in the tableau at step k (Oₖ)

p. 100 fig. 5.20 Introduction of new rows (constraints) in the tableau at step k (Oₖ)

p. 101 fig. 5.21 Comparison between reinforcement minimization and ultimate load maximization

p. 102 fig. 5.22 Alternatives for ultimate load maximization transformed into a minimization problem

p. 102 fig. 5.23 Adopted tableau for the ultimate load analysis

chapter 6

p. 105 fig. 6.1 Basic stages towards optimum reinforcement dimensioning
p. 106 fig. 6.2 Flow diagram for “Creating and solving the linear elastic finite element model“

p. 107 fig. 6.3 Modularity of ORCHID

p. 109 fig. 6.4 Graphical User Interface (GUI) of ORCHID
List of Figures

<table>
<thead>
<tr>
<th>Page</th>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>112</td>
<td>fig. 6.5</td>
<td>Definition of rotational degree of freedom in membrane free formulation elements</td>
</tr>
<tr>
<td>114</td>
<td>fig. 6.6</td>
<td>Adopted integration points coordinates</td>
</tr>
<tr>
<td>115</td>
<td>fig. 6.7</td>
<td>Dimensionless coordinate systems for triangular elements for (a) membrane (b) bending</td>
</tr>
<tr>
<td>116</td>
<td>fig. 6.8</td>
<td>Local coordinates for a quadrilateral element</td>
</tr>
<tr>
<td>118</td>
<td>fig. 6.9</td>
<td>Distinction between panel and common nodes</td>
</tr>
<tr>
<td>119</td>
<td>fig. 6.10</td>
<td>Introduction of fictitious beams for adjoining membrane panels</td>
</tr>
<tr>
<td>120</td>
<td>fig. 6.11</td>
<td>Master–Slave specification</td>
</tr>
<tr>
<td>121</td>
<td>fig. 6.12</td>
<td>Eccentric panel connections</td>
</tr>
<tr>
<td>123</td>
<td>fig. 6.13</td>
<td>Modification of the stiffness matrix for slave nodes</td>
</tr>
<tr>
<td>124</td>
<td>fig. 6.14</td>
<td>Types of loads and boundary conditions implemented in ORCHID</td>
</tr>
</tbody>
</table>

**Chapter 7**

<table>
<thead>
<tr>
<th>Page</th>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>128</td>
<td>fig. 7.1</td>
<td>T-beam modelled with continuum elements and two panels</td>
</tr>
<tr>
<td>129</td>
<td>fig. 7.2</td>
<td>Distribution of principal membrane stresses of the T-beam</td>
</tr>
<tr>
<td>130</td>
<td>fig. 7.3</td>
<td>Required reinforcement for the linear elastic stress distribution</td>
</tr>
<tr>
<td>130</td>
<td>fig. 7.4</td>
<td>User-defined reinforcement fields for the T-beam</td>
</tr>
<tr>
<td>131</td>
<td>fig. 7.5</td>
<td>Required and provided reinforcement for the linear elastic stress distribution</td>
</tr>
<tr>
<td>131</td>
<td>fig. 7.6</td>
<td>Reinforcement for the optimized elasto-plastic stress distribution</td>
</tr>
<tr>
<td>132</td>
<td>fig. 7.7</td>
<td>Reinforcement comparisons for the T-beam</td>
</tr>
<tr>
<td>133</td>
<td>fig. 7.8</td>
<td>Structural geometry and deformed mesh</td>
</tr>
<tr>
<td>134</td>
<td>fig. 7.9</td>
<td>Stress distribution for the linear elastic solution</td>
</tr>
<tr>
<td>134</td>
<td>fig. 7.10</td>
<td>Required reinforcement</td>
</tr>
<tr>
<td>135</td>
<td>fig. 7.11</td>
<td>Elasto-plastic load transfer no. 1</td>
</tr>
<tr>
<td>136</td>
<td>fig. 7.12</td>
<td>Elasto-plastic load transfer no. 2</td>
</tr>
<tr>
<td>137</td>
<td>fig. 7.13</td>
<td>Elasto-plastic load transfer no. 3</td>
</tr>
<tr>
<td>138</td>
<td>fig. 7.14</td>
<td>Simple one way slab with two vertical panels</td>
</tr>
<tr>
<td>139</td>
<td>fig. 7.15</td>
<td>Required reinforcements for linear elastic and redistributed cases of a one way slab</td>
</tr>
</tbody>
</table>
Literature


Literature


<table>
<thead>
<tr>
<th>Reference</th>
<th>Description</th>
</tr>
</thead>
</table>