ALBERTO J. M. FIGUEIREDO; ANTÓNIO ADÃO DA FONSECA; ALVARO F. M. AZEVEDO

Analysis and Optimization of Reinforced Concrete Slabs

Computer Aided Optimum Design of Structures VI
S. Hernandez, A. J. Kassab & C. A. Brebbia (editors)
WIT Press, Southampton, U.K.
pp. 237-246, 1999

Analysis and optimization of reinforced concrete slabs

A.J.M. de Figueiredo, A.M. Adão da Fonseca, A.F.M. Azevedo
Department of Civil Engineering, Faculty of Engineering, University of Porto, Rua dos Bragas, 4099 Porto Codex, Portugal
Email: ajfiguei@fe.up.pt

Abstract

A mathematical model for the analysis and optimization of reinforced concrete slabs is presented. The formulation is based on the yield line theory and requires the discretization of the slab into a mesh of triangular finite elements. The yield lines are located on the sides of the finite elements and their identification is automatically achieved. The reinforced concrete exhibits an elastic perfectly plastic behavior and the static, kinematic and constitutive relations may be organized into mathematical programs. Constraints expressing the serviceability limit states or any other requirement may be added to the mathematical programs.
In the analysis problem, the collapse load parameter and the variables that define the collapse mechanism are calculated by a computer code that solves the corresponding non-linear program (NLP). When the most economical design is required, the variables of the NLP are the thickness of the slab and the flexural reinforcement. In both cases, the geometry of the finite element mesh is optimized and thus a small number of finite elements provide good results.
Examples illustrating the characteristics of the model are presented.

1 Introduction

The yield line theory (Johansen[1, 2]) is most commonly used in the plastic limit analysis of reinforced concrete slabs, and its application is suggested by many structural codes.

According to that theory, when loads applied to a reinforced concrete slab increase, plastic deformations occur along lines. When the number of lines is such that the structure is reduced to a mechanism, collapse occurs. In the yield line method, potential collapse mechanisms are postulated and the corresponding load parameters are evaluated.

The model presented in this paper is based on a formulation developed by Munro[3] and Da Fonseca[4]. A triangular finite element model in which potential yield lines are restricted to element boundaries represents the reinforced concrete slab. The fundamental relations of that model are equivalent to a pair of dual mathematical linear programs leading, in conformity with an optimization criterion, to the safest solution.

Introducing some new developments, the aforementioned linear model becomes non-linear and the plastic limit analysis or design (synthesis) of reinforced concrete slabs has to be formulated as a non-linear mathematical program.

2 Mathematical programming

The mathematical formulation of an optimization problem may be written as follows:

$$\text{Minimize } f(\underset{\sim}{x}) \qquad \underset{\sim}{x}=(x_1,\ldots,x_n) \tag{1}$$

subject to:

$$\underset{\sim}{g}(\underset{\sim}{x})\leq\underset{\sim}{0} \qquad \underset{\sim}{g}=(g_1,\ldots,g_m) \tag{2}$$

$$\underset{\sim}{h}(\underset{\sim}{x})=\underset{\sim}{0} \qquad \underset{\sim}{h}=(h_1,\ldots,h_l) \tag{3}$$

where x_i $(i = 1,\ldots, n)$ are the *design variables*, $g_j \leq 0$ $(j = 1, \ldots, m)$ are the *inequality constraints*, $h_k = 0$ $(k = 1, \ldots, l)$ are the *equality constraints* and f is the *objective function*.

This non-linear mathematical program is solved by a computer code called NEWTOP (Azevedo[5]), using the Lagrange-Newton method.

3 Mathematical model for plastic limit analysis

In *plastic limit analysis* problems, the collapse load parameter and the corresponding mechanism are determined. First, the slab is discretized into a mesh of triangular elements, taking into consideration that yield lines will be restricted to element boundaries. In order to establish the fundamental relations (kinematic, static and constitutive), boundary conditions, loads (multiplied by a load parameter) and material properties are given as data.

The kinematic variables are associated with mechanism modal deformations, regardless of prior elastoplastic deformations. These variables (see Figure 1) consist of the vertical displacements of the nodes and the rotations of the triangular finite element sides. Static variables (see Figure 1) consist of the corresponding nodal forces and bending moments. Constitutive relations express the elastic perfectly plastic behaviour of the material, by limiting the bending moments on the elements sides to be less or equal to the plastic moments and stating that mechanism deformations are only possible at plasticized element sides.

The most innovative feature of the model presented in this paper is the "optimization" of the yield line layout. In fact, the nodal coordinates can be changed in order to obtain improved solutions, even with coarse finite element meshes.

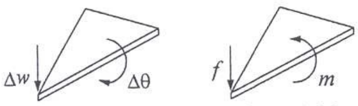


Figure 1. Kinematic variables and static variables of the model.

3.1 Finite element geometry

The static and kinematic relations involve the geometric variables defined in Figure 2.

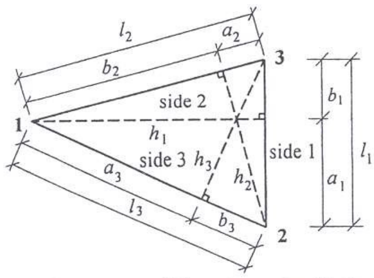


Figure 2. Geometry of the triangular finite element.

3.2 Kinematic relations

When the collapse mechanism is formed, the slab is divided in several panels with rigid behaviour. Angular discontinuities develop, but compatibility is satisfied by continuity of vertical displacements. Rotations of the outward normals to the three sides of the single element, $\Delta\theta_i$ (i=1,2,3), can be expressed in terms of the corner vertical displacements, Δw_j (j=1,2,3), as follows:

$$\begin{bmatrix} \Delta\theta_1 \\ \Delta\theta_2 \\ \Delta\theta_3 \end{bmatrix} = \begin{bmatrix} -\frac{1}{h_1} & \frac{b_1}{l_1 h_1} & \frac{a_1}{l_1 h_1} \\ \frac{a_2}{l_2 h_2} & -\frac{1}{h_2} & \frac{b_2}{l_2 h_2} \\ \frac{b_3}{l_3 h_3} & \frac{a_3}{l_3 h_3} & -\frac{1}{h_3} \end{bmatrix} \begin{bmatrix} \Delta w_1 \\ \Delta w_2 \\ \Delta w_3 \end{bmatrix} \tag{4}$$

Assembling eqns (4) for all elements of the slab, the finite element kinematic relations, expressing the *compatibility conditions*, can be written in the following form, where E is the kinematic transformation matrix:

$$\underset{\sim}{\Delta\theta} = \underset{\sim}{E}\,\underset{\sim}{\Delta w}\,. \tag{5}$$

3.3 Static relations

Each element has two types of loads: the body forces and the generated stress-resultants along the element sides. The former are expressed by statically equivalent *vertical nodal forces* applied to the element nodes, f_j $(j=1,2,3)$. The latter correspond to the *total bending moments*, m_i $(i=1,2,3)$, equivalent to the bending moment distribution along each side, and to the *vertical nodal forces*, q_j $(j=1,2,3)$, equivalent to the shear force and twisting moment distributions along each side, and applied to the element vertices. For each element, static relations expressing *equilibrium conditions* are given by:

$$\begin{bmatrix} f_1+q_1 \\ f_2+q_2 \\ f_3+q_3 \end{bmatrix} = \begin{bmatrix} -\dfrac{1}{h_1} & \dfrac{a_2}{l_2h_2} & \dfrac{b_3}{l_3h_3} \\ \dfrac{b_1}{l_1h_1} & -\dfrac{1}{h_2} & \dfrac{a_3}{l_3h_3} \\ \dfrac{a_1}{l_1h_1} & \dfrac{b_2}{l_2h_2} & -\dfrac{1}{h_3} \end{bmatrix} \begin{bmatrix} m_1 \\ m_2 \\ m_3 \end{bmatrix} \tag{6}$$

The static relations for the entire system of finite elements are the assemblage of eqns (6) and can be written in the form

$$\underset{\sim}{f} = \underset{\sim}{E}^T \underset{\sim}{m} \tag{7}$$

where E^T is the static transformation matrix and forces q_j are not present because their sum at each node is equal to zero.

The static transformation matrix being the transpose of the kinematic transformation matrix illustrates the *static-kinematic-duality*.

3.4 Constitutive relations

Constitutive relations modeling the structural material express relations between static and kinematic variables. In the present model, they consist of *yield conditions*, *flow rule* and *complementary conditions*, these last ones ensuring that mechanism deformations are only possible at the plasticized element sides (Da Fonseca[4]).

Yield conditions are based on the *Johansen's yield criterion* (Johansen[1]). They impose limiting values on the magnitude of the total bending moments on the element sides. For the i^{th} side:

$$-m_i^- \le m_i \le m_i^+\,. \tag{8}$$

The plastic normal bending moment of resistance per unit length of an element side, $\overline{m}_{pi}$, when the steel reinforcement is orthogonal to the side (see Figure 3), can be obtained by the following expression (Figueiredo[6]):

$$\overline{m}_{pi} = 0.90\,f_{yd}\,d\,A_{si}\,. \tag{9}$$

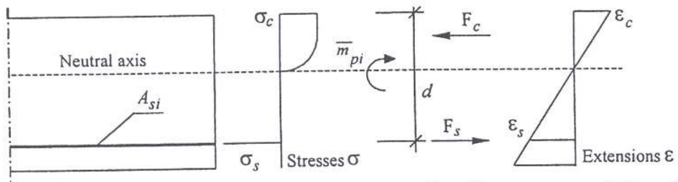


Figure 3. Plastic normal bending moment of resistance per unit length.

When the steel reinforcement is oriented in two orthogonal directions (see Figure 4), the plastic normal bending moment of resistance per unit length on the yield line t is given by eqn (10) (Figueiredo[6]).

$$\overline{m}_p = \overline{m}_{px} + \overline{m}_{py} = 0.90 f_{yd}\, d_x A_{sx} \cos^2\left(\frac{\pi}{2} - \alpha_t\right) + 0.90 f_{yd}\, d_y A_{sy} \cos^2(\alpha_t). \quad (10)$$

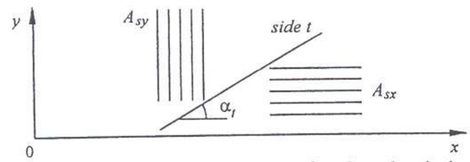


Figure 4. Contribution of the reinforcement for the plastic bending moments.

3.5 Mathematical programs

The plastic limit analysis governing relations are recognized (Da Fonseca[4]) as a set of Karush-Kuhn-Tucker conditions that are equivalent to a pair of *dual mathematical programs,* the primal program (unsafe) and the dual program (safe), both leading to a unique solution.

It must be noticed that the unsafe nature of the solution is related to the finite element model and not to the selected mathematical program. In fact, the mathematical model satisfies kinematic conditions but may not fulfil yield conditions at the interior of the elements. Since is not guaranteed to be *statically admissible*, the solution is *unsafe*. If the dual mathematical program is used, the yield line analysis problem takes the form

Maximize:	Load parameter	(11)
Subject to:	Equilibrium conditions	(12)
	Yield conditions	(13)
	Supplementary code conditions	(14)

Besides the static admissibility constraints (eqns 12-13), *supplementary code conditions* (eqn 14) may be added in order to validate the use of the yield line method, as stated in structural codes. For example, some structural codes limit the ratio of the support moments to the mid-span moments and/or the steel ratios.

3.6 Variable geometry of the finite element mesh

Even in a simple case, in which the designer knows the topology of the collapse mechanism, uncertainties may arise in the geometry of the finite element mesh. For example, in a rectangular simply supported slab with an uniformly distributed load, the selection of the mesh shown in Figure 5a) is straightforward. However, the location of the nodes *A* and *B* that correspond to the most critical mechanism (see Figure 5b)) is not known *a priori*.

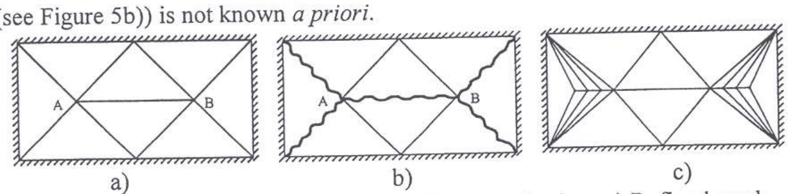


Figure 5. a) Finite element mesh. b) Collapse mechanism. c) Refined mesh.

Obviously, a refined mesh might be used (Figure 5c)), but a more powerful technique is to include these coordinates as variables of the optimization process. However, if this possibility and the selection of the collapse mechanism are simultaneously present in the mathematical program (eqns 11-14), during the optimization of the load parameter the yield lines will be moved away from the locations where the bending moments reach higher values. This problem may be overcome with the decomposition of the optimization process in two phases. The first phase corresponds to the solution of a maximization problem (eqns 11-14), where the nodal coordinates are kept unchanged. The second phase is a minimization problem (eqns 15-19), where some node coordinates may vary. Some *mechanism conditions* must be added (Figueiredo[6]).

Minimize:	Load parameter	(15)
Subject to:	Equilibrium conditions	(16)
	Yield conditions	(17)
	Supplementary code conditions	(18)
	Mechanism conditions	(19)

4 Mathematical model for plastic limit synthesis

The solution of a plastic limit synthesis problem also implies the establishment of kinematic, static and constitutive relations. In the analysis problem, the slab thickness and steel reinforcement are fixed and the design variable is the value of the load parameter corresponding to the maximum loads that can be applied to the slab. In a synthesis problem, the loads are known and the slab thickness and steel reinforcement are the design variables. If the optimization criterion is expressed in terms of a function Z, the synthesis problem may then be formulated as follows (Da Fonseca[4]):

Optimize:	Z	(20)
Subject to:	Equilibrium conditions	(21)
	Yield conditions	(22)

When the objective is to find the lowest cost solution for the slab, the function Z must express the cost associated to each solution, in terms of volumes and unit costs per unit volume for both concrete and steel.

4.1 Mathematical program

The solution of such a mathematical program (eqns 20-22) verifies the ultimate limit states only. However, since the slab thickness and the steel reinforcement are variables of the problem, it is possible to establish additional conditions in order to satisfy the serviceability limit states. Limits on the slab thickness and on the reinforcement area are examples of *serviceability conditions*. It is also useful to establish *technological conditions*. For instance, a solution with distinct thickness for all elements should be avoided. *Supplementary code conditions* may also be included.

Plastic limit synthesis problems may then be expressed in the following way:

Minimize:	Cost	(23)
Subject to:	Equilibrium conditions	(24)
	Yield conditions	(25)
	Serviceability conditions	(26)
	Technological conditions	(27)
	Supplementary code conditions	(28)

4.2 Variable geometry of the finite element mesh

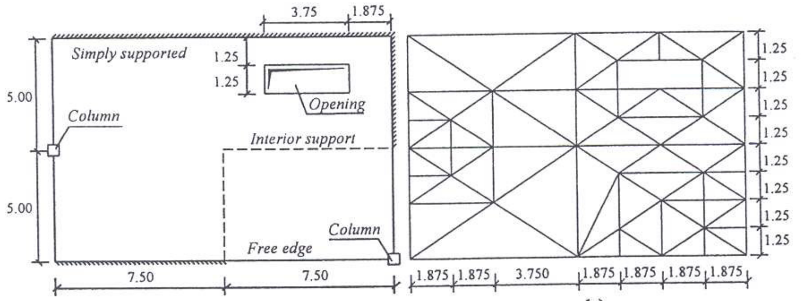
As described in Section 3.6, when the nodal coordinates are not fixed, the optimization algorithm tries to move the yield lines away from the locations where the bending moments reach higher values. For this reason, in plastic limit synthesis the optimization is also decomposed in two phases. The first phase corresponds to the resolution of the minimization mathematical program (eqns 23-28), with fixed nodal coordinates and loads (unit load parameter). The solution defines the concrete thickness, the areas of steel reinforcement and the collapse mechanism configuration. The second phase consists in the resolution of the minimization mathematical program described by eqns (15-19). Slab thickness and steel reinforcement obtained in the first phase are fixed. Nodal coordinates and load parameter are now variables of the problem. A new load parameter, lower or equal to one, and a new geometry for the mesh are obtained.

It is then necessary to repeat the first phase, considering the new geometry of the mesh and a unit load parameter. The process becomes *iterative*, being terminated when the variation of the load parameter becomes negligible.

5 Examples

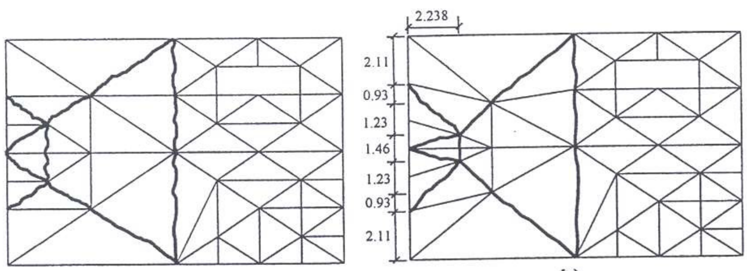
5.1 Plastic limit analysis problem

The following example has been studied by several authors and has become a test to models and computer codes. The slab, represented in Figure 6a), is isotropic and uniformly loaded. Using a mesh of 60 finite elements (see Figure 6b)), Da Fonseca[4] got to a ratio of the load parameter, λ_c, to the bending moment of resistance per unit length, $\overline{m_p}$, equal to 0.396, while Lúcio[7] obtained 0.4046.



a) b)
Figure 6. a) Geometry and support conditions. b) Mesh geometry.

When the "first phase" mathematical program for the plastic limit analysis (eqns 11-14) is used to formulate this problem, the solution $\lambda_c / \overline{m_p} = 0.4046$ is obtained. The corresponding collapse mechanism is shown in Figure 7a).



a) b)
Figure 7. a) Collapse mechanism b) Optimized mesh geometry.

The solution of the "second phase" mathematical program (eqns 15-19) describes the new optimized geometry for the finite element mesh, (see Figure 7b)). The ratio of the load parameter to the bending moment of resistance per unit length is then 0.3930.

5.2 Plastic limit synthesis problem

Figure 8a) shows a rectangular slab, isotropic (equal reinforcement in both orthogonal directions and in both the upper and lower layers), with all edges built in. Under an uniform load, the collapse mechanism topology is known (Figure 8a)), but coordinate x may be "optimized". Since two axes of symmetry exist, the study is performed for one-fourth part of the slab (see Figure 8b)).

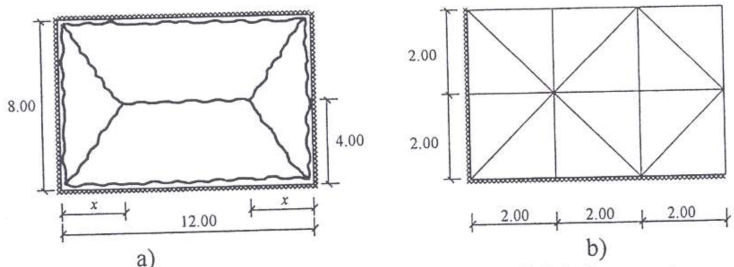


Figure 8. a) Geometry and collapse mechanism. b) Mesh geometry.

In the optimal solution of the "first phase" mathematical program (eqns 23-28) the slab thickness is 0.2020m and the steel reinforcement is 7.8237cm^2/m, corresponding to a cost of 138132 "cost units" (c.u.). In the data definition, a concrete density of 25kN/m^3 and a live load of 10kN/m^2 were considered. Dead and live loads were multiplied by a safety factor of 1.5. Concrete strength (f_{cd}) is equal to 13300kPa and its cost is 13000 c.u./m^3. Steel reinforcement has a tensile strength (f_{yd}) of 348000kPa and costs 1000000 c.u./m^3. In order to verify the serviceability limit states, a minimum value of 18 cm for the slab thickness, a minimum steel reinforcement ratio of 0.15 and a reinforcement cover leading to d=0.03m (see eqn (10)) were admitted. The "second phase" mathematical program (eqns 15-19) gives a load parameter of 0.98476, corresponding to the optimized mesh geometry represented in Figure 9.

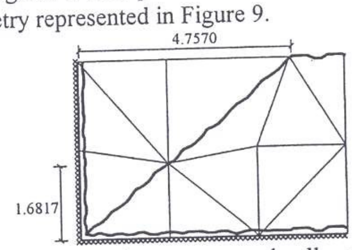


Figure 9. Optimized mesh geometry and collapse mechanism.

Since the new load factor is less then one, the first solution is unsafe and a second iteration must be solved. When the "first phase" mathematical program eqns (23-28) is solved again, considering fixed the new geometry (see Figure 9), the solution corresponds to a slab thickness of 0.2028m, to a steel reinforcement of 7.8751cm^2/m and to a cost of 138875 c.u.. The second mathematical program (eqns 15-19) gives now a load parameter of 1.0, stating that design is achieved.

6 Conclusions

Traditionally, the implementation of the yield line method requires a tedious analysis of potential collapse mechanisms. Formulations involving the yield line method and a model of finite elements produce good results, but involve the use of relatively dense meshes. The model presented in this paper, permitting the automatic selection of the collapse mechanism together with the possibility of changing nodal coordinates, allows for the determination of safer solutions, even if coarse finite element meshes are used. Examples show the validity of the model.

References

1. Johansen, K.W., *Brudlinieteorier* (in Danish), Thesis, Jul Gjellerups Forlag, Copenhagen, 1943. English Translation: *Yield-Line Theory*, Cement and Concrete Association, London, 1962.

2. Johansen, K.W., *Pladeformler* (in Danish), Polyteknisk Forlag, Copenhagen, 1946. English Translation: *Yield Line Formulae for Slabs*, Cement and Concrete Association, London, 1972.

3. Munro, J. & Da Fonseca, A.M.A., *Yield Line Method by Finite Elements and Linear Programming*, Structural Engineer, 56 B, pp. 37-44, 1978.

4. Da Fonseca, A.M.A., *Plastic Analysis and Synthesis of Plates and Shells by Mathematical Programming*, Ph.D. Thesis, University of London, 1980.

5. Azevedo, A.F.M., *Second-order structural optimization,* Proceedings of OPTI95 – Computer Aided Optimum Design of Structures, Computational Mechanics Publications, 1995.

6. Figueiredo, A.J.M., *Análise e Optimização de Lajes de Betão Armado* (in Portuguese), Ms. Thesis, Faculdade de Engenharia da Universidade do Porto, Porto, 1996.

7. Lúcio, V.J.G., *Lajes Fungiformes - Análise Plástica Limite* (in Portuguese), Ms. Thesis, Instituto Superior Técnico, Lisboa, 1985.