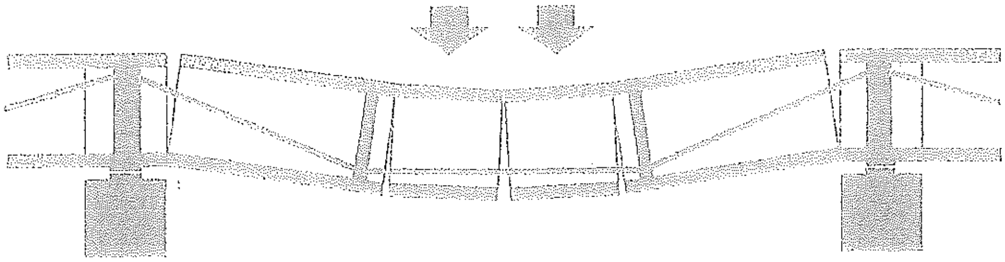
AFPC

Association Française
Pour la Construction

EXTERNAL PRESTRESSING IN STRUCTURES



NON-LINEAR CALCULATION TESTS OF PRESTRESSED BEAMS

Under the direction of
Eric CONTI & Rémi TARDY

ISSN 0761-3458

ASSOCIATION FRANÇAISE
POUR LA CONSTRUCTION
46, avenue Aristide Briand
92223 BAGNEUX
FRANCE

AFPC

EXTERNAL PRESTRESSING IN STRUCTURES

NON-LINEAR CALCULATION TESTS OF PRESTRESSED BEAMS

Calculation tests
performed for the Workshop on
Behaviour of External Prestressing in Structures
Saint-Rémy-lès-Chevreuse , France
June 9-12th , 1993

Edited by :
Eric CONTI & Rémi TARDY

Organized by :

AFPC — Association Francaise Pour la Construction

Co-sponsored by :

ACI	American Concrete Institute
ASBI	American Segmental Bridge Institute
CEB	Comité Euro-International du Béton
FIP	Fédération Internationale de la Précontrainte
PTI	Post-Tensioning Institute

with the support of :

CEBTP — Centre Expérimental de Recherches et d'Etudes du Bâtiment et des Travaux Publics

LCPC — Laboratoire Central des Ponts et Chaussées

SETRA — Service d'Etudes Techniques des Routes et Autoroutes

CONTENTS

FOREWORDS
Michel VIRLOGEUX 7

PRESENTATION OF THE CALCULATION TEST AND COMPARISON OF RESULTS
Eric CONTI, Rémi TARDY 9

COMPARATIVE CURVES 17

DATA SHEETS 27

PRESENTATION OF CALCULATIONS BY THEIR AUTHORS

FEMAS - A Structural nonlinear analysis program.
G. König, L. Qian, Germany 41

A new modelling of prestressing in CESAR-LCPC.
F.J. Ulm, P. Humbert, V. Magnat, France 51

Non-linear analysis of prestressed concrete beams.
J. Schlaich, T. Kuchler, Germany 63

SAFIR applied to prestressed beams at ambient temperature.
J.M. Franssen, Belgique 67

A computer program for the nonlinear analysis of prestressed concrete continuous beams.
V.K. R Kodur, T.I. Campbell, Canada 75

RAHM - A non-linear analysis program.
C. Stadler, M. Wicke, Austria 87

Computation of external prestressed structures.
J. Eibl, I. Retzepis, J. Huang, Germany 93

CONCRETEMAC - Program for the nonlinear finite element analysis of concrete structures.
N. Ojdrovic 105

Nonlinear analysis of prestressed concrete beams with unbonded tendons.
G. Iványi, W. Buschmeyer, Y. Hu, Germany 115

Prestressed concrete beams serviceability and ultimate limit states.
J.L. Vitek, P. Vitek, Czech Republic 123

Non-linear analysis of prestressed concrete beams.
A.E. Naaman, S.M. Jeong, USA 133

Non-linear analysis of externally prestressed concrete beams with SNOBdev
J.A. Garcia-Vargas, N. Menezes, B. Fouré, France 135

Finite element program for analysis of prestressed concrete structures.
M.H. Mohd-Yassin, F.C. Filippou, USA 149

Nonlinear analysis of the externally prestressed concrete beam.
M. Morimoto, M. Komuro, Japan 167

SAFIR APPLIED TO PRESTRESSED BEAMS AT AMBIENT TEMPERATURE

Jean-Marc FRANSSEN

University of Liège, Belgique

SUMMARY

This paper describes some general features of the code SAFIR and gives the main hypotheses of its plane beam and truss finite elements. Some particular points of the discretization of the prestressed beams defined by AFPC are discussed. The description of the results that can be obtained is given.

RESUME

Cet article décrit certains aspects généraux du programme SAFIR et reprend les principales hypothèses de ses éléments finis de poutre plane et de barre de treillis. Certains points particuliers de la discrétisation des poutres précontraintes proposées par l'AFPC sont discutés. On donne une description des résultats qui peuvent être obtenus.

1. INTRODUCTION

Since many years, had the author been involved in the numerical simulation of structures submitted to fire. With the financial support of Arbed and of the ECCS, the program CEFICOSS had been developed to analyse steel and composite steel-concrete plane frames. The evolution of the 2D temperature distribution in the sections was calculated by finite differences and the structural behaviour via 2 nodes plane beam finite elements.
As this code had been written specifically for the case of plane frames, it was decided that a new program should be written in order to offer more general possibilities. This code received the name of SAFIR.
When Professor Dotreppe was aware of the bench mark test proposed by AFPC, it suggested the author to participate, this test being a good validation exercise, although it concerns a structure at ambient temperature. The results of the bench mark test have been reported in [1].
This paper describes the code SAFIR, in the features that have been used for this test, and explains how the prestressed beams have been modelled.

2. THE CODE SAFIR

2.1. Plane beam.

The 2D beam is a displacement based element. It has two nodes at the ends, each one supporting two translations and one rotation, plus one intermediate node supporting the non linear part of the longitudinal displacement. This longitudinal displacement of the node line is a second order power function of the longitudinal co-ordinate, whereas the transverse displacement of the node line is a third order power function of the longitudinal co-ordinate.
The hypothesis of Bernoulli is used when evaluating the displacement field within the element and the hypothesis of Von Karman when evaluating the axial strain. In order to eliminate locking, the non linear part of the strain is averaged on the length of the element.

$$\varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2L} \int_{-\frac{L}{2}}^{+\frac{L}{2}} \left(\frac{\partial v}{\partial x}\right)^2 dx \qquad (1)$$

where u : longitudinal displacement
v : transverse displacement
x : longitudinal co-ordinate
L : length of the element.

The system of equilibrium equations linking the incremental nodal displacements to the incremental nodal forces as well as the energetically equivalent nodal forces are established via the principle of virtual displacements, written in a total corrotationnal description.
The volumetric integration that must be performed when evaluating the stiffness matrix and the internal nodal forces is split into a longitudinal and a sectional integration.

- The cross section is divided into several fibres. At ambient temperature, the strain in a section is constant at all the points that have the same transverse co-ordinate. In this case, the fibres are horizontal rectangles, each of them having the width of the section at this level. The fibre model is here used as a layer model. The strain, tangent modulus and stress in the layer is evaluated at mid level of the layer. The integration is made by the method of the rectangles, i.e. that each layer is geometrically characterised by its area and the position of its center.

The sectional stiffness are evaluated by ;

$$EA_i = \sum_{j=1}^{N} E_{ij} A_j$$

$$ES_i = \sum_{j=1}^{N} E_{ij} A_j y_j$$

$$EI_i = \sum_{j=1}^{N} E_{ij} A_j y_j^2 \qquad (2)$$

The internal forces are evaluated by

$$N_i = \sum_{j=1}^{N} \sigma_{ij} A_j$$

$$M_i = \sum_{j=1}^{N} \sigma_{ij} A_j y_j \qquad (3)$$

where	EA_i, ES_i, EI_i	:	sectional stiffness
	N_i, M_i	:	axial and bending internal forces
	E_{ij}	:	tangent modulus of a layer
	A_j	:	area of a layer
	y_j	:	transverse co-ordinate of a layer
	i	:	refers to the section
	j	:	refers to the layer
	N	:	number of layers.

It has to be mentioned that this method underestimates the flexural stiffness in a way that is significant when the number of layer is too low.
For a rectangular section, for example,

$N = 1$	provides	$EI_i = 0$!!
$N = 2$	"	$EI_i = 0{,}75 \times EI_i$ exact
$N = 10$	"	$EI_i = 0{,}99 \times EI_i$ exact

- The longitudinal integration is made by the traditional method of Gauss with the number of points of integration ranging from 2 to 6.

2.2. Plane truss.

The 2D truss element is also based on displacements. Its formulation is so well known that it will not be repeated here. If large displacements are considered, the non linear strain is calculated according to :

$$\varepsilon_x = \frac{1}{2} \frac{L^2 - L_i^2}{L_i^2} \qquad (4)$$

where L : length of the deformed element
L_i : initial length of the element

i.e. Van Karman hypothesis is not present.
No provision is made to distinguish between tension and compression, i.e. the element has no notion such as buckling and must be used in compression only for stocky sections.

2.3. Material characteristics.

The material characteristics have been used as described in the calculation forms sent to the participants i.e. ;
Concrete : parabolic-plateau. No tension strength.
Passive rebars : elastic-plastic.
Prestressing steel : Ramberg-Osgood.
All the laws account for plasticity, i.e. unloading is parallel to the original modulus when the strain decreases.

2.4. General features.

A general Newton-Raphson method is used in the structural calculation, i.e. the stiffness matrix is recalculated after each iteration.
Dynamic effects are not considered.
Post peak behaviour of the structure can be tracked by means of displacement control.

In addition to the elements presented here, SAFIR also has a 3D beam element for structural analyses, and a plane 3 or 4 nodes element for transient conductive thermal analyses or to determine torsionnal stiffness and warping function in composite steel-concrete sections.

3. DISCRETIZATION OF THE CALCULATION TESTS.

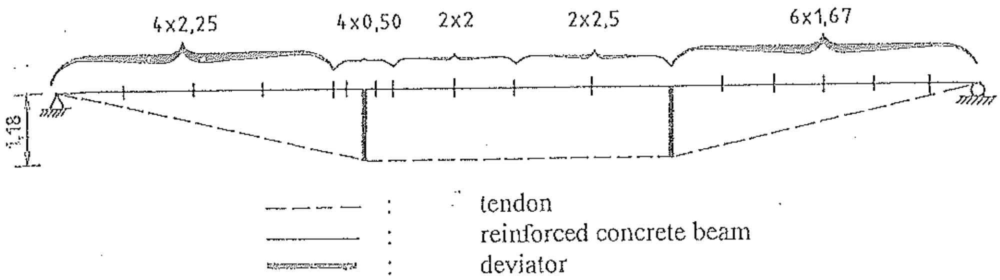


Figure 1 : Isostatic beam, external prestressing

Each beam has been discretized by means of (see Fig. 1),
- beam elements accounting for concrete and passive rebars,
- truss elements accounting for prestressing strands,
- very stiff beam elements rigidly fixed to the reinforced beam in order to introduce the eccentricity of the tendons. Those elements could be easily suppressed if the geometrical eccentricity of the tendon was directly introduced in the truss element, which would be the normal way to formulate a prestressing finite element.

The externally prestressed isostatic beam is represented by means of
- 18 reinforced concrete beam elements,
- 3 tendon elements
- 2 stiff deviator elements.

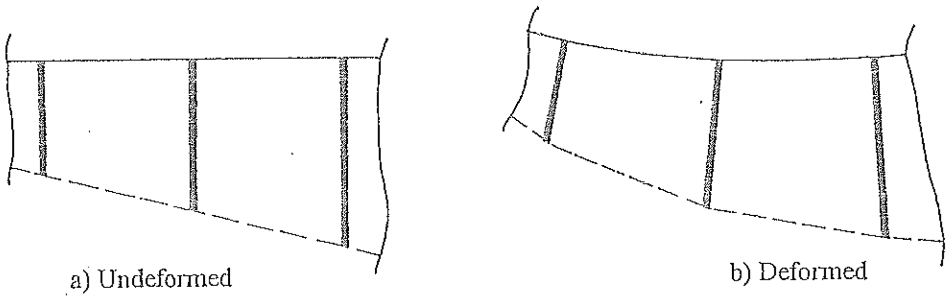


Figure 2 : Internal prestressing.

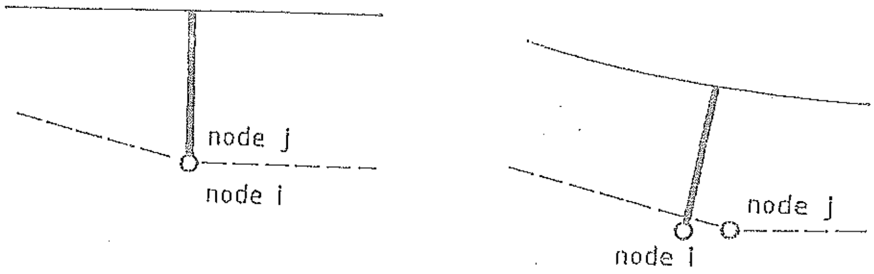
The internally prestressed isostatic beam is represented by means of
- 18 reinforced concrete beam elements,
- 18 tendon elements,
- 17 deviator elements (see Fig. 2a).

The tendon elements remain straight between 2 fictive deviators in the deformed state (see Fig. 2b) and their length is therefore slightly shorter than the length that could be calculated in a normally written prestressing element accounting for the curvature of the beam. This effect has been shown to be of very limited influence within the frame of a Ph. D. thesis now under completion in Liège.

Two longitudinal points of integration in every beam have been used.

The reinforced cross section of the isostatic beam is represented by
- 20 layers of 1 cm. in the deck,
- 14 layers of 10 cm.,
  18 layers of 2 cm.,
  4 layers of 1 cm. in the webs,
- 1 layer of 40 cm² as passive steel.

The node line has been placed at 62 cm. from the top surface in order to have no eccentricity of the tendon at the supports.



a) Undeformed   b) Deformed

Figure 3 : Free split at deviators.

The case of free split at the deviators of externally prestressed beams has been represented by means of two different nodes having the same co-ordinate at the lower end of the deviators (Fig. 3a).
One node is fixed to the deviator,
one note is fixed to the tendons.
The horizontal displacements of the two nodes are independent whereas the vertical displacements are forced to be equal via a master-slave relation (Fig. 3b).

The prestressing is introduced by means of initial strains in the tendons. The strain ε_i introduced will produce a load $P_i = E_p \cdot \varepsilon_i \cdot A_p$. This load corresponds to the load in the tendons before elastic shortening of the reinforced concrete beam. As the prestressing load of 6000 kN is described as the tension in tendon after losses, a first step of the simulation is to find the initial strain that produces this tension of 6000 kN;

A strain calculated as $\varepsilon_i = \dfrac{6000\ 10^3}{5000 \quad 190000} = 6,32\ 10^{-3}$ was first applied and a first simulation gave a first value of the prestressing load after elastic losses. The initial strain was then corrected to 6,49 10^{-3} and a second simulation yielded a load in the tendon equal to 6000 kN in the inclined parts of the tendon and 5930 kN in the central part, after the self-weight has been applied and the elastic losses have been considered.

The section of the hyperstatic beam is represented by means of ;
42 reinforced concrete elements
7 tendon elements
6 stiff deviator elements.

4. RESULTS PRESENTED.

Results can be provided after every time step or, if required, after each iteration. This possibility produces enormous amounts of results and is generally used for research purposes in order to establish a convergence criterium for example or to understand why convergence has not been detected in cases showing numerical problems.
After each time step of this structural simulation, the following results have been printed ;
- The duration of time since the beginning of the loading.
- The number of iterations performed within this time step.
- The minimum Eigen value of the stiffness matrix at time t divided by the minimum Eigen value of the stiffness matrix of the unloaded undeformed structure. This ratio has the value 1 at t=0 and decreases to 0 at failure time. Each significant decrease of the ratio indicates a loss of stiffness. This ratio is linked to the first vibration frequency of the structure. It can be used to automatically adapt the time step and also to detect numerical failure. If failure is detected when the ratio had no yet reached a very low value, there is a good probability that some problem caused the numerical simulation to stop whereas the structure had still a significant degree of stiffness and presumably of resistance.
- The displacements at the nodes.
- The axial and bending force at the longitudinal points of integration in the beam elements;
- The axial load in the tendon elements.
- The stresses in the layers of one beam element (close to the load).
- The reactions.

At the time being (sept.93), post processing capacity is limited to a sort of the output files in order to produce EXCEL files. It is therefore possible to draw the evolution of one or several results as a function of time. The possibility is therefore present to detect yielding or cracking in different parts of the structure.

Drawings of the structure on which the deformed body or the diagrams of solicitations is superimposed has not yet been programmed.

REFERENCE.

[1] Calculation tests.,
Workshop on Behaviour of external prestressing in structures, Saint-Remy-les-Chevreuse, France, June 9-12, 1993. Association Française Pour la Construction.