Transactions on the Built Environment vol 29, © 1997 WIT Press, www.witpress.com, ISSN 1743-3509

CALEB: A COMPUTER PROGRAM FOR GEOMETRIC AND MATERIAL NONLINEAR ANALYSIS OF OFFSHORE PLATFORMS AND GENERAL FRAMED STRUCTURES

Adilson C. Benjamin
PETROBRAS/CENPES/DIPREX

Breno P. Jacob and Nelson F. F. Ebecken
COPPE/UFRJ - Civil Engineering Department

Abstract - CALEB is a nonlinear finite element program for geometric and material nonlinear analysis of offshore platforms and general framed structures. The finite element formulations implemented in CALEB assumes large displacements and small strain. Geometric nonlinearity is taken into account by means of an updated Lagrangian formulation. Material nonlinearity is taken into account by means of an elastic perfectly plastic model derived from plastic hinge theory. Capabilities of CALEB version 1.4, the current release of CALEB, include: pre-processor for mesh refinement, problem-oriented language input, distributed member loads, automatic generation of environmental loads, joint failure check, save/restart procedure, eigenvalue buckling analysis, nonlinear stress analysis, and graphical post-processing. In this paper a brief description of CALEB version 1.4 and of its main features is presented.

INTRODUCTION

CALEB is a nonlinear finite element program for geometric and material nonlinear analysis of offshore platforms and general framed structures. Presently, the main application of the CALEB program is the collapse analysis of fixed offshore platforms. The need of performing a collapse analyses of such structures may arise both in the design and in the operation phases, to determine the behaviour of the damaged or undamaged structure, under operational, extreme or accidental loads. Collapse analyses are also performed to generate data for the study of the reliability of the platform.

The CALEB program was originally developed to fulfill academic requirements only[1,2,3], and, therefore, did not offer resources for automatic data generation and visualization of results. It was not suitable for the collapse analysis of large-scale structures such as an offshore platform, since it also lacked other important features, such as an automatic procedure for the refinement of the element mesh, and for the introduction of geometric imperfections. In order to remove such drawbacks and make the program more user-friendly, Petrobras[4,5,6] with the cooperation of COPPE/UFRJ, started in 1991 software development activities to enhance the CALEB program and to develop a suite of pre- and post-processor programs.

The current version of the CALEB program (version 1.4) has the following features[7] :

- Problem-oriented language input
- Distributed member loads
- Automatic generation of environmental loads
- Joint failure check according to API RP2A
- Save/restart procedure
- Scalar element
- Nonlinear beam elements
- Nonlinear stress analysis
- Eigenvalue buckling analysis
- Incremental displacement control algorithm for nonlinear stress analysis

CALEB's input data file is a free format file in which data is specified through a problem-oriented language. This problem-oriented language is built around groups of *commands* stored in 80-bytes physical records. Each command is composed by real numbers, integer numbers, keywords and data label. Details of this problem-oriented language were presented elsewhere[8].

PRE- AND POST-PROCESSORS

The pre- and post-processor programs that comprise the CALEB suite are: RAMER, PRESOL, POSPAN, VESPAE and POSPER.

RAMER is a pre-processor program that, beginning from a basic mesh of beam elements, automatically generates a refined mesh according to user-specified criteria. It features also a module for automatic nodal reordering, to optimize the bandwidth of the global stiffness matrix of the refined structure.

PRESOL is another pre-processor program that generates P-Y, T-Z and Q-V curves to represent the soil behaviour in soil-structure interaction problems, according to the API code. The resulting curves can then be used as nonlinear force-displacement curves for the scalar element of the CALEB element library.

POSPAN is a numerical post-processor that reads a bynary data file generated in an analysis performed by the CALEB, containing all input data and results from selected steps of the nonlinear analysis. The POSPAN program manipulates the data read from this file, and, according to free-field, problem-oriented commands supplied by the user, generates some of all of the following files:

- a listing file for print-out of selected results;
- a new data file that can be input to the CALEB program, allowing a restart of the analysis;
- a file with the nodal coordinates of a selected deformed configuration (corresponding to a step of a nonlinear analysis, or to one buckling mode or a combination of selected modes). This file can then be inserted in the input file of the original CALEB analysis, to consider geometrical imperfections of the structure;
- Input data files for the graphical post-processors VESPAE and POSPER.

The POSPAN program is designed to allow a clear separation between distinct phases of a structural analysis - the numerical analysis itself and the interpretation of the results. It performs functions that are equivalent to those of a database manager program, since its commands provide the user a series of options and functions that can be activated at will, while the examination of the results proceed.

VESPAE is a graphical post-processor that represents the original configuration of the structure, selected deformed configurations or buckling modes, and stress maps. It presents an interactive user-interface, through option menus activated by mouse operations and keystrokes.

POSPER is another graphical post-processor, intended to trace response curves for selected degrees of freedom or force components, and stress diagrams. It also presents an interactive user interface.

The development of CALEB program and its pre and post-processors has been performed in IBM mainframe computers, with the MVS/ESA operational systems. The migration of this suite of programs to personal workstations is scheduled in the near future. The conversion of the numerical programs will be relatively simple, since they are coded in standard FORTRAN 77; the conversion of the graphical post-processors will, however, require additional efforts since the graphical libraries are not compatible.

FINITE ELEMENT LIBRARY

A scalar element and a beam element are available. Both elements have straight axis and two nodes. Six unknowns are associated to each one of these nodes, namely : 3 translations (u, v and w) and 3 rotations (ru, rv and rw).

Scalar Element

The element associates a nonlinear spring to each nodal d.o.f. (degree of freedom). The force-displacement relations that defines the nonlinear springs are given through pairs of points and admit intervals with null stiffness in the origin.

The force-displacement relations are defined at the local reference system of the element. This reference system can be the global system, or anyother one defined by the two nodes of the element and a third auxiliary node. The position of the reference system can be maintained fixed or can be updated according to the displacements of any two nodes of the structure.

Beam Element

The beam element is a displacement based element, in which the transverse bending displacements are interpolated by cubic functions and the longitudinal and torsional displacements are interpolated linearly.

This element, called SLCB (Standard Linear/Cubic Beam) in the present paper, is the standard beam element. It is a widely used element because of the simplicity of its formulation, ease of use and accuracy of results.

The element is available in three versions: the MBEAM element (material nonlinear only), the GBEAM element (geometric nonlinear only) and the GMBEAM element (material and geometric nonlinear). These nonlinear formulations preserve all the qualities of the linear formulation.

Element formulation assumes large displacements, small strain and elastic-plastic material. The geometric nonlinearity is taken into account by means of an updated Lagrangian formulation[12]. The yield criteria is a function of the nodal axial force and the nodal bending moments[2,13].

The element material stiffness matrix, derived using concepts from plastic hinge theory and matricial analysis of structures, relates in an elastic-plastic way the incremental nodal displacements and forces[2,13]. This matrix is explicitly integrated at the local moving coordinate system.

The element geometric stiffness matrix is derived using the principle of virtual work and a complete nonlinear strain tensor[1]. This matrix is a function not only of the element axial force but also of the nodal bending moments[1,14]. It is explicitly integrated at the local moving coordinate system. The position of this element local moving coordinate system depends on the rigid body motion of the element and is continuously updated.

NONLINEAR STRESS ANALYSIS

Nonlinear stress analyses can be performed using a standard load-control strategy, or a displacement-control strategy. This latter strategy is a modified version of the algorithm proposed by Batoz and Dhatt[9], according to Justino[10]. This algorithm allows, in an analysis with several load patterns, the selection of a load pattern that will be incremented while the others remain constant.

At the end of each step of the nonlinear analysis, the punching-shear collapse of the tubular joints is verified using the API-RP2A code.

Equilibrium Equations

The nonlinear problem is solved by means of an incremental-iterative algorithm. The equilibrium equations used in this algorithm have the following form :

$$ {}^{t+\Delta t}\mathbf{K}^{(k-1)}\,\Delta\mathbf{u}^{(k)} = \Delta\mathbf{R} = {}^{t+\Delta t}\mathbf{F} - {}^{t+\Delta t}\mathbf{R}^{(k-1)} \tag{1} $$

$$ {}^{t+\Delta t}\mathbf{u}^{(k)} = {}^{t+\Delta t}\mathbf{u}^{(k-1)} + \Delta\mathbf{u}^{(k)} \tag{2} $$

$$ {}^{t+\Delta t}\mathbf{K}^{(k-1)} = \mathbf{K}_M + \mathbf{K}_G \tag{3} $$

where the left superscript indicates the current step of the incremental analysis, and the right superscript is the iteration count; and

${}^{t+\Delta t}\mathbf{K}^{(k-1)}$ - Tangent stiffness matrix;

$\mathbf{K}_M$ - Material stiffness matrix;

$\mathbf{K}_G$ - Geometric stiffness matrix;

$\Delta\mathbf{u}^{(k)}$ - Vector with incremental nodal displacements between iterations k and $k-1$;

${}^{t+\Delta t}\mathbf{u}^{(k)}$ - Vector with total nodal displacements at iteration k of step $t+\Delta t$;

$\Delta\mathbf{R}$ - Vector with out-of-balance nodal forces;

${}^{t+\Delta t}\mathbf{F}$ - Load vector, applied in all iterations of step $t+\Delta t$;

${}^{t+\Delta t}\mathbf{R}^{(k-1)}$ - Vector of updated internal forces.

The discretization of the equilibrium equations is performed in the usual way via the Finite Element Method[12]. Equations (1) and (2) hold for the whole structure, the global stiffness matrix and force vector being assembled from the contribution of the corresponding quantities from the individual elements. The element stiffness matrix is the result of the summation of the material and the geometric stiffness matrices as indicated by Equation (3).

Iteration Procedures

Three options of iteration procedures are available, namely:

- the Full Newton-Raphson Method (FNRM),
- the Modified Newton-Raphson Method (MNRM) and
- the Full Initial Stress Method (FISM).

The difference between these procedures resides in the strategy employed for the reavaliation of the stiffness matrix of the structure. In the FNRM the stiffness matrix is updated in each iteration; in the MNRM the stiffness matrix is updated only in the first iteration of each load step, and in the FISM all iterations are performed with the elastic stiffness matrix of the structure.

The performance of each of these methods is dependent upon the type of problem being solved. In general, the FNRM and the MNRM are recomended for geometric and geometric and material nonlinear analyses, and the FISM is recomended for material nonlinear analyses.

Convergence Criteria

The convergence of iteration cycles can be checked through three criteria: a residual force criteria, a displacement criteria and an internal energy criteria. These criteria can be used either isolately or simultaneously.

EIGENVALUE BUCKLING ANALYSIS

The "linearized buckling analysis"[12,15] is a procedure that allows the estimation of buckling loads, and the corresponding buckling modes. The results will be accurate whenever it can be assumed that the behaviour of the structure prior to buckling is strictly linear. Of course this assumption does not hold for the collapse of offshore structures, but nevertheless this procedure will be useful for the introduction of geometric imperfections on the finite element mesh, as will be shown later.

The linearized buckling procedure consists in assembling and solving an eigenvalue problem given by

$$(\mathbf{K}_E + \lambda\, \mathbf{K}_G)\, \Phi = 0 \qquad (4)$$

where $\mathbf{K}_E$ is the elastic linear stiffness matrix, and $\mathbf{K}_G$ is the geometric stiffness matrix.

This eigenvalue problem is solved using the Subspace Iteration Method[12,16] for the first m eigenpairs; as a result we obtain the eigenvalues λ_i, $i=1,m$ that represent the buckling loads, and the corresponding eigenvalues Φ_i that represents the buckling modes.

Transactions on the Built Environment vol 29, © 1997 WIT Press, www.witpress.com, ISSN 1743-3509

USE OF THE STANDARD BEAM ELEMENT IN COLLAPSE ANALYSES OF OFFSHORE PLATFORMS

Mesh refinement with SLCB element

When performing a nonlinear collapse analysis of a fixed offshore platform employing the standard beam element (SLCB element), each compressed member should be discretized with several elements to allow the deformed configuration of the member to be adequately represented.

Moreover, geometric imperfections should be incorporated in the model, in the form of an initial curvature of the members. This avoids that a compressed member behave like an ideal, perfect column, with its equilibrium path remaining in the unstable branch, absorbing axial loads even after the buckling load having been reached. The introduction of initial curvatures not only avoids the numerical problems associated with such unstable behaviour, but also lends a more realistic modelling since every member of the actual structure is prone to have such geometric imperfections and therefore to be subjected to bending stresses from the beginning of the loading.

This need to generate a refined mesh of SLCB elements incorporating geometric imperfections may represent a major drawback in the modelling of a fixed offshore platform In such a three-dimensional framed structure, the task of introducing initial curvatures in a member represented by several elements is far from trivial, since an adequate plane should be chosen, as well as the compatibility with the boundary conditions at the member ends, that are elastically clamped.

The more apparent and “modern” solution of employing an interactive graphical pre-processor may turn out to be unfeasible, considering the drawbacks mentioned above and the large amount of members of a typical platform. The graphical processing may become very slow and the work becomes rather tedious and prone to errors.

A more efficient solution for this problem is provided by the numerical tools available in the CALEB system[11]. This solution consists in performing the generation of the refined/imperfect mesh in three steps, beginning from a “basic” mesh where each member is discretized by only one beam element. Such basic mesh is usually available, since it comprises the mesh that had been employed previously in preliminar linear analyses of the platform.

The first step of the generation of the refined/imperfect mesh employs the RAMER pre-processor program to perform a selective, non-uniform refinement of the mesh, according to some relevant criteria such as the slenderness ratio of the members. The result is a refined mesh with no imperfections.

Transactions on the Built Environment vol 29, © 1997 WIT Press, www.witpress.com, ISSN 1743-3509

In the second step a linearized buckling analysis is performed using this refined/perfect mesh and the CALEB program The results are the linear estimates of the buckling loads, and the corresponding buckling modes. Of course those linear estimates are not adequate to predict the actual collapse load, since it is known that the pre-collapse behaviour is far from linear, but the buckling modes will be useful in the next step for the generation of the refined/imperfect mesh.

The third step, finally, employs the POSPAN program to post-process the results of the linearized buckling analysis. A displacement field is generated by taking a combination of the first buckling modes (local buckling modes). The nodal coordinates of the refined/imperfect mesh is then automatically generated by adding that displacement field to the nodal coordinates of the refined mesh that resulted from the first step of this process.

These steps comprise not only a computationally more efficient solution, but also avoids errors that would be introduced by direct user-intervention in a standard graphical mesh generator. It is also consistent from a theoretical point of view, and, finally, results in an important by-product for the user: a deeper insight of the nonlinear behaviour of the platform.

Other Elements

There are other approaches in the literature for the execution of collapse analyses, with other types of elements (e.g. the Marshall Strut[17], and the "Idealized Tubular Structural Unit" ITSU[18]), that employ only one element to represent the geometric and material nonlinear behaviour of a compressed member. However only the ITSU element modelling is known to reach the same level of accuracy attained by the SLCB element modelling, with several elements per compressed members.

The advantage of the ITSU element in relation to the SLCB element resides in the use of one element per member, that reduces the size of the mesh and the time necessary to the generation of the numerical model. If the two elements had the same computational cost per element, the analyses performed with ITSU elements would be extremely cost effective when compared with the ones performed with SLCB elements. Nevertheless, the formulation of the ITSU element is considerably more complex and consequently the computational cost of generation of the element matrices is higher in the ITSU element than in SLCB elements in which the element matrices are explicitly integrated. The smaller computational cost per element reduces the computational cost of collapse analyses performed with refined meshes of SLCB elements and make them competitive.

Also, the use of refined meshes of SLCB elements is in accordance with a celebrated practical rule of the application of the Finite Element Method in nonlinear analyses, that states that "refined meshes of simple elements are better than coarse meshes of sophisticated elements". Refined meshes may also allow the introduction of elements to represent the behaviour of tubular joints, of dented members, and local buckling of the walls of the tubular members.

Another point that speaks in favor of the use of standard elements like the SLCB with refined meshes is the accelerated evolution in the performance of the computer hardware, that resulted in the introduction of high-performance computers with vector/parallel architectures. The efficient use of such architectures may reduce the CPU requirements for an analysis to a fraction of the time required in a standard scalar processor.

These remarks lead us to the conclusion that the use of refined meshes, that until recently could be considered a drawback of simple elements like the SLCB, can now be viewed as an important advantage, since it allows a more accurate representation of the behaviour of the structure, not only globally but also locally.

CONCLUSION

The CALEB system has a broad area of application in the analysis of offshore structures, and can be used as a valuable tool for the optimization and extension of lifetime of fixed platforms and other offshore equipments.

It is seen that, with the computational tools implemented in the CALEB system allied to the high performance of the currently available hardware, there is practically no limits for the application of SLCB elements in the collapse analysis of fixed offshore platforms.

This paper is intended to briefly describe the main characteristics of the CALEB system. A more detailed treatment of the methodology for collapse analysis of offshore platforms using the standard beam element, as well as the results of the application of the system to selected case studies, will be presented in a subsequent paper (to be published).

REFERENCES

[1] BENJAMIN, A. C. - "Nonlinear Geometric Analysis of Framed Structures by the Finite Element Method", M. Sc. Thesis, COPPE/UFRJ (1982), Brasil (In Portuguese).

[2] BENJAMIN, A. C. - "Application of the Simple Plastic Beam Theory in the Formulation of an Elastic-Plastic Beam Element", Doctorate Seminar, COPPE/UFRJ, (1985), Brasil (In Portuguese).

[3] BENJAMIN, A. C. - "A Study of the Elastic-Plastic Geometric Nonlinear Behaviour of Framed Structures Using the Finite Element Method", Doctorate Seminar, COPPE/UFRJ, (1985), Brasil (In Portuguese).

[4] BENJAMIN, A. C. - "Collapse Analysis of Rigid Risers and General Framed Structures Using Caleb Program", Project 279800, Final Report, SEDEM Report No. 026/92, PETROBRAS/CENPES/DIPREX, (1992), Brasil (In Portuguese).

[5] BENJAMIN, A. C. - "Maintenance and Improvement of Pre-Processors and Post-Processors Programs of Caleb Program", Project 284800, Final Report, SEDEM Report No. 002/93, PETROBRAS/CENPES/DIPREX, (1993), Brasil (In Portuguese).

[6] BENJAMIN, A. C. - "Implementation in Caleb System of Computational Tools for the Collapse Analysis of Offshore Platforms", Project 295600, 1st Partial Report, SEDEM Report No. 01/97, PETROBRAS/CENPES/DIPREX, (1997), Brasil (In Portuguese).

[7] CALEB: Geometric and Material Nonlinear Analysis of Framed Structures - Version 1.4, User's Manual, PETROBRAS/CENPES/DIPREX, December 1995.

[8] LIMA, B. S. L. P., JACOB, B. P., EBECKEN, N. F. F. and BENJAMIN, A. C. - "Portable FORTRAN Programming Tools in the Development of a Structural Analysis Program", Computers & Structures, vol. 57, No. 6, pp. 1109-1125, (1995).

[9] BATOZ, J. L. and DHATT, G. - "Incremental Displacement Algorithms for Nonlinear Problems", Int. J. Num. Meth. Engng., No. 14, pp. 1262-1267, (1979).

[10] JUSTINO FILHO, M. R. - "Collapse Analysis of Offshore Structures on the INTEL iPSC/860 Hypercube Supercomputer", D. Sc. Thesis, COPPE/UFRJ (1993), Brasil (In Portuguese).

[11] BENJAMIN, A. C. - "Collapse Analysis of Offshore Platforms Using CALEB System", Project 295600, 2nd Partial Report, SEDEM Report No. 02/97, PETROBRAS/CENPES/DIPREX, (1997), Brasil (In Portuguese).

[12] BATHE, K. J. - "Finite Elements Procedures for Engineering Analysis", Prentice-Hall, New Jersey, USA, (1982).

[13] BENJAMIN, A. C. and EBECKEN, N. F. F. - "Application of the Simple Plastic Beam Theory in the Formulation of an Elasto-Plastic Beam Element", Revista Brasileira de Engenharia (RBE), Caderno de Engenharia Estrutural, Vol. 3, No. 2, pp. 29-57, (1985), Brasil (In Portuguese).

[14] BENJAMIN, A. C. and EBECKEN, N. F. F. - “On Large Displacements Elasto-Plastic Analysis of Three-Dimensional Beam Structures Using Explicitly Integrated Stiffness Matrices”, Revista Brasileira de Engenharia (RBE), Caderno de Engenharia Estrutural, Vol. 4, No. 2, pp. 45-73, (1986), Brasil (In Portuguese).

[15] COOK, R.D., MALKUS, D.S. and PLESHA, M.E., “Concepts and Applications of Finite Element Analysis”, John Wiley & Sons (1989)

[16] BATHE, K. J. AND WILSON, E. L., “Large Eigenvalue Problems in Dynamic Analysis”, J. Engng Mech. Div. ASCE, No. 98, pp. 1471-1485, (1972).

[17] MARSHALL, P.W., GATES, W.E. and ANAGNOSTOPOULOS, S. - "Inelastic Dynamic Analysis of Tubular Offshore Structures", OTC Paper 2908, pp. 235-246, (1977).

[18] UEDA, Y., RASHED, S.M.H. and NAKACHO, K. - "New Efficient and Accurate Method of Nonlinear Analysis of Offshore Tubular Frames (the Idealized Structural Unit Method)", Trans. ASME, J. Energy Resources Technology, Vol. 107, pp. 204-211, (1985).