Transactions of the Korean Nuclear Society Spring Meeting
Chuncheon, Korea, May 25-26 2006

The Nonlinear Finite Element Analysis Program NUCAS for Prestressed Concrete Containment Building

Hong-Pyo Lee,a Young-Sun Choun ,b Young-Chul Song a
a Korea Electric Power Research Institute, Integrity Assessment Group, 103-16 Munji-Dong Yuseong, Daejeon, Korea
b Korea Atomic Energy Research Institute, Integrated Safety Assessment Division, P.O. Box 105,Yuseong, Daejeon, Korea,
hplee@kepri.re.kr

1. Introduction

Containment buildings in nuclear power plants are the last barriers to protect the spread of radiation materials to the environment. It is important to evaluate the performance of containment building during its service life. During the several years, a nonlinear finite element (FE) analysis program NUCAS[1], which can evaluate the ultimate pressure capacity of the prestressed concrete containment building has been developed in KAERI. This paper is mainly focused to describe research progress on the nonlinear FE analysis program NUCAS for a reinforced concrete containment buildings.

2. Program Organization

NUCAS has been created with 8-node, 9-node assumed strain shell element and lower order(8-node) solid element on the top of the research purpose FE analysis program FEAP[2](Figure 1).

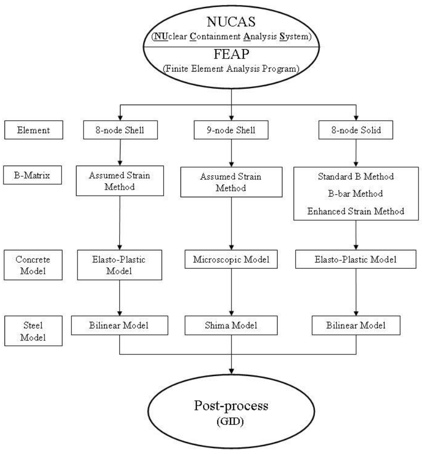

Figure 1. Layout of the NUCAS

2.1 Finite Element

2.1.1 Degenerated Shell Element

The code applied into the development Reissner-Mindlin degenerated shell element which is considered the following two basic assumptions:

- The normal to the mid-surface of shell remains straight after deformation but not necessarily normal to the deformed mid-surface.
- The normal transverse stress is negligible as in Kirchhoff-Love theory.

To avoid finite element deficiencies such as shear and membrane locking phenomena, the substitute strains based on the assumed strain method are used in the shell formulation.

2.1.2 Lower Order Solid Element

The lower order solid finite elements are formulated with three different ways such as the use of standard strain-displacement relationship, B-bar method and enhanced assumed strain method[3,4].

2.2 Material Model

2.2.1 Concrete

The inelastic characteristic of concrete material is well represented by using the so-called elasto-plastic and fracture material model for uncracked situation and several existing material models such as shear transfer model, reduction model for compressive strength, strain softening model and tension stiffening model are adopted to represent cracked concrete with developed cracking criteria from the experimental results(Figure 2).

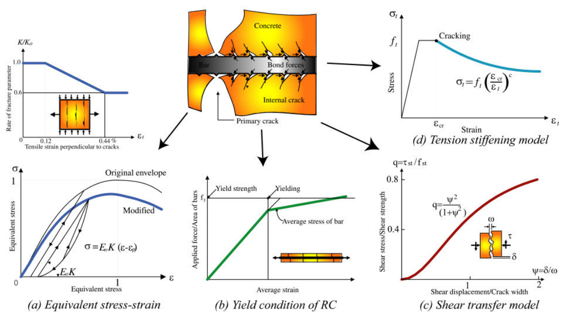

Figure 2. Concept of the concrete material model

2.2.2 Steel

The stress-strain relationship of mild steel is usually assumed to be elasto-plastic with a distinct yield stress of bare bar's yield stress. However, when the reinforcing bars are surrounded by concrete, the average stress-strain relationship exhibits very different behavior than that exhibited by a bare bar without concrete. To consider this phenomenon, we generally underestimate the yield stress, this is the apparent yield stress of an embedded bar in concrete. In the program, the apparent yield stress for this situation is represented by Hsu's model[5].

2.2.3 Tendon

In three-dimensional(3D) nonlinear FE analysis of prestressed concrete structures, three methods available for the simulation of tendon are the smeared, discrete and embedded method. The smeared and discrete formulations are dependent on the concrete element mesh. In 3D applications, this can lead to prohibitive computational costs due to the use of many unnecessarily small elements or inaccuracies caused by elements with undesirable aspect ratios. To remedy these problems, an embedded formulation is preferable. Linear and curved embedded approaches with inverse mapping by Elwi and Hrudey[6] are formulated in the NUCAS code.

3. Benchmark Test Suite

It has been used to check the capability of NUCAS and provided as the future benchmark tests for the FE analysis of prestressed concrete containment. The nonlinear FE analysis using the present NUCAS-version is tested and demonstrated with several numerical examples such as Vecchio panel, cylindrical shell, KAERI panel, EPRI panel McNeice slab and 1/4 scale containment to verify the performance and numerical accurate. Figure 3 shows one of the examples developed benchmark test suite. Reference 1,3,4 are described more detail.

4. Conclusion

This paper describes the current version of NUCAS which is mainly focused to evaluate nonlinear behavior and ultimate pressure capacity of nuclear containment building in KAERI. From the benchmark tests, the FE analysis results by the program agree very well with the experimental data and other numerical results. Finally, NUCAS will be essential to develop the state of art FE analysis code for accurate safety assessment of containment building and our consistent research on safety analysis techniques.

Acknowledgement

The Research grant from the Ministry of Science and Technology, Korea, for the Nuclear Research & Development Program is gratefully acknowledged.

REFERENCES

[1] S.J. Lee, H.P. Lee and J.M. Seo, The Nonlinear Finite Element Analysis Program for Reinforced Concrete Containment Building NUCAS, KAERI, KAERI/TR-2076/2002, 2002.

[2] R.L. Taylor, FEAP - A Finite Element Analysis Program, User's Manual, Version 7.1, 1999.

[3] Y.J. Lee, S.J. Lee, Y.S. Choun and J.M. Seo, A comparison Study on the Performance of Lower Order Solid Finite Element For Elastic Analysis of Plate and Shell Structures, KAERI, KAERI/TR=2489/2003, 2003.

[4] H.P. Lee, Y.S. Choun and J.M. Seo, A Study on the Standard 8-node Solid Finite Element for Nonlinear Analysis of Reinforced Concrete Containment, KAERI, KAERI/TR-2963/2005, 2005

[5] T.T.C. Hsu, Unified Theory of Reinforced Concrete, CRC, 1992.

[6] A.E. Elwi and T.M. Hrudey, Finite Element Model for Curved Embedded Reinforcement, Journal of Engineering Mechanics, Vol. 115, No. 4, 1989.

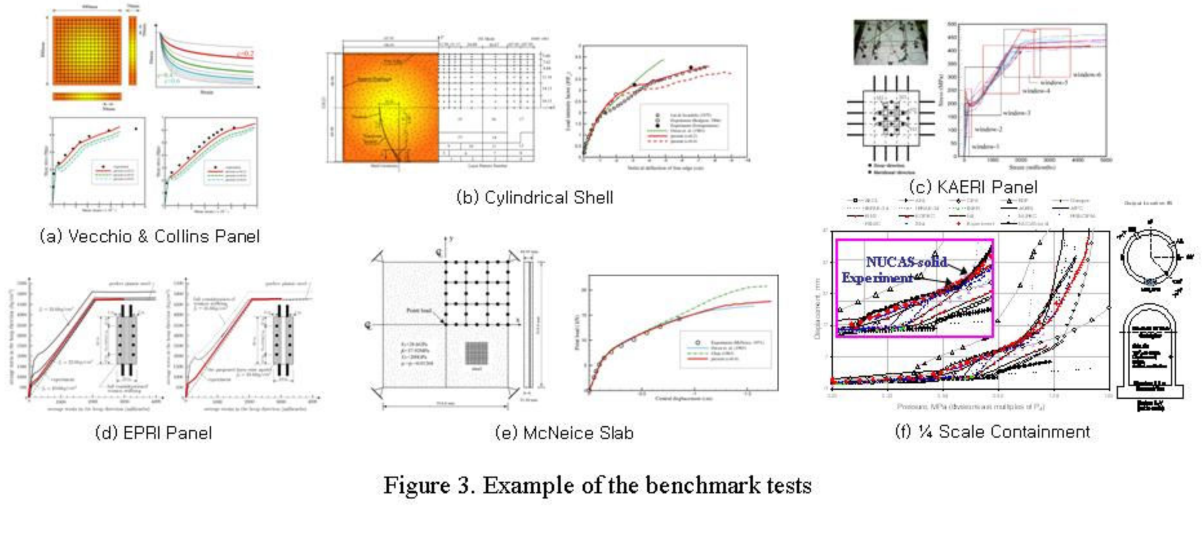

Figure 3. Example of the benchmark tests