3C Computer Programs Used in the Design and Analysis of Seismic Category I Structures

3C.1 Introduction

The following Seismic Category I structures and their foundations of the Nuclear Island are analyzed and/or designed using the computer programs described in this appendix:

(1) Reactor Building

(2) Concrete Containment Structure

(3) Control Building

3C.2 Static and Dynamic Structural Analysis Systems (STARDYNE)

3C.2.1 Description

STARDYNE is a large-scale, finite-element program that has a broad range of analysis types and many different structural elements. The STARDYNE system of structural analysis programs is segmented into individual programs. A variety of static or dynamic analyses may be performed by using one or more of the individual programs in a coordinated series of computer runs.

3C.2.2 Validation

STARDYNE is written and maintained by the System Development Corporation of Santa Monica, California, and is available on the Control Data Corporation (CDC) system. Program validation documentation is available at CDC in San Francisco, California.

3C.2.3 Extent of Application

This program is used for the static analysis of the Reactor Building and containment structure.

3C.3 Concrete Element Cracking Analysis Program (CECAP)

3C.3.1 Description

CECAP computes stresses in a concrete element under thermal and/or nonthermal (real) loads, considering effects of concrete cracking. The element represents a section of a concrete shell or slab, and may include two layers of reinforcing, transverse reinforcing, prestressing tendons, and a liner plate.

The program outputs stresses and strains along the element in the concrete, reinforcement, and liner plate, and resultant forces and moments for the composite concrete element.

CECAP assumes linear stress-strain relationships for steel and for concrete in compression. Concrete is assumed to have no tensile strength. The solution is an iterative process, whereby

tensile stresses found initially in concrete are relieved (by cracking) and redistributed in the element. The equilibrium of nonthermal loads is preserved. For thermal effects, the element is assumed free to expand inplane, but is fixed against rotation. The capability for expansion and cracking generally results in a reduction in thermal forces and moments from the initial condition.

3C.3.2 Validation

CECAP is written and maintained by Bechtel Power Corporation (BPC) of San Francisco, California. Program validation documentation is available at BPC in San Francisco.

3C.3.3 Extent of Application

This program is used for the analysis of the Reactor Building and containment structure.

3C.4 Finite Element Program for Cracking Analysis (FINEL)

3C.4.1 Description

FINEL is a proprietary computer program of Bechtel Power Corporation, San Francisco, California. The FINEL program performs a static analysis of stresses and strains in plane and axisymmetric structures by the finite element method. The program performs the non-linear static analysis utilizing a stepwise linear iteration solution technique. Within each solution cycle, status of all elements is determined and their stiffness adjusted by the program prior to the next iteration cycle. The Von Mises yield criterion is used to determine the status of all ductile materials and brittle materials which are in compression. A ductile material is assumed to yield in all directions when the yield criterion is exceeded. A brittle material is assumed to be cracked in the direction in which the maximum principal stress exceeds the specified tensile stress. The modulus of elasticity for each material is adjusted for the next solution cycle to conform to the secant modulus corresponding to the calculated strain in the element following the bilinear stress strain relationship specified. The numerical algorithm assumes that the state of stress which exists when converged solution is achieved is independent of the stress history of the loading.

3C.4.2 Validation

The FINEL code has been extensively used in the past to design reinforced concrete containment structures and to predict the strains and deflections during their structural integrity tests (SIT). The correlation between the predicted and observed deflections and strains during such tests has been very satisfactory. FINEL code predictions have been on the conservative side. The FINEL code was validated by comparing results based on FINEL with results based on classical solutions and/or experimental results existing in literature. A further confirmation of its validity is provided by comparison of predicted strains and deformations based on FINEL with experimental results obtained during SIT on various Bechtel designed containments.

3C.4.3 Extent of Application

This program is used for the static load analysis of the reactor building and containment to determine stresses and strains in the various structural elements and resultant forces and moments at selected sections, and is used to evaluate the ultimate strength of the containment.

3C.5 ANSYS

3C.5.1 Description

ANSYS is a large, finite element program for a broad range of analyses types. The structural analysis capabilities include material and geometric nonlinear analysis, static analysis, and a variety of dynamic analyses.

The element for a concrete cracking analysis allows a full-nonlinear analysis of reinforced concrete with cracking and crushing of concrete.

3C.5.2 Validation

ANSYS, Inc. of Canonsburg, Pennsylvania developed ANSYS. The program validation documentation is available at ANSYS, Inc.

3C.5.3 Extent of Application

This program is used for the containment dynamic analysis of containment loads, for the containment ultimate capacity analyses and for containment seismic margin analysis.