

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

3C.1 Introduction

The following Lungmen NPS Units 1 & 2 Seismic Category I structures and their foundations are analyzed and/or designed using the computer programs described in this appendix:

- (1) Reactor Building
- (2) Concrete Containment Structure
- (3) Control Building
- (4) Auxiliary Fuel Building

The computer program used for the Safety Service Water Pump House structure will be supplied with FSAR.

3C.2 ANSYS

3C.2.1 Purpose

ANSYS is a general purpose finite element computer program with interactive capabilities by ANSYS, Inc. The analysis method is based on standard displacement formulation of the finite element method.

ANSYS can be used to perform detailed finite element analyses of numerous items. Primary use of the program will be in the analysis of mechanical equipment and concrete. The program is capable of performing static, dynamic, and thermal gradient analyses.

3C.2.2 Data Input

Input to ANSYS consists of the finite element model geometry (i.e., nodes, elements material properties, thermal properties), boundary conditions, structural loads, or thermal conditions.

3C.2.3 Data Output

Output of ANSYS consists of finite element geometry plots and analysis results which includes plots and tables.

3C.2.4 Computer Hardware and Operating Software Requirements

Hardware platform: Hewlett Packard Series 700 work station.

Operating system: HP-UX 9.01 or higher (UNIX).

3C.2.5 Computer Hardware and Supporting Software Requirements

The program is run on a UNIX based platform. GE currently runs the program on their Hewlett Packard work stations.

3C.3 Shimizu Section Design Program-2D (SSDP-2D)

3C.3.1 Purpose

Section design of Reactor Building Concrete Structures, such as RCCV, shear walls, and slabs is performed using SSDP-2D.

SSDP-2D computes stresses in a thick 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 orthogonal reinforcing. It does not include the effect of the orthogonal liner.

SSDP-2D calculates the stresses considering two-dimensional equilibrium conditions of section forces with the existence of thermal loads and concrete cracking. It is assumed in the code that concrete has an anisotropic property and that cracked concrete does not carry tensile forces. Concrete is assumed to have no tensile strength.

3C.3.2 Data Input

Section forces and moments resulting from NASTRAN analysis are combined according to corresponding load combination tables in order to obtain design section forces and moments.

3C.3.3 Data Output

The program outputs stresses and strains along the element in the concrete and reinforcement. It assumes linear stress-strain relationships for steel and concrete in compression. 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 reduction in section stiffness and a reduction in thermal forces and moments from the initial values. SSDP takes this phenomenon into consideration and to calculate concrete and rebar stresses that equilibrate with section forces and moments.

3C.3.4 Computer Hardware and Operating Software Requirements

Hardware platform: Hewlett Packard Series 700 work station.

Operating system: HP-UX 9.01 or higher (UNIX)

3C.3.5 Computer Hardware and Supporting Software Requirements

The program is run on UNIX based platform. Shimizu currently runs the program on their Hewlett Packard work stations.

3C.4 NASTRAN**3C.4.1 Purpose**

NASTRAN is a GE in-house version of the MSC/NASTRAN program which is developed by the MacNeal Scwendler Corporation. NASTRAN is a general purpose computer program for finite element analysis; its capabilities include: static response to concentrated and distributed loads, to thermal expansion and to enforced displacements; dynamic response to transient loads, to steady-state sinusoidal loads, and to random excitation; and determination of eigenvalues for use in vibration analysis.

3C.4.2 Data Input

Input to NASTRAN consists of a finite elemental model (nodes, elements, constraints, material properties) and applied loads.

3C.4.3 Data Output

Output to NASTRAN consists of the finite element geometry plots and analysis results which include plots and tables.

3C.4.4 Computer Hardware and Operating Software Requirements

Hardware platform: Hewlett Packard Series 700 work station.

Operating system: HP-UX 9.01 or higher (UNIX).

3C.4.5 Computer Hardware and Supporting Software Requirements

The program is run on a UNIX based platform. GE currently runs the program on their Hewlett Packard work stations.

3C.5 A System for Analysis of Soil-Structure Interaction - SASSI**3C.5.1 Purpose**

This computer program consists of a number of interrelated program-modules which can be used to solve a wide range of dynamic soil-structure interaction (SSI) problems in two or three dimensions. The program is used to obtain seismic design loads and in-structure floor response spectra for the seismic Category-I buildings accounting for the effects of SSI. The program is based on the finite-element method formulated in the frequency domain using a substructuring technique.

The program SASSI was developed at the University of California, Berkeley, in 1982 under the technical direction of John Lysmer. The version of SASSI used is a modified version of the program obtained from University of California Berkeley, in 1990 and implemented by International Civil Engineering Consultants, Inc. (ICEC) on the IBM/RISC-6000 computer workstation in 1994. The version of SASSI obtained from the University of California, Berkeley, in 1990 was the same version of the program implemented on the IBM mainframe computer and applied to the Diablo Canyon Power Plant, Long Term Seismic Program by Bechtel Corporation in 1988. During the course of installation, testing, and validation of the program by ICEC to make it suitable for application on the IBM/RISC-6000 computer workstation, some modifications to the program were made to correct known bugs in the program discovered since 1990, to improve the program performance, and to enhance the program output to facilitate post-processing. Thus, in addition to the modifications and enhancements made to the program by Bechtel prior to 1988 (which included correcting the motion phases in the free-field site Rayleigh wave calculation, replacing the plate/shell element, modifying the spring element to include element damping capability, and providing the option for local end release for the beam element), further modifications and enhancements were made by ICEC which include correcting two bugs discovered in 1994 in the stress calculations of the two-dimensional plane-strain and three-dimensional brick solid elements, correcting maximum soil-layer number that can be specified in the point-load solution (POINT3) module to the same number as that allowed in the free-field site calculation (SITE) module, modifying the membrane-stress calculation of the plate/shell element to output results in stress-resultant-per-unit-length form instead of stresses per unit area, modifying the transfer-function output to a file format that permits fast post-processing of results, and providing an option for selection of frequencies to be used for combining the transfer function results.

The ICEC version of SASSI used in the SSI analysis for the seismic Category-I buildings has been extensively tested and verified against benchmark solutions for a wide range of SSI problems published in literature, including a verification using the field test results obtained from the 1/4-scale containment model test program conducted in Lotung, Taiwan, sponsored jointly by Taiwan Power Company (TPC), Electric Power Research Institute (EPRI), and the U. S. Nuclear Regulatory Commission (USNRC).

3C.6 Free-Field Site Response Analysis - SHAKE

3C.6.1 Purpose

This program is used to perform the free-field site response analysis to generate the design-earthquake-induced strain-compatible free-field soil properties and site response motions required in the seismic SSI analysis.

SHAKE is a computer program developed at the University of California, Berkeley, by B. Schnabel, John Lysmer and H. B. Seed in 1972. The program is based on the theory of one-dimensional propagation of shear waves in the vertical direction in a horizontally-layered visco-elastic soils system overlying an elastic half-space medium. Non-linearities in soil shear

modulus and hysteresis damping as functions of shear strain in soil are accounted for by the use of equivalent-linear soil properties using an iterative equivalent linearization procedure to obtain constant values of shear modulus and hysteresis damping ratio compatible with the effective shear strain in each soil layer. The final iterated, strain-compatible shear moduli and damping ratios obtained from the SHAKE analysis are used as the equivalent-linear soil properties for the free-field site soil in seismic SSI analysis. The free-field site response motions obtained from the SHAKE analysis using the final iterated, strain-compatible equivalent linear soil properties are used as the free-field seismic input motions for the seismic SSI analysis.

3C.6.2 Computer Software Requirements

This program is run on PC on DOS/WINDOWS.

3C.7 GT STRUDL

3C.7.1 Purpose

GT STRUDL (Structural Design Language) is a subsystem of GTICES (The Georgia Tech Integrated Civil Engineering System). It solves structural engineering problems in frame analysis, finite element analysis, static and dynamic analysis, as well as steel and concrete design.

GT STRUDL is a widely accepted analysis and design program developed by and licensed to Black and Veatch by:

Georgia Tech Research Corporation (GTRC)
Computer-Aided Structural Engineering Center (Case Center)
Georgia Institute of Technology
Atlanta, Georgia 30332-0355, U.S.A.

3C.7.2 Data Input:

Input to GT STRUDL consists of finite element model geometry, boundary conditions, structural loads, or thermal conditions.

3C.7.3 Data Output:

Output of GT STRUDL consists of F.E.M. geometry plots and analysis results which include plots & tables.

3C.7.4 Computer Hardware and Operating Software Requirements

Hardware platform: Sun SPARC work station or server
Operating system: Solaris 2.4/SunOS 5.4

3C.7.5 Computer Hardware and Operating Software Requirements

Hardware platform: 386/486/Pentium PC
Operating system: PC DOS/WINDOWS 3.1 or WINDOWS NT

3C.8 TEMCOM

3C.8.1 Purpose

TEMCOM is a heat transfer analysis program which is developed by Shimizu. TEMCOM solves a temperature distribution in a two-dimensional model based on a finite differential method. It performs analyses under the following conditions.

- Element: triangle and quadrilateral elements
- Surface heat transfer: convection and radiation
- Temperature condition: steady-state and transient temperature conditions

3C.8.2 Data Input

Input to TEMCOM consists of a finite element model (nodes, elements, material properties), boundary conditions (surface heat transfer coefficients, temperatures), and other control data (analysis interval time, output interval time).

3C.8.3 Data Output

Nodal temperatures are obtained by a TEMCOM analysis.

3C.8.4 Computer Hardware and Operating Software Requirements

Hardware platform: IBM 9021-740
Operating System: MVS version 4.2.2

3C.8.5 Computer Hardware and Supporting Software Requirements

TEMCOM is developed with computer languages of PL/I and IBM based Assembler. Therefore, an IBM compatible mainframe machine is required for its execution. Shimizu currently runs TEMCOM on their IBM mainframe.

3C.9 SCOTH

3C.9.1 Purpose

SCOTH is a computer program that generates an acceleration time history whose response spectrum matches a user-defined target acceleration response time spectrum. The algorithm implemented in the program is based on the fact that a small variation in a time history results in small changes in the spectrum values without affecting the spectrum time (the time at which

a spectrum is achieved). Using the stationary of spectrum times, it is possible thus to derive the required small adjustments to the time history from the small changes in its spectrum. The solution scheme proceeds from a time history selected as an initial approximation, which is altered in a sequence of iterations resulting in uniform convergence to the target spectrum until a desired level of accuracy is achieved.

3C.9.2 Data Input

Input to SCOTH consists of user-defined target acceleration response spectrum.

3C.9.3 Data Output

Acceleration time history whose response spectrum matches the user-defined target acceleration response spectrum.

3C.9.4 Computer Hardware and Operating Software Requirements

Hardware platform:	386/486/Pentium PC
Operating System:	PC DOS