3D Computer Programs Used in the Design of Components, Equipment and Structures

3D.1 Introduction

As discussed in Subsection 3.9.1.2, this appendix describes the major computer programs used in the analysis of the safety-related components, equipment and structures. The quality of the programs and the computed results is controlled. The programs are verified for their application by appropriate methods, such as hand calculations, or comparison with results from similar programs, experimental tests, or published literature, including analytical results or numerical results to the benchmark problems.

Updates to Appendix 3D will be provided to indicate any additional programs used, especially by vendors of components and equipment, or the later version of the described programs, and the method of their verification.

3D.2 Fine Motion Control Rod Drive

3D.2.1 Fine Motion Control Rod Drive—FMCRD01

The FMCRD01 program is used to obtain scram performance data for various inputs to the fine motion control rod drive (FMCRD) stress analysis for both code and non-code parts. The use of this program is addressed in Subsection 3.9.1.3.2. Experimental data on pressure drops, friction factors, effects of fuel channel detection, etc., are used in the setting up and perfecting of this code. Internal drive pressures and temperatures used in the stress analysis are also determined during actual testing of the prototype FMCRD.

3D.2.2 Structural Analysis Programs

Structural analysis programs, such as NASTRO4V and ANSYS, that are mentioned in Subsections 3D.3 and 3D.5 are used in the analysis of the FMCRD.

3D.3 Reactor Pressure Vessel and Internals

The following computer programs are used in the analysis of the reactor pressure vessel, core support structures, and other safety class reactor internals: NASTR04V, SAP4G07, HEATER, FATIGUE, ANSYS, CLAPS, ASSIST, SEISM03, SASSI and ACSTIC2. These programs are described in Subsection 4.1.4.

3D.4 Piping

3D.4.1 Piping Analysis Program—PISYS

PISYS is a computer code for analyzing piping systems subjected to both static and dynamic piping loads. Stiffness matrices representing standard piping components are assembled by the program to form a finite element model of a piping system. The piping elements are connected to each other via nodes called pipe joints. It is through these joints that the model interacts with the environment, and loading of the piping system becomes possible. PISYS is based on the linear elastic analysis in which the resultant deformations, forces, moments and accelerations at each joint are proportional to the loading and the superposition of loading is valid.

PISYS has a full range of static dynamic load analysis options. Static analysis includes dead weight, uniformly-distributed weight, thermal expansion, externally-applied forces, moments, imposed displacements and differential support movement (pseudo-static load case). Dynamic analysis includes mode shape extraction, response spectrum analysis, and time-history analysis by modal combination or direct integration. In the response spectrum analysis [i.e., uniform support motion response spectrum analysis (USMA) or independent support motion response spectrum analysis (ISMA)], the user may request modal response combination in accordance with NRC Regulatory Guide 1.92. In the ground motion (uniform motion) or independent support time-history analysis, the normal mode solution procedure is selected. In analysis involving time-varying nodal loads, the step-by-step direct integration method is used.

The PISYS program has been benchmarked against Nuclear Regulatory Commission piping models. The results are documented in a report to the Commission, “PISYS Analysis of NRC Benchmark Problems”, NEDO-24210, August 1979, for mode shapes and USMA options. The ISMA option has been validated against NUREG/CR-1677, “Piping Benchmark Problems Dynamic Analysis Independent Support Motion Response Spectrum Method,” published in August 1985.

3D.4.2 Component Analysis—ANSI7

ANSI7 is a computer code for calculating stresses and cumulative usage factors for Class 1, 2 and 3 piping components in accordance with articles NB, NC and ND-3650 of ASME Code Section III. ANSI7 is also used to combine loads and calculate combined service level A, B, C and D loads on piping supports and pipe-mounted equipment.

3D.4.3 Area Reinforcement—NOZAR

The Nozzle Area Reinforcement (NOZAR) computer program performs an analysis of the required reinforcement area for openings. The calculations performed by NOZAR are in accordance with the rules of ASME Code Section III, 1974 edition.

3D.4.4 Dynamic Forcing Functions

3D.4.4.1 Relief Valve Discharge Pipe Forces Computer Program—RVFOR

The relief valve discharge pipe connects the pressure-relief valve to the suppression pool. When the valve is opened, the transient fluid flow causes time-dependent forces to develop on the pipe wall. This computer program computes the transient fluid mechanics and the resultant pipe forces using the method of characteristics.

3D.4.4.2 Turbine Stop Valve Closure—TSFOR

The TSFOR program computes the time-history forcing function in the main steam piping due to turbine stop valve closure. The program utilizes the method of characteristics to compute fluid momentum and pressure loads at each change in pipe section or direction.

3D.4.5 Response Spectra Generation

3D.4.5.1 ERSIN Computer Program

ERSIN is a computer code used to generate response spectra for pipe-mounted and floor-mounted equipment. ERSIN provides direct generation of local or global acceleration response spectra.

3D.4.5.2 RINEX Computer Program

RINEX is a computer code used to interpolate and extrapolate amplified response spectra used in the response spectrum method of dynamic analysis. RINEX is also used to generate response spectra with nonconstant model damping. The nonconstant model damping analysis option can calculate spectral acceleration at the discrete eigenvalues of a dynamic system using either the strain energy weighted modal damping or the ASME Code Class N-411-1 damping values.

3D.4.6 Piping Dynamic Analysis Program—PDA

The pipe whip dynamic analysis is performed using the PDA computer program, as described in Appendix 3L. PDA is a computer program used to determine the response of a pipe subjected to the thrust force occurring after a pipe break. It also is used to determine the pipe whip restraint design and capacity.

The program treats the situation in terms of generic pipe break configuration, which involves a straight, uniform pipe fixed (or pinned) at one end and subjected to a time-dependent thrust force at the other end. A typical restraint used to reduce the resulting deformation is also included at a location between the two ends. Nonlinear and time-independent stress-strain relations are used to model the pipe and the restraint. Using a plastic hinge concept, bending of the pipe is assumed to occur only at the fixed (or pinned) end and at the location supported by the restraint.

Effects of pipe shear deflection are considered negligible. The pipe-bending moment-deflection (or rotation) relation used for these locations is obtained from a static nonlinear cantilever beam analysis. Using moment-angular rotation relations, nonlinear equations of motion are formulated using energy considerations, and the equations are numerically integrated in small time steps to yield the time-history of the pipe motion.

3D.4.7 Not Used

3D.4.8 Thermal Transient Program—LION

The LION program is used to compute radial and axial thermal gradients in piping. The program calculates a time-history of ΔT_1, ΔT_2, Ta, and Tb (defined in ASME Code Section III, Subsection NB) for uniform and tapered pipe wall thickness.

3D.4.9 Not Used

3D.4.10 Engineering Analysis System—ANSYS

The ANSYS computer program is a large-scale general-purpose program for the solution of several classes of engineering analysis problems. Analysis capabilities include static and dynamic, plastic, creep and swelling, small and large deflections, and other applications.

This program is used to perform non-linear analysis of piping systems for time varying displacements and forces due to postulated pipe breaks.

3D.5 Pumps and Motors

Following are the computer programs used in the dynamic analysis to assure the structural and functional integrity of the pump and motor assemblies, such as those used in the ABWR ECCS.

3D.5.1 Structural Analysis Program—SAP4G07

SAP4G07 is used to analyze the structural and functional integrity of the pump/motor systems. This program is also identified in Subsections 4.1.4.1.2, 3D.3 and 3D.6. This is a general structural analysis program for static and dynamic analysis of linear elastic complex structures. The finite-element displacement method is used to solve the displacement and stresses of each element of the structure. The structure can be composed of unlimited number of three-dimensional truss, beam, plate, shell, solid, plane strain-plane stress and spring elements that are axisymmetric. The program can treat thermal and various forms of mechanical loading. The dynamic analysis includes mode superposition, time-history, and response spectrum analysis. Seismic loading and time-dependent pressure can be treated. The program is versatile and efficient in analyzing large and complex structural systems. The output contains displacement of each nodal point as well as stresses at the surface of each element.

3D.6 Heat Exchangers

The following computer programs are used in dynamic and static analyses to determine the structural and functional integrity of the heat exchangers, such as those used in the ABWR RHR System.

3D.6.1 Structural Analysis Program—SAP4G07

The structural integrity of the heat exchanger is evaluated using SAP4G07. This program is described in Subsection 3D.5.1.

3D.7 Soil-Structure Interaction

3D.7.1 A System for Analysis of Soil-Structure Interaction—SASSI

This program consists of a number of interrelated computer program modules which can be used to solve a wide range of dynamic soil-structure interaction (SSI) problems in two or three dimensions. This program is used to obtain enveloped seismic design loads based on the finite-element method using substructuring technique, as described in Section 3A.5 of Appendix 3A of this document. A description of this program is included in Subsection 4.1.4.1.9.

The SASSI computer program was developed at the University of California, Berkeley, under the technical direction of Prof. John Lysmer. The Bechtel version of the program was obtained from the University of California, Berkeley. During the course of installation, testing, and validation of the Bechtel version of the program on the IBM System, some modifications and enhancements were made to the program to improve the performance. These include correcting the motion phases in Rayleigh wave calculation, replacing the plate element, modifying the spring element to include damping capability, and providing the option for local end release condition in beam element. The program was verified against benchmark results reported by various investigators in the technical literature.

3D.7.2 Not Used

3D.7.3 Free-Field Response Analysis—SHAKE

This program is used to perform the free-field site response analysis required in the seismic SSI analysis (Subsection 3A.6).

SHAKE is a computer program developed at the University of California, Berkeley, by Schnable, Lysmer and Seed (Reference 3A-5 of Subsection 3A.11). The program uses the principle of one-dimensional propagation of shear waves in the vertical direction for a system of horizontal, visco-elastic soil layers to compute soil responses in the free-field. The nonlinearities in soil shear modulus and damping are accounted for by the use of equivalent linear soil properties using an iterative procedure to obtain values for modulus and damping compatible with the effective shear strains in each layer. The final iterated, strain-compatible properties are used as equivalent linear soil properties in seismic SSI analysis.