GE Energy

James C. Kinsey
Project Manager, ESBWR Licensing

PO Box 780 M/C J-70
Wilmington, NC 28402-0780
USA

T 910 675 5057
F 910 362 5057
jim.kinsey@ge.com

MFN 07-225

Docket No. 52-010

April 18, 2007

U.S. Nuclear Regulatory Commission
Document Control Desk
Washington, D.C. 20555-0001

Subject: **Response to Portion of NRC Request for Additional Information Letter No. 67 Related to ESBWR Design Certification Application – DCD Tier 2 Appendix 3D – RAI Numbers 3.9-12, 3.9-13, 3.9-14, 3.9-16 and 3.9-100**

Enclosure 1 contains GE's response to the subject NRC RAIs transmitted via the Reference 1 letter.

If you have any questions or require additional information regarding the information provided here, please contact me.

Sincerely,

Kathy Sedney for

James C. Kinsey
Project Manager, ESBWR Licensing

D068

Reference:

1. MFN 06-378, Letter from U.S. Nuclear Regulatory Commission to David Hinds, *Request for Additional Information Letter No. 67 Related to ESBWR Design Certification Application,* October 10, 2006

Enclosure:

1. MFN 07-225 – Response to Portion of NRC Request for Additional Information Letter No. 67 Related to ESBWR Design Certification Application – DCD Tier 2 Appendix 3D – RAI Numbers 3.9-12, 3.9-13, 3.9-14, 3.9-16 and 3.9-100

cc:

| | |
|---|---|
| AE Cubbage | USNRC (with enclosures) |
| DH Hinds | GE (with enclosures) |
| RE Brown | GE (w/o enclosures) |
| eDRF | 0000-0063-8617 |

**Enclosure 1**

**MFN 07-225**

**Response to Portion of NRC Request for**

**Additional Information Letter No. 67**

**Related to ESBWR Design Certification Application**

**DCD Tier 2 Appendix 3D**

**RAI Numbers 3.9-12, 3.9-13, 3.9-14, 3.9-16 and 3.9-100**

**NRC RAI 3.9-12**

*In accordance with Standard Review Plan Section 3.9.1, provide the following information for each computer program listed in DCD Tier 2,Appendix 3D: 1) The author, source, dated version and facility, 2) the extent and limitation of its application, and 3) the method used to demonstrate its applicability and validity.*

**GE Response**

DCD Tier 2 Appendix 3D was updated in revision 3 with the latest engineering computer programs (ECPs) description, author, source, extent and limitations of the program's application and the method(s) used to demonstrate the program's applicability and validity. Additional changes to DCD Tier 2 Appendix 3D revision 4 will be made as noted in the attached markup.

Facility and version information is provided in the table below:

| Program | Company – Location | Version | Facility |
|---|---|---|---|
| ANSYS | Hitachi – Japan | 5.6, but may be updated to later versions. | PC (WINDOWS 2000, XP) |
| ANSYS | GE – San Jose, Sunol | v8.1, v8.1A1, v9.0, v9.0A1, v10.0, and v10.0A1 | SGI Server |
| ANSYS | GE – Wilmington | v8.1 | LINUX server |
| ASHSD2 | Hitachi – Japan | 0 | Engineering Work Station 3050RX/230 |
| EVAST | Hitachi – Japan | 0 | Engineering Work Station 3050RX/230 |
| TACF | Hitachi – Japan | 0 | Engineering Work Station 3050RX/230 |
| ABAQUS | Hitachi – Japan | 6.5 | PC (WINDOWS 2000, XP) |
| FEMFL | Hitachi – Japan | 0 | Engineering Work Station 3050RX/230 |
| SEISM | GE – All locations | 03V, October 1998 | VAX Main Frame |
| PISYS | GE – All locations | 07D, January 1998 | DEC Alpha workstation |
| ANSI7 | GE – All locations | 713D, January 1998 | DEC Alpha workstation |
| RVFOR | GE – All locations | 06D, January 1998 | DEC Alpha workstation |

| Program | Company – Location | Version | Facility |
|---|---|---|---|
| TSFOR | GE – All locations | 01D, January 1998 | DEC Alpha workstation |
| ERSIN | GE – All locations | 03V, October 1998 | VAX Main Frame |
| RINEX | GE – All locations | 01V, October 1998 | VAX Main Frame |
| PDA | GE – All locations | 01 | PC |
| LION | GE – All locations | 401, March 1994 | Workstation and PC |
| EZPYP | GE – All locations | 07D, January 1998 | DEC Alpha workstation |
| SAP4G07 | GE – All locations | 07V, December 1995 | VAX Main Frame |

**DCD Impact**

DCD Tier #2, Appendix 3D was revised in revision 3 with the requested information and additional information will be added as noted in the attached markup.

**NRC RAI 3.9-13**

*In DCD Tier 2, Appendix 3D verify that all computer programs used for calculating stresses and cumulative usage factors for Class 1, 2, and 3 components include staff endorsed environmental effects on the fatigue curves.*

**GE Response**

The following DCD Tier 2 Appendix 3D computer programs are used for calculating stresses and cumulative usage factors for class 1, 2 and 3 components.

- ANSYS. When used in this capacity, environmental effects on fatigue curves are accounted for outside of the program.
- EVAST. Environmental effects are not included when using fatigue curves from ASME Code Section III Appendix I. Prior to use of the program results, the environmental effects will be factored into the final output.
- ANSI7. Includes environmental effects on the fatigue curves according to DG1144 and NUREG/CR-6909.

**DCD Impact**

No DCD changes will be made in response to this RAI.

**NRC RAI 3.9-14**

*For each of the components listed in DCD Tier 2, Section 3D, provide the computer program that was used to evaluate the stresses for determining that the American Society of Mechanical Engineers Code Section III, Appendix F, limits were met.*

**GE Response**

The following 4 computer programs listed in DCD Tier 2 Appendix 3D will determine stresses:

- ANSYS, but hand calculations are used to evaluate the stresses for determining if ASME Code Section III, Appendix F, limits were met.
- ASHSD2, FEMFL, ABAQUS, but only Elastic Analysis as defined in ASME Code Section III, Appendix F F1321.3 requirements is designed within this computer program when used to evaluate the stresses for determining if ASME Code Section III, Appendix F, limits were met.

**DCD Impact**

No DCD changes will be made in response to this RAI.

**NRC RAI 3.9-16**

*Identify the components where the inelastic Service Level D limits were met, under these postulated events in DCD Tier 2, Section 3.9.1.4.*

**GE Response**

Control rod guide tube, control rod drive housing and control rod drive outer tube components met the inelastic Service Level D limits, under the postulated events in DCD Tier 2, Section 3.9.1.4.

If any additional components are identified requiring inelastic analysis to meet Service Level D limits, the components will be evaluated using computer programs identified in DCD Tier 2 Appendix 3D. The deformations under faulted conditions will be evaluated in critical areas to assure the necessary design deformation limits, such as clearance limits, are satisfied.

**DCD Impact**

No DCD changes will be made in response to this RAI.

**NRC RAI 3.9-100**

*Provide a listing and description of the computer programs and calculational procedures used for the analysis of the reactor pressure vessel and the reactor pressure vessel internals, including the core support structures.*

**GE Response**

DCD Tier 2 Appendix 3D lists engineering computer programs (ECPs) and their description, used for the design of the reactor pressure vessel and the reactor pressure vessel internals, including the core support structures. Appendix 3D does not include ECPs for fuel, those ECPs can be found in Subsection 4.1.4.1. Calculational procedures used for the design of the reactor pressure vessel and the reactor pressure vessel internals, including the core support structures have reduced significantly with the introduction of computer programs. Calculational procedures or GE technical design procedures (TDPs) that may be used for ESBWR, except fuel analysis, are listed in the table below. TDPs are prepared to govern an analytical design process that warrants documentation to assure product quality and traceability and to standardize the design process.

| Procedure | Description |
|---|---|
| TDP-0143 Pressure-Temperature Curve Process | The method for calculating Pressure-Temperature (P-T) Curves for BWR reactor pressure vessels. The P-T curves define the pressure-temperature limits of operation and are used by an operating plant to meet 10CFR50 Appendix G. |

**DCD Impact**

No DCD changes will be made in response to this RAI.

# 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.

## 3D.2 FINE MOTION CONTROL ROD DRIVE

### 3D.2.1 (Deleted)

### 3D.2.2 ANSYS

#### *3D.2.2.1 Description*

See Subsection 3D.3.1.1.

#### *3D.2.2.2 Validation*

Hand calculations for theoretical equations published in literature.

#### *3D.2.2.3 Extent of Application*

This program is used for the elastic stress analysis and vibration analysis of the FMCRD. The program calculates elastic stresses for level D, faulted limits, but the ASME Code Section III Appendix F limits are not within the program. Evaluation for level D limits are performed by hand calculation.

## 3D.3 REACTOR PRESSURE VESSEL AND INTERNALS

Computer programs used in the analysis of the reactor pressure vessel, core support structures, and other safety class reactor internals are described below and in Subsection 4.1.4.1.

### 3D.3.1 ANSYS

#### *3D.3.1.1 Description*

The ANSYS computer program is a finite element 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 like thermal analysis, material non-linearities, contact analysis, etc.

### *3D.3.1.2 Validation*

~~ANSYS is maintained by ANSYS INC., located at 275 Technology Drive, Canonsburg, PA, 15317.~~ ANSYS applicability and validity is demonstrated by running a series of verification cases that exercise the elements and options used in the finite element code. The verification cases provided by ANSYS, Inc. are extracted from textbooks in which classical or theoretical solutions are published or can readily be obtained by simple hand calculations.

### *3D.3.1.3 Extent of Application*

This program is used for the elastic and inelastic stress analysis and vibration analysis of the reactor pressure vessel and internals. The extent and limitation may be determined by the verification cases that are performed to qualify ANSYS as an Approved Production Program that is verified and documented for design applications or for all technical activities used in developing design related information. The program calculates elastic and inelastic stresses for level D, faulted limits, but the ASME Code Section III Appendix F limits are not within the program. ~~Evaluations for level D limits are performed by hand calculation.~~The computer program may be used for calculating stress and cumulative usage factors for Class 1, 2 or 3 components, but the environmental effects are addressed outside ANSYS.

## 3D.3.2 Dynamic Stress Analysis of Axisymmetric Structures Under Arbitrary Loading - ASHSD2

### *3D.3.2.1 Description*

This FORTRAN program was created at the Earthquake Engineering Research Center, University of California, Berkeley. A finite element method is presented for the dynamic analysis of complex axisymmetric structures subjected to any arbitrary static or dynamic loading or base acceleration. The three-dimensional axisymmetric continuum is represented either as an axisymmetric thin shell or as a solid of revolution or as a combination of both. The axisymmetric shell is discretized as a series of frustrums of cones and the solid of revolution as triangular or quadrilateral "toroids" connected at their nodal point circles. Hamilton's variational principle is used to derive the equations of motion for this discrete structure. This leads to a mass matrix, stiffness matrix and load vectors that are all consistent with the assumed displacement field. But to minimize computer storage and execution time a diagonal mass matrix has been assumed in writing the computer program (with the input diagonalized accordingly by coordinate system transform). These equations of motion are solved numerically through the time domain either by direct integration or by modal superposition. In both cases, a step-by-step integration procedure was used.

### *3D.3.2.2 Validation*

Hand calculations using theoretical equations published in literature are performed to demonstrate the program's applicability and validity.

### *3D.3.2.3 Extent of Application*

This program will be used to calculate elastic stresses in the reactor pressure vessel and shroud support using axisymmetric shell and solid elements for axisymmetric and non-axisymmetric

static loading. The program calculates stresses for level D, faulted limits, but only ASME Code Section III elastic analysis defined in F1321.3 requirements.

### 3D.3.3 EVAST

#### *3D.3.3.1 Description*

This FORTRAN program was created by Babcock-Hitachi K.K. to calculate stress intensities, perform fatigue evaluation and evaluate thermal ratcheting.

#### *3D.3.3.2 Validation*

Hand calculations are performed to demonstrate the program's applicability and validity.

#### *3D.3.3.3 Extent of Application*

This program will be used to evaluate the primary stress intensities, fatigue and thermal ratcheting of the shroud support. Where fatigue curves are used, environmental effects are not considered.

### 3D.3.4 TACF

#### *3D.3.4.1 Description*

This FORTRAN program was created by Babcock-Hitachi K.K. to evaluate temperature distribution.

#### *3D.3.4.2 Validation*

Hand calculations using theoretical equations published in literature are performed to demonstrate the program's applicability and validity.

#### *3D.3.4.3 Extent of Application*

This program will be used to evaluate the steady and non-steady state axisymmetric thermal conduction of the reactor pressure vessel and shroud support.

### 3D.3.5 ABAQUS

#### *3D.3.5.1 Description*

This PC based program was created by ABAQUS, Inc. ABAQUS solves traditional implicit finite element analyses, such as static, dynamics, thermal, all powered with a wide range of contact and nonlinear material options. ABAQUS also has optional add-on and interface products with address design sensitivity analysis. ABAQUS enables a wide range of linear and nonlinear engineering simulations.

#### *3D.3.5.2 Validation*

Hand calculations using theoretical equations published in literature are performed to demonstrate the program's applicability and validity.

#### *3D.3.5.3 Extent of Application*

This program will be used for elastic and plastic stress analysis in addition to temperature distribution analysis of the reactor pressure vessel. The program calculates stresses for level D, faulted limits, but only ASME Code Section III elastic analysis defined in F1321.3 requirements.

### 3D.3.6 FEMFL

#### *3D.3.6.1 Description*

This FORTRAN program was created by Babcock-Hitachi K.K. to evaluate elastic stresses.

#### *3D.3.6.2 Validation*

Hand calculations using theoretical equations published in literature are performed to demonstrate the program's applicability and validity.

#### *3D.3.6.3 Extent of Application*

This program will be used for elastic stress analysis using axisymmetric structural shell and solid elements. The static loading may be axisymmetric or non-axisymmetric. The program is used in the analysis of the reactor pressure vessel. The program calculates stresses for level D, faulted limits, but only ASME Code Section III elastic analysis defined in F1321.3 requirements.

### 3D.3.7 SEISM03

#### *3D.3.7.1 Description*

See Subsection 4.1.4.1.3.

#### *3D.3.7.2 Validation*

Test cases analyzed and compared with previous design problems are performed to demonstrate the program's applicability and validity.

#### *3D.3.7.3 Extent of Application*

In addition to those applications outlined in Chapter 4, this program will be used to evaluate non-linear fuel lift analysis with spring, stop and friction elements.

## 3D.4 PIPING

### 3D.4.1 Piping Analysis Program – PISYS

#### *3D.4.1.1 Description*

PISYS is a computer code for analyzing piping systems subjected to both static and dynamic piping loads. Finite element models of a piping system formed by assembling stiffness matrices represent standard piping components. 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 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.

#### *3D.4.1.2 Validation*

The PISYS program has been benchmarked against NRC piping models. The results are documented in Reference 3D-1 for mode shapes and USMA options. The ISMA option has been validated against NUREG/CR-1677 (Reference 3D-2).

Subsequently, the PISYS07 program, which is used for ESBWR piping analysis, has been benchmarked against NUREG/CR-6049.

#### *3D.4.1.3 Extent of Application*

This program will be used for elastic stress analysis of piping systems.

### 3D.4.2 Component Analysis - ANSI7

#### *3D.4.2.1 Description*

ANSI7 is a computer code for calculating stresses and cumulative usage factors and to calculate combined stresses for service levels A, B, C and D loads in accordance with ASME Section III article NB-3650. The program includes environmental fatigue effects on fatigue curves according to DG-1144 and NUREG/CR-6909.

#### *3D.4.2.2 Validation*

Hand calculations are performed to demonstrate the program's applicability and validity.

#### *3D.4.2.3 Extent of Application*

This program will be used 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 levels A, B, C and D loads on piping supports and pipe-mounted equipment. The program calculates elastic stresses for level D, faulted limits, but the ASME Code Section III Appendix F requirements are not within the computer program.

### 3D.4.3 (Deleted)

### 3D.4.4 Dynamic Forcing Functions

#### *3D.4.4.1 Relief Valve Discharge Pipe Forces Computer Program – RVFOR*

##### 3D.4.4.1.1 Description

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 GE developed FORTRAN computer program computes the transient fluid mechanics and the resultant pipe forces using the method of characteristics.

##### 3D.4.4.1.2 Validation

Hand calculations and experimental tests are used to demonstrate the program's applicability and validity.

##### 3D.4.4.1.3 Extent of Application

This program will be used to calculate the time-dependent forces on the wall of the discharge pipe.

#### *3D.4.4.2 Turbine Stop Valve Closure – TSFOR*

##### 3D.4.4.2.1 Description

This GE developed FORTRAN program utilizes the method of characteristics to compute fluid momentum and pressure loads at each change in pipe section or direction due to the fast closure of the turbine stop valve.

##### 3D.4.4.2.2 Validation

Hand calculations are used to demonstrate the program's applicability and validity.

##### 3D.4.4.2.3 Extent of Application

The TSFOR program computes the time-history forcing function in the main steam piping due to turbine stop valve closure.

#### *3D.4.4.3 (Deleted)*

#### *3D.4.4.4 (Deleted)*

### 3D.4.5 (Deleted)

## 3D.4.6 Response Spectra Generation

### *3D.4.6.1 ERSIN Computer Program*

#### 3D.4.6.1.1 Description

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

#### 3D.4.6.1.2 Validation

Hand calculations and test cases analyzed are used to demonstrate the program's applicability and validity.

#### 3D.4.6.1.3 Extent of Application

Equipment Control Panels, Local Equipment Racks, Main Steam Isolation Valves (MSIVs), Safety Relief Valves (SRVs) and Hydraulic Control Units (HCUs) are some of the components ~~that are~~ analyzed using ERSIN computer code.

### *3D.4.6.2 RINEX Computer Program*

#### 3D.4.6.2.1 Description

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 non-constant 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.4.6.2.2 Validation

Hand calculations and test cases analyzed are used to demonstrate the program's applicability and validity.

#### 3D.4.6.2.3 Extent of Application

This program will be used to generate multiple damping spectra for piping.

### *3D.4.6.3 (Deleted)*

### *3D.4.6.4 (Deleted)*

### 3D.4.7 Piping Dynamic Analysis Program – PDA

#### *3D.4.7.1 Description*

PDA is a FORTRAN 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.4.7.2 Validation*

PDA output pipe whip restraint force and displacement benchmarked with ANSYS nonlinear time history analysis results.

#### *3D.4.7.3 Extent of Application*

Pipe break analyses.

### 3D.4.8 Thermal Transient Program – LION

#### *3D.4.8.1 Description*

LION is a FORTRAN program used to compute radial and axial thermal gradients in piping. The program calculates a time-history of $\Delta T_1$, $\Delta T_2$, Ta, and Tb (defined in ASME Code Section III, Subsection NB) for uniform and tapered pipe wall thickness.

#### *3D.4.8.2 Validation*

LION was compared to analytical results published in literature.

#### *3D.4.8.3 Extent of Application*

Pipe thermal analyses.

### 3D.4.9 Engineering Analysis System - ANSYS~~05~~

#### *3D.4.9.1 Description*

See Subsection 3D.3.1.1.

#### *3D.4.9.2 Validation*

See Subsection 3D.3.1.2.

#### *3D.4.9.3 Extent of Application*

This program is used to perform non-linear analysis of piping systems for time varying displacements and forces due to postulated pipe breaks. Also, this program is used to perform finite element analysis of pressure retaining components and civil structures against the loads and events postulated in the design specifications.

### 3D.4.10 Piping Analysis Program – EZPYP

#### *3D.4.10.1 Description*

EZPYP is a GE FORTRAN program that links the ANSI7 and PISYS program together. The EZPYP program can be used to run several PISYS cases by making user-specified changes to a basic PISYS pipe model. By controlling files and PISYS runs, the EZPYP program gives the analyst the capability to perform a complete piping analysis in one computer run.

#### *3D.4.10.2 Validation*

No calculations are performed in the program.

#### *3D.4.10.3 Extent of Application*

Sorting of data output from PISYS and ANSI7.

### 3D.4.11 (Deleted)

## 3D.5 PUMPS AND MOTORS

Following are the computer programs used in the dynamic analysis to assure the structural and functional integrity of the ESBWR pump and motor assemblies.

### 3D.5.1 Structural Analysis Program - SAP4G07

#### *3D.5.1.1 Description*

This program 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.5.1.2 Validation*

Hand calculations and test cases analyzed are used to demonstrate the program's applicability and validity.

### *3D.5.1.3 Extent of Application*

SAP4G07 is used to analyze the structural and functional integrity of the pump/motor systems.

## 3D.5.2 (Deleted)

## 3D.6 (DELETED)

## 3D.7 REFERENCES

3D-1 General Electric Co., "PISYS Analysis of NRC Benchmark Problems," NEDO-24210, August 1979.

3D-2 USNRC, "Piping Benchmark Problems Dynamic Analysis Independent Support Motion Response Spectrum Method," NUREG/CR-1677, August 1985.