Transactions of the 15th International Conference on
Structural Mechanics in Reactor Technology (SMiRT-15)
Seoul, Korea, August 15-20, 1999



# Use of Advanced Tools for Design and Analysis of CANDU 9 Equipment and Structures

S. K. W. Yu, S. Soliman and D. Dikic

*Atomic Energy of Canada Limited (AECL), Canada*

## ABSTRACT

Design optimization and detailed stress analysis has been done for all of the key CANDU 9 equipment, structures and components including the reactor building, the reactor structure assembly; the fuelling machine and carriage; and the nuclear system piping and supports. The effective use of powerful computers combined with state of the art analysis methods, such as the use of 3D CAD, has resulted in high quality work with significant time and cost savings. This paper will describe the effective use of these advanced design and analysis tools for the CANDU 9 NPP. The improvements in design and analysis work process will be illustrated with samples from the analysis and design work performed.

## INTRODUCTION

The CANDU 9 reactor design is based on the reference operating CANDU units in Canada and has a gross output capacity of 935MWe. Evolutionary improvements have been incorporated into the design to enhance constructability, safety, operability and maintenance.

The continuous evolution of the computer technology and software has opened the door to a revolution in the design and analysis. This has enabled engineers and designers to get their products with a short cycle time and meet tight project schedules. The effective use of advanced design software with user friendly Graphical Users Interface (GUI) combined with state of the art analysis methods has resulted in increased efficiency. By integrating the design and analysis processes, such as the use of the design database from the 3D CADD models as design input and the feedback of the analysis results to further design optimization, has resulted in high quality work with significant time and cost savings. Continuous advancement in engineering tools has also led to faster evolution of the engineering products.

The most significant enhancement of the design engineering methodology is the integration of design and analysis. Powerful and affordable high performance workstations enable the designer to optimize the design faster. CAD systems and analysis software were linked and the same data bases were utilized.

## STRUCTURAL SEISMIC ANALYSIS

Finite element computer models were used in performing the seismic analyses of the CANDU 9 Reactor and Fuel Handling structures. Seismic models for the Reactor Assembly and the Fuelling Machine carriage were developed. Each model was made up of a number of Macros that were linked together by a main program. All Macros used to generate solid models were developed using Boolean functions and a parametric design language. A Graphic User Interface (GUI) was used to view the resulting seismic responses including animation of the outputs; such as mode shapes, frequencies, deformation and stress patterns of the structure. The introduction of macros and parametric languages coupled with the powerful Graphic User Interface (GUI) enables designers to implement efficient product development cycles. The finite element programs were equipped with powerful solvers that were twenty times faster than the traditional frontal and banded solvers. This feature helped the analyst and designer to determine weak points of the design and find fast and efficient solutions to achieve design optimization. The modified design could then be incorporated in the final analysis model by modifying only the input Macros under consideration.

## Seismic Analysis of the Calandria and Shield Tank Assembly (CSTA)

The CSTA was made up of the Calandria Vessel (CV), two End Shields (ES), the Shield Tank (ST), supports and embedment. The components were joined together to form an integral multi compartment structure. The CV consists of a horizontal, cylindrical, single walled, stepped shell, closed by a tube sheet at each end and spanned horizontally by 480 calandria tubes. Flexible annular plates welded to the main shell and to the sub-shell form the shell wall at each step, and accommodate differential thermal expansion between the calandria tubes and calandria shell. The CV was filled with heavy water. The two ES were integral part of the CSTA. Each ES consists of a horizontal, cylindrical shell, closed at each end by a tube sheet and spanned by 480 lattice tubes. The ES was filled with light water and carbon steel balls. The ST is a cylindrical, welded, carbon steel shell enclosed at each end with end walls. The CV and ES were installed inside the shield tank to form an integral structure. The ST was filled with light water. The ST was supported by plate and shell elements that were attached to the perimeter of the ST end walls. The supports were then joined to embedment rings that were set into the concrete of the reactor vault end walls.

CSTA seismic analysis was a relatively complex process because of the intricate geometry with many different discontinuities, and due to the hydraulic attached mass which follows the structure during its vibration. In order to reasonably simulate an accurate behavior of the reactor structure assembly, detailed finite element models were generated from the CAD solid model using EMS Intergraph [1] and ANSYS [2] codes. The attached masses, which account for the fluid masses affected by the vibrating structure, were calculated. These masses of the attached components supported partly or totally by the assembly which includes piping, reactivity control units, end fittings, etc. were also considered in the analysis.

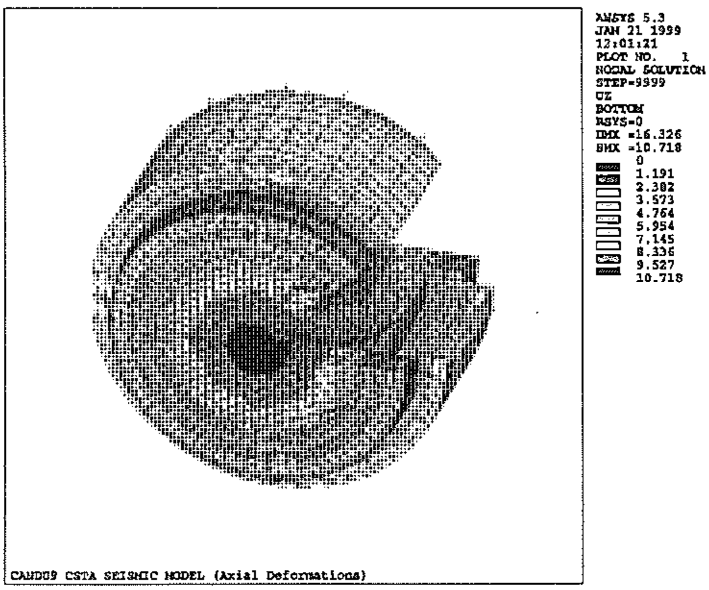



**Figure 1 CANDU 9 CSTA Seismic Model (Axial Deformations)**

Several ANSYS finite element types such as flat plate STIFF63, plastic shell STIFF43, elastic BEAM4 and the mass STIFF21 were used to model different components of the assembly. CSTA model was supported at the vault wall and the stiffness and mass effect of the vault were considered in the analysis. Guyan reduction dynamic condensation method was then used to condense the behavior of the model at certain nodes and directions that were called the master degrees of freedom. The HBI

(Householder-Bisection-inverse Iteration) modal extraction technique as available in the ANSYS code was used to extract enough frequencies and corresponding modes of vibration to represent the dynamic behavior of the model. Spectrum analysis was finally performed to calculate the structure response for a given FRS (Floor Response Spectrum). Figure 1 shows the finite element model and deformed shape of the CSTA for FRS based on a Peak Ground Acceleration of 0.2g.

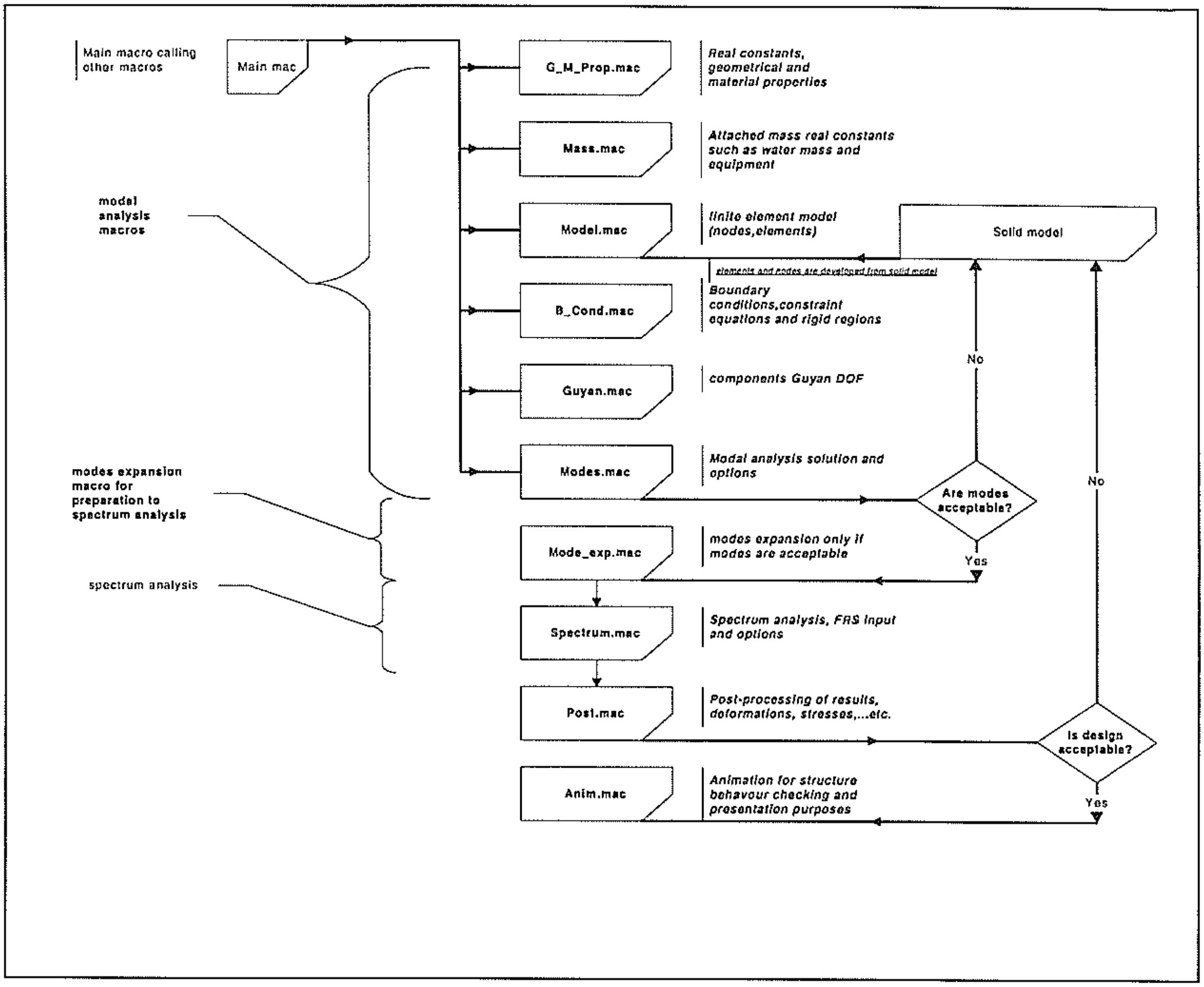



**Figure 2 CSTA Seismic Analysis Process and Macros**

Analysis process is shown in the flow chart given in Figure 2. Thus, the analysis was performed in three phases; modal analysis, modes expansion and spectrum analysis. There were eleven macros, each with a unique function. Each component in the CSTA structure was assigned a unique real and material number in the property macro. Post macro called each component by its material number and processed the results in similar fashion resulting in well organized output in a remarkably short time. Model macro had processed the elements and the nodes from the solid model using EMS finite element interface package IFEM, [1]. Guyan macro defines the master degrees of freedom and the modes macro lists the modal analysis solution and options. As seen in Figure 2 after the modal analysis was performed, the resulting frequencies and mode shapes can be investigated. If the response was acceptable, the analysis proceeded to the next mode expansion, otherwise the process was repeated including the interface between the finite element and solid models.

Third phase of the analysis was the spectrum analysis using the specified FRS where the stresses, deformations, accelerations and other structural responses were calculated and evaluated. If the structure response was not acceptable, the whole process was repeated, otherwise the analysis is documented and presented to the reviewers for final approval. It should be pointed out that the arrangement given in Figure 2 has many benefits. The analysis can be repeated for any set of loading such as higher seismicity by changing only the spectrum macro and repeating the analysis. Parametric and sensitivity analyses were performed by changing material properties, boundary conditions, loads, etc. With minor input changes, the same macros can be used for static and thermal analysis.

## Seismic Analysis of the Fueling Machine Carriage and Head

The CANDU 9 Fuelling Machine (F/M) carriage supports and transports the F/M head from the reactor face through the maintenance lock to the service ports. It also enables the F/M head to align with any required Fuel Channel (F/C) or service port. Each F/M carriage consists of seven main components; a two wheel base, a lower turntable, two columns, an upper guide beam, a U-frame and elevator, a cart and upper turntable, and cradle and F/M head. Seismic analysis of the fuelling machine involved the development of Lumped Mass Models (LMM) for all the seven sub-assemblies of the F/M structure, the F/C and the CSTA. This analysis accounts for different configurations in the unattached case where the F/M was disconnected to the reactor face and the attached case where the F/M was connected to the reactor.

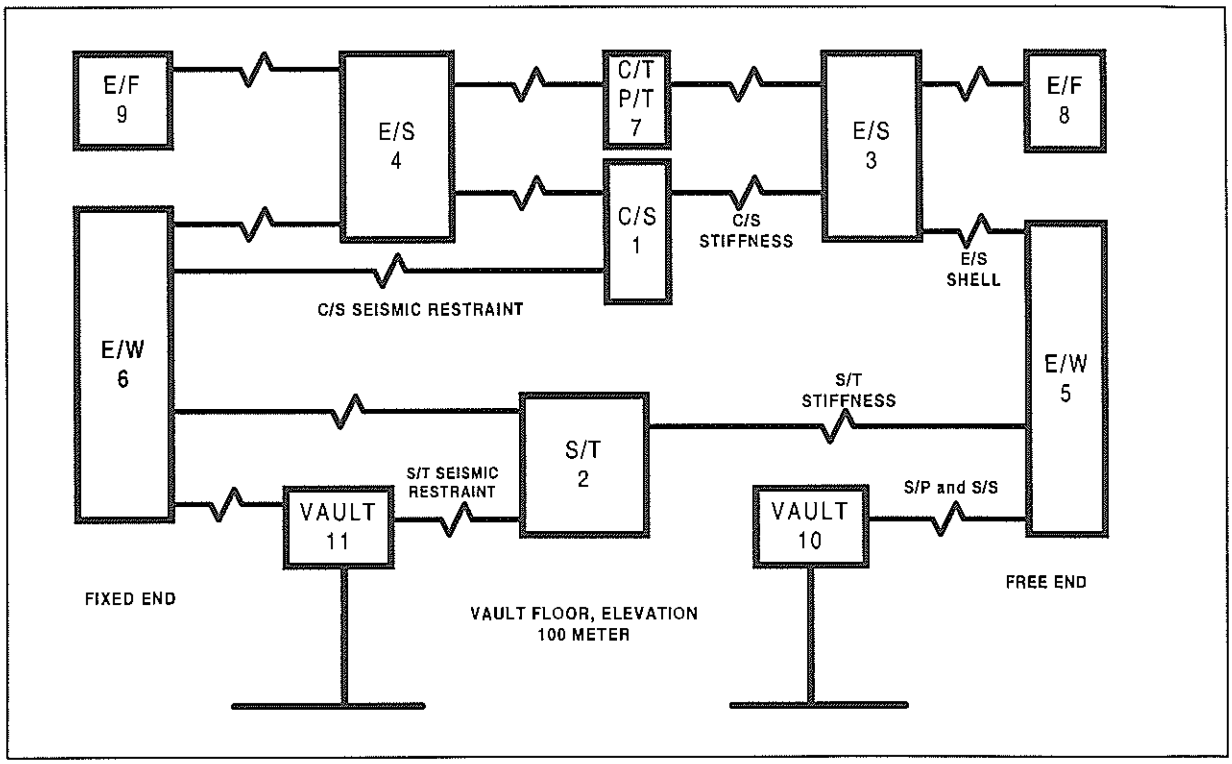



**Figure 3 CSTA Lumped Mass Model**

Attached case represents the on-reactor scenario where two F/Ms were clamped on the end fittings of a given F/C on both sides of the reactor. Mass and geometrical properties for the F/M components were extracted either from the EMS solid model or by building separate models in ANSYS using Boolean functions and operations. These solid models were used as a means of interface between the designer and the analyst and allow for visual inspection of the components. Also, they facilitate the understanding of the design of various components and how they fit together. Finite element models and seismic Lumped Mass Models (LMM) were developed for each component of the CSTA, F/C, and F/M carriage using ANSYS computer code. A typical lumped mass model of the calandria shield tank assembly is shown in Figure 3.

The models were used to conduct seismic analyses for the CANDU 9 fuelling machine carriage structure. Each model was made up of a number of macros linked together by a main program. Each macro has a single function and forms a part of the assembled finite element model. All macros were built parametrically, which allows for an efficient implementation of any design modifications, since any design change affects specific input macros and not all models. As with the CSTA analysis, the F/M analysis was performed in three phases; the modal analysis, mode expansion and the spectrum analysis. Modal analysis was carried out to calculate the natural frequencies and mode shapes of the structure. Mode shape for any frequency less than or equal to 33 Hz was animated and reviewed to ensure that the overall motion of the structure was acceptable. Typical values of natural frequencies of the attached cases are given in Table 1. The spectrum analysis was conducted using FRS values calculated for the vault walls at the highest elevation where the F/M was supported. Sensitivity analyses were conducted on all parts of the carriage, and the stiffness of various members was modified (subject to designer's approval) to obtain the desirable dynamic behavior. After a few iterations between design and analysis, an optimized F/M design was achieved.

## THE PIPING AND SUPPORT DESIGN AND ANALYSIS TOOLS

A powerful 3D CADDS system has been used for the CANDU 9 nuclear power plant (NPP) design including the three dimensional modeling of the piping system, the concrete and steel structures. AECL has developed additional software design tools to comprehensively integrate the analysis, piping and support design activities.[3] The Piping System Analysis Interface (PSAI) tools will reliably transfer piping system data from the 3D CADDS model as inputs for the analysis software in performing thermal-hydraulic and stress analysis. The Pipe Support Design System (PSDS) tool will permit selections from standard pipe support designs and will perform analysis checks against the loading input and sizing of the auxiliary steel members.

With these tools the engineering team can reliably transfer model data to the design/analysis tools via the CADDS interfacing tools to perform thermal-hydraulic, and stress analysis calculation to design auxiliary steel members and piping supports. In addition, these basic tools are complemented with archival tools, graphical and data reporting packages, analysis post processors, etc.

### The Piping System Analysis Interface (PSAI) Tool

The piping system analysis interface tool is a multifaceted graphical and alpha-numerical code capable of reading the 3D CADDS 'neutral file' which is an alpha-numerical rendition of both the graphic data and the 3D CADDS database. It is a file containing the intelligence of the equipment and piping models. The piping analysis interface tool is capable of extracting all the information it needs from the CADDS 'neutral file' and building input data sets for submission to the various analysis and design tools of the integrated design environment. These include the thermal-hydraulic design and verification tools, the piping stress analysis solver and its post processor capable of performing code evaluations.

The piping analysis interface tool is the mortar, as it were, of the integrated design environment in the process engineering area. It provides two major benefits from the point of view of the economy and speed of operation in a design milieu: automation and quality control. Automation, since it eliminates labour intensive tasks and manual intervention in the creation of analysis data sets and quality, because it eliminates the possibility of input errors and ensures data consistency across the various disciplines and tools of the integrated environment

The piping analysis interface tool provides the required links between the CADD system and the piping stress analysis program ADLPIPE. They are capable of managing data stored in the task database and in the associated computational model and they provide graphical support in any interaction with the users.

The improved piping analysis process includes: to select the pipe runs to be extracted from the 3D CADDS environment; to modify or fine tune the engineering elements of the extracted model; and to generate analysis data sets in the format required by the analysis solvers and to initiate analysis execution.

The graphical capability of the piping interface tool provides an on-line graphical representation of the pipe run, similar to the traditional stress isometric drawing.

## The Piping Support Design System (PSDS) Tool

The piping support design system (PSDS) tool interfaces with the CADDS suite of plant design system software and with the piping stress analysis code. From the former it can activate the relevant 3D model files using the PSDS graphical and database system. From the latter it reads the files containing piping stresses and load results at each restraint. It then selects and extracts relevant piping support load results for each of the service conditions. Following this automatic data transfer the operator can then easily model in the CADDS three-dimensional environment a whole variety of possible combinations of elements and structures to graphically represent a pipe support. The designer uses the system library of parametric steel members as well as a comprehensive catalogue of piping support elements and hanger components to facilitate the task. In most cases, the software can semi-automatically place load-rated components and evaluate the piping support against its rated loads. Where rated loads are not available, stress evaluation is performed to ASME Section III NF or ANSI B31.1

For non-standard components, PSDS tool controls steel member capacity and links the design data to an integrated stress analysis solver capable of verifying members and fasteners in special assemblies. PSDS tool designs and checks welds according to the ASME and AISC design rules.

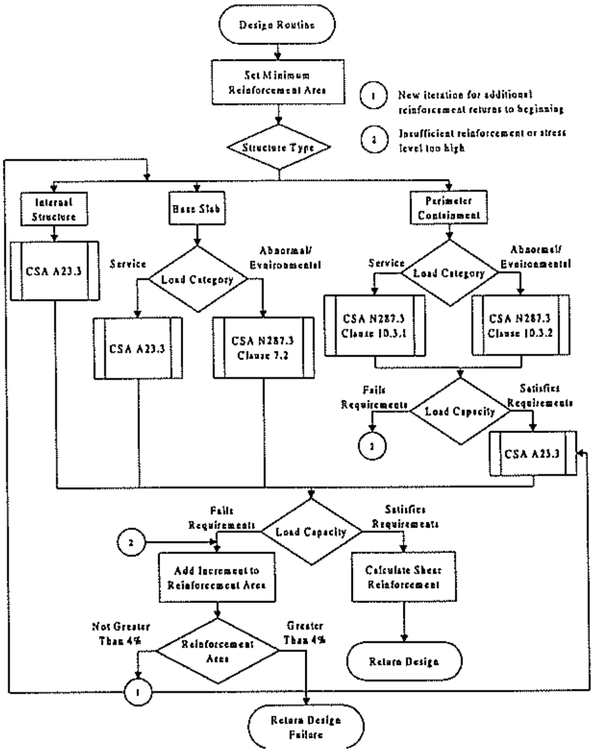
If the designer attempts to select incompatible components, PSDS possesses enough intelligence to reject irreconcilable combinations of catalogue elements. The software also possesses good reporting capabilities. It can print a stress summary, prepare 2D drawing, and extract Bills of Material. Finally PSDS is integrally connected to the 3D CADD system clash check capability and hence guarantees a total interference free design.

## CONCRETE DESIGN SOFTWARE

Design of reactor building structure, including internal structure, has been conducted using an in house developed software. The software program reads results of analysis conducted by ANSYS finite elements program, and creates the design load (factors based on the CSA A23.3-M94 or N287.3-M93). Depending on the part of the reactor building structure type being designed, the software calculations are either based on service load level requirements (allowable stress level in prestressed concrete wall) or limited strains level (as defined by N287.3-M94) or requirements of the CSA A23.3-M94 Standard. Additional strength investigation for pre-stressed concrete structure has been conducted to confirm safety factor based on limit state design methodology as required by code. The flow chart of design routine is shown in Figure 4.

With an assumed reinforcing and depending on structure type and load category, the program calculates the section load capacity (service and limit states, or limit states only) and then checks whether the section has a capacity higher than the applied load. If not, the program automatically increases the reinforcing amount and calculates again the capacity until the requirements for larger capacity than design is met or reinforcing is greater than allowable by code requirements. A sample of the interaction curve for abnormal / environmental load category of base slab is shown in Figure 5. Similar interaction curves have been calculated for each type of structure depending on code requirements and load category. The samples of stress/strain limits applicable to this portion of the interaction curve are shown in Figures 6 and 7.

**Figure 4 Concrete Design Flow Chart**




The software program can provide the design of a single structure or a group of structures analyzed by ANSYS program. If the analysis work needs to be corrected or additional design is required to analyze higher or modified load level, it is not necessary to repeat the whole analysis process. It can be done by changing the load multiplication factor in the load combination database file, and then running the design program again. The software program will print out for each element to be designed, specified load category, design load and result of design. In the case of prestressed structure, it will also provide serviceable load with reinforcing strength, and load combination based on limit state design methodology.

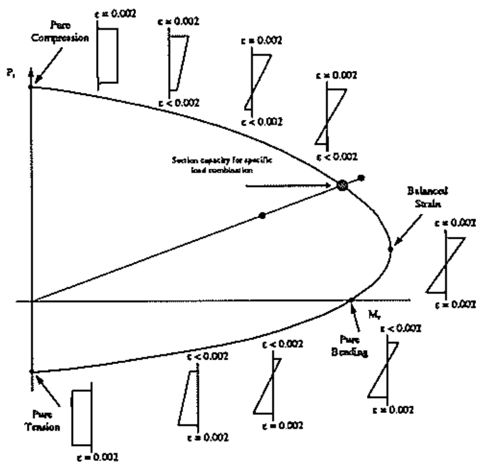



**Figure 5 Interaction Curve**

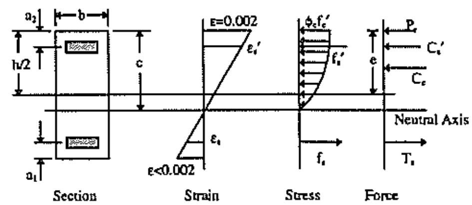
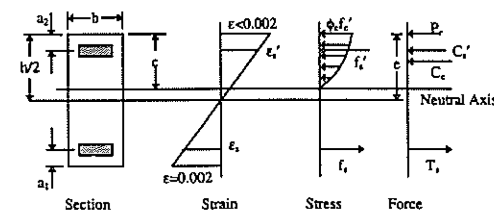



**Figure 6 Case of Compression strain of 0.002**




**F**

**igure 7 Case of tension strain of 0.002**

## CONCLUSION

Key CANDU 9 systems, structures and equipment have been designed and analyzed using advanced engineering tools and state-of- the art analysis methodology. The engineering design and analysis process has been improved by using the design database in 3 dimensional computer aided design models in the analysis methodology, as shown in the CANDU 9 reactor and fuel handling equipment analysis. The synergy between the analysts and designers has resulted in high quality work with much confidence in the optimized product. AECL has also developed additional software either to integrate the design and analysis process such as the piping system analysis interface software. In addition, engineering tools have been developed to perform routine design and analysis faster such as that used for pipe support design and analysis, and for concrete design and analysis.

## REFERENCES


1. Intergraph Engineering Modeling System (EMS), REVISION 2.3 , January,1996
2. ANSYS , Multi-purpose finite element program, Engineering analysis system, Rev.5.3
3. F.Nuzzo, S.K.W.Yu and K.R.Hedges, "Mechanical Engineering Utilizing Advanced Engineering Tools for the CANDU 9 Project", Annual Meeting of the Korean Society of Mechnical Engineers, Seoul, Korea, 1998 May.


**Table 1 Attached Case Scenario-Modal Analysis, frequencies and mode shapes**

| Mode No. | Frequency Hz. | Direction and Comments |
|---|---|---|
| 1 and 2 | 3.7917 , 3.7966 | Yaw of F/M head and Cradle (for F/M # 1 and F/M # 2). |
| 3 and 4 | 3.8055 , 3.9271 | Pitch of F/M head and Cradle (for F/M # 1 and F/M # 2). |
| 5 and 6 | 4.3259 , 4.3486 | Single F/C (X and Y directions). |
| 7 | 4.9879 | Z movement of the F/M head (F/M # 2, at the free end of the F/C) coupled with pitch of the second F/M (at the fixed end of the F/C) |
| 8 and 9 | 5.9298 , 5.9308 | 480 F/C (X and Y directions), Reactor Model. |
| 10* | 8.3835 | Z movement of the whole reactor components |
| 11and 12 | 9.0124 , 9.0128 | X movement of U frames (at the floating ends). |
| 13and 14 | 10.060 , 10.060 | Y movement (vertical) of upper guide beams. |
| 15 | 10.629 | movement of the F/M head (F/M # 1, at the fixed end of the F/C). |
| 16 to 19 | ≈ 11.5 | 480 E/F's (in X and Y directions). |
| 20and 21 | 12.704 , 12.764 | Deflection of the column in Z direction (for F/M # 1 and F/M # 2). |
| 22 to 29 | ≈ 16.6 | Y and X movement of the pipe support beams for the upper guide beam. |
| 30 | 17.798 | Z movement of the equivalent 480 F/C's. |
| 31and 32 | 20.597 , 20.987 | Yaw motion of the columns around Y axes (for F/M # 1 and F/M # 2). |
| 33 | 21.143 | C/S, movement is X direction |
| 34and 35 | 21.775 , 21.796 | Y movement of the carriage base (for F/M # 1 and F/M # 2). |
| 36 | 21.975 | C/S, movement in Y direction |
| 37and 38 | 24.074 , 24.079 | X movement of the columns at the floating end of the U frames. |
| 39and 40 | 25.762 , 25.787 | Z movement of the Carriage base (free end) |
| 41 | 26.717 | C/S and S/T axial movement (out of phase) |
| 42 | 28.965 | C/S movement in Z direction |
| 43and 44 | 29.262 , 29.643 | Swing of the U frames around X axis for F/M # 1 and F/M # 2). |
| 45and 46 | 30.394 , 30.555 | X movement of the columns (not at the floating ends) |
| 47and 48 | 32.005 , 32.419 | S/T movement is X and Y direction |