SECTION 3 Stress Analysis of Reactor Structure Assembly (RSA)

3.1 The Role of Analysis in the Design Process - Preliminary design

As noted above, the fundamental purpose in performing stress analyses is to ensure the design meets the requirements for strength, stiffness and durability. But as noted in the Outline of the Design Process, in Session 1, in the early stages of design, approximate *preliminary* calculations are made to provide guidance to the designer in determining acceptable sizes and configurations. This is especially useful where compromises must be made, or where alternatives are being considered. At this point, the structures and components may be represented by simple elements to permit rapid calculation and assessment, and only approximate values are sought. The designer will assess these results and apply judgement (and sometimes, speculation) to arrive at a satisfactory concept.

Figures 3-1, 3-2, 3-3 & 3-4 show an example of this, where **Figure 3-1** shows the cross-section of a proposed support system for a large CANDU reactor. The Shield Tank End Wall is a double walled, webbed ring surrounding the end shield. It is rigidly joined to the end shield perimeter by welding to the tube sheets. It is flexibly joined to the vault wall embedment outside it by means of an annular shell. The shell transmits axial and transverse loads rigidly, but is flexible for radial thermal expansion. **Figure 3-2** shows a *segment* of the End Wall, comprised of inner and outer plates and a web, as if the End Wall were comprised of a series of such radial, cantilevered I-beams, joined edge to edge. The bending stiffness of this segment was calculated by simple beam formulae, then the "dishing" stiffness of the entire End Wall was determined by multiplying by the number of segments in the entire Wall. This stiffness was then used to make a comparison of two different support schemes. The comparison is shown in **Figure 3-3.** On the left is shown the design as described above, which can be seen to behave as a simply-supported plate. On the right is an alternative design, where the End Wall is rigidly attached to the vault wall outside it, by welding to the re-enforcing bars. This design behaves like a "built-in" plate. The formulae for "dishing" deflection of simple flat plates are shown below the figures, and it is clear the built-in version has 1/4 the deflection, meaning it is four times as stiff. Furthermore, observation of the deflected shape reveals there is an inflection very close to the end shield/ end wall joint (ie, where the direction of bending reverses, which means there is zero bending moment acting, only pure axial load). Accordingly, an annular plate can be inserted at that diameter without changing the system stiffness. This design is shown on **Figure 3-4**, and would be a superior design for seismic design purposes. It was thought the inclusion of the annular shell would minimize the radial thermal expansion to be transmitted into the re-enforced concrete vault wall. Unfortunately, it was not found to be acceptable; although the radial thermal loads were easily taken into the re-enforcing bars, the corresponding "hoop" stress acting tangentially in the concrete was excessive.

It is also noted that, with the very great capability of modern modelling and analysis programs, and the great capacity and speed of modern computers, it is relatively easy and economical to analyse two or three preliminary design options by the more exact methods at the outset. Furthermore, the possibility of immediately selecting one of those options provides an incentive to choose this route. However, since these analyses are usually performed by the analyst on essentially automatic machines, the designer becomes less directly involved, and clues which might provide better understanding, from which more optimal concepts might arise, may not be seen.

When the designer believes the design is complete, the *formal* analyses are done to confirm for the record that the design fully meets all the requirements. Occasionally, some shortfalls may be found, necessitating further adjustments to the design. On the other hand, it is often simpler to refine the analysis methodology or assumptions, to obtain a more precise evaluation of the design. This permits reducing excessive conservatism included in generous safety margins included to allow for imprecision in the method, or uncertainties in data values initially assumed. This process is often iterative, where the results of an early assessment indicate which factors are dominant and which are less influential, and it accordingly indicates the most effective adjustment to make .

3.2 Design by Rule and Design by Analysis.

Design by Rule

In designing for pressure retaining items, a component may be designed by selecting the configuration, component, joint details, and material thickness specified in a set of rules laid out in the ASME code. This approach generally applies to design of Class 3 systems (where the requirements of ASME Section III, Division 1, Section ND-3000 apply) and Class 6 systems (where the requirements of CSA standard B51 apply). (Previously, ASME Section VIII, for non-nuclear vessels, was also used for systems having low nuclear content in the fluid enclosed.) The designer only makes some simple calculations to verify compliance or determine minimum acceptable thicknesses etc, and he may not determine actual stresses for his design. As noted above, the particular rules to be applied depend on the code class, and the nature and function of the item. Note that, if the "design by rule" approach is used, the designer only determines that the design is acceptable for that specified service; he will obtain no useful information about the state of stresses in that item. Any calculated stress is expressed as simply direct stress and compared to an allowable stress S_A = the lesser of: 5/8 yield strength; or 1/4 ultimate strength. This is shown in **Figure 3-5**. Because of the nature of the method, this approach is conservative, and results in lower allowable stress values being applied; and this is reflected also in the applicable materials, manufacturing processes and inspection.

Design by Analysis

For more demanding design situations, once the configuration has been essentially finalized, detail stress analysis is done, to confirm the design meets the standards and codes. This approach must be applied to Class 1 and Class 2 systems, (where the requirements of ASME Section III, Division 1, Section NB-3000 and ND-3000 apply, respectively). For these items, the calculated value of resulting *stress intensity* P is compared to the allowable value. Stress intensity is determined by first calculating the *principal stresses* acting on an element, then finding the *difference* between the *algebraically largest and smallest principal stresses*. Because the method of analysis is more precise, and corresponding manufacturing and inspection methods are more rigorous, the *allowable stress intensity* S_M is a less conservatively factored fraction of the material's yield strength or ultimate strength than applies for S_A.

The allowable stress applied depends on the category of loading being applied. Loading cases are generally grouped as Design loads and Operating loads, as shown on **Figures 3-6 & 3-7.** Specifically, CSA N285.0 (Clause 7) and ASME define service levels A through D, depending on the probability of occurrence and severity of the loads, as shown in **Figure 3-8.** This terminology was adopted in 1974, and a comparison to the previous version is shown in **Figure 3-9**. For each category of service load, the allowable value used depends the nature of the applied stresses, ie, whether they are purely tensile (membrane) or including bending. Furthermore, it depends also on whether the stress is due to primary (externally applied) load or secondary (self-induced because of thermal restraint). **Figures 3-10, 3-11 & 3-12** show these alternative cases.

3.3 The Analysis Process

The first step in the analysis process is to list and categorize the several key *loading cases* which result from the several *operating conditions*. These include: normal steady state operation, and normal start-up and shutdown transients (Level A); frequent off-normal or upset conditions or transients (Level B); and unusual emergency and accident conditions (Levels C & D). Unusual conditions applied during manufacture, installation or pressure testing must also be included. A typical set of design cases, for a CANDU 6, are shown in **Appendix I**.

For each defined *loading case* that the reactor structure is subjected to, the total load applied is a *combination* of the *basic loadings*, ie, the weight of structural members and fluids and other components enclosed and supported, plus pressure and buoyancy loads, plus loads due to thermal restraint, plus dynamic loads due to acceleration acting on the mass of the components and contents. Weight loads will be the same for all cases, but other basic loads (eg, temperatures, pressures, seismic loads) will take on different values for the different cases.

The net stresses resulting from each of these *combinations* are the sums of the stresses for each of the basic loads for this loading case. The method of analysis used therefore, is to calculate the stresses for a *unit load value* for each basic load, using a Finite Element Model (FEM), or a classical stress analysis formula if appropriate) and then multiply all the stresses by the value of that load, for each loading case. The net stress for that loading case, is therefore the sum of the stresses for all the factored basic loads. Typical sets of stresses for each load and for several design cases are shown in **Appendix II**, for a CANDU 6 reactor structure assembly.

3.3 Finite Element Modelling (FEM) & Analysis

3.3.1 The Concept of Finite Element Analysis

Stresses in simple elements are readily determined by applying fundamental stress analysis equations for such basic elements as struts in tension or compression, bars in bending, shear or torsion, pressurized cylinders under hoop tension, and wires and strips under catenary tension. Complex structures are assumed to be made up of such elements, joined together such as to transfer parts of the external load between themselves. The stresses throughout such structures can be determined if the patterns of internal load transfer are known.

In structures comprised of simple, pin-jointed struts, and which do not have redundant members, these loads on internal members are found by applying the principals of equilibrium of forces at each joint. **Figure 3-13** shows such a simple structure. The loads are in the same directions as the struts so that the horizontal and vertical components of each strut load can be determined; and equations can be written for equilibrated horizontal and vertical sets of components. When all joints have been assessed, a set of consistent simultaneous equations will have been prepared, which will include all the unknown loads. All the external loads will have been found by applying equilibrium conditions to the overall structure. This set of simultaneous equations can then be solved by fundamental mathematical methods, to provide values for all unknowns..

However, if the system has added struts or added support points, it has redundant load paths, and the above simple approach doesn't work. The system will yield fewer equations than the number of unknowns. **Figure 3-14** shows such a structure. In this case, the patterns of load transfer can be determined by first calculating the *elastic deformation* each element will undergo as a result of the load acting on it, and then applying the principals of continuity between the elements. That is, if one considers any one joint, it is clear that neighbouring elements must endure exactly the same horizontal and vertical components of deformation at that point, because of their being joined. If the structure is comprised totally of pin-jointed struts, there will be one unknown load and one unknown deflection for each element. There will be known values at the external boundaries where the external loads are applied, and where the attachment points will have zero (or specified) deflection. These two sets of simultaneous equations can then be solved together to provide values for all unknowns.

Furthermore, this same approach can be applied when the structure is comprised of other kinds of elements, such as beams and torsion bars, ie, when the joints are rigid rather than pinned. In this case, the continuity conditions at each joint will include equilibrium for bending moments and torques as well as forces in all directions, and there will be continuity for angular deflections as well as for linear ones. The method is the same, but there will be a much larger number of simultaneous equations to solve, making the solution a very onerous task. This is the basis for Finite Element Modelling (FEM) and Analysis. It is particularly useful in analysing large trusses, shell structures and vessels, especially where they have discontinuities, such as nozzles, changes of thickness or discrete support points. These structures can be arbitrarily sub-divided into a large number of beam or plate elements and analysed with great precision. Each joint on each element might possibly have six kinds of load applied, causing six corresponding kinds of deflection. ie, it can have forces in three directions and torques or moments acting around each of the three axes at each joint, causing three corresponding linear deflections and three corresponding angles of twist/bend there. The joints are called "nodes", and the kinds of deflection are called "degrees of freedom". For some structures, only some of the forces and motions at a node need be represented, especially at boundaries and points of loading, and the model will have fewer degrees of freedom, and therefore reduced steps in the computations. Nonetheless, even simplified models for reactor structures can have many thousands of nodes and many thousands of degrees of freedom.

The method can easily produce an overwhelming amount of calculation, and can become limited by the ability to handle the large numbers of element loads and deflections. It also produces an enormous amount of information to be assessed, especially when a number of loading cases has to be assessed, each possibly including several sets of loads. For example, in dynamic load cases, such as seismic loading, each element carries an inertia load in addition to its nodal loads, as its own mass is subjected to acceleration. Fortunately, the equations can be arranged in a consistent format, and lend themselves to the technique of *matrix mathematics*, where the rules of manipulation of columns and rows of coefficients leads to more efficient solution processes. By carefully and systematically entering node and element data in tabulated formats, assembly of the matrices can be organized into routine processes, to speed up the work and minimize entry errors. Furthermore, such processes are readily handled by computer programs, so that structures with large numbers of complex elements can be efficiently dealt with.

Software programs such as Stardyne and Ansys have been commercially developed to analyse very large and complex finite element models (FEMs), represented by large numbers of nodes and many kinds of complex elements. Techniques were also developed to break a very large structure into a number of sub-structures, whereby each portion can be reduced to a simplified representation and re-assembled with other condensed portions, into a simpler 'global' model representing the entire assembly. This global model is then more amenable to analysis. The original sub-structure portions are each analysed in the computer to provide the simplified sub-structure versions, which will display similar mass and stiffness characteristics as the originals, at the selected reduced number of nodes. However, today's modern computers have the capacity to deal directly with very large, complex models having complex load conditions, and modern software programs such as Patran can construct the FEM automatically from prescribed configurational descriptions and overall structural dimensions.

Figures 3-15 & 3-16 show FEMs for typical CANDU 6 reactor assembly items. Modern computers also have sufficient capacity and speed to produce the analysis outputs very quickly. **Appendix III** shows the parametric calculations and data tabulations compiled in modal analysis using the FEM for a CANDU reactor structure assembly. It shows the large number of modes and their participation factors, as well as the modal masses of the major structural components. **Figure 3-17** shows the seismic FEM for the CANDU 6 calandria structure assembly. In addition to overwhelming volumes of output data listings, modern computer programs can produce graphical displays in colour, and in a number of formats, to show deflections and stresses pictorially, which greatly ease the task of handling and assessing the enormous volume of output data produced. **Figures 3-18 & 3-19** show some typical output displays.