Transactions, SMiRT-22
San Francisco, California, USA - August 18-23, 2013
Division VI Design and Construction Issues

Integrated Analysis and Design of Nuclear Power Plant Concrete Structures

Peter R. Barrett, P.E. [1] and Joseph K. Harrold, P.E. [2]

[1]Vice President, CAE Associates, Middlebury, CT (barrett@caeai.com)
[2] Engr. Supervisor / Advisory Engr, AREVA Inc., Charlotte, NC (joseph.harrold@areva.com)

ABSTRACT

Nuclear power plants consist of multiple geometrically complex concrete structures. It is common practice for several of the primary structures to be concentrated onto a common basemat (i.e., nuclear island) requiring a single comprehensive finite element model. While this practice results in several benefits, it can also amplify the difficulty in performing analysis and design activities. Since the various structures forming the nuclear island typically fall under the jurisdiction of different design codes, it is not unusual for the overall number of load combination permutations to be large (e.g., over ten thousand). The number of load combinations, coupled with the typically large number of elements (e.g., tens of thousands) results in a vast amount of data to be processed during concrete design activities.

Detailed stress analysis and local concrete member reinforcement design are two major tasks for structural engineers evaluating these facilities. These tasks are often separated between different engineering groups and computer codes making it difficult to implement and synchronize design changes between departments. This difficulty is compounded by the fact that analysis of complex nuclear structures requires large scale FEA software (Example - ANSYS 2013) that was not developed specifically for concrete design activities. This paper presents both current and recommended future methodologies for integrating these two tasks. The paper also illustrates how better trained engineers and smart automation of data development and transfer from analysis to design with an ultimate goal of enveloping constructible reinforcement recommendations that meet design codes (ASME, ACI349) will shorten the analysis/design cycle.

ENGINEER TRAINING

One of the critical items in creating a better connection between analysis and design is through the education of the engineers that are bridging the two disciplines. Often different groups perform analysis and design where the analysis group is focused on the finite element simulations while the design engineers are focused on design details. The primary data passed from the analysis to design group consists of forces and moments of critical members for a large suite of design cases. In the past finite element models were much simpler and less loading conditions were considered, hence the quantity of data passed between engineering divisions was manageable. Each group worked more or less independently of each other and neither had to be an expert in the others job. This is no longer the case.

Today with the increased computational resources we have significantly larger finite element models that provide a high level of fidelity to the computer simulations. We also have the ability to simulate an entire earthquake time history including for example nonlinear basemat-to-soil contact in real time where thousands of time points of analysis results are available at each active degree of freedom in the model. The downside is these analyses create terabytes of data. This data can contain detailed stress results for every element integration point for every load combination which for a complex structure can consist of over a million elements and thousands of load combinations. How to effectively deal with this data is a major problem that requires joint skills from both the analysis and design teams.

The analysts are well versed in creating the analysis results and verifying that the simulation accurately models the response of the structure. It is typically not their responsibility to perform actual design calculations. Their job is complete when they dump terabytes of data on the desk of the design engineer. Since the analyst may not be versed in the details of design they will often error on the side of caution by including all possible results to make sure that the designer has all the possible data he can use, especially in the area of local connections. His responsibility ends after the analysis is completed, checked and documented.

The design engineers are responsible for the detailed design of walls, slabs, and connections. They are experts on local design, but are often lost in this sea of data. They need forces and moments for the worst case enveloping loads provided in a format fit to design the reinforcement. They may have performed some finite element work in school, but are not prepared to digest the terabytes of results. Even the best automated design software requires engineering interpretation. When asked to extract data from the finite element model to better understand local over stresses, it can be a very intimidating task. Where do I start? How do I access the data? If I want to average the local stress concentrations how do I perform these tasks? Training can help both sides to better understand each other's job in order to develop a better process.

Analysis Training for the Designer

Software tools used to develop detailed models of large complex structures such as an entire Nuclear Island are focused on modeling efficiency, fast processing and efficient calculation of displacements, strains and stresses. The codes are also adapt in dealing with nonlinearities including nonlinear frictional contact. The calculations can be distributed to multiple processors on a single machine or over a number of machines in a network. The efficiency of the computational tool allows one to calculate discrete stresses at each element integration point in the model over the entire time history and/or load combinations. However it is impossible to save all this data for every time point of an earthquake analysis for example and thus judgment is needed in determining which data to store and the frequency of storing results. Also, the fidelity of the model still requires many simplifying assumptions during the geometry modeling and meshing process. For example, by including local sharp corners the FEA calculation will result in singular stress states that cannot be used in design. The mesh density is typically created for FEA efficiency but this might not necessarily be the best for the designer. Stresses often must be averaged to be accounted for in the design process. Engineering judgment is required in best interpreting the results. The more the design engineer is aware of the analysis process the more it will make the designer adapt at best leveraging the finite element calculation.

Training classes for the designer in finite element simulations should be focused on their specific needs. The designer does not have to be able to mesh complex geometry, determine the best time stepping setup for an earthquake simulation or define the best settings for the contact elements to enhance convergence. However, what they do need training in is a better understanding in how the finite element methods computes and stores displacements, stresses and strains. For example they need to know the difference between centroidal data, integration data and the internal finite element assumptions for nodal stresses. Figure 1 illustrates the typical types of stress results available in most finite element software programs. Three different stress plots are shown for the same load case. These three plots illustrate the large variation in predicted local stress that the designer must deal with. Although these might be very accurate stresses they are not in a suitable form for the designer. The designer would prefer critical section forces and moments.

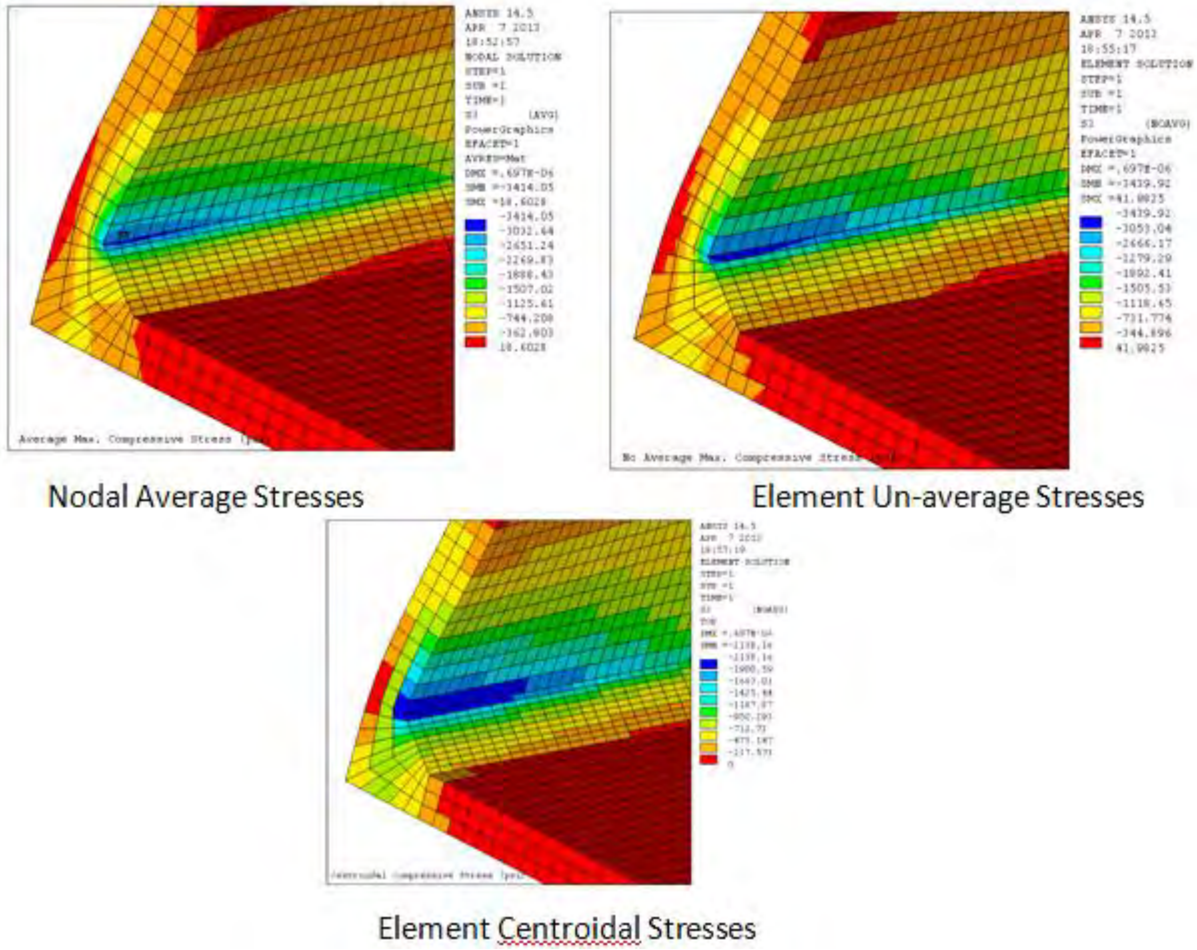

Figure 1. Example stress values including nodal averaged vs. unaveraged vs. element centroidal values

The designer also needs to be adapt at extracting local data as needed using isolation tools such as select logic or defining a local coordinate system to extract stresses, forces or moments at any orientation. Other useful tools include the ability to isolate individual vs. combined load cases in order to determine the worst case load combinations and in the case of overstress what is the driving load. They also should have the skill of making a local submodel for conditions such as when the differences between averaged and unaverged stress values are large. Automation tools can be developed to aid the designer in this process. In this case a submodel is defined as an independent finite element model that maps the displacement response from the global model onto an independent local finite element grid. Using this skill one can increase the fidelity of the finite element results without the need for rerunning large scale analyses as long as the stiffness in the area of interest does not change. While the designer can easily use these tools as a black box, their value increases if they fully understand the process and can investigate questionable results or special cases on their own. There is a large time savings if the designer does not have to reengage the analysis each time one needs clarification or expanded results.

Design Training for the Analyst

Training classes for the analyst should emphasize on the design process and how the analysis results are input into the code checks. An emphasis on design rules for example would include standard practice on where it makes the most sense to splice rebar. Having this data upfront would be useful for the analyst when he is laying out his finite element mesh so that he can implement element boundaries, groupings and orientations that are advantageous for design. For example when forcing a mapped mesh on floor slabs, make the orientation align with the axes most likely to be in the direction of the main span. Also at intersections between floor and walls where 3d elements are used, assure that the fidelity of the mesh is sufficient to capture the peak moments that occur at these intersections. Understand the design process so that output results controls can be defined to minimize the amount of data stored on the results files. For example if there is a plan mapped out for the designer, then the analyst can isolate the stress results that are stored to the results file to only contain data for the elements that will be used in design.

An additional benefit of the analyst understanding how the results will be used in design is that it will help him decide how to model certain attributes of the structure. For example, there are several acceptable methods of modeling concrete drop beams under a concrete slab (e.g., a T-beam) that are sufficient to properly distribute the stress throughout the slab and T-beam. However, if the results are to be used for design of the T-beam, then the manner which the T-beam portion of the model is created becomes more crucial, if the goal is to minimize the amount of additional analysis required to obtain the proper forces and moments. For example, if the T-beam is modeled as a flat plate with either a beam element or a shell element (oriented 90 degrees from the plane of the slab), then the designer must first determine how to combine the resulting forces and moments from the flat plate and beam portions of the model, before performing the actual design. Whereas, if the T-beam is modeled from the start as a beam element with properties representing a T-beam, the design process is greatly simplified. In general, for portions of the model intended to be used as direct input for design, the analyst must be trained to think not just of how to model the structure to properly distribute loads in a global manner, but also to consider what particular forces and moments are required by the designer at various locations, and include the appropriate modeling techniques to obtain those forces and moments.

EXISTING SOFTWARE TOOLS

Desktop workstation with multiple CPU cores can now readily solve static and dynamic, linear and nonlinear time history analyses of large scale finite element models using commercial FEA software such as ANSYS (2013). Nuclear structure models can consist of 3d brick, shell and beam element meshes with enough detail to extract design forces and moments. Modeling the superstructure, foundation and soil for simulating soil-structure interaction is an example of the most complex analyses performed. The nonlinear response while more accurate also requires many additional load steps to be solved since superposition cannot be used for these cases. FEA software can also now simulate soil-foundation frictional contact including uplift to determine the true building motion and at the same time extract high fidelity finite element results in the building superstructure. Stress data from these analyses is readily available for the entire building for all individual load cases and over the entire time history solution in each element, but the output created is an extremely large amount of data that is not in the best format for concrete design.

With extremely thick walled nuclear power plant concrete structures, it is common for analyses to model reinforced, pre-stressed or post-tensioned concrete floors and wall structures using 3D solid elements. The 3d solid elements output detailed stresses at the integration points of these elements. For example an 8-node brick element will have 8 internal integration point stress values that for linear analyses are extrapolated to the nodes. However, codes and standards in the majority of the cases utilize forces and moments for the design calculations. CivilFEM (2013), a civil engineering design tool integrated into ANSYS, includes a utility that integrates the continuum element stresses and converts them into forces and moments for design. The unique post processing routine converts this data into equivalent forces and moments for design evaluation on an integration point by integration point level. This point-by-point basis is then mapped onto pseudo shell elements that are automatically generated at the wall centerline (see Figure 2). The force and moment data can then be used for concrete design activities.

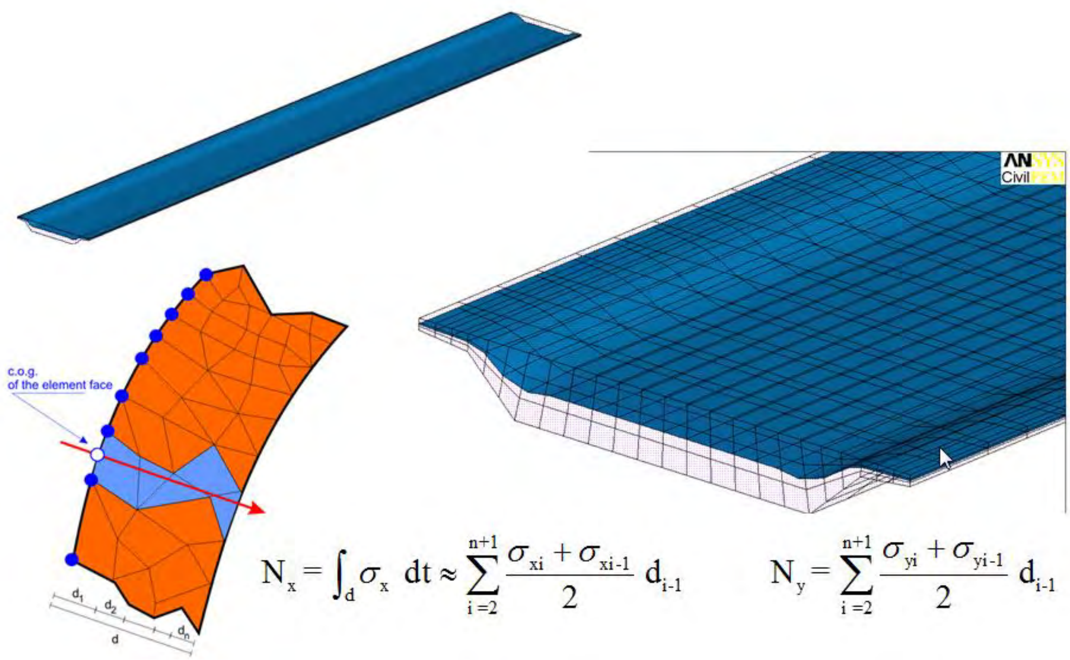

Figure 2. Pseudo shell element used to develop force and moments per unit length for code evaluation

From the forces and moments, CivilFEM will calculate, based on a provided concrete cover, the amount of steel needed to satisfy the specified design code. For concrete slabs as an example, the steel area is determined by evaluating the forces and moments, material properties, and steel location using one of several design methods including Wood-Amer, CEB-FIP, Orthogonal Directions and Most Unfavorable Direction. A thorough evaluation is performed for specific code requirements, for example for ACI359 radial shear, tangential shear and the combination of flexure and membrane force are accounted for in the rebar design.

If one could use different rebar amounts and spacing at each node location in the finite element model the designer's job would be easy! Obviously this is not the case. While the design software is a very valuable tool it does not complete the design process. The designer has to develop a defendable strategy on how to take this nodal steel area per unit length data and convert it into actual design drawings. Questions that have to be addressed include:

- What size and spacing of bars most efficiently meets these calculated steel areas?
- Where can I average the stress data?

- How do I know I am conservative or conversely am I being overly conservative by using the extreme design values (e.g., which high stresses are real, and which are a result of modeling singularities)?

Customized tools are often also developed on a project by project basis that help reduce the finite element data and/or results from the automated code checking into a form more favorable form for design. This is where the collaboration between the analyst and the designer can greatly enhance the design efficiency. For example, creating named isolated groups of elements and isolating the critical design quantities required for these regions allows the analyst to extract this data and transport into spreadsheets, in typical design units (e.g., forces in kips/foot, moments in kip*feet/foot), that are often more attractive to the designer to process. Porting data into spreadsheets provides most designers a comfort zone for design iterations. As part of this exporting of data, a variety of plotting tools can enhance the designers understanding of the finite element analysis results. For example:

- Stress component plots on the top and bottom surfaces of the elements provide direction and magnitude of stresses.
- Force and moment contours illustrate the distribution of forces and moments on a per unit length basis. Figure 3 illustrates Moment per unit length data available for design.
- After code evaluation the top and bottom reinforcement can also be plotted. Figure 4 illustrates the rebar required in the longitudinal direction for the example connection.

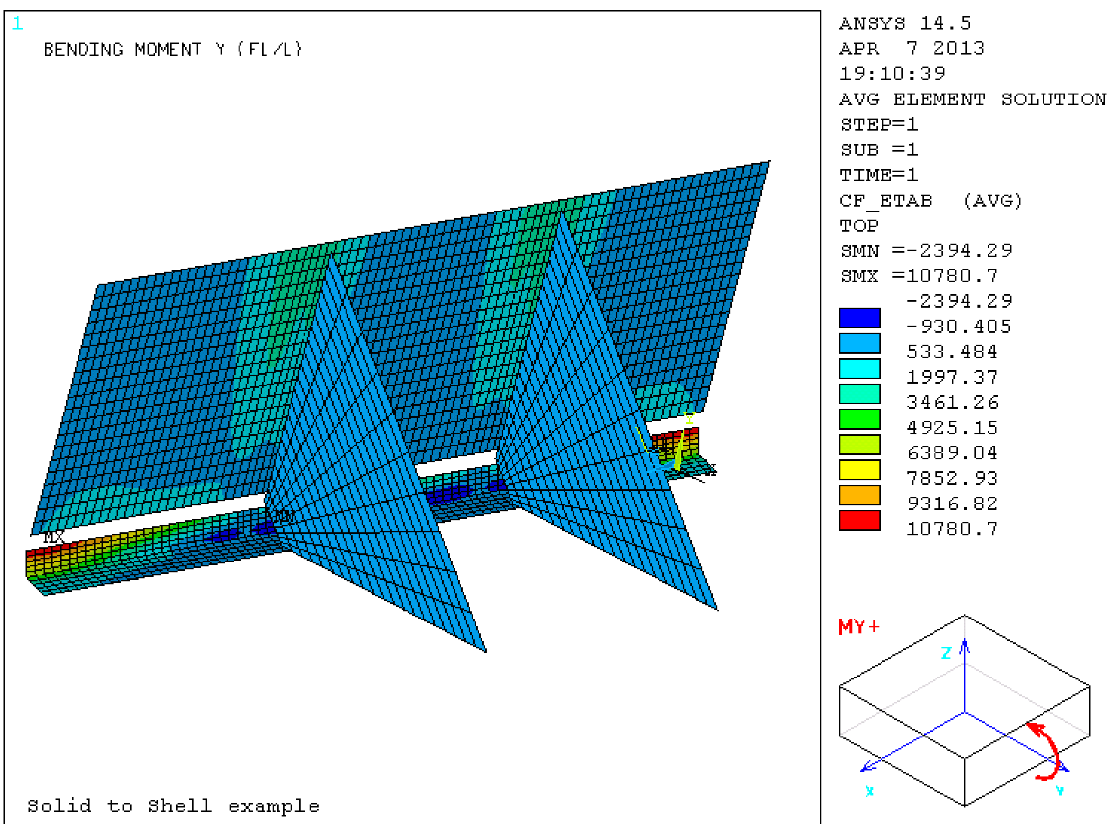

Figure 3 Example bending moment per unit length plot

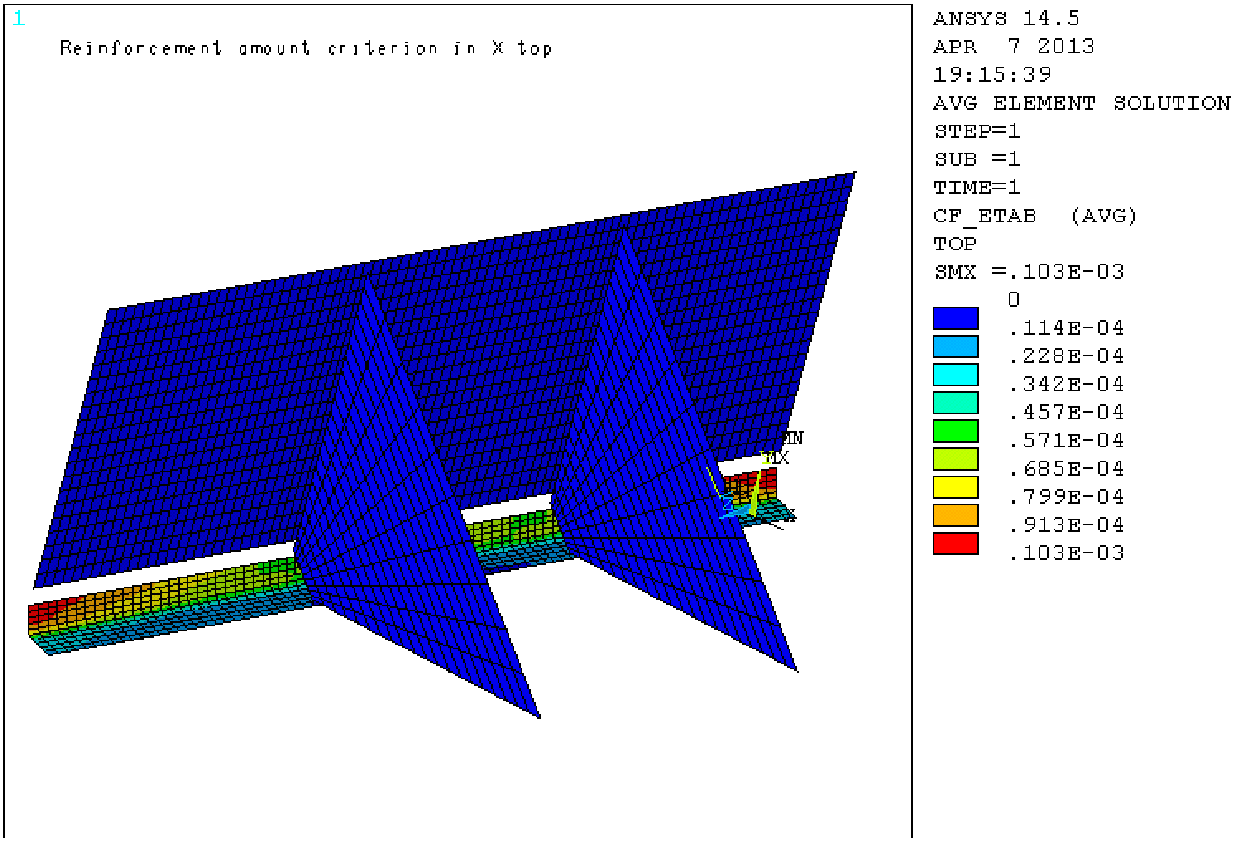

Figure 4 Example reinforcement ratio on the top shell surface per unit length

FUTURE FEA BASED CONCRETE DESIGN SOFTWARE DEVELOPMENTS

Existing concrete design software can provide detailed design for specific regular geometry such as one-way slabs, two-way flat plates, two-way flat slabs (with drop panels), slab band systems, one-way pan joist systems and one-way skip joist systems. These codes however are not suited to directly interface with detailed finite element data. NESCC 2011 recommends that ACI Committee 447 develop guidelines on the post-processing and interpretation of finite element output typically produced by detailed finite element modeling used in nuclear power plant design. These proposed guidelines will be a welcome addition to the analysis/design process.

Concrete design requires the development of reinforcement that meets code requirements, but also meets constructability requirements. As mentioned earlier, post-processing routines in CivilFEM have been written to facilitate processing of this data, however many opportunities for improvement still exist. Such opportunities include:

- An automated manner of reviewing the total population of load combinations and accurately determining the controlling load combinations taking into account appropriate averaging where applicable.

- An automated manner of averaging “hot spots” during the design process, to allow the designer to take into account force/moment redistribution of cracked concrete.

- An automated manner of designing for multiple load combinations at one time, while maintaining an efficient, constructible design.
- Automated confirmation that the proposed reinforcement is sufficient assuming the redistribution of load where the stress state in the model is updated based on the proposed reinforcement.

Future software development which addresses such opportunities, resulting in a constructible design which meets applicable design requirements, would be beneficial to the nuclear industry. The developers of these tools would benefit from specific guidelines spelled out in the design codes to aid in their development.

FUTURE ANALYSIS SOFTWARE TOOLS

While the concrete design tools will benefit from the simplification of the detailed analysis data, the analysis tools could also be improved with pre-defined templates that can be quickly implemented for project specific tasks. A direct integration into the design tool and automated updates to the finite element models in a parametric setup could greatly reduce the design cycle. While this type of automation exists on a single part level, expanding parametric modeling to the entire building is not possible at this time, but should be considered in future software developments.

CONCLUSIONS

More intelligent design software will reduce the concrete design cycle. The advances is software will need to be developed to specifically meet the demographics of the end users. In summary:

- Better software tools are needed to bridge the gap between analysis and design
- More education should be provided for the analyst in design and the designer in analysis
- Incorporating design aspects into the development of the computational analysis model will shorten the design cycle.

.

REFERENCES

ANSYS® 2012, Release 13 Help System, ANSYS Inc, Canonsburg, PA.

ASME Boiler & Pressure Vessel Code Section III Division 2, "Rules for Construction of Nuclear Facility Components: Code for Concrete Containments", 2004 code, no addenda.

ACI 349, "Code Requirements for Nuclear Safety Related Concrete Structures", 2001.

CivilFEM® 2012, Release 13 Theory Manual, Solid to Shell Design Procedure, Ingeciber, Madrid, Spain

"Recommendations for Future Development". Concrete Codes and Standards for Nuclear Power Plants, NESCC Concrete Task Group (CTG) Report Version 26; February 22, 2011 – Final Draft – submitted to NESCC 2011