Development of a Finite Element Model of ATUCHA II NPP Reactor Pressure Vessel for Pressurized Thermal Shock Analysis

Paolo Ferrara, Dino Araneo, Fabio Moretti, Francesco D’Auria
Department of Mechanics Nuclear and Production Engineering
Via Livornese 1291, San Piero a Grado, (Pisa) Italy
Tel: +39-050-2210352, Fax: +39-050-2210384
p.ferrara@ing.unipi.it, d.araneo@ing.unipi.it, f.moretti@ing.unipi.it, dauria@ing.unipi.it

ABSTRACT

The present work deals with the development of a detailed Finite Element (FE) model of the Atucha II Nuclear Power Plant (NPP) Reactor Pressure Vessel (RPV) for Pressurized Thermal Shock (PTS) analysis. This model simulates both the base and the cladding metal, the Cold Legs (CL), the Hot Legs (HL), the emergency systems penetrations with the corresponding nozzles. A complete RPV FE model is needed for the calculation of the stresses in the undamaged structure to be used as input for the calculation of the stress intensity factor $K_I$ by means of the “weight function” method. This is one step of the methodology developed at University of Pisa (UNIPI) for the deterministic analysis of PTS scenarios. Such methodology is based on the use of a chain of codes and starts with the thermal hydraulic analysis of the NPP with a system code (such as Relap5, Cathare2, etc.) for a selected transient scenario. The goal of this step is to estimate the cooling load induced on the internal RPV wall surface by the Emergency Core Coolant (ECC) injection, and provide boundary conditions for the next step. The region of interest is represented by the welding lines in the down-comer region. The next step consists of the 3D analysis of the down-comer flow and of the temperature field in the RPV wall, using a CFD code with Conjugate Heat Transfer (CHT) capabilities. Once the pressure of the system and the temperature in the RPV wall is known, a stress analysis can be performed by means of a Finite Element (FE) structural mechanics code. The last step of the methodology is the Fracture Mechanics (FM) analysis using weight functions, to calculate the stress intensity $K_I$ factor at crack tip to be compared with the critical stress intensity factor $K_{IC}$.

1 INTRODUCTION

The RPV has long been considered one of the most reliable components in Pressurized Water Reactors (PWR). Nowadays a general target for the countries that produce nuclear energy is to extend the operation life of existing plants. From this point of view, the RPV is one of the major components that may limit the useful life of the nuclear power plant. The risk for the RPV structural integrity is connected to the presence of a flaw of relatively large size, a high level of embrittlement due to radiation damage, and the occurrence of a thermal-hydraulic transient inducing strong stresses in the vessel wall. Severe loading conditions are produced during a Pressurized Thermal Shock (PTS), in which an over cooling may induce strong thermal stresses, the internal pressure being maintained at high thermal level or the
system being repressurized during the transient. The main initiating PTS events identified in literature are Small Break Loss of Coolant Accident (SBLOCA), Main Steam Line Break (MSLB), Loss of Main Feed Water (LOFW), Steam Generator Tube Rupture (SGTR) and Loss Of Heat Sink (LOHS). The transients leading the Nuclear Power Plant (NPP) to a PTS scenario are intrinsically asymmetric, so it is important to avoid any simplification by symmetry in the model used for the analyses. In recent years important progress has been made in the development of analysis methods for the best estimation of the thermal loads and the structure stresses on the vessel wall, needed for an in-depth fracture mechanics study. A research activity carried out at Department of Mechanics Nuclear and Production Engineering of the University of Pisa is aimed at developing a computational tool able to perform parametric analysis, assuming various shapes and locations of the flaw in a RPV [6].

2 UNIPI METHODOLOGY FOR PTS ANALYSIS

University of Pisa developed a methodology (summarized in figure 1) concerning the use of a chain of codes for the deterministic analysis of PTS scenarios, [1], [2].

The methodology starts with the thermal hydraulic analysis of the NPP with a System Thermal-Hydraulic (SYS TH) code (such as Relap5-3D, Cathare2, etc.) during a selected transient scenario. The goal of this step is to (roughly) calculate the cooling load induced on the internal RPV wall surface by the Emergency Core Coolant (ECC) injection, and to provide boundary conditions for the next step. The region of interest is represented by the RPV welding lines facing the core region. The next step consists, for transients with the fluid in single phase, of a 3D analysis of the down-comer flow and of the temperature field in the RPV wall, using a CFD code (possibly with CHT capabilities). An interesting feature of the methodology is represented by the possibility to transfer the RPV wall temperature profile calculated by the SYS TH code to the ANSYS FE model (green arrow in figure 5.a). Once the pressure of the system and the temperature distribution in the RPV wall are known during the

Figure 1: UNIPI methodology for PTS analysis

The methodology starts with the thermal hydraulic analysis of the NPP with a System Thermal-Hydraulic (SYS TH) code (such as Relap5-3D, Cathare2, etc.) during a selected transient scenario. The goal of this step is to (roughly) calculate the cooling load induced on the internal RPV wall surface by the Emergency Core Coolant (ECC) injection, and to provide boundary conditions for the next step. The region of interest is represented by the RPV welding lines facing the core region. The next step consists, for transients with the fluid in single phase, of a 3D analysis of the down-comer flow and of the temperature field in the RPV wall, using a CFD code (possibly with CHT capabilities). An interesting feature of the methodology is represented by the possibility to transfer the RPV wall temperature profile calculated by the SYS TH code to the ANSYS FE model (green arrow in figure 5.a). Once the pressure of the system and the temperature distribution in the RPV wall are known during the
selected transient, (with the support of SYSTEM TH and CFD codes respectively), a stress analysis can be performed by means of a FE structural mechanics code (such as ANSYS) applied to a complete model ANSYS grid, where both pressure and thermal loads are considered. For this purpose it is needed to transfer main CFD results (in particular: temperature profile) to the ANSYS model, in order to evaluate the thermal stresses inside the RPV. Once the stress profile in the RPV wall is known, FM analysis is then performed by means of the weight function method for the calculation of the stress intensity factor (KI) to be compared with the critical stress intensity factor curve (Klc) of the material.

The present paper discusses relevant achievements from the application of this complex activity to a specific case; the following topics are covered:

- an overview of ATUCHA II Nuclear Power Plant and RPV model;
- a detailed complete ANSYS model and grid of the Atucha II RPV for the structural analysis of PTS scenarios;
- the interfaces between the different codes involved in the methodology;
- results of a MSLB test-calculation.

3 OVERVIEW OF ATUCHA II NUCLEAR POWER PLANT

Atucha II is a pressurized, heavy water cooled, heavy water moderated reactor with a net electrical power of 692 MW (2161MWth). ATUCHA II NPP has been designed by the KRAFTWERKS UNION AG Erlangen, West Germany, with participation of Empresa Nuclear Argentina de Centrales Electricas S.A., Buenos Aires (ENACE). Natural uranium is used as fuel; therefore heavy water (D\textsubscript{2}O) is used in the reactor as coolant and moderator.

3.1 Primary components of the Reactor Coolant System (RCS)

The reactor system consists of the RPV, its internals and the control shutdown equipment. The coolant is circulated through the two parallel reactor coolant loops by two Main Coolant Pumps (MCP), it enters the RPV via two inlet nozzles and flows through the annular down-comer between the pressure vessel and moderator tank walls. After arriving to the RPV bottom, the coolant flows upward through the coolant channels and, after joining with the closure head bypass and gap flows, leaves the RPV from two reactor outlet nozzles. The coolant flows from the RPV through the reactor coolant piping to the two Steam Generators (SG), where the power supplied by the reactor and coolant pumps is removed from the coolant. The primary side pressure of the reactor, reactor coolant and moderator system is maintained by the pressurizing system. The pressurizer (PRZ) is connected to the reactor coolant system by the surge line, which is connected to the Hot Leg (HL) of reactor coolant line. The reactor coolant system consists further of two loops, each comprising one SG, one MCP and the interconnecting reactor coolant piping together the pressurizing and PRZ relief system. The two identical, parallel reactor coolant loops have the task of transferring the power generated in the RPV to the SG.
Figure 2: ATUCHA II RPV vertical and horizontal sections

4 ATUCHA II RPV MODEL

ATUCHA II RPV model is a shell made of two materials: carbon steel in the external part and stainless steel liner in the inner part. The RPV is equipped with hemispherical closure and bottom heads, which contain hemispheres full of carbon steel, called “Filler”. Thickness through down-comer wall is constant between lower and upper welding lines, as shown by red lines in figure 4.a; thickness increases above the upper welding line and then it remains constant up to the upper head. The RPV has 12 penetrations:

- 2 coolant inlet nozzles;
- 2 coolant outlet nozzles;
- 2 moderator outlet nozzles;
- 2 hot safety injections nozzles;
- 2 cold safety injections nozzles;
- 2 nozzles for the instrumentations.

RPV is fixed to the basement through 4 double-T beams, which are welded to the RPV at about the same height of Cold/Hot Legs, as shown in figure 3.b. The assembled RPV model is shown in figure 3.a, while some details of the nozzle region are shown in figure 4.

Figure 3: a) view of the computational domain, liner + carbon steel; b) double-T beams
Figure 4: view of the penetrations: a) hot leg, cold leg and safety injections; b) moderator outlet nozzle and fuel failure detection nozzle

5 MESH GENERATION

The Atucha II RPV has been modelled and meshed with the meshing tool ANSYS MULTI-PHYSICS 11.0. The computational grid was obtained as an assembly of several sub-meshes generated separately adopting 4 simple criteria:

1 - a new sub-mesh is used when geometry changes; each sub-mesh corresponds to a different sub-volume; total volume is the sum of all sub-volumes;
2 - considering the interface between two adjacent sub-volumes, every node of the first volume must coincide with a node of the second volume. Therefore there must be continuity in the global mesh;
3 - an interface between two adjacent sub-volumes must lie on down-comer welding lines, in order to have some nodes lying on those lines (figure 5.a).
4 - ratio of linear dimensions of two consecutive elements should not exceed 2.0, (preferably 1.5).

Only one mesh element type was adopted in order to adapt the meshing process to the different degrees of geometrical complexity associated with the different regions of the computational domain: namely, the hexahedra SOLID45 was used, which has 8 nodes, 3 degrees of freedom per node, (figure 5.b), [4], and allows thermal and structural analysis.

Figure 5: a) welding lines in RPV (in red); b) element SOLID45

Only few parameters have been fixed:
13 elements are foreseen through the carbon steel thickness: their relative thickness is calculated by a geometric progression with ratio between two consecutive element thicknesses: \( r = \frac{a_{n+1}}{a_n} = 1.25 \); (larger elements are in the external part); (figure 6.a);

3 elements (with the same thickness) are foreseen through the stainless steel liner thickness (figure 6.b);

160 elements are foreseen around the down-comer circumference (figure 7.a);

80 elements are foreseen along the arc in the bottom head (figure 7.b);

ratio between the third liner element and the first carbon steel element is 1.5 (figure 6.a).

Figure 6: mesh details: a) RPV thickness: carbon steel + liner; b) 3 elements on liner thickness

Figure 7: mesh details: a) 160 elements on the circumference; b) RPV bottom head

Figure 8: view of the complete mesh model
The complete meshed model includes 344 sub-volumes, about 0.88 million elements and about 1.1 million nodes (figure 8). Some details of the penetrations are shown in figure 9.

Figure 9: mesh details: a) cold leg and cold safety injection; b) moderator outlet

6 TEST CALCULATION

6.1 Selected scenario: Main Steam Line Break

The PTS initiating event considered in this test-calculation is a MSLB with 0.1A break area at the top of the SG #1.

6.2 Main features of TH and CFD Models

A Relap5-3D© nodalization of ATUCHA II has been developed by GRNSPG group for PTS analysis. In particular for the down-comer region, two multi-dimensional components available in the Relap5-3D© SYS TH code have been used. The multi-dimensional component was developed to allow the user to model more accurately the multi-dimensional hydrodynamic features of reactor applications, primarily in the vessel (i.e., core, down-comer) and steam generator. The multi-dimensional component defines a one, two, or three dimensional arrays of volumes and the internal junctions connecting the volumes [5].

Because of the fluid in the primary side is in single phase, the results of the Relap5 calculation of the MSLB transient have been used as boundary conditions for the CFX code for a more detailed calculation of the mixing phenomena occurring in the down-comer. A FORTRAN subroutine has been built for this operation.

The CFX model developed of ATUCHA II down-comer and RPV is also very accurate: the computational domain consists of a solid domain (Carbon steel + stainless steel liner) with a total of about 0.63 million nodes and a fluid domain (about 1.7 million nodes); the assembled computational domain, along with its components, is shown in figure 10.
6.3 Calculation set-up

Solid domain model has 4 protrusions that represent 4 welded supporting “T” beams, as shown in figure 11.a. The loads considered are:

- thermal loads;
- mechanical loads.

Thermal loads are due to the temperature difference between a reference condition \((T = 554K\) everywhere) and the actual condition in a specified time step; 8 different time steps have been analyzed during the MSLB transient: the first one is: \(t01 = 0.65s\), the last one is: \(t08 = 210.65s\) and the constant interval between two consecutive time steps is \(\Delta t = 30s\).

Mechanical loads are mainly due to:
- pressure of fluid on the internal walls (look at fig. 11.b, for example);
- weight of filler in the lower part (bottom head): \(W \approx 4.55 \times 10^6N\);
- gravity: \(g = 9.81m/s^2\).

Two typical nuclear grade materials have been used for our test-calculation:
- Carbon steel: Ferritic Steel 22 NiMoCr 37;
- stainless steel: 1.4550.
6.4 CFX-ANSYS temperature transfer

CFX and ANSYS code use different optimized meshes of the same solid domain and different numerical approaches: Finite Volumes (FV) and FE respectively. So a flexible CFX-ANSYS coupling technique is needed, for transferring temperature values calculated in the CFX node mesh to ANSYS nodes, wherever they are placed: the following two methods can be adopted:

- **First approximation method**: ANSYS node gets the temperature of the nearest CFX point. This approach is used when the two meshes have about the same number of nodes;
- **Accurate method**: the temperature in each ANSYS node is calculated as the weighted average temperature of the 8 nearest CFX points (distance being used as weight factor).

In the case discussed in this paper, CFX and ANSYS meshes are accurate, so that, the first method can be applied. Two subroutines were created for the assignment of the temperature to the ANSYS nodes from the closest CFX node. Due to the large number of nodes involved in this operation (about 1 million), this operation takes about three days running on a Pentium Core2 E4500 with 2 GB of ram. In order to reduce the computational time, a Perl script has been developed in order to run the above subroutine in parallel on eight nodes (bi-processor) of an IBM AMD-OPTERON© based cluster. With this improvement the computational time has been decreased to eight hours.

6.5 ANSYS calculation performance

The structural mechanic calculations were run on an AMD-OPTERON© based Linux-cluster. Each calculation run took around 6 hours; in other terms: 12 hours of calculation for every minute of problem simulated.

6.6 Results of the ANSYS calculations

- The result of the temperature transfer process from CFX to the ANSYS model is plotted in figure 12 a;
- Von Mises stresses of the ANSYS calculation (for the loads specified in the chapter 6.3) are shown in figure 12.b.
7 CONCLUSIONS

The paper describes an application of the methodology developed at UNIPI for PTS analyses, together with some results of a MSLB test-calculation for Atucha II NPP. The attention has been focused on one step of the methodology concerning the development of the RPV FE model for the structural mechanic analysis of ATUCHA II reactor. One of the key features of the methodology is the possibility to perform the calculation of the stress intensity factor $K_I$ with the weight function method by mean the stresses calculated using a complete model of the RPV without any simplification or approximation because of the asymmetric nature of the PTS phenomena.

Different codes (Relap5-3D, CFX, ANSYS) have been used for a complete PTS analysis, therefore, starting from the boundary conditions of the SYS TH code till the calculation of the $K_I$, a number of interfaces have been built for transferring the data from one code to another. As a future activity, the upper part of the vessel can be introduced and coupled with the bottom part by mean of bolts and nuts.

In addition, an upgrade of this methodology is in progress aiming at the implementation of each step of the analysis into an integrated software tool, called “PTS Platform”, able to execute automatically the complete PTS analysis, once the boundary conditions have been given in input to the software.

REFERENCES


