Transactions, **SMiRT-22**
San Francisco, California, USA - August 18-23, 2013
Division IIII

ANALYSIS STRATEGY OF THE ITER VACUUM VESSEL FOR THE MANUFACTURING DESIGN

Joan Caixas[1], Angel Bayon[1], Elena Fernandez[1], Jesús Izquierdo[1], Marc Jimenez[1], José María Zarzalejos[4], Julio Guirao[2] and Joaquin Polo[3]

[1]F4E, c/Josep Pla, n.2, Torres Diagonal Litoral, Edificio B3, E-08019, Barcelona, Spain
[2]Numerical Analysis Technologies, S L., Marqués de San Esteban 52, Entlo, 33209 Gijon Spain
[3]Iberdrola Ingeniería y Construcción, Avenida de Manoteras 20, 28050 Madrid Spain
[4]External at F4E, c/Josep Pla, n.2, Torres Diagonal Litoral, Edificio B3, E-08019, Barcelona, Spain

ABSTRACT

The Vacuum Vessel is the first radiological barrier of plasma classified as nuclear component class 2, following the RCC-MR 2007 code (equivalent to ASME Section III) and will require to be approved by the Autorité de Sureté Nucléaire.

It is necessary a global analysis strategy able to justify and optimize the detailed manufacturing design of the component. To treat this problem, the strategy proposed is based on Finite Elements Models associated with reduction techniques, instead of the complete entire detailed model with all components and details.

The entire model using reduction techniques has reduced significantly the number of degrees of freedom of the entire Vacuum Vessel model and the computing time needed with high accuracy results.

The different computation techniques used are an efficient solution for the validation of the manufacturing design and the optimisation of large and complex models such as the Vacuum Vessel.

1. INTRODUCTION

The ITER vacuum vessel (VV) (Figure 1a) is responsible of enclosing the plasma of the ITER tokamak. The primary VV functions are to provide radiation shielding against plasma neutrons and a high quality vacuum. The VV is the first radiological barrier for tritium and activated dust classified as nuclear component class 2, following the RCC-MR 2007 code (Equivalent to ASME Section III) and will have to be approved by the Autorité de Sureté Nucléaire.

The VV is a toroidal structure with an outer radius of 19.4 m, an inner radius of 6.5 m and a height of 11.3 m, assembled by welding nine sectors (400 Tones per sector), manufactured with a D shape double wall (60mm thick) with stiffening ribs between.

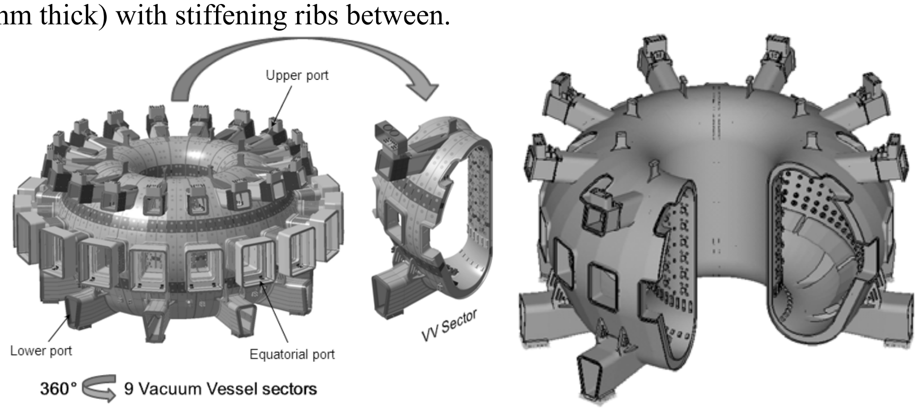

Figure 1. a) ITER VV Sectors b) 360º VV FE model

Each sector contains over 1.78km of full penetration welds in an area of 67m2. This yields a large average of 26m of weld per square meter of VV wall, each centimeter of which has to be 100% volumetric NDT inspected, according to RCC-MR nuclear code.

In each sector, there are attachment features for in-vessel components, 400 highly loaded (blanket, divertor and invessel coils - each resisting forces up to 55 Tons) and 115 lightly loaded (cooling manifolds < 2 Tons), with an average of 8 support lugs/m2. The interspace of the double-wall VV is subjected to high pressure loads (3.1MPa during pressure test), variable complex electro magnetic loads coming from plasma disruptions, thermal fatigue (plasma core temperature 150 milion °C), nuclear heating, etc

The VV sector is required to have tight tolerances (≤±10mm) of the overall profile to allow the sectors to be welded later in-situ with the other sectors (Tolerances ≤ 0.1% compared to the global VV dimensions). Therefore a mistake made early in the detailed manufacturing design stage can greatly impact the overall cost and release schedule of the VV. The greater the gap between detailed manufacturing design and resources committed, the greater the risk to the project, since could cause unnecessary delays and expenditures.

FE simulation plays an essential role in this strategy to meet these challenging tolerances of the VV sectors and optimization of the detailed manufacturing design to achieve the RCC-MR inspection requirements. It is necessary a global analysis strategy (Figure 2) capable to deal with all detailed manufacturing design challenges. This strategy has to be able to anticipate all the analysis needs during the detailed design phase to avoid impacts on the schedule. Those analyses should be a tool to use during the manufacturing design process, rather than a tool to use in checking the suitability of a near-final manufacturing design.

Complex models that are subjected to transient or complex loads (Ioki, K et al. 2009) such as the VV are becoming a computational challenge. FEM of an entire machine (Fig.1b), even without all the details, is complicated and time consuming (13 milion DOF and large amounts of computational time).

To solve this problem, the strategy proposed is based on a FEM model associated with reduction techniques (e.g. submodeling, component mode synthesis, welding condensation, etc) instead a complete entire detailed model with all components and details (Figure 3). The methodology with reduced computation time was developed to enable the simulation and optimisation of many different designs alternatives quickly.

2. METHODOLOGY

A global analysis strategy has been developed, capable to justify and optimize the detailed manufacturing design following the RCC-MR 2007 code. The strategy followed is based on a Finite Elements Models associated with reduction techniques and divided in 4 stages (Figure 2).

The strategy depicts the procedure followed to achieve the final detailed manufacturing design, including the optimized welding sequence, starting from a global FEM model approach to a detailed FEM model approach, as explained in the following sections.

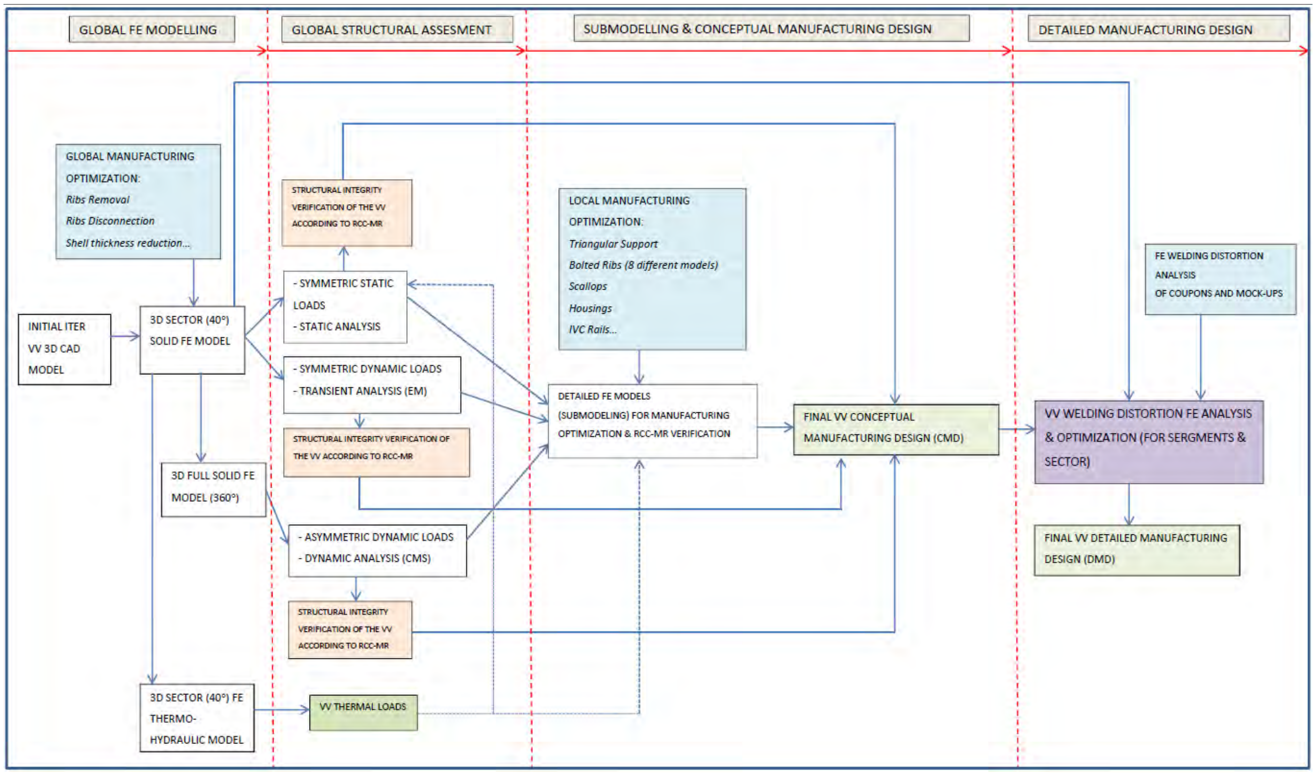

Figure 2. Analysis strategy scheme

2.1. GLOBAL FINITE ELEMENT MODELS

Using the cyclic simmety of the component, different global FEM models of the VV have been built in order to be used in diferent type of analysis, according to the loads to be applied and outputs to be obtained, as detailed in the section 2.2. The VV FEM models built comprise 40° sector and 360° VV for mechanical anlaysis, and 20° half sector for thermo-hydraulic analysis.

Finite Element Model of the VV Sector (40°)

The VV Sector is modeled very precisely, including all ribs, housings, gusset, etc (Figure 3). The whole mesh is structured so that the total node number of the model is kept relatively low while having an accurate mesh able to represent VV structural behavior. The main purpose of the 40° sector model is to have an accurate tool to assess all the loads presenting cyclical symmety and detect all the critical areas (Figure 7b) in terms of primary loads. In a second step, the sector model allows using submodeling techniques on smaller models where the different construction details can be assessed and other failure mechanisms such as fatigue or ratcheting can be studied.

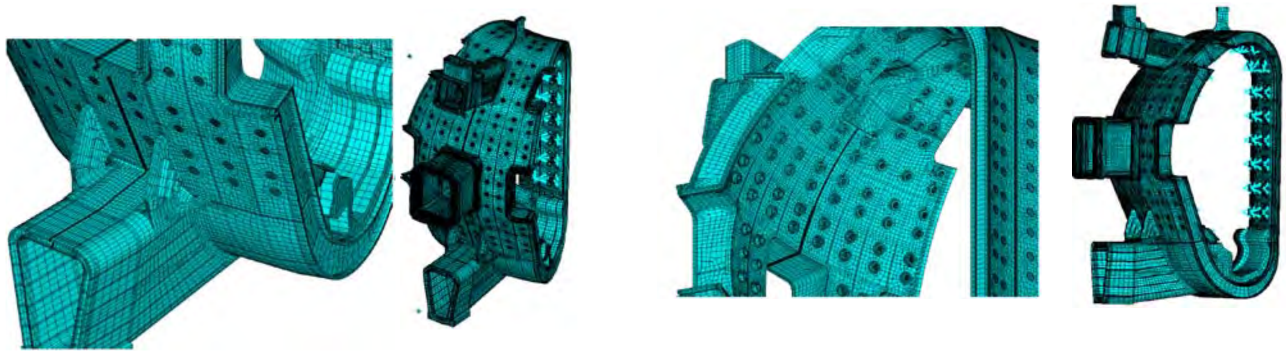

Figure 3. Vaccumm Vessel, Finite Element Model (40° Sector)

The 40º sector model includes all structural elements of the VV and also all the different components interfering with it. They have been included in a simplified way, using punctual masses and other types of elements as complex constrains in order to have a proper spatial distribution of all the masses that may interact with the VV in terms of inertial actions. The size of the model was kept around 1.5 million DOFs leading to reasonable computational times during the solution of all the different load combinations specified. At the same time the level of discretization (mainly structured mesh) easily allows an efficient postprocessing for the application of the integrity criteria in the elastic route according to RCC-MR code, requiring stress linearization for the stress assessment.

Finite Element Model of the Sector (360°) - Seismic Analysis

In order to perform a complete seismic analysis of the whole VV structure two approaches can be followed. The first one consits in developing simplified model using simple mass and stiffness elements which reflects the spatial mass and stiffness distribution of the whole system. While the outlined approach can reasonably be used in regular structural systems (ie: structures of beams, plates, etc), its application to the particular situation of the VV is rather difficult considering the special geometric particularities of this structure. On the other hand the application and combination of the effects calculated with other load cases soved in more complex models becomes a problem. The second approach lays on the idea of developing the mentioned analysis over the same detailed model used to analyze the rest of load cases. This implies additional difficulties derived to the fact that while in general applicable load cases are cyclically symmetric (allowing the associated simplifications in the FE model) the nature of the seismic load and the kind of analysis to be performed (spectrum analysis) makes the application of the symmetry advantage not feasible. Thus, it is necessary build a FE model with a high number of degrees of freedom (DOF), around 13.5 million (1.5 Million DOF per sector), which includes the nine single symmetric sectors (Figure 4b). Obviously, handling a model of these characteristics requires special computational means not available in general. However other can be applied in order carry out this analysis.

In large models, reducing the complete model can be done dividing the complete structure into several substructures (components) and perform the analysis component which is called sub-structuring (Figure 4c). The so-called Component Mode Synthesis (CMS) technique is both a sub-structuring and a reduction method. Given a cyclic symmetric (periodic) structure, a dynamic analysis can be performed for the entire structure by modelling only one sector.

The Craig-Bampton formulation (Craig, 1968) can be used for Modal synthesis in this 360° VV model of 9 structures (sectors) that have a common interface (each called a sub-structure) may be coupled together for an efficient analysis of the combined structure.

The structural dynamics uses finite element models initially built for static analysis.

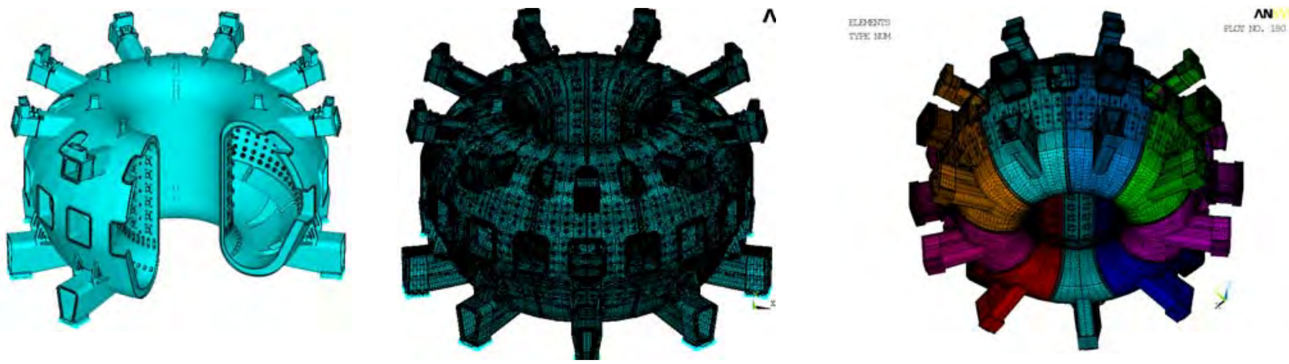

Figure 4. a) VV Geometry b) VV - FEM (360° model) c) VV – CMS model

Finite Element Model of the Sector (20°) – Thermo-HydraulicAnalysis

The VV is an active cooled component submitted to cyclic thermal loads. The temperature distribution and their evolution during the operational cycle of the machine (400s of plasma every 1800s) is an essential input for the mechanical analysis.

A thermal hydraulic model has been built to implement the CFD analysis of the VV regular sector. The model corresponds to one of the VV cooling loops, i.e. to a half sector (20° model). It accommodates a wide range of spatial scales, from the millimetric gap between some IWS plates to the 10m height of the VV itself. A computational mesh with 50 millions of elements has been generated and used to compute the steady-state pressure and flow fields from a Reynolds-Averaged Navier-Stokes model with k-ω SST turbulence closure. The nuclear heat load is modeled as a volumetric heat deposition with a fine poloidal distribution. The steady state is solved in both the solid and fluid domains and the heat transfer coefficient is estimated on the shells wetted surfaces. The temperatures found are later (Figure 6) to be used as input to a stress analysis.

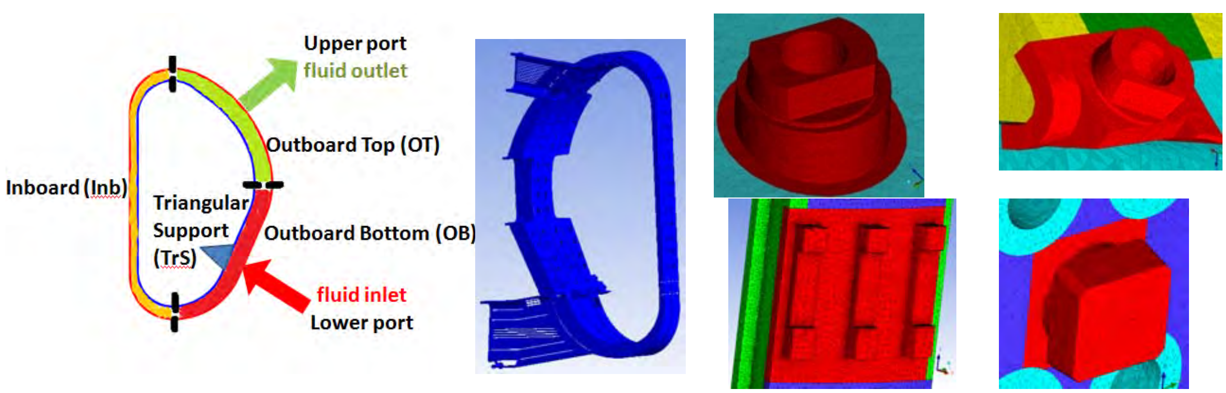

Figure 5. VV Finite Element Model for Thermo-Hydraulic Analysis

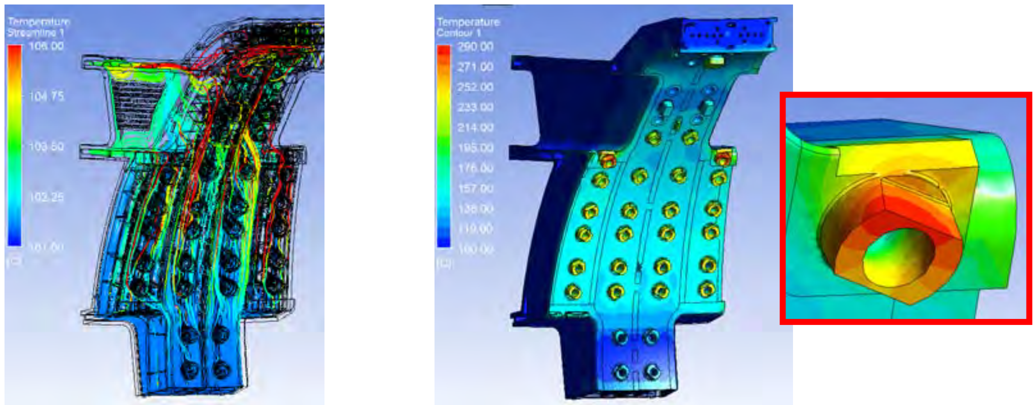

Figure 6. Thermo-Hydraulic Analysis results

2.2. GLOBAL STRUCTURAL ASSESSMENT

Using the FEM models built, a stress assessment has been performed for the elements involved in the VV, considering possible types of failure and including stress classification and categorization techniques to analyse the critical sections. In particular, studies to optimise the manufacturing design on each specific VV area have been developed to assess the compliance with RCC-MR 2007 requirements.

Static Analysis of Symmetric Loads

On the 40º sector model, 34 simple load cases and 36 load combinations are solved (Figure 7a). In a first step, a graphical postpocessing has been defined to better detect the critical areas. The VV has been divided in 7 different zones (Figure 7b) and on each of those zones several critical areas have been highlighted. Stress Classification Lines are defined at each critical area identified in order to assess code compliance at each of those critical areas for each of the load combinations studied. The size of the FE model although remaining low enough to solve all the load combinations has a sufficient level of discretization with at least 3 elements through the thickness. Stress assessment for primary loads is done based on the stresses extracted at each of the stress classificaiton lines defined.

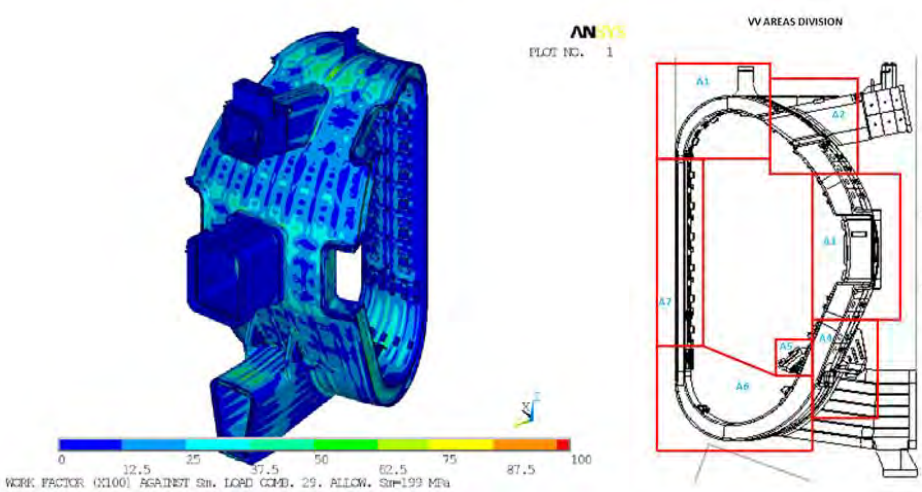

Figure 7 a) VV 40° Sector results of a load combination (*Pressure Test and gravity*) b) VV analysis areas division

The sector model has been also used to assess some manufacturing optimisations. For example, it was proposed to optimize the structure by removing several inner ribs, and the results be compared with the baseline to compare safety margins.

Transient Analysis of Symmetric Loads

The 40º sector model has been used to assess the Dynamic Amplification Factor due to symmetric transient electromagnetic loads and stress analysis of the sector model under electromagnetic loads (Figure 8a). For that, static, modal and transient dynamic analyses have been performed. First, results of modal analysis have been studied identifying the main frequencies and modes. In a second step the supplied EM forces and moments resultants have been introduced in the model. From the modal analysis results the size of time integration step has been also determined to capture properly the dynamic behaviour.

The next step has been determining the transient system response through numerical step by step integration. Then, after each integration step the response has been analysed in detail storing only the information of interest. Finally, the post-processing of the displacements at certain control points and resultant forces and moments and its comparison with the static solution allowed proposing a DAF associated to the global system (Figure 8b).

Additional runs have been solved to study the influence of each zone (structural and non-structural zones) on the global results. These additional runs have also been run statically to compare the results with the dynamic simulations.

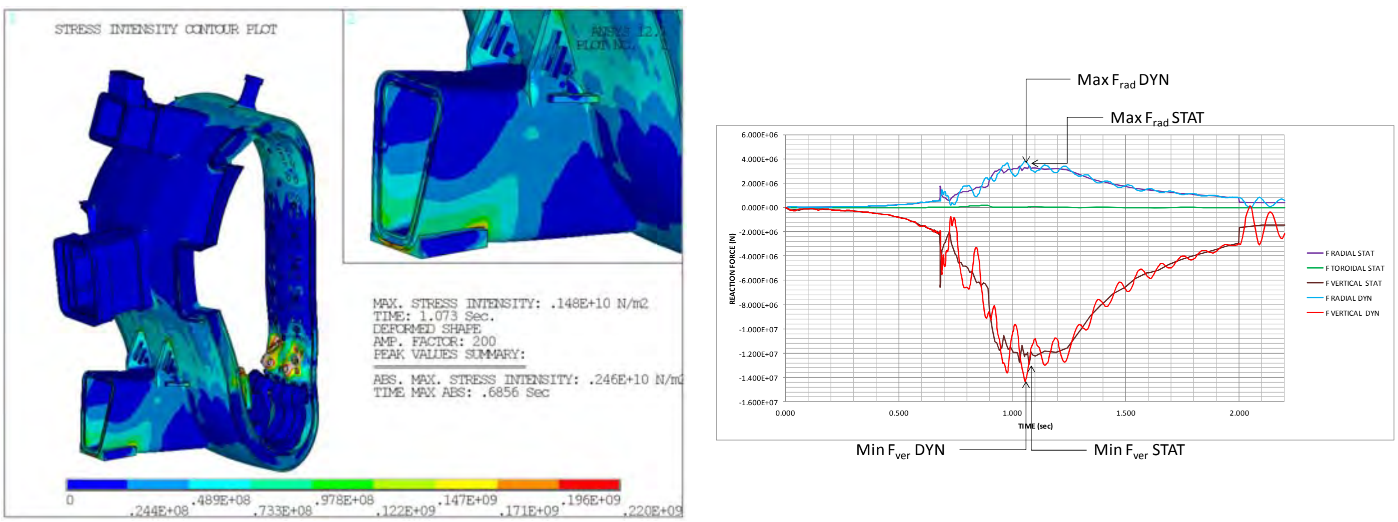

Figure 8 a) VV 40° Sector stress results. b) Transient system response vs Static system to find out DAF

Dynamic Analysis of Asymmetric Loads

Due to the limitations of the 40º sector model which has cyclic symmetry boundary conditions and cyclic symmetric loads some effects cannot be caught. In particular in horizontal direction the VV will present two main modes which are dual.

Therefore two modal analyses have been carried out, one on the 40º sector model and the other onte on a 360º model using CMS techniques as explained in previous paragraphs. The main frequencies have been identified on both models (Figure 9a), showing both the same vertical main mode, but the 360º model was able to catch the dual horizontal modes. A response spectrum analysis (RSA) on the 360º model has been carried out in order to avoid the limitations of the cyclic symmetry models (40º sector model) in two orthogonal horizontal directions and one vertical. The RSA carried out in each horizontal direction allowed considering only two orthogonal modes with above 99% of mobilized mass. The RSA in vertical direction has been done taking into account only the fundamental mode (mobilizing 80% of the system's total mass). The contribution of the modes with frequencies above the main one is taken into account with a pseudomode through the residual mass method. This way will allow to maintain the sign of displacements and stresses in the calculated response and its direct combination with other static load cases. Responses have been combined according to Newmark rule to identify the worst seismic response. Seismic response has been later combined with other loads such as pressure to perform an assessment of the system (Figure 9b).

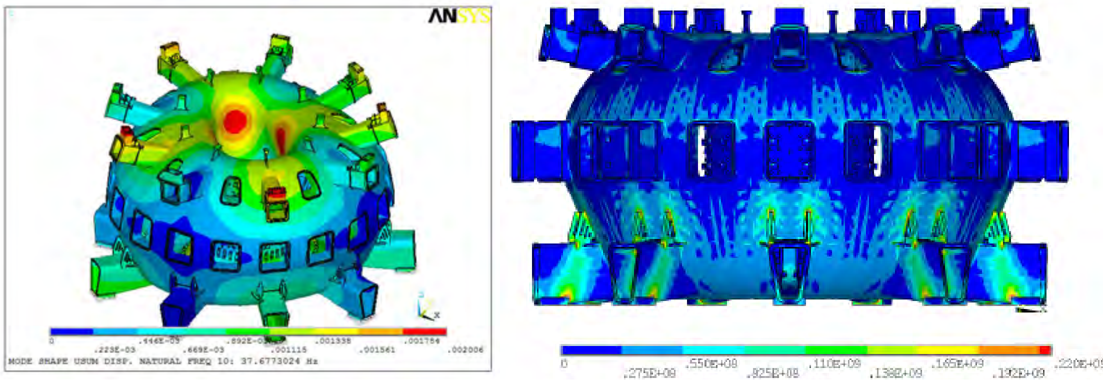

Figure 9. a) VV Modal Analysis b) Stress results of the load combination (seismic and pressure)

2.3. LOCAL MODELS FOR CONCEPTUAL MANUFACTURING DESIGN OPTIMIZATION

The development of a FE model of the ITER VV including all its components in detail is unaffordable from the computational point of view due to the large number of degrees of freedom it would require. This limitation can be overcome by using submodeling techniques to simulate the behaviour of the detailed local areas.

Submodelling is a FE general widely used technique which allows getting more accurate results in a given region of a coarse model by generating an independent, more finely meshed model of the region under study. Results of the coarse model in the cut boundaries are used as boundary conditions of the submodel. The principle behind submodelling assumes that the cut boundaries are far enough away from the stress concentration regions. Analyses on detailed submodels of these regions have been developed (Figure 10), accounting both for the detailed geometry of their components and their interaction with the rest of the structure. Moreover this is a code requirement RB-3214 (AFCEN, 2007) for local models.

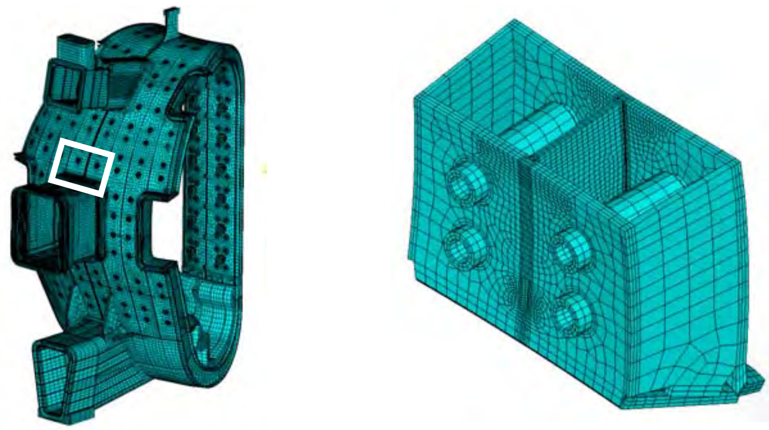

Figure 10 a) VV 40° global model b) Submodel of a local area

2.4. WELDING DISTORTIONS ANALYSIS FOR WELDING SEQUENCE OPTIMIZATION

In order to achieve the tolerances without a full prototype, the manufacturing route must rely on welding distortion FEM analysis (Guirao, 2007). The VV sector distortions are not controlled with the use of stiff jigs, but by calculation and pre-compensation. The results of the calibration of the mock-ups, analysed with FEM models, to calibrate the heat input and distortion, are fed into fully compliant FEM models of the segments and the segment joining.

The method was used to analyze the distortions (Jones L. et al., 2011) in the inboard segment prototypes. The modelling of tack welds and the selection of sequence made a difference up to 10 times to the level of distortion. Due to the cylindrical shape the stiffness is 3 times greater in the convex direction than the concave.

The weld distortion simulation, based on non lineal thermo-mechanical transient analysis (Figure 11) and allows various supporting tools and boundary conditions to be applied or removed at different assembly steps. The FE simulation is calibrated and validated with the same welding technique parameters and comparing distortions and real measurements as described in Caixas J. et al (2012).

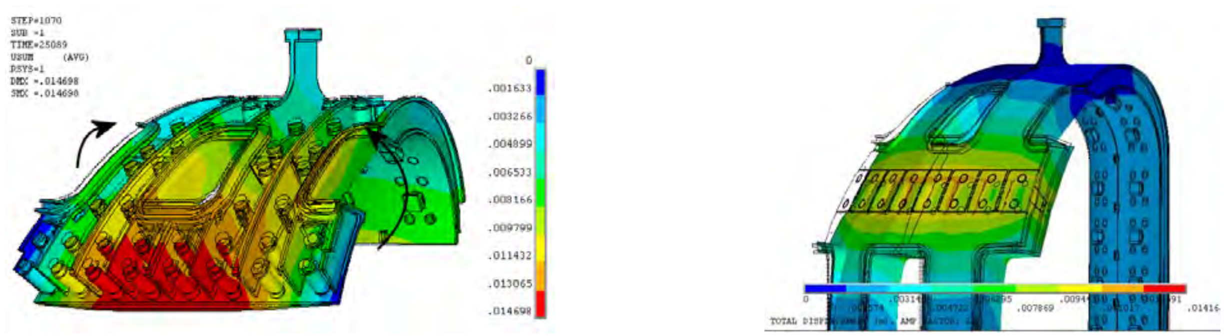

Figure 11 Welding distortion analysis of a) Toroidal rib b) segments joining

3. DISCUSSION

The development of global FE model of the ITER VV including all its components in detail is unaffordable from the computational point of view due to the large number of degrees of freedom it would require. This limitation can be overcome by using submodelling techniques, which is according to RCC-MR code (Section RB 3214).

A methodology used for dynamic analysis, is capable to perform a simulation of a 3D solid model of the 360º VV. This method reduces significantly the computing time needed, which make it suitable for large dynamic models with high accuracy results

A stress assessment has been performed for the elements involved in the assembly considering possible types of failure and including stress classification and categorization techniques to analyse the critical sections. In particular, studies to optimise manufacturing on each specific VV area have been developed to assess the compliance with RCC-MR 2007 requirements.

The applied methodology for welding distortions simulation is suitable to establish weld sequences which minimizes welding distortions in big components as the ITER sector poloidal segments giving a reasonable approximation in quantitative results.

4. CONCLUSIONS

The different computation techniques used are an efficient solution for the manufacturing design and optimisation for large and complex models such as the VV.

The VV FE entire model using reduction techniques has reduced significantly the number of DOF of the entire VV model and the computing time needed with high accuracy results.

Additionally if during fabrication unexpected welding distortion occurs, then using the condensed welding simulation technique, another welding sequence can be calculated quickly starting from the distorted geometry to correct the distortions and meet the final tolerances.

ACKNOWLEDGEMENTS

The work carried out by the agents mentioned herein has been supported by the European Union within the European Fusion Technology Programme, first under the auspices of EFDA, Garching and F4E Barcelona. Special acknowledgement to the contributions of Iberdrola and NATEC (Spain) and AMW (Italy).

DISCLAIMER

The views expressed in this publication are the sole responsibility of the authors and do not necessarily reflect the views of Fusion for Energy. Neither Fusion for Energy nor any person acting on behalf of Fusion for Energy is responsible for the use which might be made of the information in this publication. The views and opinions expressed herein do not necessarily reflect those of the ITER Organization

REFERENCES

AFCEN (2007) "Design and Construction rules for Mechanical Components of Nuclear Installations applicable to High Temperature Structures and to the ITER Vacuum Vessel", RCC-MR Edition 2007.

Caixas J. et al (2012) 'Weld Distortion Prediction and Control of the ITER Vacuum Vessel using Finite Element simulations' SOFT 2012 - 27th Symposium on Fusion Technology, Liège (2012)

Craig, R. a. (1968). Coupling of Substructures for Dynamic Analyses. *AIAA* , 1313-1319.

Guirao, J., Rodriguez, E., Bayon, A., Jones, L., (2009) "Use a New Methodology for Prediction of Weld Distortion and Residual Stresses Using FE Simulation Applied to ITER Vacuum Vessel Manufacture", Fusion Engineering. 84 (12) 2187–2196.

Ioki, K., Bachmann C., Chappuis P.,, Cordier, J.-J., Giraud, B., Gribov, Y., Jones, L., et al. (2009), "ITER Vacuum Vessel: Design Review and Start of Procurement Process", *Fusion Engineering and Design* 84 (2) 229-235.

Jones L. et al., (2011) "Manufacturing Preparations for the European Vacuum Vessel Sector for ITER", ISFNT-10, Portland, USA.