International Journal of Emerging Engineering Research and Technology
Volume 3, Issue 8, August 2015, PP 37-40
ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online)

Optimization of the Diagnostical Cabling of a Fusion Power Plant with the Help of Modeling of Thermo-mechanical Analysis

Judit Szalai[1]

[1]*Department of Machine and Industrial Product Design, Budapest University of Technology and Economic, Budapest, Hungary*

ABSTRACT

In the course of our work the objective was to develop the charge and the construction of diagnostic cable ducts accommodated in the vacuum vessel of fusion reactors. The main charge of risk which affects the components is nuclear radiation as well as temperature charge and excitation of disturbing electrical signals. This was placed into the center of analysis, respectively, we have completed a survey of effects influencing the structure involving all components where the main aspect of design was to assess the thermal tensions resulting from the heat charge. We made a structural optimization resulting in compensating the charges accordingly and providing for an appropriate safety reserve. With the aid of the results we have developed an optimization program supporting the design.

Keywords: ITER, Coupled thermo-mechanical processes, Design, Numerical analysis

INTRODUCTION

ITER (International Thermonuclear Experimental Reactor) is toroid-shaped tokamak of double-wall construction which development is aimed at achieving the development of the technology for future electric power plants by widening our sphere of plasma-physical knowledge. In the power plant planned, a part of 400 MW will leave the plasma from the fusion power of 500 MW in the form of neutrons which means a power density of several hundred kW/m2 arising as a charge on the wall of the vacuum vessel of the power plant.

The abduction of the heat charge is provided by water-cooled shielding blocks designed into the wall of the vacuum vessel. Due to the developing heat, also a solution for the minimization of the heat charge of every component allocated on the wall of the vacuum vessel, respectively in the vicinity of the plasma, therefore exposed to the heat charge has to be found and care shall be taken of a heat abduction of appropriate extent.

The highest heat charge to be taken into account when composing the structural design – affecting in the course of the fusion process the vacuum vessel and the shielding blocks – amounts to 360 ºC in the case of the shielding blocks from the side of the plasma and to less than 150 ºC in the case of the wall of the vacuum vessel. In the cables and cable ducts of the diagnostic system the heat development in the internal volume stemming from neutron and gamma radiation will take up a value between 0,024 – 0,12 W/cm^3.With the reactor operating, providing the appropriate signal transmission is of basic importance for a flawless operation of the data cables running in the cable ducts.

The change of temperature distribution emerging on approaching stabilized heat conduction considered stationary may not exceed the value of 10°K as in the opposite case the accuracy of the measured data will not be adequate. In designing the diagnostic cabling mounted on the wall of the vacuum vessel these processes entailing a sizeable heat development pose several tasks to be solved in the course of designing.

Development of the Diagnostic Cabling of the Vacuum Vessel

In recent years several concepts have appeared to implement the cable ducts mounted on the wall of the vacuum vessel. In the course of designing our objective was to create new concepts in which

Address for correspondence:

szalai.judit@gt3.bme.hu

abduction of heat developing in the cable ducts, respectively, superficial heat transfer of appropriate extent between the cable duct and the wall of the cooled vacuum vessel are solved i.e. the heat tensions arising would charge the structural elements to the least extent possible. The main problem to face in the examined structure is that the heat expansion of the various materials employed is of different extent, resulting in a heat tension at higher temperatures among the elements in the structure which had been assembled free of tensions (at room temperature).

Description of the Models of Enhanced Construction

When designing the new cable ducts geometrically and structurally we were using simplified geometrical models for the basis of examinations. In the course of our work several thermal simulations were completed to be able to assess the extent of heat development arising in the cable ducts and that of the heat tensions serving as guidance for the geometrical setup of the new cable duct and for choosing materials. Table 1 shows the thermal and mechanical properties used for the analysis [1-4].

Table1. *Parameters for FEM simulation*

	Thermal Property					*Mechanical Property*
Material	Heat Generation (W/m^3)	Thermal Conductivity (W/mK)	Density (kg/m^3)	Specific Heat Generation (J/kg^0C)	Linear Coefficient of thermal Expansion ($10^{-6} K^{-1}$)	Young's modulus (GPa)
Inconel 718	$6x10^5$	45	7872	448	7,31	202
Aluminum	$3x10^5$	237,5	2689	951	23,8	69

Our main objective is to handle stresses in operating mode caused by thermal expansions due to thermal conductivity and other design considerations made up various constituent materials. For the compensation of this stress in the mounting parts first we investigated simple rigid coupling. The thermal-mechanical FE analysis is displayed in Fig. 1.

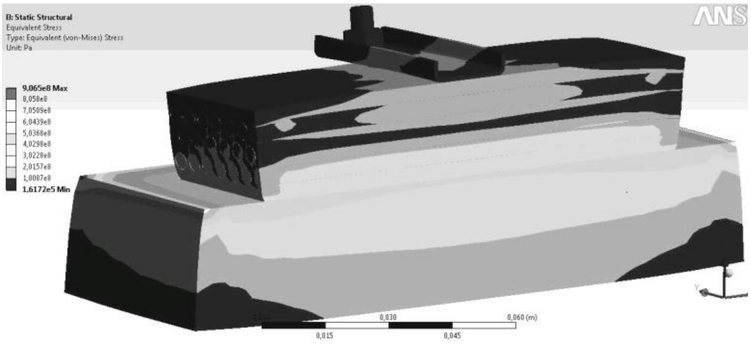

Figure1. *Resulting summary of maximum thermal stress intensity of the FEM analysis with half symmetry*

Development of the Lab-View Based System Optimizing Planning

The elaborated program uses graphical interface for the calculation of exerted forces in the elements of the construction. The target sequence for performing automated calculation of parametric design practice method and software development for calculating the load-dependent behavioral characteristics.

We use in this case these steps of the developing method:

1. Engineering safety analysis,
2. Time-dependent stress distribution,
3. Strength calculation on the finite element model of the design,
4. Calculations to make the design withstand the operational transients.

Plant data pertaining to individual vessel elements, based on individual vessel elements data, the program conducts a condition-oriented lifetime analysis for various degradation mechanisms which may occur in power plants. This process is supported via an intelligent user interface, with transparent display of the design and key data powerful analysis functions (stress analysis, thermal-mechanical analysis functions), and a module for management and evaluation of examination results.

The user interface contains input fields where each material, condition and geometry variable can be specified for the calculation's equations. With pressing the start button under the input fields the program calculates and shows the numeric results for the exerted forces with the defined parameters. The block diagram of Figure.2 shows the main features of the system.

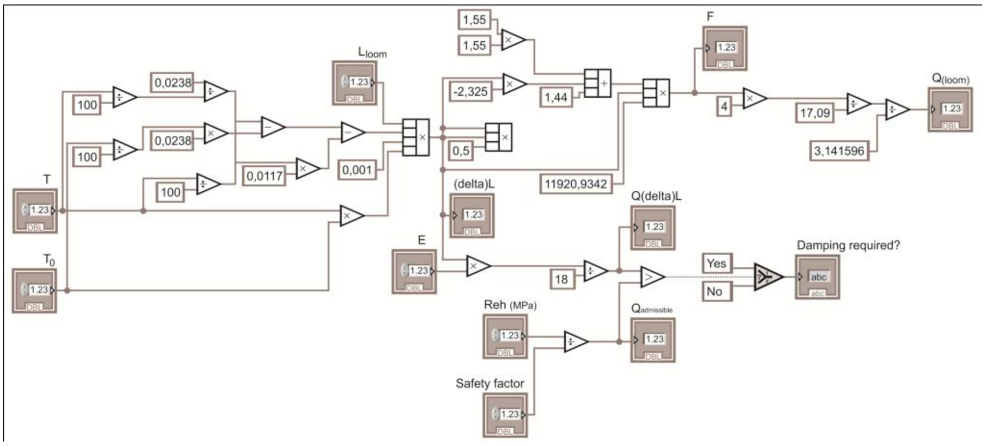

Figure2. *The block diagram of the optimization system*

The purpose of the software system:

- Transparent display of the design and data, as well as the continually updated as-is condition of the plant,
- Concentration of examination and maintenance activities in relevant system areas,
- Assessment of the effects of refitting work prior to its performance through the use of simulation calculations.

Checking Up on Results with Finite Element Analysis

The data used for the finite element simulation is also used for the analytic calculation as well. The finite element model's parts were heated up to 350°C. The model's bottom was totally fixed. The fixing was only providing vertical clamping due to the fact that the vacuum vessel's wall is built from the same material which expands with negligible difference. The results from the infinite model and the analytic calculations were similar where the difference was between 5-20%. Peak tensions were at the fixture of the bottom parts which was caused by the fixture which was necessary for the model's error-free operation.

CONCLUSIONS

In the course of planning we have solved the constructional implementation and the choice of materials taking account of the heat development in volume resulting from neutron radiation int he way that these meet the prescriptions for design. To meet technical requirements we made a steady-state thermal analysis using the software ANSYS, displaying the temperature allocations. We have made a structural analysis to evaluate the distribution of the intensity of tension. Using the results we have developed a program for creating a database to aid design which will support a transparent analysis of tensions arising at the effect of charges no later than at the stage of design.

REFERENCES

[1] C. Constanda, E.J. Bardo: Integral Methods in Science and Engineering, Springer Science & Business Media, 2013

[2] Mats G: The Finite Element Method: Theory, Implementation, and Applications, Springer, Berlin, 2013

[3] E. Onate: Structural Analysis with the Finite Element Method Linear Statics, Springer, Dordrecht 2013

[4] Y. Nakasone, S. Yoshimoto, T. A. Stolarski: Engineering analysis with ANSYS software, Elsevier, Oxford, 2008

AUTHOR'S BIOGRAPHY

Judit Szalai received her MSc degree in Industrial Engineering at the Budapest University of Technology and Economics (BME), Budapest, Hungary in 2011. Currently she is a PhD student at the Department of Machine and Industrial Product Design, at BME.

Lately Ms. Szalai is participating in a fusion engineering research and implementation projects focusing on thermal-mechanical analysis and method development for engineering systems including stress analysis, structural optimization, failure, and fatigue analysis.