SAND2007-4018C

Finite Element Stress Analysis OF ITER Module 13

Joe Garde[a], Dennis Youchison[a], Greg Natoni[a], Jim Bullock[a], T.J. Tanaka[a], Manmeet Narula[b], Alice Ying[b], M. Sawan[c], P. Wilson[c]

[a]*Sandia National Laboratories, Albuquerque, NM, USA*
[b]*University of California, Los AngeleS, CA, USA*
[c]*Fusion Technology Institute University of Wisconsin-Madison, MADISON, WI, USA*

***Abstract*— Of the 18 module designs in ITER, the US is responsible for three. Each of these modules will be designed to meet requirements established by the ITER international organization (ITER IO). Finite element analysis (FEA) is being utilized to ensure that the module designs are in compliance with the strength requirements established by ITER IO. The strength requirements are defined in terms of maximum allowable stress and strain conditions under loading scenarios determined by ITER IO. These allowable conditions are based on material properties and the expected frequency of the specific loading condition being investigated. This paper will present the FEA approach applied to the design of Module 13. The thermally induced stress distributions caused by ITER operating conditions and internal pressure of cooling fluid will be presented. Stresses caused by electromagnetic forces on the module will also be presented if available. The stress levels under these conditions will be compared to the allowable limits defined by the ITER IO.**

I. Introduction

The US ITER team is currently designing modules 7,12 and 13 of the ITER blanket system. Each module consists of a first wall (FW) section, which faces the plasma and a shielding section located behind the first wall. The modules will be subjected to thermonuclear, electromagnetic and pressure loads. The modules will contain internal cooling passages to accommodate the high thermal loads [1]. The design of a shield module must demonstrate acceptable stress levels as defined by the ITER international organization (ITER IO)[2]. The temperature and coolant pressure distributions in module 13 during operating conditions have been analyzed using computational fluid dynamics (CFD) software. Heating loads used in the CFD analysis were determined by neutronics analysis performed using operational conditions defined by ITER IO[3]. Finite element analysis (FEA) is being used to determine the stress levels in the US module designs. The current design iteration of module 13 has been analyzed using pressure and temperature distributions calculated for ITER

II. ITER IO Elastic Analysis Stress Limits

For the purpose of designing ITER components, ITER IO has defined analysis methods, acceptable stress levels and material properties [2]. Allowable stress limits, are defined for each material as a function of temperature, fluence and loading scenario. The Sm stress limit, defined for each material, applies to primary stresses calculated using elastic analysis. Stress levels caused by the combination of primary and secondary stresses are limited to 3Sm. Other stress limits are provided by ITER IO to address irradiation effects on material properties, geometric stress discontinuities and cyclic and dynamic loads [4]. For the purpose of analyzing and refining the current design of module 13, primary stresses are compared to the Sm stress limit and combined primary and secondary stresses are compared to the 3Sm limit.

III. Finite Element Analysis

A. FEA Approach

The preliminary FEA elastic stress analysis of module 13 has been performed using Abaqus FEA software[5]. The meshes used were created with CFdesign software[6] which was used to solve the temperature distributions and coolant pressure and flow distributions during operating conditions. The calculated temperature distributions were transferred along with the mesh into Abaqus in order to solve for the thermally induced stresses in the components. To investigate the stress levels caused by the coolant flow, the maximum coolant pressure was applied to all internal coolant-channel surfaces.

B. Geometry and Mesh

The three-dimensional geometry of the modules has been modeled using CATIA V5, R16. Figure 1shows the CATIA model of the module 13 shield block.

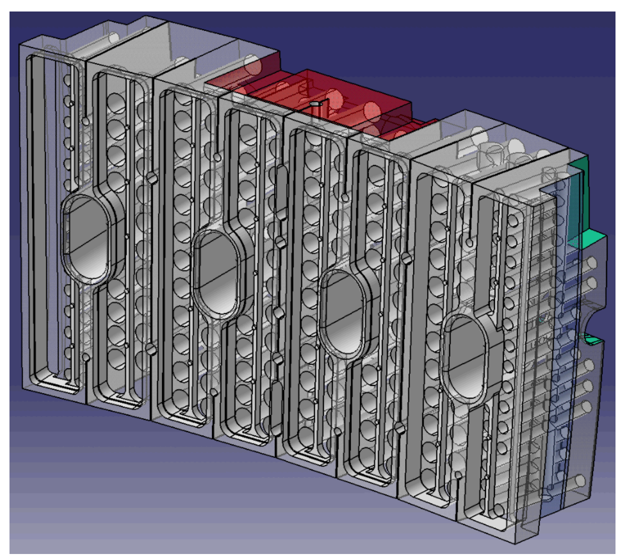

Figure 1. CATIA model of module 13 shield block.

Two sections of the shield block were chosen for the elastic FEA. A corner section of the geometry and a section in the upper center region were selected under the assumption that the analysis results would be representative of the entire shield block. Figure 2 shows the corner portion of the block that was used as the geometry for an FEA mesh.

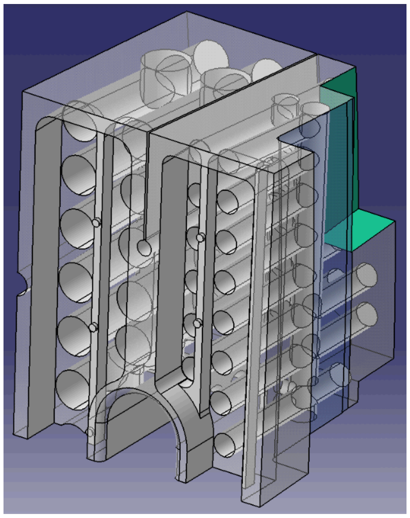

Figure 2. Geometry of module 13 shield block corner used for FEA mesh.

Meshes used for FEA stress analysis were taken from the computational fluids and thermal analyses performed using CFdesign. The meshes consisted of linear tetrahedral meshes with a nominal element size of 3mm. The meshes used for the shield block sections and the first wall section are seen figures 3-5.

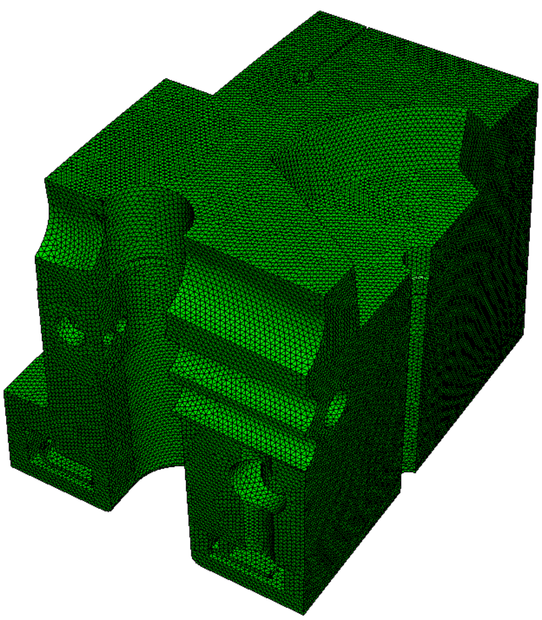

Figure 3. Tetrahedral mesh of module 13 shield block corner section.

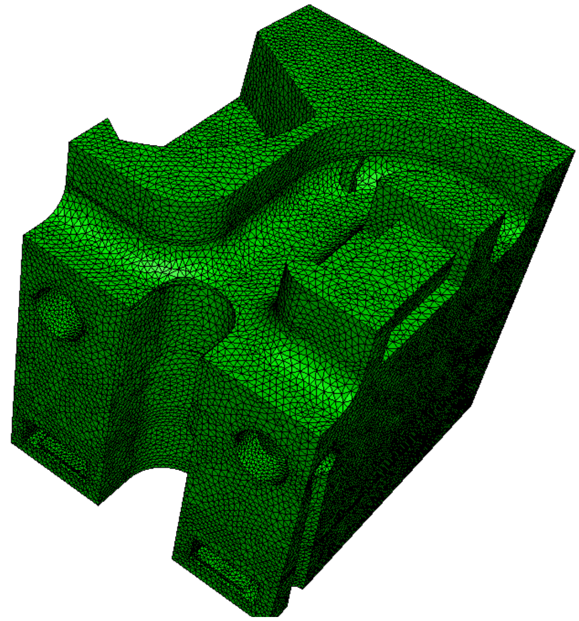

Figure 4. Tetrahedral mesh of module 13 shield block middle section.

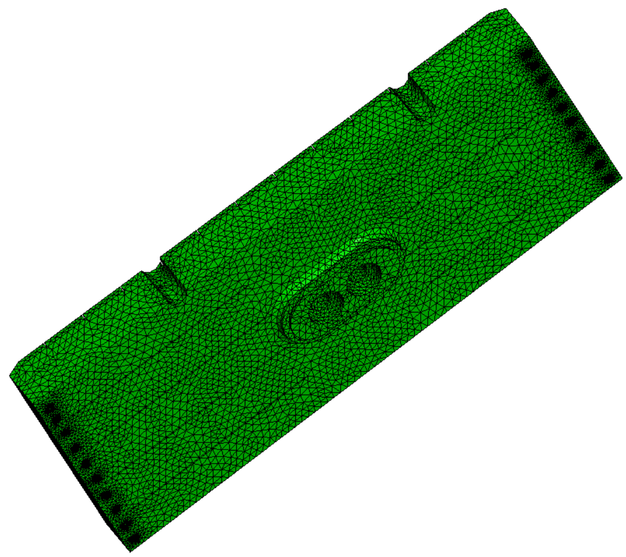

Figure 5. Tetrahedral mesh of module 13 first wall section.

C. Materials

Material properties used were consistent with ITER IO specifications [4]. Temperature dependencies of properties were accounted for in the analysis.

The shield block is austenitic stainless steel type 316L(N)-IG. The IG suffix is used to denote ITER grade which has additional specifications dictated by ITER IO[4].

The first wall is constructed of three materials. The plasma facing layer is Beryllium (S-65C). The beryllium tiles are joined to a layer of copper-chromium-zirconium alloy (CuCrZr-IG). The CuCrZr-IG layer is bonded to a stainless steel 316(N)-IG base. Stainless steel 316(N)-IG coolant tubes are integral to the CuCrZr-IG layer [1] .

D. Loads and Boundary Conditions

Faces where the shield block sections were cut from the model geometry were restrained in one degree of freedom, as is typical with symmetry. The corner node shared by these faces was fixed in three degrees of freedom. While the modeled geometries do not meet the requirements of symmetry, these restraints allow for an initial investigation of stress levels and distributions.

The first wall model represents a section that is structurally independent from neighboring components. Analysis of the first wall section was based only on stresses caused by temperature gradients. These parameters allowed the first wall analyses to utilize the stabilize feature in Abaqus. No additional restraints were required.

To find thermally induced stresses, temperature distributions were taken directly from the CFD results. A temperature was applied to each node in the mesh. Stresses caused by coolant pressure in the shield block were investigated by applying a 3MPa pressure to all internal coolant channel surfaces.

CFD results of temperature distributions during operating conditions are shown in figures 6-8.

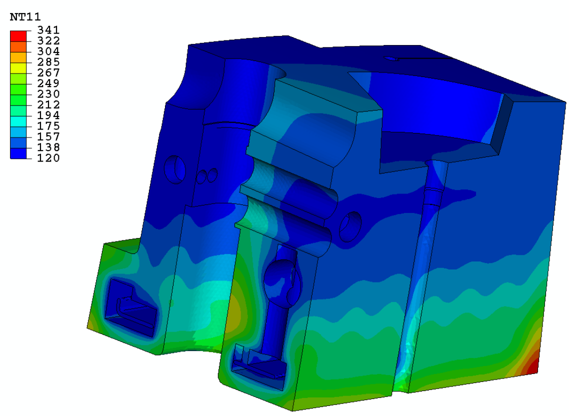

Figure 6. Temperature distribution of shield block corner section (^{0}C).

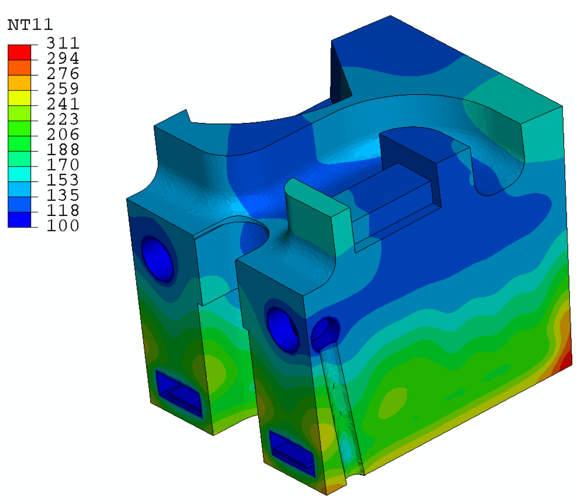

Figure 7. Temperature distribution of shield block middle section (^{0}C).

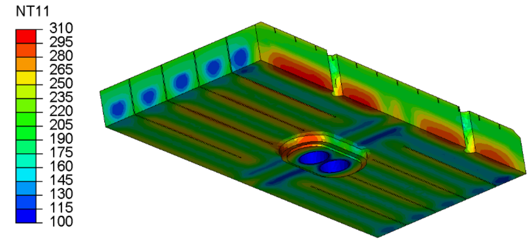

Figure 8. Temperature distribution of first wall section (^{0}C).

E. Stress Results

The FEA results showed acceptable stress levels in most of the analyzed geometry. Tresca stress intensities were compared to the Sm and 3Sm stress limits given for each material. The maximum temperature in each analysis was used to define Sm for the entire material volume rather than comparing local temperatures to local stresses. Sm stress limits for each analysis are listed in table 1.

TABLE I. SM STRESS LIMITS

FEA section	Max temperatur e (^{0}C)	Sm at max temperature (MPa)	3Sm at max temperature (MPa)
Shield block corner ss316L(N)-IG	340	109	327
Shield block middle ss316L(N)-IG	311	113	339
First wall ss316L(N)-IG	305	114	342
First wall CuCrZr-IG	224	104	312
First wall Be(S-65C)	262	75	225

The maximum stress in each analysis was caused by local mesh or geometric discontinuities. Figures 9-11 show stress plots displaying only results with Tresca stresses over 3Sm. These results are from the FEA analyses using steady state temperature distributions at operating conditions. The analyses performed with 3MPa pressure on the internal coolant channel surfaces revealed no stresses over the Sm limit for primary stresses.

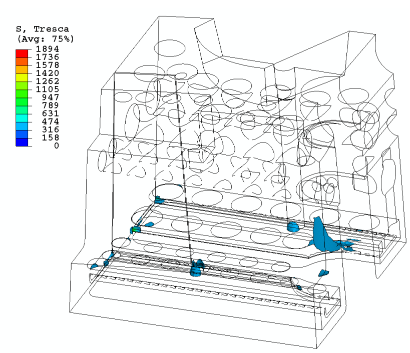

Figure 9. Thermally induced Tresca stress intensities above 3Sm limit in shield module corner section.

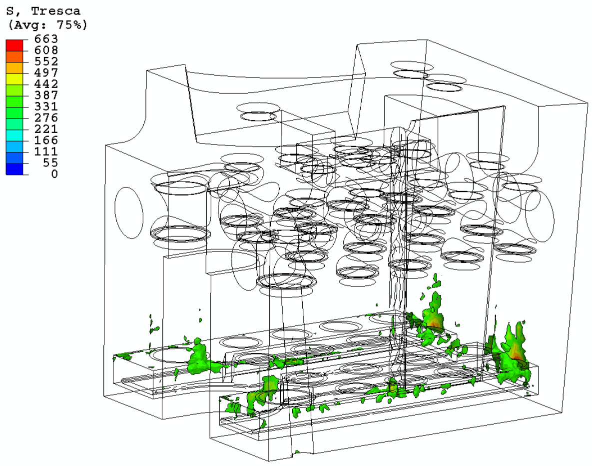

Figure 10. Thermally induced Tresca stress intensities above 3Sm limit in shield module mid section.

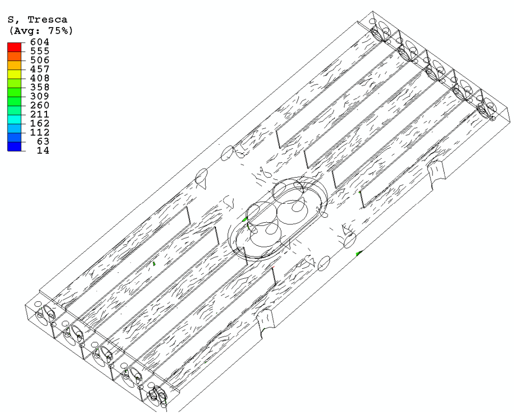

Figure 11. Thermally induced Tresca stress intensities above 3Sm limit in stainless steel section of first wall.

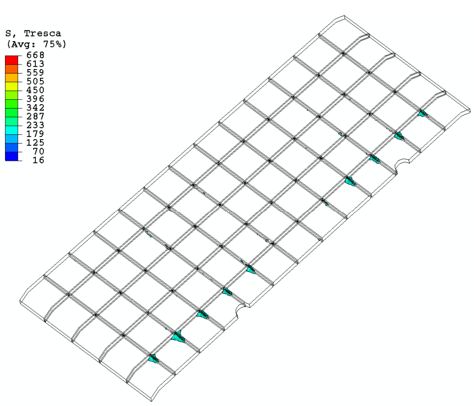

Figure 12. Thermally induced Tresca stress intensities above 3Sm limit in Be(S-65C) section of first wall.

IV. Discussion

This initial series of analyses has served two purposes. The results of nuclear heating calculations have successfully been used to solve temperature and pressure distributions using CFD software. These CFD results have in turn served as loading conditions used in FEA analyses to determine stress levels in module 13. This effort has demonstrated the ability to determine stress intensities from nuclear heating and coolant flow. Results of the initial elastic FEA analyses of module 13 also indicate that the stress intensities are generally under the Sm and 3Sm limits dictated by ITER IO for primary and secondary stresses during operating conditions.

The results of these analyses will contribute to the next design iteration of module 13. The areas of high stress intensities will be examined to determine if design changes can bring all of the stress below the Sm and 3Sm limits.

The next series of stress analyses will require greater rigor. Temperature and pressure loads will be included along with electromagnetic forces to determine the stress levels on modules 7, 12 and 13. The CFD results from the next iterations will be mapped onto meshes better suited for FEA of mechanical stresses. These meshes will model larger portions of the modules while refining element sizes near smaller geometric features.

The Sm and 3Sm limits for elastic analysis are appropriate for refinement of initial designs. As the design matures additional stress limits dictated by ITER IO will require investigation. These include specific rules for stress concentrations, cyclic loading, irradiation hardening, dynamic forces and other design considerations.

V. Conclusion

An initial finite element analyses has been performed on the current design iteration of ITER module 13. This effort has demonstrated the ability to utilize neutronic calculations and CFD analyses to produce FEA stress results. Examination of the results reveals that the stress distributions in module 13 under operating conditions are generally at an acceptable level. Upcoming design iterations will be analyzed using this general method with the inclusion of electromagnetic forces. As the design evolves, compliance with other ITER IO specified loading scenarios and the applicable stress limits will be pursued.

VI. References

[1] "ITER detail design descriptions 1.6 Blanket", edition 2004.

[2] "ITER structural design criteria for in-vessel components (SDC-IC)", 2004.

[3] M.E. Sawan, B. Smith, P Wilson "Neutronics assessment of the ITER first wall/shield" unpublished.

[4] "ITER materials properties handbook (MPH-IC)".

[5] ABAQUS is a software product developed by ABAQUS, INC, a sbsidiary of Dassault Systems S.A., URL http://www.abaqus.com.

[6] CFdesign is a software product of Blue Ridge Numerics, Inc., URL http://www.cfdesign.com.