Design Optimization of Table Top Tokamak Support Structure using Optistruct

[1]Ravi Pragash.N, [2]C.D.Sankhavara
[1]PhD Research Scholar, [2]Director
[1]RK University, Rajkot, Gujarat, India
[2]Insight Education, Rajkot, Gujarat, India

Abstract : ***In general, Tokamaks are operated for various plasma confinement experiments all around the world since many decades. However, the complexity of such machines are too high to handle in terms of design and operation. Being the fourth state of matter, the plasma state is very challenging in engineering terms when it is produced and confined in such tokamaks. The major constraints are being the machine volume itself to accommodate enough diagnostic and other subsystems, and the support structure of the machine. The support structure requires a full scale design optimization if possible to maximum possible levels in both weight and size but still withstanding the operational structural loads due to plasma instabilities. Such an attempt is made in optimizing the support structure of a Table Top Tokamak (TTT) that is designed as a prototype. The software used is the Optistruct module from Hyperworks software which helps to carry out Design optimization using Finite Element Methods on any defined model. This paper reveals the description of using such software capability to optimize the structural design of supports for the TTT machine, so that the procedure can be applied to any large tokamak or even a Fusion reactor. In this case study, the Optistruct software is used to reduce a mass of about 12 kg from a 27.5 kg designed structure by means of optimization technique. In many applications, it is observed that a reduction of mass of about 45 % is achieved from the mechanically designed structure when the Optistruct method is used. The major mass reduction of components plays major role in rocket payloads, aerospace components and in novel applications like Robotics and tokamak applications. The procedure of applying Optistruct is explained in a way that the readers will be able to optimize any structure required after learning the modules and steps.***

IndexTerms – Optimization, Table Top Tokamak, Optistruct, Tokamak Support Structure

1. INTRODUCTION

The Table top tokamak (TTT) is designed using defined physics and engineering parametrical design procedures. The tokamak is supported using a frame-like support structure that is designed to withstand various plasma loads and dead weight of the vessel and magnets and other subsystems. Although, the support structure is designed using the standard procedures of engineering calculations, the structure is required to undergo detailed process of optimization.

2. DESIGN OPTIMIZATION :

The conceptual design of the frame for the Table top Tokamak is arrived at using the estimated load specification [1]. However, an approach towards the optimization of the frame design is carried out using Optistruct, a finite element Optimization Software from Hyperworks Software. This paper brings out the necessary steps that are involved in optimization of the frame. First step is to import the geometry file as shown in Fig.1. Then using 2D panel shown in right hand side shell Meshing is to be done as in Fig.2.

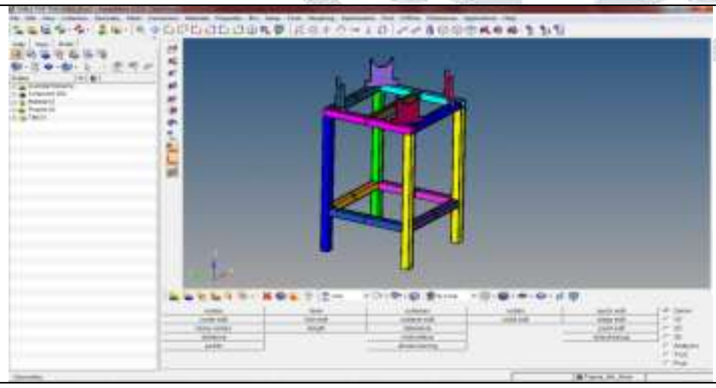


Figure 1 - Import of the basic frame geometry

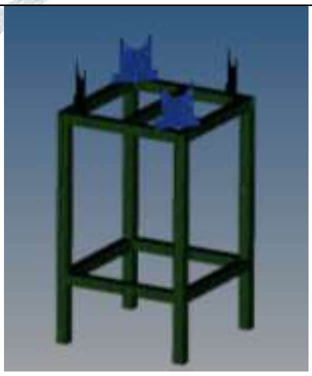


Figure 2- 2D Shell Meshing

For Optimization purpose, the frame is considered as a Box open in bottom side and thickness 50 mm & upper mounting thickness taken as 16 mm as designed as in Fig.3 and meshing is done as per the consideration. Then, the next task is to create Material properties and assign material properties. Give the thickness to individual component collector & then Assign the material and property to generated mesh as shown in Fig.4. The upper Mountings connected with RBE2 elements to apply the required load and moment at a single point by creating the Load collectors for LOAD, GRAVITY, MOMENT and CONSTRAINTS menu to give combination of Load & moment together. As shown in Fig.5, apply the dead load of 200kg & moment of 100 N-mm at the single point joining all RBE2 elements and constraint all DOF (rotational and translational) at the base of the frame [2]. Apply Gravity to the structure, that is self weight of the frame as the second load step.

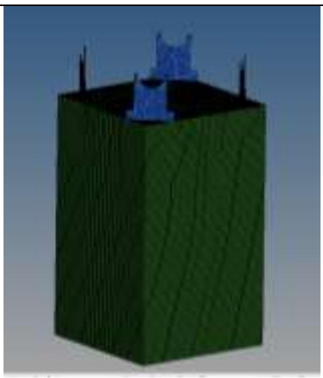


Figure 3 Box Modeling of frame

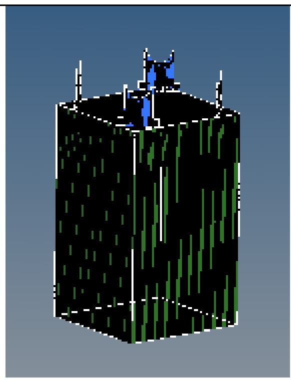


Figure 4 Assigning Material properties

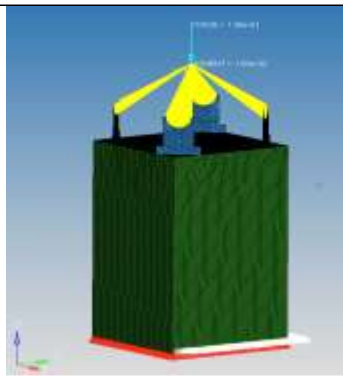


Figure 5 Gravity Loading on Model

The next major step is to do the Analysis using Optistruct module to run the analysis. When the analysis is completed successfully, Hyperview module has to be used to view the results of Von-Mises stresses and displacement as shown in Fig.6 & Fig.7 respectively.

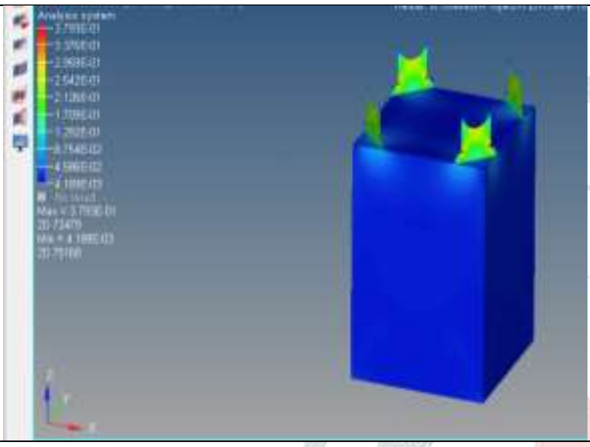


Figure 6 Von-Mises Stress = 0.3793 MPa

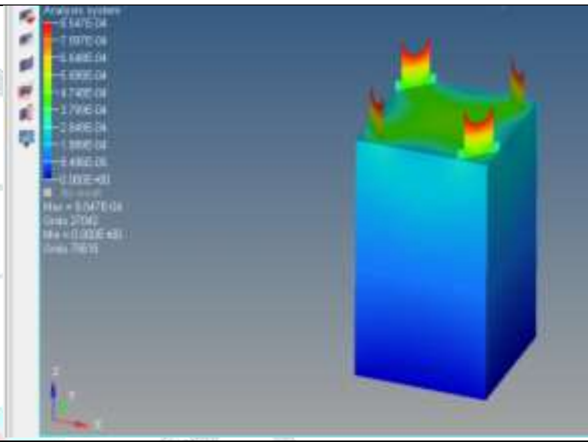


Figure 7 Displacement = 8.54 E -04 mm

3. SIZE OPTMIZATION :

In order to initiate the size Optimization of the assembly, only the thicknesses of box open at bottom side are considered whereas upper four mountings thicknesses are not considered. Next step is to give the properties of thicknesses that are to be optimized & the range of thicknesses minimum and maximum as 2mm to 3 mm. Similarly, constraint of parameter stress value of maximum 8 MPa is given. Now the two responses for optimization namely displacement & Compliance are created on model by selecting all nodes. After checking the parameters of size optimization prepared, the process of optimization in the Optistruct module is started. When the optimization is completed, a message will be displayed as " FEASIBLE DESIGN WITH ALL CONSTRAINTS SATISFIED " if the optimization process is successfully done. If displayed message is " INFEASIBLE DESIGN SOME CONSTRAINT VIOLATED " means, more iterations are to be done to achieve the desired results [3].

Further, the results are viewed in the Hyperview by same method explained above and the results are shown below. The thickness plot shown below in Fig.8, satisfies the given constraint with the 3 mm thickness of the BOX open at bottom side keeping the constant thickness (16 mm) for the upper four mountings.

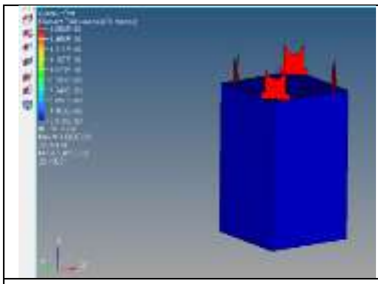


Figure 8
Constraints of Thickness

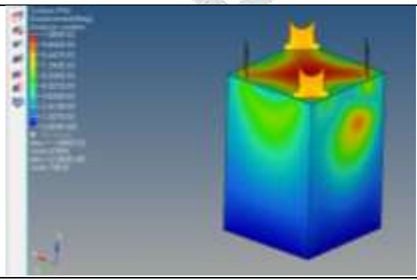


Figure 9
Plot showing thickness

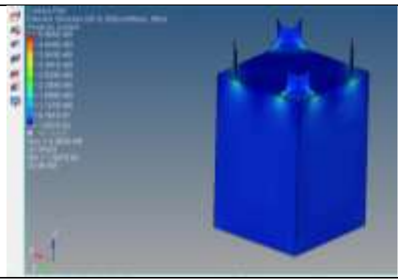


Figure 10
Plot showing Von-Mises

The displacement obtained as 0.01 mm as shown in Fig.9 and the Von-Mises stress of 5.05 MPa shows that the FEASIBLE DESIGN through size optimization is completed. Keeping the thickness 3mm for the box, and turning the topology optimization in which the design and non-design areas are defined in which the unnecessary material will be removed by satisfying constraints. If constraints are not satisfying or better FEASIBLE DESIGN is considered, then more number of modifications in design areas and number of iterations are initiated to get the FEASIBLE DESIGN as the final result.

4. TOPOLOGY OPTIMIZATION:

In this topology optimization process, the material removal areas are defined. The design area is the volume in which material should be removed whereas non-design area is the volume where the material can be removed in the process of material optimization as shown in

Figure-11. The maximum value of stress constraint parameter is given as 10 MPa. Now as defined in above section, two responses namely Compliance & Volumefraction are defined. The response volume & upper bound 0.4 as per the iso-surface standard value are defined and thus the deck for topology optimization is prepared.

Now, running the analysis for the optimization process is initiated to get the Feasible Design. The results of Feasible design in different iterations after the optimization using FEA are shown below figures. The iterations of design optimization for various frame designs are explained in the following table, referring Fig.11 to Fig.31.

Iteration – 1		

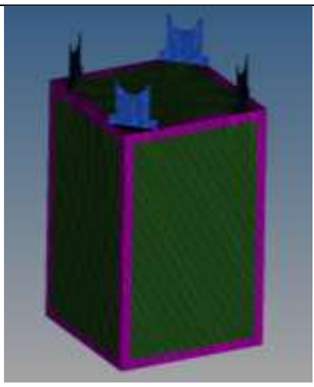
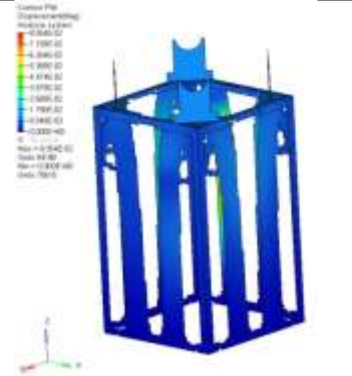
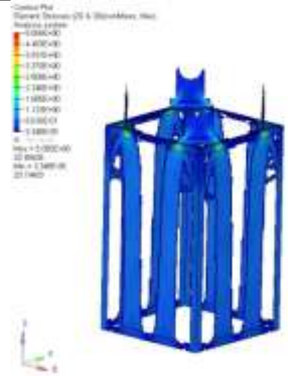

		

Figure 11 Design and Non-Design Areas	**Figure 12** Displacement=0.08 mm	**Figure 13** Von-mises stress=5.05 MPa
Iteration - 2		

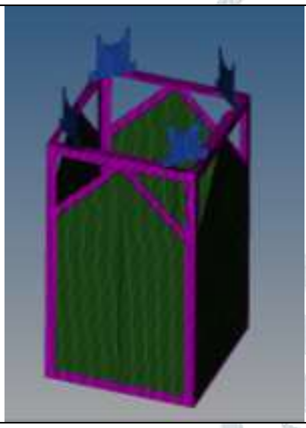
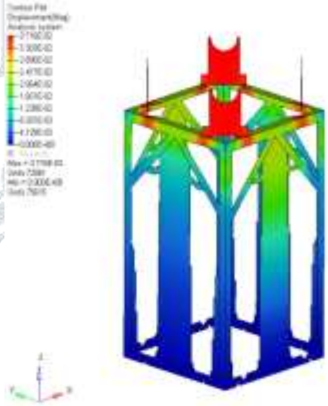
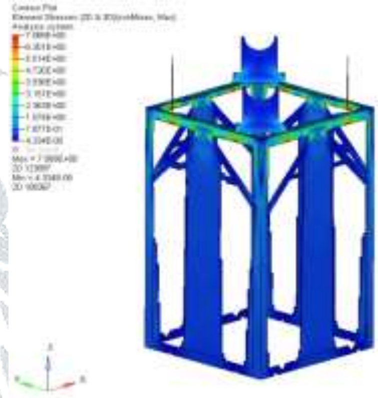

		

Figure 14 Design and Non-Design Areas	**Figure 15** Displacement = 0.03716 mm	**Figure 16** Von-Mises stresses = 7.089 MPa
Iteration – 3		

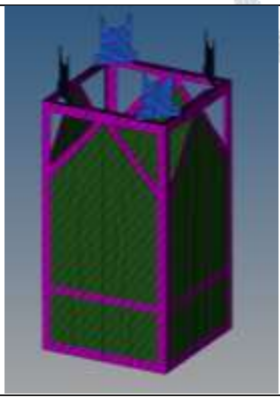
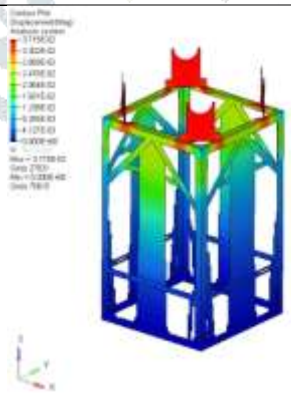
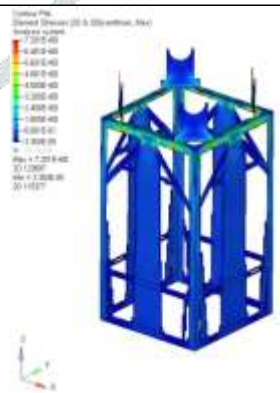
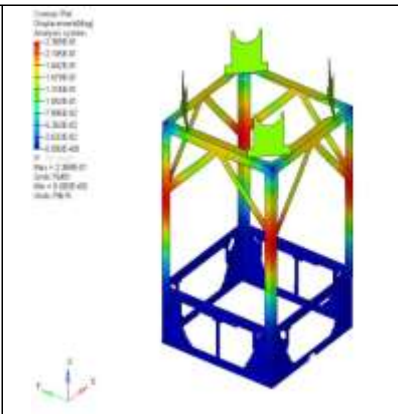
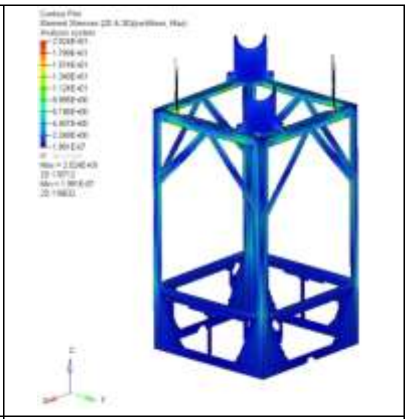

		

Figure 17 Design and Non-Design Areas	**Figure 18** Displacement = 0.037 mm	**Figure 19** Von-Mises stresses = 7.201 MPa
Iteration – 4		

		
Figure 20 Design and Non-Design Areas	**Figure 21** Displacement == **0.2369** mm	**Figure 22** Von-Mises stresses =**20.24** MPa
Iteration – 5		

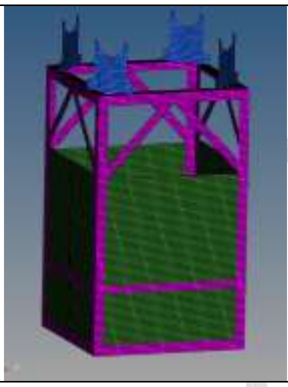
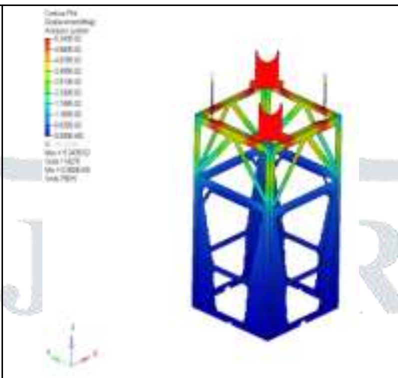
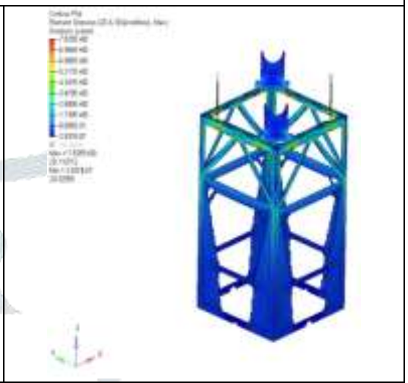

		

Figure 23 Design and Non-Design Areas	**Figure 24** Displacement = 0.0524 mm	**Figure 25** Von-Mises stresses = 7.825 MPa
Iteration – 6		

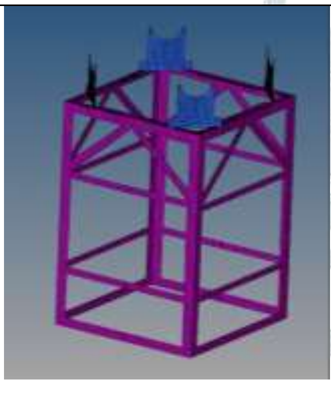
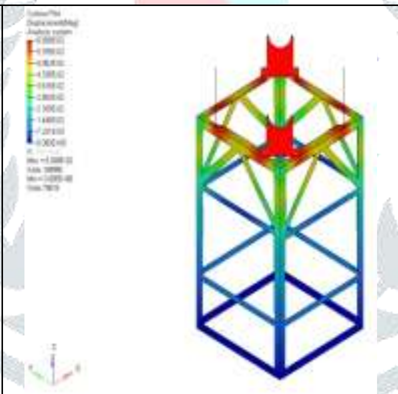
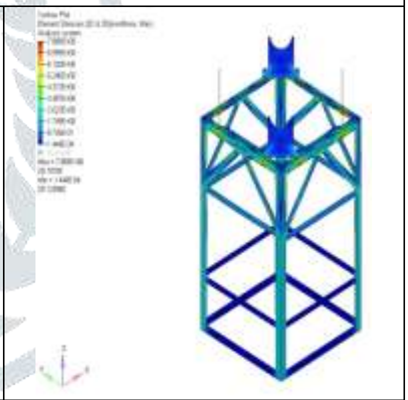

		

Figure 26 Design and Non-Design Areas	**Figure 27** Displacement = 0.065 mm	**Figure 28** Von-Mises stresses = 7.869 MPa
Iteration – 7 (Final Feasible Design)		

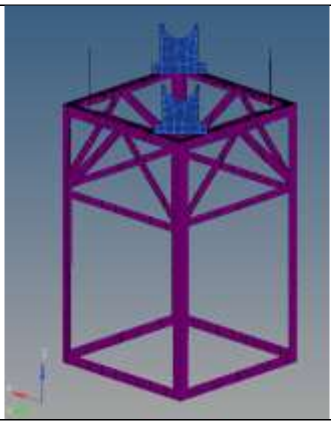
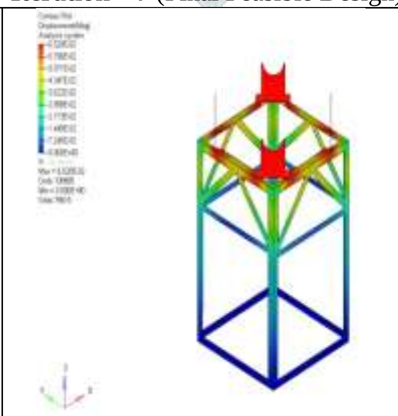
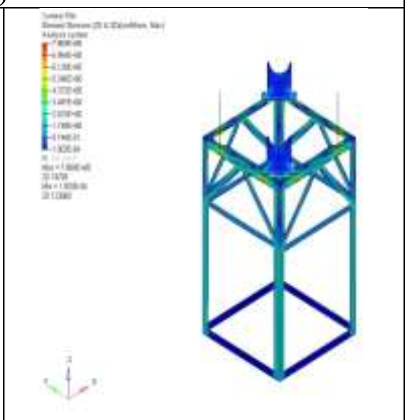

		

Figure 29 Design and Non-Design Areas	**Figure 30** Displacement = 0.0652 mm	**Figure 31** Von-Mises stresses = 7.869 MPa

IV. RESULTS AND DISCUSSION

The process of design optimization on the support structure has resulted in reducing the weight of the frame by 12 kg with almost 45% weight reduction in the designed frame structure. Using L-angles , the new frame design is satisfying the given constraints with the

Feasible design structure criteria. The material being Aluminum is found to have an allowable stress of about 35 MPa. The Maximum stress observed on new frame structure 7.869 MPa, which is well within the allowable limits. Hence the design optimization is successful in optimizing the structure of such complicated structures with complexity in loading conditions during the machine operations. Further, such optimization techniques can be repeated for more load combinations and other structural loads. If applied for large tokamaks like ITER or DEMO like fusion reactors [4], these design optimization techniques will result in reducing structural parameters and heavy structures, resulting in the only required physical structural masses to withstand the defined load combinations which the other way not possible by hand calculations of design. Apart from these applications, this technique will be highly effective if used in Robotics, Remote Handling and Aerospace applications where weight reduction is considered to be very critical criteria.

REFERENCES

[1] M.P.Gryazevich,E.Del Bosco,A.Malaquias,G.Mank,G.Van He,A.Hirose,M.hron, B.Kuteev Oost, Yexi,G.O.Ludwig,C.Silva,G.M.Vorobyev,"Joint Research using Small Tokamaks",2005

[2] Anatoly Panina , Wolfgang Biela , Philippe Mertensa , Francois Nuniob , Louis Zanic "Mechanical pre-dimensioning and pre-optimization of the tokamaks' toroidal coils featuring the winding pack layout", Preprint of Paper to be submitted for publication in Proceedings of 29th Symposium on Fusion Technology (SOFT 2016)

[3] D. S. Lee, Y. D. Kwon, J. S. Han, "Design Optimization for Shear Key on the TF Coil Structure of the KSTAR Tokamak", Strength of Materials, January 2016, Volume 48, Issue 1, pp 32–38

[4] YOUNG-DOO KWON, DAE-SUEP LEE, "A Study On Design Optimization Of Conical Bolt In The TF Coil Structure Of The KSTAR Tokamak", International Journal of Modern Physics B, Volume 20, Issue 25n27, 30 October 2006