International Journal of Engineering Science Invention
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 4 Issue 8|| August 2015 || PP.33-38

Static Analysis of Mobile Launch Pedestal

P. Devkiran[1], Dr.B.Jayachandriah[2], KNVSN Kumar[3], P.Selvaraj[4]
[1]*M.Tech Student, CAD/CAM, SKIT, Srikalahasti, Chittoor dist, A.P. India*
[2] *principal and Professor, Mechanical Engineering SKIT, Srikalahasti, Chittoor dist, A.P. India*
[3] *Scientist/Engineer, SCEND & ASG, SDSC SHAR, ISRO*
[4] *Group Director, SCEND & ASG, SDSC SHAR, ISRO*

ABSTRACT*: This Heavy Launch Vehicle is assembled over a structure called the Mobile Launch Pedestal (MLP) inside the Vehicle Assembly Building (VAB). After assembly, the rockets are to be transported to the Launch Pad for launching Vehicle. Since, the vehicle is supported on the pedestal till launch, the pedestal is designed for the stiffness criteria and the design is checked for the stress developed. As the pedestal is designed based upon the stiffness, it is very high. The weight of the launch pedestal is 800 Tons. The pedestal deck is fabricated using steel plates of various thicknesses ranging from 6 mm to 120 mm. This design is already realized and also used for an experimental launch. The Static Analysis of Mobile Launch Pedestal (MLP) deck is done by using ANSYS 15. The objective is to finding the stress and deflection values. This value can be compared with the previous achieved values. This work is used weight optimization of the Mobile Launch Pedestal (MLP) in future.*

Key words: *VAB, MLP*

I.INTRODUCTION

The Mobile Launch Pedestal (MLP) is a large steel structure positioned inside VAB during integration of launch vehicle. The pedestal will also provide as a working platform and supporting system for integration activities of the vehicle. After assembly in VAB, the vehicle is to be transported to the Launch Pad via track for lift-off at launch pad. The MLP also provides rigid support to the vehicle during the launching operation, thereby exposing itself to large thrust load for a short time span.

A bogie system is provided for mobilizing MLP to launch pad. The MLP is anchored to ground to withstand wind conditions and launch loads. The bogie assembly is moved back to storage area after anchoring the MLP structure at launch pad. After completion of launching, the bogie system is brought to launch pad and attached to MLP structure and then is transported to VAB.

II.CONFIGURATION OF MLP

The MLP contains of two strap-on support rings over the deck to support the S-200 strap-on motors. The overall size of the MLP is 19.2mX19.2mX6.8m. The top deck is 19.2 x 19.2 m in plan and provided with two circular opening diameters of 4.1 m. The general height of the deck is 4 m. The distance between the two legs is 18 m and the distance between bogies is 14m as shown in fig 1 & 2. The overall weight is 800 T.

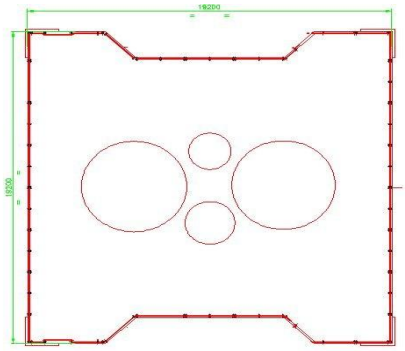

Fig:1 MLP (PLAN)

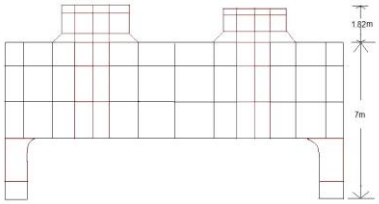

Fig:2 MLP (ELIVATION)

III.MODELING:

The quarter model of MLP deck is analyzed using ANSYS Design Modular (DM). The structure and the boundary condition are symmetric on both the axes (X and Z). The quarter model of MLP is created as shown in fig.3 and later this quarter model i s used for Analysis with symmetry boundary conditions. Shell63 element [2] is used for modeling. This element has both bending and membrane capabilities. The element is defined by four nodes and each node has six degrees of freedom. The geometric model is discretized to finite element model and is shown in fig.4. The plate thickness values of MLP are assigned after modeling is completed.

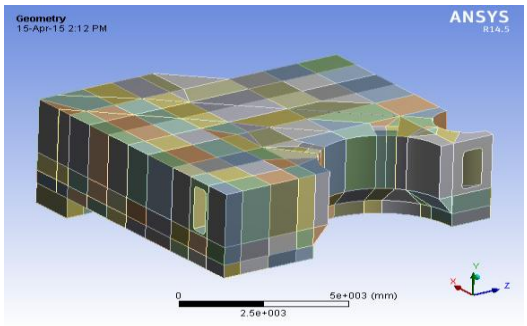

Fig: 3 MLP (quarter model)

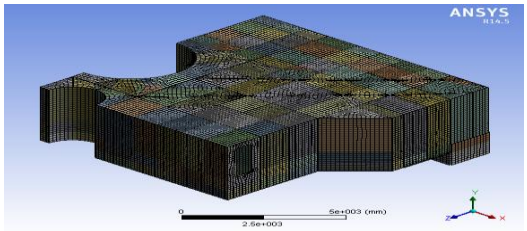

Fig: 4 MLP(meshed model)

The original thickness of the steel plates ranges from 6 mm to 120 mm.

IV. MATERIAL PROPERTIES:

The material properties used for MLP is Structural steel plates conforming to IS 2062. The properties are listed in the following Table No. 1.

Table:1 structural steel details

Material	Property	value
Structural Steel	Young's Modulus(kg/mm^2)	210000
	Density (Kg/m3)	7850
	Poisson's ratio	0.3
	Ultimate Strength (MPa)	410
	% Elongation	21

V. RESULTS

STATIC ANALYSIS

The quarter model of MLP is analyzed by using Finite Element Method [4].The symmetric boundary conditions are applied at the boundary of the symmetric planes X and Y. The structure is assumed as fixed at leg portion and loads are applied on MLP as show in fig 5

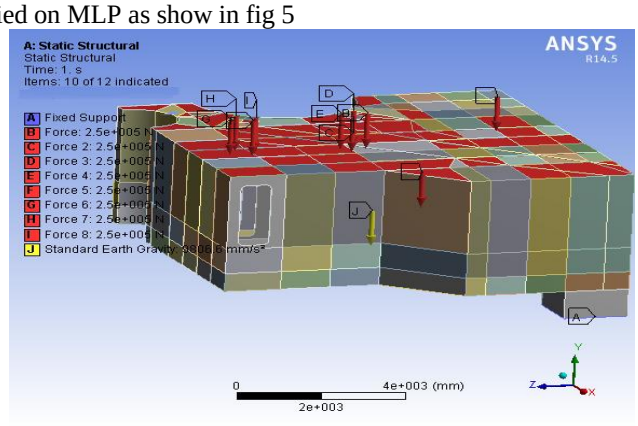

Fig: 5 Loads Acting On MLP

The stresses as well as deflection recorded when the total load is acting are shown in fig 6&7

Maximum Von-mises stress =121 MPa Maximum deflection =3.94mm Initial Volume of
mobile launch pedestal =1.93e10

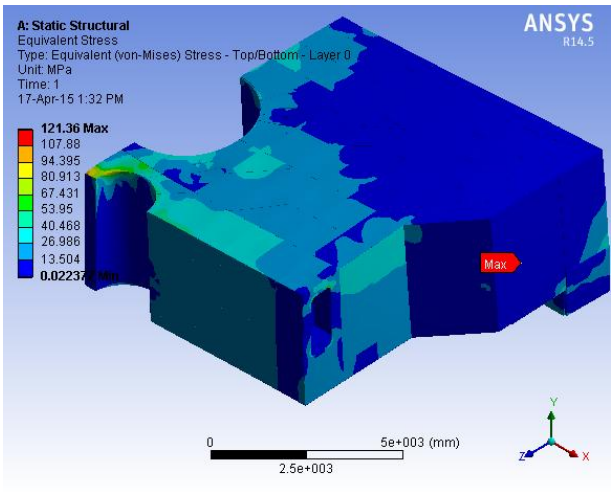

Fig:6 Von-Mises Stress

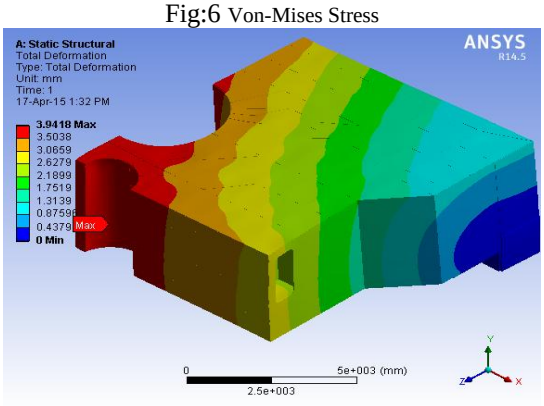

Fig:7 Total deflection

VI. COMPARING RESULT'S WITH

AUFRG'S:

The Geometric & analyzed project model is to be validated in order to perform the optimization. To evaluate this model, the present project model static analysis results are compared with the earlier available MLP analysis results which were carried out by the AUFRG team of Anna University, Chennai. The analysis of the AUFRG's model is also carried out in Ansys and the results for the same loading & boundary conditions are brought here. The comparison of stresses and deflections for validation are as illustrated in table 3.

Table:2 Comparison of Analytical models for stress & deflection

Description	Project model	AUFRG Model
Weight in (t)	158	150
Maximum von-mises stress (MPa)	121.8	121
Deflection(mm)	3.972	3.93

From the above table, it is seen that the deflection value obtained from the project model analysis is 0.42% more than the value from AUFRG model. Similarly, the weight is more by 0.2% than the AUFRG model. Since these values are well within 5% variation, the project model can be considered for the optimization. The deflection plots of the project model and the AUFRG model are shown in Figs 13&14. And while coming to the stress value obtained from the project model analysis is 0.8% more than the value from AUFRG model. The stress plots of the project model and the AUFRG model are shown in Figs 11&12.

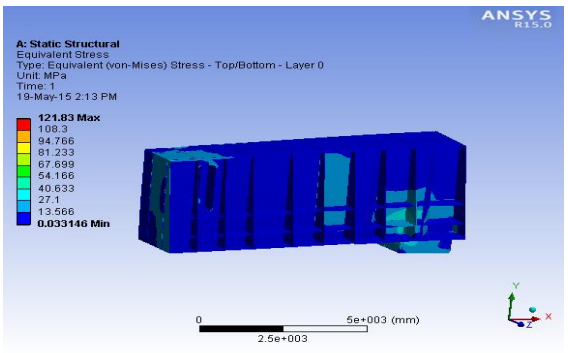

FIG: 8 Stress Value For Value Of Project Model

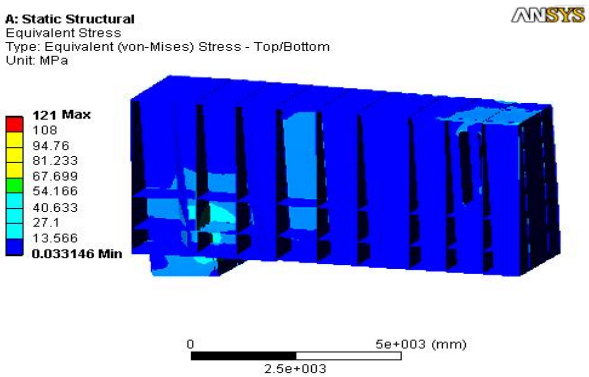

FIG: 9 Stress Value Of AUFRG Model

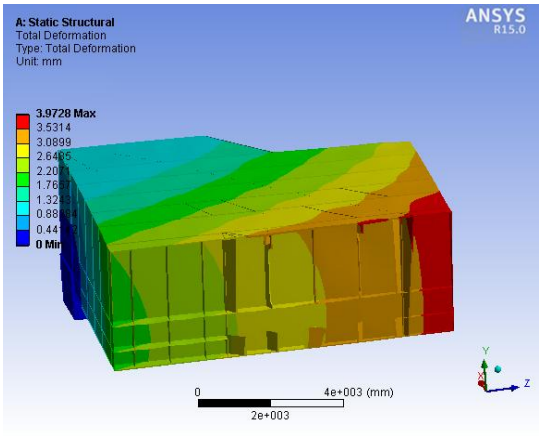

FIG: 10 Total deflection from project model

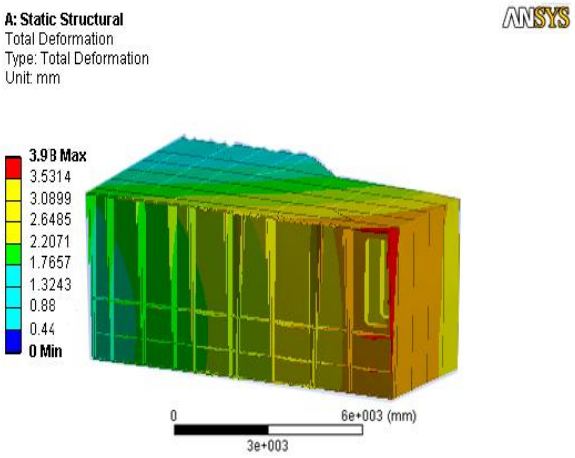

FIG: 11 Total deflection from AUFRG model

The above four figures 8,9,10& 11 are showing that the project model value is accurate, by comparing project model values of stress and deflection with the stress and deflection of the AUFRG model.

VII. CONCLUSION

1. The quarter symmetry model of MLP deck is carried out using Ansys design modeler.
2. The deck model is also validated in comparison with the AUFRG model and the results are satisfactory.
3. The static structural analysis of the model is carried out and the max. von mises stress is 121MPa and max. Z-deflection is 3.9mm.
4. The static structural analysis of Mobile Launch Pedestal (MLP) is used for future to reduce the weight of the MLP. By using the stress and deflection values we can do weight optimization.

.ACKNOWLEDGEMENT : The authors acknowledge the constant support and encouragement given by the Director, SDSC SHAR to publish this paper. Also the authors would like to acknowledge the extended support by the Technical Paper Review committee for their suggestions in improving the technical content of this paper.

REFERENCES

[1] Preliminary Design Document for MLP Structure.Doc No: MEC / 09E9 / SA / V1-2 / 34/01252Rev.1, Dated August, 2000 by MECON Ltd, RANCHI.
[2] ANSYS. Theoretical Manual for Design analysis.
[3] Identification Of Shear Cracks In Reinforced Beams Using Finite Element Method (Ansys) H.U. Khan, M.N. Rafique, S. Karam, K. Ahmad and A. Bashir

[4] Finite Element Stress Analysis Of Truck Chassis Using Ansys: Review Ahmad O. Moaaz1 and Nouby M. Ghazaly2 1Mechanical Engineering Dept., Faculty of Engineering, Beni-Suef University, Beni Suef -62511, Egypt 2Mechanical Engineering Dept., Faculty of Engineering, South Valley University, Qena-83523, Egypt

[5] Finite Element Analysis for Various Structures Made of Classic and Composite Material by Using ANSYS Software Nguyen Viet Hung, Thai The Hung, Luu Quang Thin, Bui Tran Trung, Luu Chi Hieu Hanoi University of Technology, Hanoi, Vietnam.

[6] Structural Weight Optimization Of A Bracket Using ANSYS Prashant Kumar Srivastava1 , Rajive Gupta2 , Simant3

AUTHORS:

Dr. B. Jayachandriah Professor and Principal at Srikalahasteeswara Institute of Technology, Srikalahasti. He has completed his masters in mechanical engineering from BITS Pilani College, specialization in IC Engines. He completed his PhD from JNTUH, Hyderabad specialization in IC Engines CFD. The author has more than 25 years' experience in Teaching & research in various subjects of Mechanical Engineering and he has guided about 25 Master's Thesis & number of B.Tech projects. He has credit to 50 publications in various national and International journals and conferences.

P. Dev Kiran M.Tech student at Srikalahasteeswara Institute of Technology, Srikalahasti. Perusing M.Tech 2nd year CAD/CAM branch with distinction marks. Project done under the guidance of Dr. B. Jayachandriah.
.