International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 11, November - 2013

Mechanical Performance of Test Fixture during Thermo Elastic Deformation Studies

P R Praveen[1]
M.TECH (Machine Design)
BMS College of Engineering
VTU Belgaum
India

T. S. Sriranga [2]
Scientist SG ISAC
ISRO Bangalore
India

H. Abdul Hameed [3]
Scientist SG ISAC
ISRO Bangalore
India

Dr. J. SharanaBasavaraja[4]
Mechanical
Associate Professor
BMS College of Engineering
India

Abstract

The Mechanical performance of test setup support is carried in MSC Patran and is mainly due to the thermal effect on the model due to on-orbit temperature excursion, the temperature gradient is obtained from the thermocouple during TED test process. The mechanical support is modeled in UNIGRAPHICS and MSC Patran and post processed in MSC PATRAN and NASTRAN. The model is optimized in OPTISTRUCT with mass as the objective and Frequency as the response in analysis.

Keywords: TED (Thermo Elastic Deformation), UNIGRAPHICS, MSC PATRAN, MSC NASTRAN, OPTISTRUCT, Frequency as the response in analysis.

“1. Introduction”

Tests are done to validate the design of shape sensitive structures in special environmental chambers that simulate the on-orbit environmental conditions. The support is the main part in the communication satellite and antenna reflector sits on the it also help the antenna in unfuring and help it to get the requried shape. The reflector can be rotated to the requried angle with the help of the support. It also plays an important role in obtaining the isostatism between the reflector fixture and support. Advanced noncontact measurement technique, namely, Close Range Photogrammetry (CRP) is used to measure shapes of structures to evaluate Thermo Elastic Distortions (TED). During test and measurement, the article under test (AUT) has to be mounted on a special fixture that provides iso-static support that effectively decouples the differential deformations of the AUT and the fixture. Mechanical analysis of the fixture is done in Patran and Nastran Software. That this is indeed the case is examined and results are presented in the work.

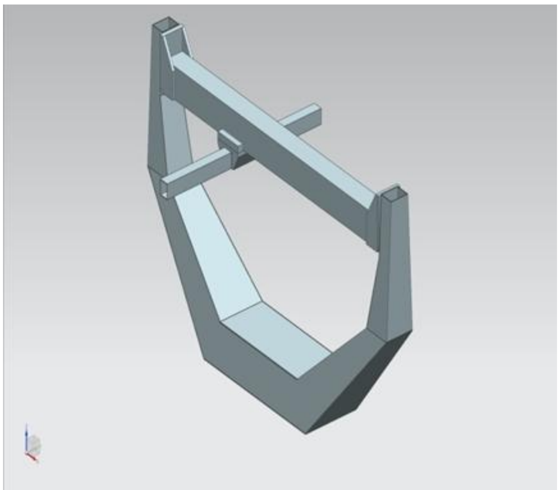
The structure of support is made of sheet metal which are welded to form a hallow solid or shell type structure, so as to reduce the mass of the structure. The boundary conditions are captured in the Patran model. For the thermal soak conditions, the likely frequency and deformation are computed and presented here.

The structure to be tested can be a part of a spacecraft. Surface accuracy of such structures is controlled at fabrication stage. The surface shape is expected not distort beyond specified values during the spacecraft service life.

“2. Modeling of Support Structure”

The modeling of the support is carried out in UNIGRAPHIC, MSC PATRAN as per the required condition. The modeling of the support in UNIGRAPHICS is done as per the specifications but the inner slots in the structure are not considered in the modeling along with the connecting parts i.e. nuts, bolts because the meshing of the inner slots surface is very cumbersome process and was causing fatal error during postprocessor of MSC PATRAN, thus to compensate we optimized the model for the mass.

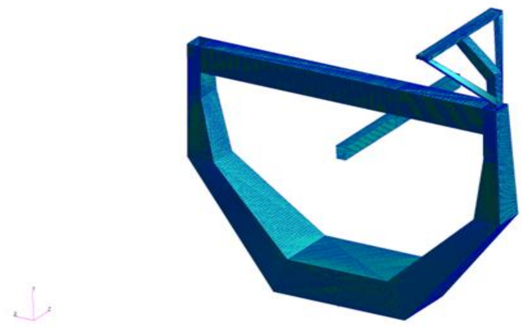
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 11, November - 2013



"Figure 1: Initial Support Model in Unigraphics"

In the modeling [1], firstly we have modeled the basic structure of the support keeping in mind the shape, structure and dimension. In the modeling we have not taken the parts which are connecting the two sheet metal rather we have united the two different metal sheet into one hollow solid and removed the nuts and bolts used in the model as the use of the nut and bolts may cause problem in the meshing of these parts. The material that is mainly used for this structure is aluminium which is a light weight material and best situated for satellite parts.

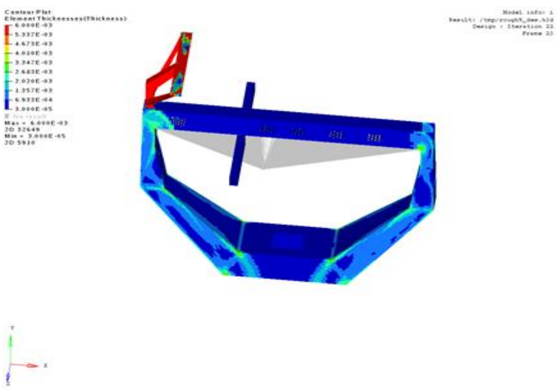
The support modeled in UniGraphics software is converted into IGES format so that it can be easily exported into PATRAN. In PATRAN [2] the model is carried out for meshing, as the mesh seeds are created on the curve. The number of seeds on a curve depends on the size of the curve, as a minimum element size of 5 has to be maintained in meshing between the seeds. The scale bar is also modeled with the support structure.



"Figure 2: Support model with Scale bar"

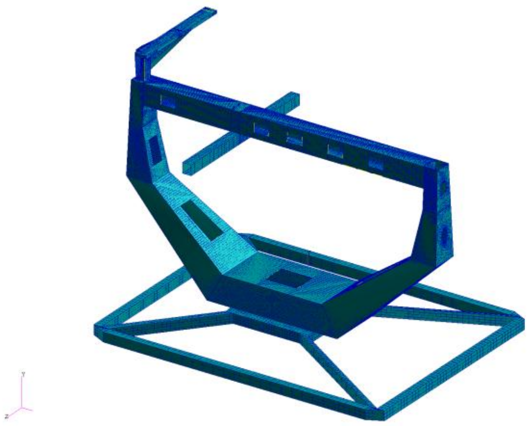
"3. Optimization of Model"

Optimization of the support is carried out in OptiStruct of HYPERMESH [3] package to get the optimized model for an objective of mass with frequency as the response. The optimization is considered in the work mainly to consider the inner slots and connecting parts which are not taken in modeling of the support in UNIGRAPHICS. The penalty by this we are taking into consideration is the frequency of the model. In this process the mass which are omitted in modeling are applied through the CG.



"Figure 3: HyperView of support"

The required mass is removed by making the slots on the surface of the model as shown where the size of the slot in turn depend on the surface size and they are taken in the ratio of seeds given to the surface curves. The stand is modeled for the optimized support

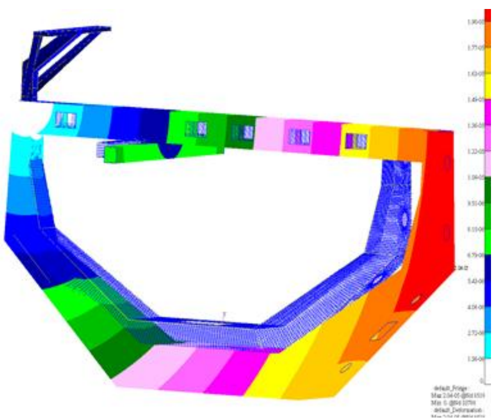


"Figure 4: Optimized Support model."

"4. Analysis of the support"

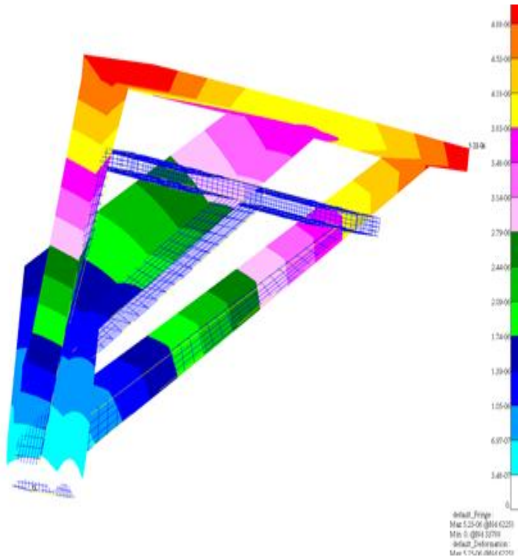
The optimized model is subjected to linear elastic [4] and Normal mode analysis in Patran. The linear elastic analysis is used to know the amount of deformation generating in model due to the temperature gradient obtained from the thermoelastic deformarion test and normal mode analysis is know modal shape and freqency of the support.

In case of liner elastic analysis, the requried temperature gradients are applied to the model in an uniform element manner and all requried properties are given to the model for analysis. The boundary condition taken in the analysis is same as the test, were the stands are fixed to the rails in the chamber.



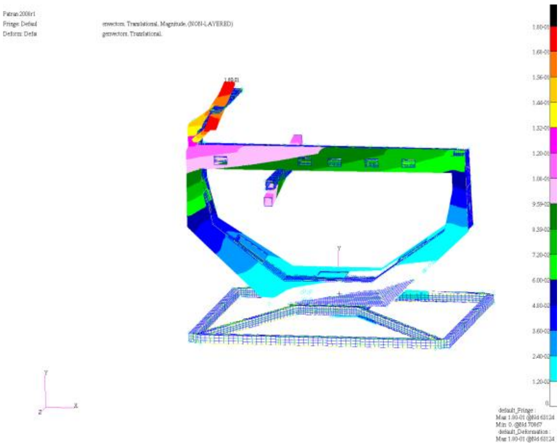
"Figure 5: Deformation of support to test temperature."

The deformation of the scale bar is also carried out from the same value of the temperature by grouping the requried part into one group separate from the default group and the deformation obtained from the linear static analysis of the scale bar.

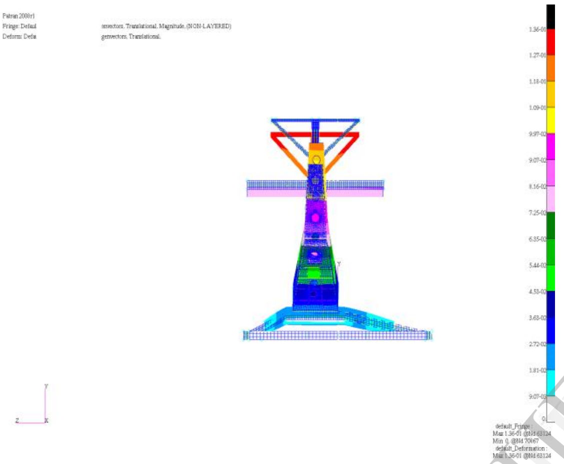


"Figure 6: Deformation of Scale bar to test temperature condition"

[5]During simulation on ground, the chamber is likely to be a vibration source. Hence dynamics of the fixture was studied to have an estimate of the dynamic displacements at critical points on the fixture thus the normal mode analysis is done. In case of Normal mode analysis, the boundary constraint of the model doesnot play any role in this analysis. If the model is free with no constraint then it is called as free vibration analysis with the first six freqency set to value zero or close to zero and from 7^{th}set onwards we get the freqency of the model, however in case if the model have a constrained boundary condition then in analysis we get a set of frequency where from its 1^{st} set it give us the value not as in case of the free vibrations but in the forced vibration, it includes the value of all the set of freqency as obtained in the free vibrations. The mode shape of the support with the frequency list is given below.



"Figure 7: 1st mode shape."



"Figure 8: 2nd mode shape"

"Table 1: Frequency of Support model"

MODE NO	FREQUENCY
1	10.58
2	11.45
3	15.397
4	22.266
5	23.847

"5. Test Correlation"

Temperature excursion seen during ground simulation tests was applied and the forces and dilatation generated at the supports were computed to be providing an isostatism between the reflector and support.. Technical evaluation of the measurement system was completed and the facility is capable of conducting similar thermo elastic deformation tests on structural components to an accuracy of +/- 25 microns over 2 meter test article size. Thus the system stability obtained from the analysis has direct impact on the test article.The values of test and the analysis are correlated as shown in table below

"Table 2: Test value correlation"

SL NO	TYPE	TEST VALUE	ANALYSIS VALUE
1	Deformation	25 MICRON	19.6 MICRON
2	Frequency	10 Hz	10.58 Hz

"6. Conclusion"

An effort is also made to compare the calculated values to correlate with already available Test data done during earlier test campaign. The utility of this work lies in its directly gaining confidence in the existing test setup design. It is well known that the performance of the fixture affects the measurement accuracy of TED. Hence verification of the design of support with fixture in test setup is made here and gives increasing confidence in the special test hardware.

Acknowledgements

The authors thank Dr K Renji, Group Director Structures, ISAC for permitting me to carry out this work and publish the same. We also thank Mr. Raghunath Behara, for giving general support and test data in this work.

Reference.

1. INTESPACE Documents for Modeling, 2008.
2. "Patran 2008 Release Guide", MSC Crop, 2008".
3. "Hyper Mesh 08 User Guide", MSC Corp 2008".
4. "Linear Static Analysis User's Guide", MSC Nastran 2005r2.
5. Jonathan Bryant, "Thermo Vacuum Chamber Control System" NASA, Kennedy Space Center, Florida, 2008.