IJITECH International Journal of Innovative Technologies
WWW.IJITECH.ORG
ISSN 2321-8665
Volume.06, Issue.01,
January-June, 2018,
Pages:0423-0428

Finite Element Analysis on Hot Structures of Hypersonic Vehicle at Different Boundary Conditions

M. ANIL[1], K. RAJASEKHARA REDDY[2], V. PURUSHOTHAM[3]
[1]PG Scholar, Dept of ME(CAD/CAM), IITM, Markapur, AP, India.
[2]Assistant Professor, Dept of ME, IITM, Markapur, AP, India.
[3]Assistant Professor & HOD, Dept of ME, IITM, Markapur, AP, India.

Abstract: When a hypersonic vehicle moves at intended speed it experiences aerodynamic heating. Structural integrity is affected by these high temperatures. The vehicle structures are called as hot structures or warm structures. Hot structures are fabricated to resist high temperatures more than 810^0K. Carbon– carbon composites, monolithic titanium alloys, metal – matrix composite materials are some examples of hot structures. These are used for hat – stiffened panels, honey comb sandwich panels. An example of an aircraft which uses these hot structures is Hyper – X 43 vehicle which has irregular shaped wing panels. This vehicle is subjected to nominal Mach 8. A heat transfer, thermal stress and thermal buckling analysis will be performed on three regions of the upper wing skin, 1) a fore wing panel, 2) an aft wing panel, 3) a unit panel at the middle of the aft wing panel. Heat transfer analysis will be performed on mid span of wing segment. The three panels mentioned will be subjected to four different boundary conditions with combinations of simply supported and clamped conditions.

Keywords: TPS, Hyper – X 43 Vehicle.

I. INTRODUCTION

Hypersonic flight vehicles are subjected to severe aero dynamic heating during flights. To maintain structural integrity at these high temperatures, the vehicle structural design concepts for high Mach number vehicles are different from those of low Mach number aircraft. The vehicle structure may be called “hot” structures or “warm” structures, depending on the operating temperature range. The hot structures are fabricated with high-temperature materials and are capable of operating at elevated temperatures exceeding 810.8 K. Typical hot structural components are carbon/carbon composite structures, hat-stiffened panels fabricated with either monolithic titanium alloys or metal-matrix composite materials, and honeycomb sandwich panels fabricated with high-temperature materials such as Titanium, or nickel-based Inconel (Inco Alloys International, Inc., Huntington, West Virginia) alloys. The warm structures are fabricated with light-weight materials such as aluminum and must be insulated so that the sub-structural temperatures will not exceed the operating temperature limit of 449.7 K. The space shuttle orbiter is a good example of warm structure. The entire vehicle is protected with a thermal protection system (TPS) to shield the aluminum substructure from overheating beyond the warm temperature limit. An example of a recent hot structure is the new hypersonic flight research vehicle called Hyper-X (designated as the X-43 vehicle), which has unconventional wing structures with irregular-shaped wing panels (described in the following section). The X-43 is an unmanned experimental hypersonic aircraft with multiple planned scale variations meant to test various aspects of hypersonic flight. It is part of NASA's Hyper-X program. It has set several airspeed records for jet-propelled aircraft. A winged booster rocket with the X-43 itself at the tip, called a "stack", is launched from a carrier plane. After the booster rocket (a modified first stage of the Pegasus rocket) brings the stack to the target speed and altitude, it is discarded, and the X-43 flies free using its own engine, a scramjet.

II. DESCRIPTION OF PROBLEM

Thermal stress analysis is performed on three regions of the wing skin (lower or upper); 1) a fore wing panel, 2) an aft wing panel, and 3) a unit panel at the middle of the aft wing panel. A fourth thermal stress analysis is performed on a midspan wing segment. The unit panel region is identified as the potential thermal stress initiation zone. Therefore, thermal stress analysis of the Hyper-X wing panels could be reduced to the thermal stress analysis of that unit panel Mentioned analysis is for uniform temperature loading, a dome shaped temperature loading of same temperature loading analysis will be presented. The analysis is carried with HAYNES 230 alloy and reason behind selecting this material is, it is a top-of-the-line high-performance, industrial heat-resistant alloy for applications demanding high strength as well as resistance to environment. It is a substantial upgrade in performance capabilities from common iron nickel- chromium and nickel-chromium alloys, and displays the best combination of strength, stability, environment resistance, and fabricability of any commercial nickel-base alloy. HAYNES 230 alloy can be utilized at temperatures as high as 1250°C for continuous service. Its resistance to oxidation, combustion environments and nitriding, recommends it highly for applications such as nitric acid catalyst grids, high-temperature bellows, industrial furnace fixtures and hardware, strand annealing tubes, thermocouple protection tubes, and many more.

TABLE I: Composition of HAYNES 230 Alloy

Element	Min	Max
Carbon	0.05 %	0.15%
Manganese	0.30%	1.00%
Silicon	0.25%	0.75%
Phosphorus	--	0.03%
Sulfur	--	0.015%
Chromium	20.00%	24.00%
Cobalt	--	5.00%
Iron	--	3.00%
Aluminum	0.20%	0.50%
Titanium	--	0.10%
Boron	--	0.015%
Copper	--	0.50%
Lanthanum	0.005%	0.05%
Tungsten	13.00%	15.00%
Molybdenum	1.00%	3.00%
Nickel	REM%	

III. MODELING AND MESHING

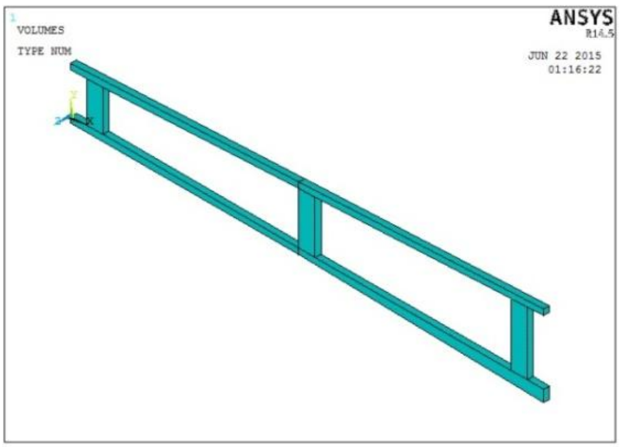

Fig.1. Wing Cross section showing stiffeners.

As mentioned in introduction design of wing structures of Hyper – X is entirely different from that of the conventional wing structures as shown in Figs.1 and 2. The conventional spar and rib system is replaced with multiple radial stiffeners. Modeling of these stiffeners was done in ANSYS. Key points were created at the starting of the modeling based on the dimensions given. These points were joined using lines. Areas are created by joining these lines. Stiffeners are created as different areas as they are separate members which are welded to sheet panels. Areas which are created are extruded to a length of 2.258mm thickness as the width of the sheet considered. Width of stiffeners was taken as 6.25mm. Three radial stiffeners were modeled with panels attached to it. As in the real condition stiffeners are line welded to wing skin. Stiffeners are attached to sheet panels by using glue command which gives welding condition. Hollow area between first two stiffeners is treated as Bay – 1 and hollow area between second and third stiffener is treated as Bay – 2. Temperature on Bay – 1 and Bay – 2 upper skin are imposed with 888.617K and lower skin temperature on Bay – 1 and Bay – 2 was given as 588.64 K. These temperatures were considered at 89th second of flight.

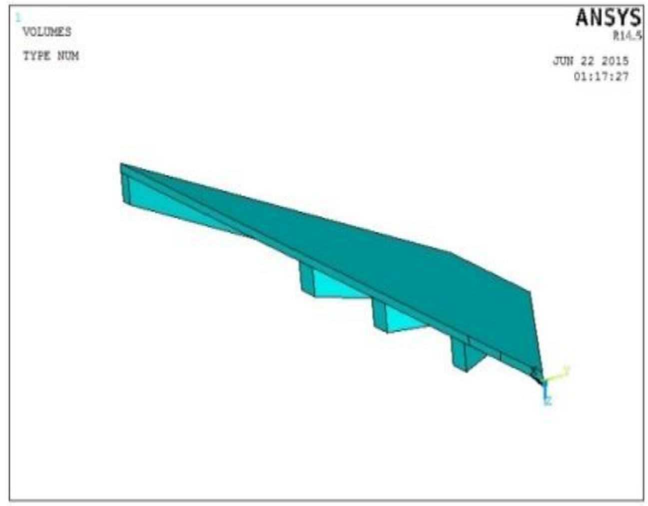

Fig.2. fore wing panel.

IV. STRESS ANALYSIS

A. Thermal Stress Analysis

The panel distribution over each bay is arch – shaped. This is the typical behavior of hot structural panels supported by the boundary heat sinks. The structural temperatures at typical points of the wing segment at t=89 sec are listed in table 2. the upper and lower wing skin temperatures listed are the peak temperatures of each bay.

TABLE II: Structural Temperatures At Typical Points of Wing Segment; t = 89 sec, K

	Fore weld site	Bay 1 skin (peak)	Middle weld site	Bay 2 skin (peak)	Aft weld site
Upper	630.3	895.8	670.8	881.3	609.7
Lower	521.4	592.5	519.7	584.2	487.5

The simulated chord-wise thermal stresses induced in the wing-segment skins under free expansion is as shown in fig.3.

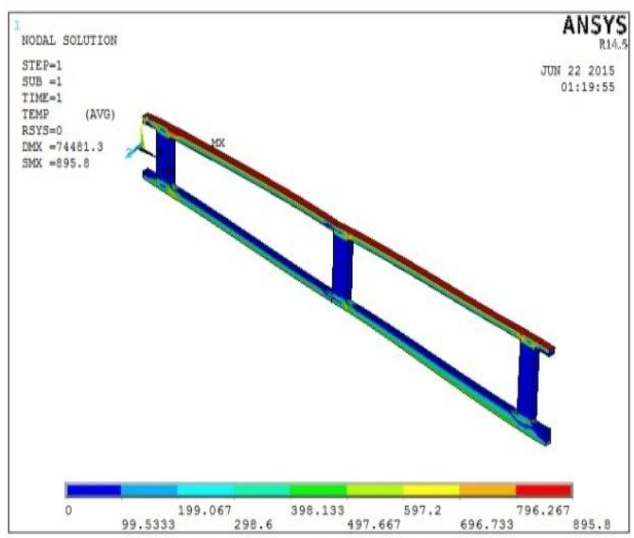

Fig. 3. Nodal solution of wing cross-section.

V. THERMAL STRESS ANALYSIS ON FORE PANEL WITH DIFFERENT BOUNDARY CONDITION

Results of this paper is as shown in bellow Figs.4 to 22.

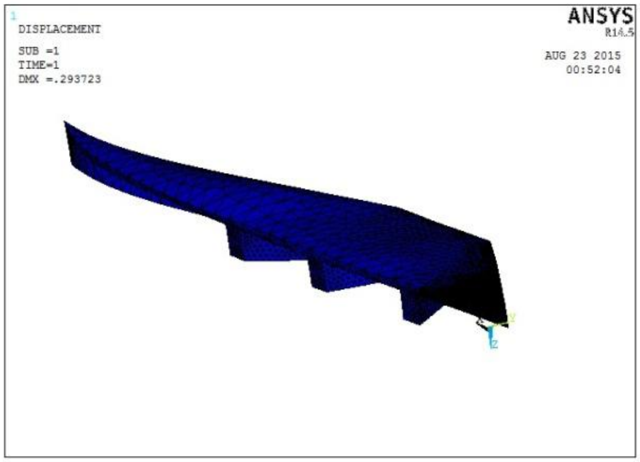

Fig. 4. Deformation of wing fore panel at 348.11 K.

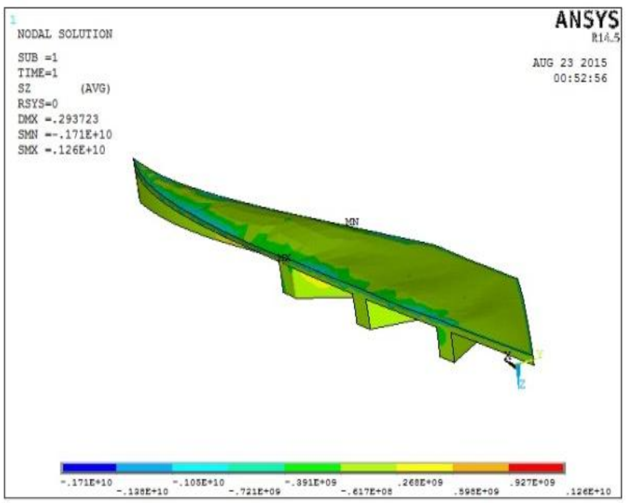

Fig.5. Stress on wing fore panel along the thickness at 348.11K.

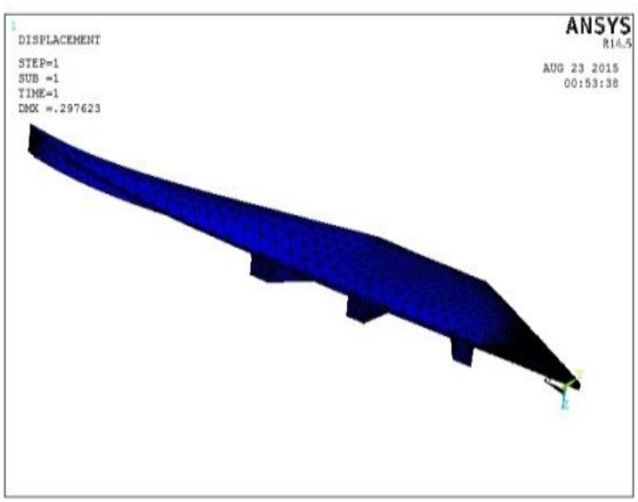

Fig.6. Deformation of wing fore panel at 353.11K.

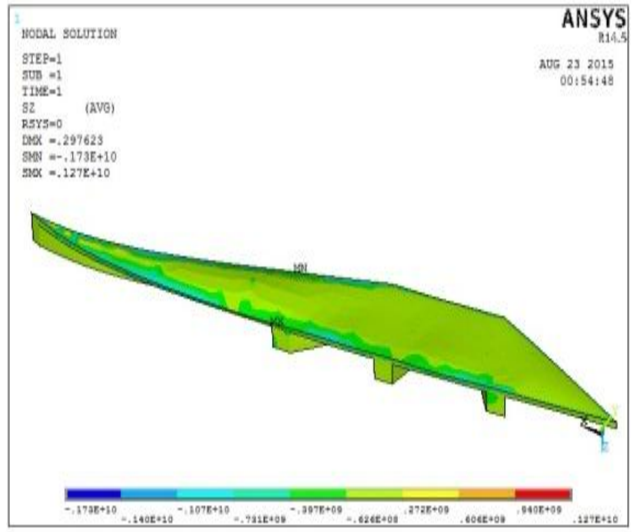

Fig.7. Stress on wing fore panel along the thickness at 353.11 K.

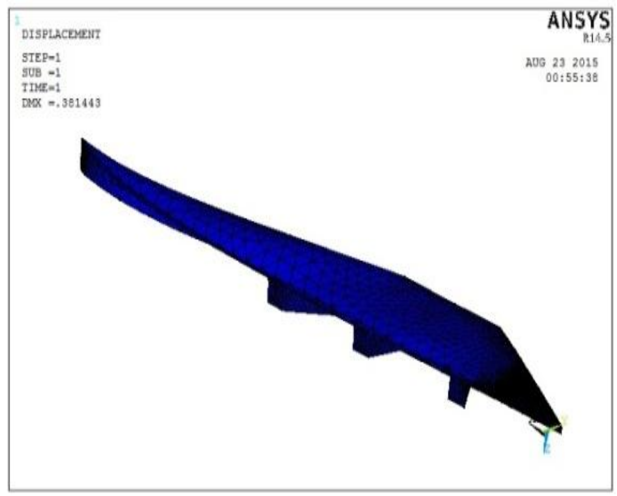

Fig.8. Deformation of wing fore panel at 452.54 K.

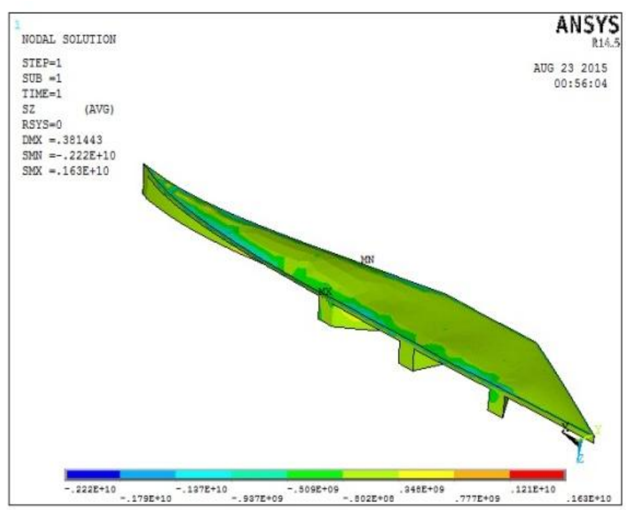

Fig.9. Stress on wing fore panel along the thickness at 452.54 K.

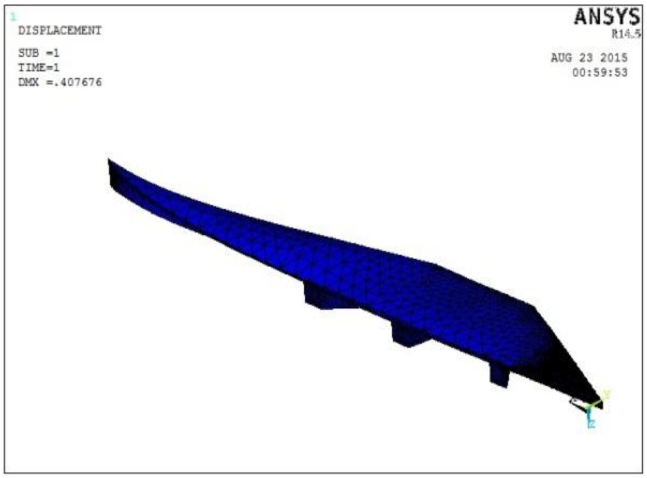

Fig.10. Deformation of wing fore panel at 483.10 K.

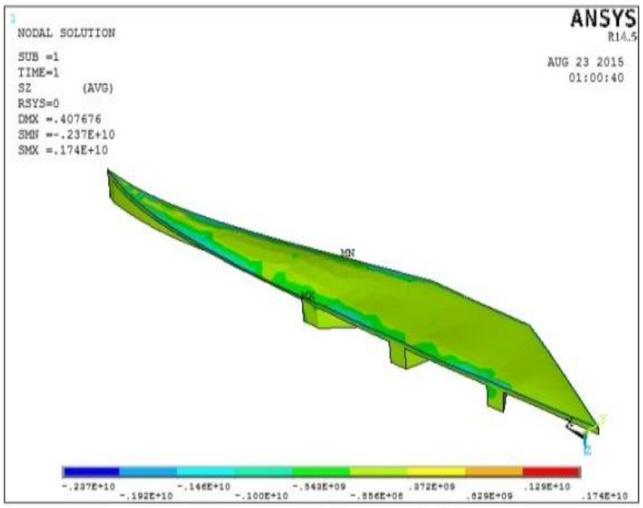

Fig.11. Stress on wing fore panel along the thickness at 483.10 K.

When different temperatures 322.56 K, 352.55 K, 359.78 K, and 418.66 K were imposed on fore panel maximum stress of 0.16 N / mm^2 was obtained. At 483 K Haynes material can resist up to 170 MPa of stress. It is seen that stress obtained is within the permissible limit.

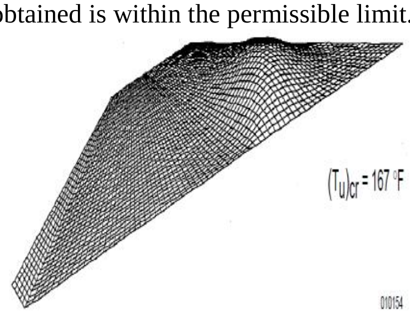

Fig.12. fore panel.

From Reference 14 when a uniform temperature of 452.54 K is experienced by fore panel maximum deformation and stress is obtained at near corner as shown. Position of maximum stress on fore panel is almost same.

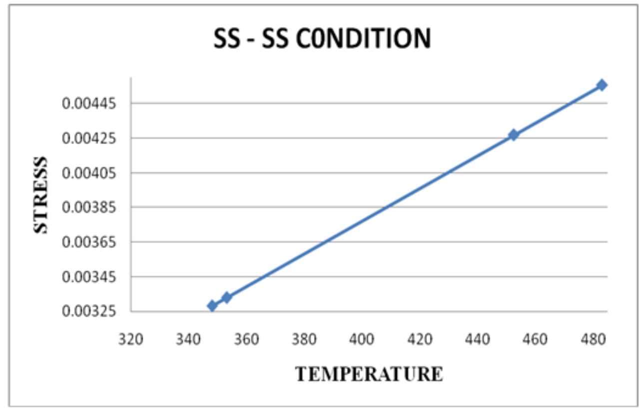

Fig.13. Graph, Stress Vs Temperature.

TABLE III: Temperature and Stress Variation On Fore Panel With SS – SS Condition

Simply supported – simply supported condition	
Temperature, K	**Stress, N/mm^2**
348.11	0.00328
353.11	0.00333
452.54	0.004268
483.1	0.004556

A. Thermal Stress Analysis On Aft Panel With Different Boundary Condition

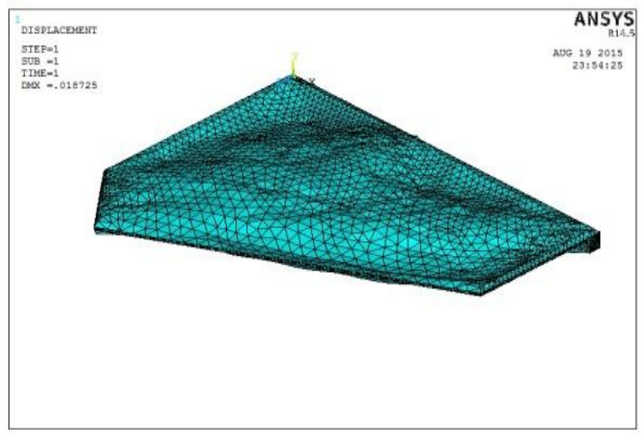

Fig.14. Deformation of wing aft panel at 322.56 K.

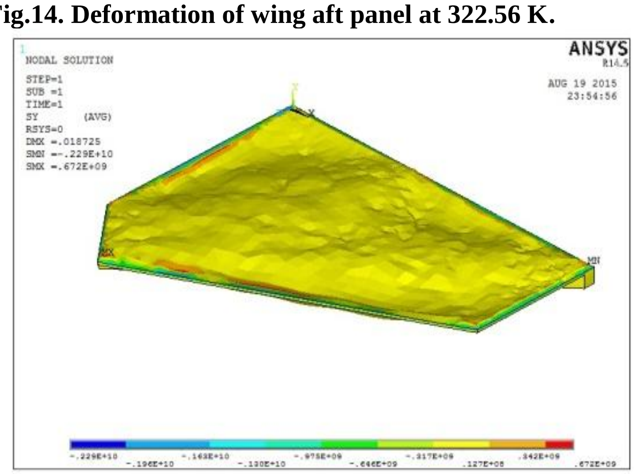

Fig.15. Stress on wing aft panel along the thickness at 322.56K.

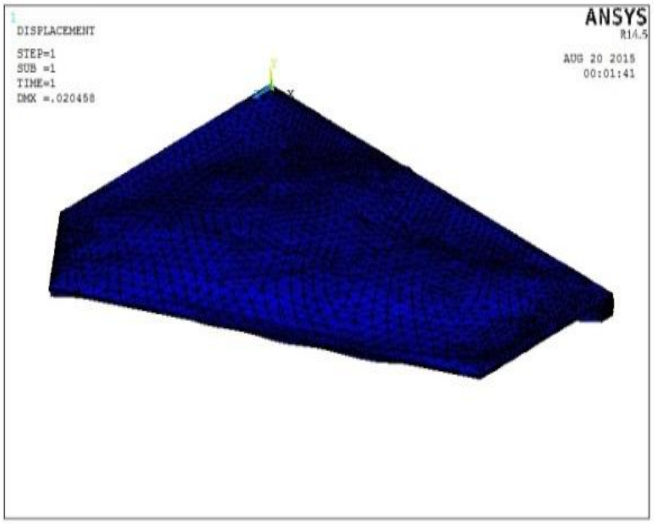

Fig.16. Deformation of wing aft panel at 352.55 K.

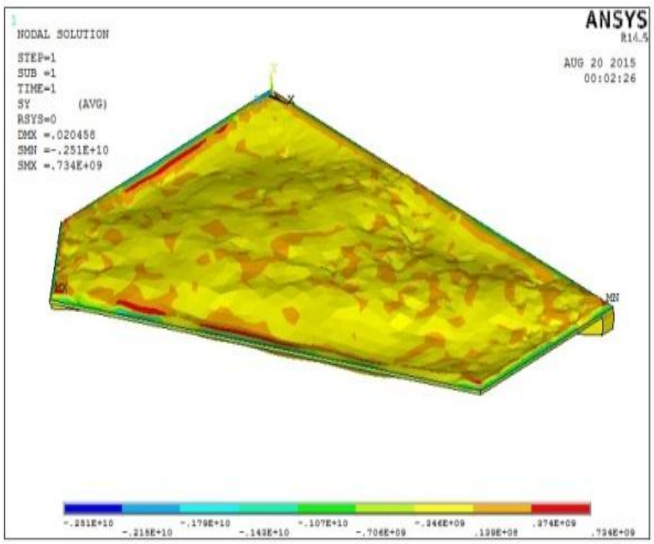

Fig.17. Stress on wing aft panel along the thickness at 352.55K.

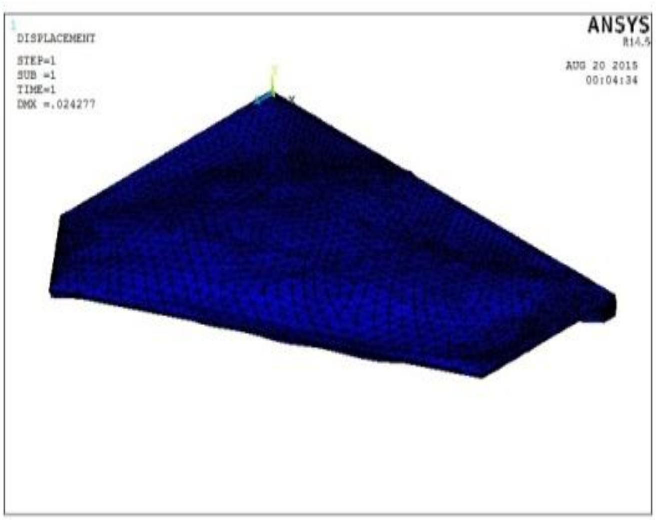

Fig.18. Deformation of wing aft panel at 418.66K.

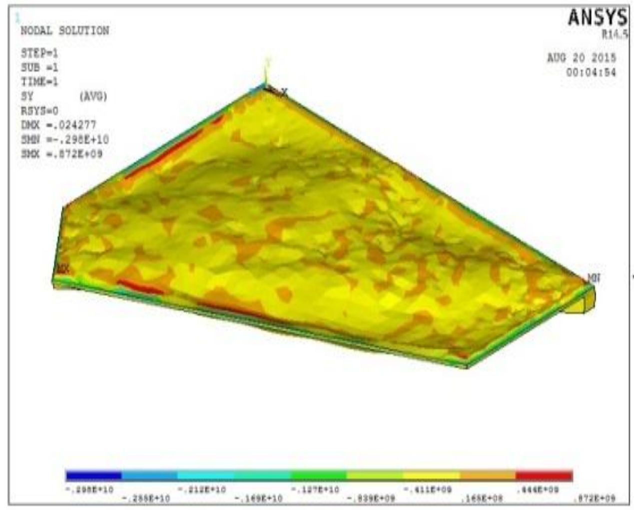

Fig.19. Stress on wing aft panel along the thickness at 418.66K.

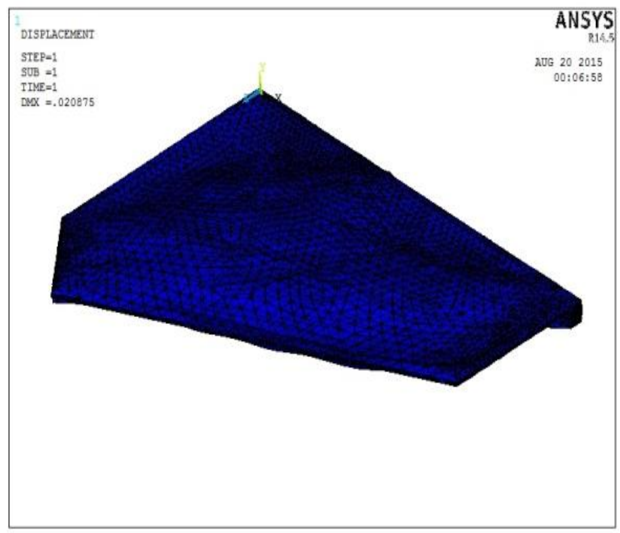

Fig.20. Deformation of wing aft panel at 359.78K.

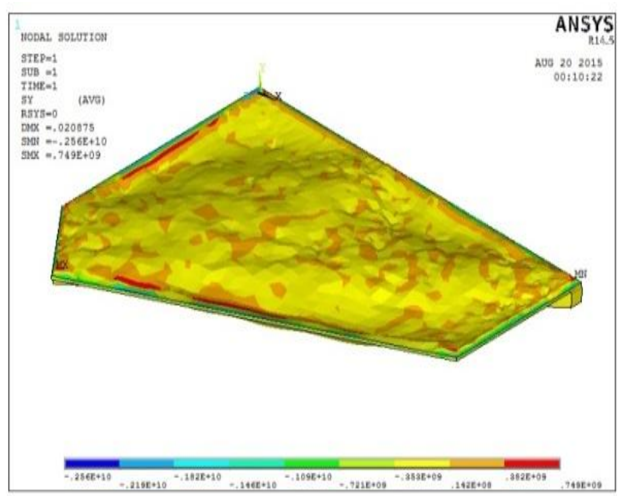

Fig.21. Stress on wing aft panel along the thickness at 359.78K.

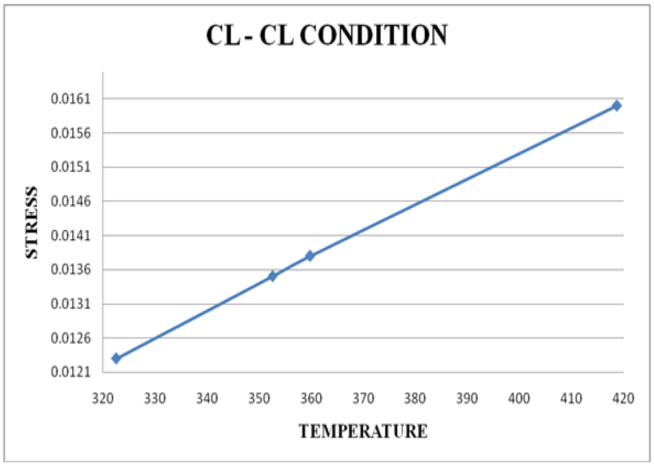

Fig.22. Graph, Stress Vs Temperature.

TABLE IV: Temperature and Stress Variation On Aft Panel with CL – CL Condition

Clamped – Clamped condition	
Temperature, K	**Stress, N/mm^2**
322.56	0.0123
352.55	0.0135
359.78	0.0138
418.66	0.016

VI. CONCLUSION

Heat transfer, thermal stress, and thermal buckling analyses were performed on the Hyper-X wing structure for the Mach 8 mission. The key results of the analyses follow. For fore panel with simply supported – simply supported condition and when it is exposed to different temperatures (348.11K, 353.11K, 452.54K, 483.10K) deformation & stress due to deformation is within the resistive value of Haynes 230 material which is used. For different boundary conditions considered for aft panel of wing is capable of resisting different uniform temperatures (322.56K, 359.78K, 352.55K, 418.66K) and 540 N/mm^2. Unit panel which is a part of aft panel whose edges are supported on stiffeners when is experiences a temperatures 302.23K, 388.11K, 318.67K, 419.21K and 540N/mm^2 its stress developed due to deformation is resistible and deformation is within the elastic limit. By identifying the unit panel region as the potential thermal buckling initiation zone, thermal buckling analysis of the Hyper-X wing panels may be reduced to the thermal buckling analysis of the unit panel without going through complex modeling of the entire wing structure. Material considered for the stiffener of the wing is capable of resisting the temperature imposed on it.

VII. REFERENCES

[1]Ko, William L., Robert D. Quinn, and Leslie Gong, Finite-Element Reentry Heat Transfer Analysis of Space Shuttle Orbiter, NASA Technical Paper 2657, December 1986.

[2]Robert D. Quinn and Leslie Gong Ames Research Center, Dryden Flight Research Facility. Real – Time Aerodynamic Heating and SurfaceTemperature Calculations for Hypersonic Flight Simulation.1990.

[3]William L. Ko Dryden Flight Research CenterEdwards, California, Thermal and mechanical BucklingAnalysis of Hypersonic Aircraft Hat-Stiffened Panels with Varying Face Sheet Geometry and Fiber Orientation.

[4]Ko, William L. and Raymond H. Jackson, Compressive Buckling Analysis of Hat-Stiffened Panel, NASA TM-4310, August 1991.

[5]Ko, William L. and Raymond H. Jackson, Shear Buckling Analysis of Hat-Stiffened Panel, NASA TM-4644, November 1994.

[6]Ko, William L. and Raymond H. Jackson, Thermal Behavior of a Titanium Honeycomb-Core Sandwich Panel, NASA TM-101732, January 1991.

[7]Ko, William L. and Raymond H. Jackson, Combined Compressive and Shear Buckling Analysis of Hypersonic Aircraft Structural Sandwich Panels, NASA TM-4290, May 1991. Also published as AIAA Paper No. 92-2487-CP, in the proceedings of the 33rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference, Dallas, Texas, April 13-15, 1992.

[8]Ko, William L., "Mechanical and Thermal Buckling Analysis of Sandwich Panels Under Different Edge Conditions," Pacific International Conference on Aerospace Science and Technology Conference Proceedings, Vol. II, Tainan, Taiwan, December 6-9, 1993.