Response of the Enhanced Polar Outflow Probe (e-POP) Instrument Under Shock Loading

M. Nejad Ensan, D.G. Zimcik
Institute for Aerospace Research, National Research Council Canada, Ottawa, Ontario. Canada.

Abstract

Spacecraft components encounter mechanical shock from a variety of sources. Components must withstand a series of flight shock pulses, and must be designed and tested accordingly to ensure reliability.

This paper presents simulation of the response of the Enhanced Polar Outflow Probe (e-POP) instrument to the shock loading, during payload separation, using LS-DYNA® nonlinear finite element analysis software. Details of the model and simulation approach and the results obtained from that analysis are included in this paper.

The mission science objective of the e-POP is to study plasma and atmospheric outflows in the polar region and the wave generation, particle interaction, and propagation associated with these outflows. The e-POP Instrument payload is a part of the "CASade, Smallsat and IOnospheric Polar Explorer" (CASSIOPE) mission. The CASSIOPE mission is a joint mission for the development and demonstration of key CASCADE technologies for future global bulk data delivery system, the development and demonstration of a generic SmallSAT Bus for future Canadian Space Agency (CSA) space missions.

Introduction

Spacecrafts may sustain various types of shock during their development and mission. From the launch to its mission end, a spacecraft may for instance sustain the following shocks:

- Launcher induced shocks such as fairing or stage separation. They propagate in the launcher structure, reach the launcher-spacecraft interface and then propagate in the spacecraft structure.
- Spacecraft release shock. They are directly generated at the launcher-spacecraft interface by the separation system.
- Appendage release shocks or subsystem actuating shocks. Some appendages which are clamped for launch have to be released to be in function (solar arrays, antenna reflectors, instruments).

Therefore, a need exists for the short duration dynamic response of structures to assess component structural integrity. This provides an approach to reduce risk with accurate dynamic structural response prediction during component design while maintaining a lower cost than the current testing to demonstrate design robustness.

Many researchers have investigated the high velocity impact problem including Goldsmith [1], Jones [2], Zukas [3] and Chang [4]. However the current approach is concerned on application to aerospace components.

Finite Element Analysis

Although different approaches for analysis of shock events are available, finite element analyses, which are based on accurate constitutive models, provide the most detailed information on the spatial and temporal distribution of shock events [5]. The objective of the present work was to simulate the transient dynamic impact resulting from the launch loads at the payload interfaces, using LS-DYNA [6] finite element analysis software. Features of this code include large material and element libraries, many contact algorithms and a high level of accuracy.

Input Load / Shock Response Spectrum

A typical Shock Response Spectrum (SRS) to define the input to the target structure resulting from launch vehicle (L/V) - spacecraft separation is shown in Fig. 1.

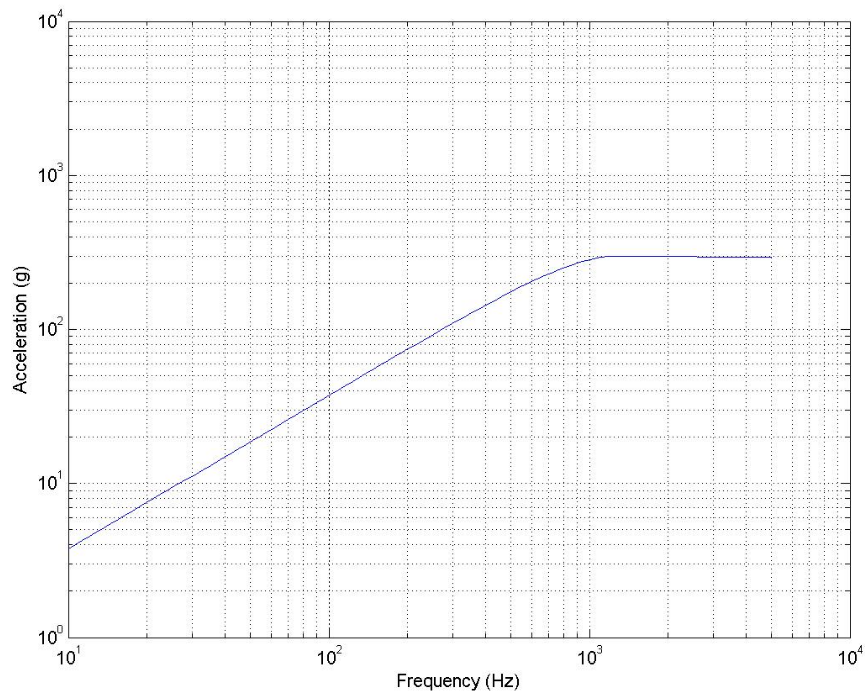

Fig. 1: Typical Shock Response Spectrum.

It is important to note that the SRS is not the shock input resulting from the L/V. Rather, as the name implies, the SRS defines the response of a structure at its undamped natural oscillation frequency to a shock and is not the input shock that generates that response. The general definition of a SRS as found in such references as MIL-STD-810 is:

"The SRS is the response of a single degree of freedom system at its undamped natural oscillation frequency with a Q of 10 to an input pulse"

where Q is defined as $\left(Q = \frac{1}{2\zeta} \right)$ and ζ defined as the damping ratio.

Application

The CASSIOPE mission is a joint mission for the development and demonstration of key Cascade technologies for future global bulk data delivery system, the development and demonstration of a generic SmallSAT Bus for future Canadian Space Agency (CSA) space missions, and the collection of e-POP Science Data for space weather studies [7]. The University of Calgary directs the e-POP project, and a team which is comprised of researchers and engineers from several Canadian universities, institutes and the private and public sectors.

Description of the e-POP Scientific Instrument

A concept illustration of one of the e-POP scientific instrument is shown in Fig. 2. Structural details of this instrument including dimension, materials and assembly were provided by CSA. The key component of interest was the dual Microchannel Plate(s) internal to the unit shown in Fig. 3.

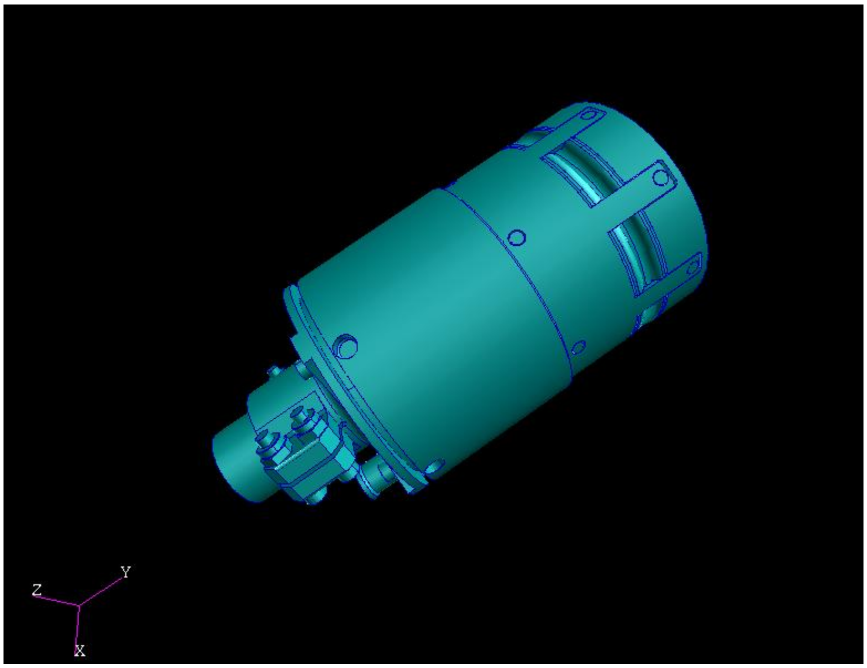

Fig. 2: Scientific instrument of e-POP solid view.

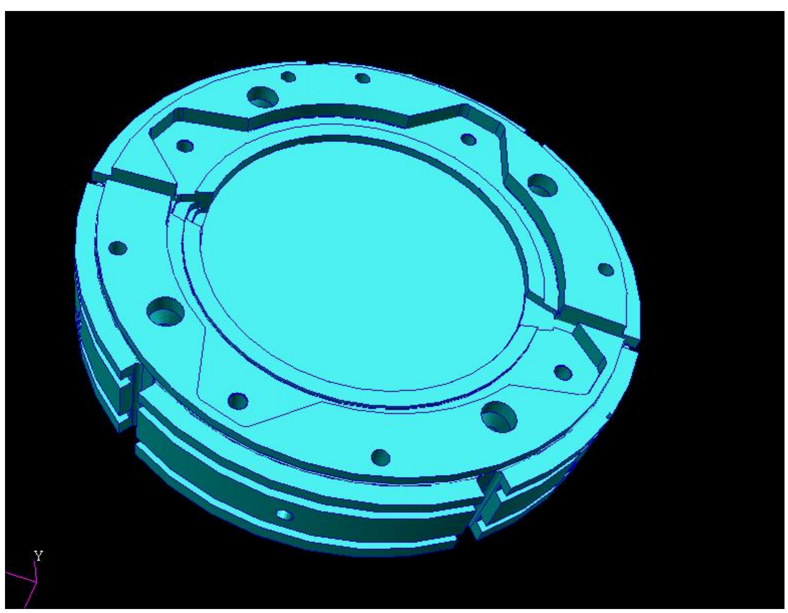

Fig. 3: Microchannel plates (MCPs) and housing.

Three-dimensional e-POP Model

A three-dimensional model to study the shock response of the e-POP scientific instrument was developed as shown in Fig. 4a-b. Assumptions were made to simplify the process to speed up the acquisition of results. This simplified model included the following major components: boom stem, e-POP scientific instrument boom adapter plate, printed circuit boards (PCBs), anode printed circuit board, microchannel plates (MCPs), lower skin, MCPs housing and screws & nuts connecting these components together.

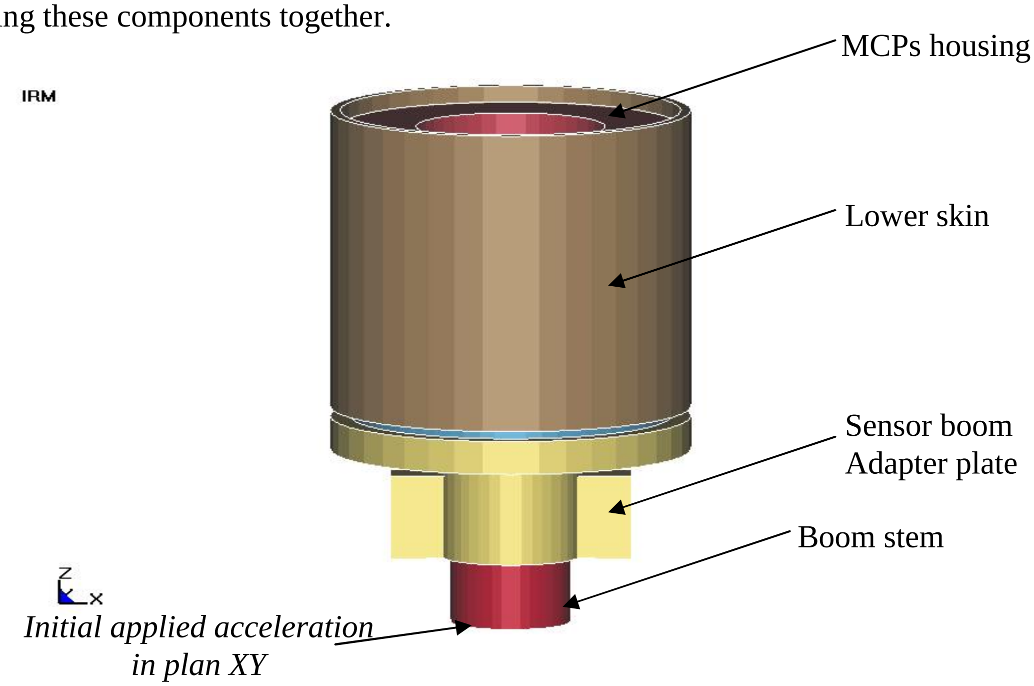

Fig. 4a: Scientific instrument of e-POP model.

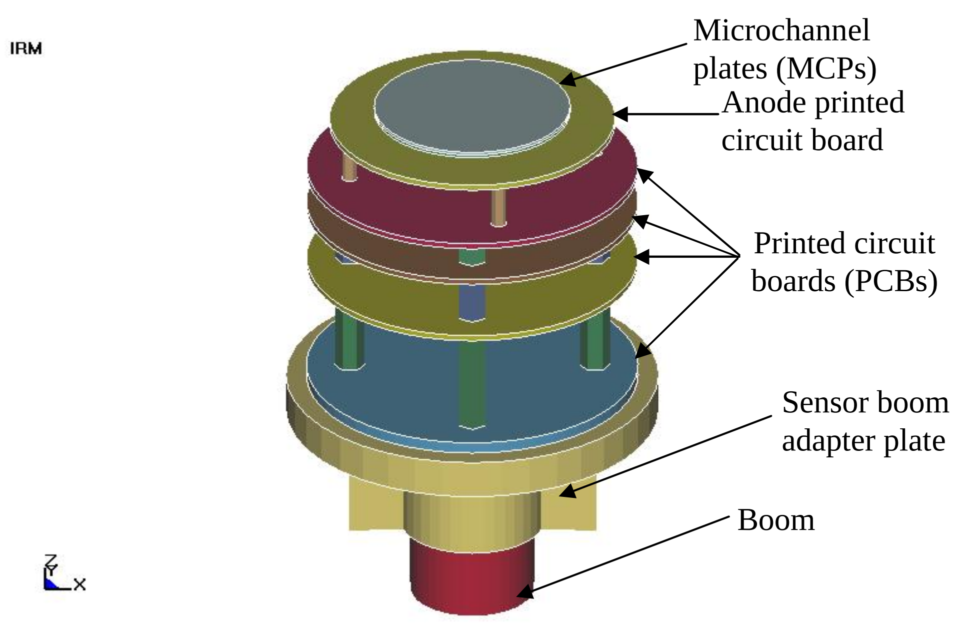

Fig. 4b: Scientific instrument of e-POP model without Lower skin and MCPs housing.

The e-POP scientific instrument was modeled using approximately 21000 solid elements each with six degrees of freedom at each node as shown in Fig. 5a-b. The screws and nuts connecting the different printed circuit boards of the e-POP scientific instrument were also modeled using solid element formulation with a total of approximately 3000 elements.

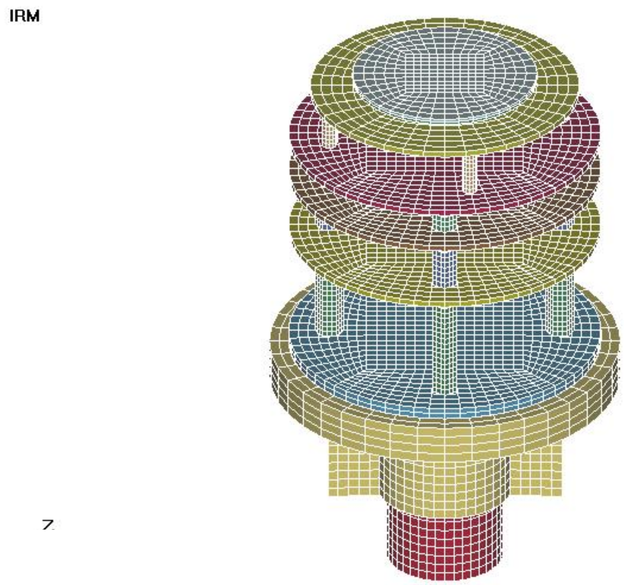

Fig. 5a: Scientific instrument of e-POP's finite element model.

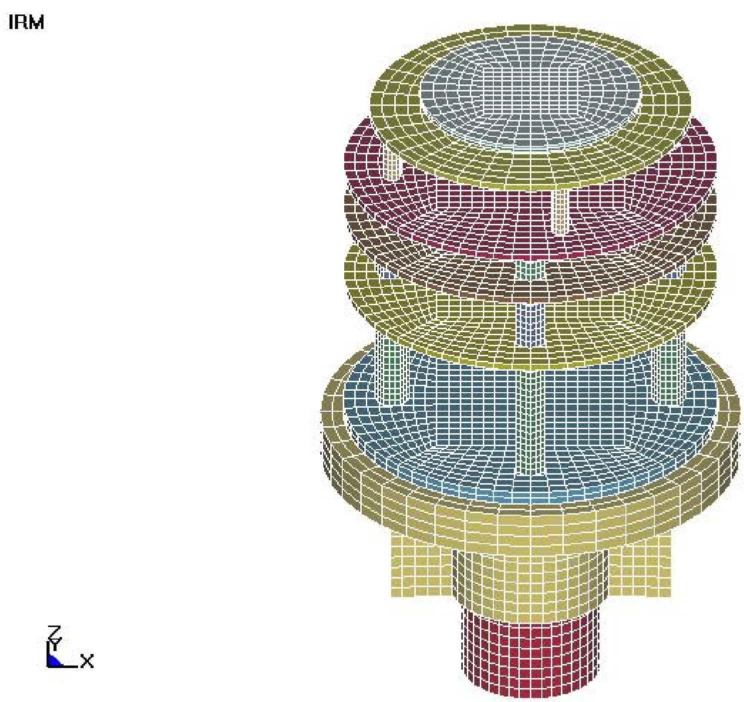

Fig. 5b: Scientific instrument of e-POP's finite element model without lower skin and MCPs housing.

The material properties used for the different components provided are listed in Table 1.

Table 1: Material Properties

Component	Density (slugs/in^3)	Elastic modulus (lb/in^2)	Poisson's ratio	Ultimate strength (psi)
Printed circuit boards (PCBs), anode printed circuit board	1.8E-4	2.0E6	1.5E-1	N/A
Boom stem, sensor boom adapter plate, lower skin and screws & nuts	2.5E-4	1.0E7	3.3E-1	N/A
Microchannel plates (MCPs)	2.1E-4	8.7E6	1.0E-1	200
MCPs housing	1.2E-4	3.6E6	1.5E-1	N/A

Numerical Simulation

Numerical simulation using this finite element model was performed assuming an initial acceleration in the z-direction at the base of the e-POP scientific instrument at the bottom surface of the boom stem as shown in Fig. 4a. Since conservatism was desired, the Von Mises (maximum effective stress) criterion has been chosen.

The impact events were analyzed for a time period of 4 ms, which was chosen to be sufficiently long to simulate the initial events of the impact. Dynamic results were recorded at one hundred equal time steps (every 0.04 ms).

Maximum Effective Stresses

The maximum effective stress (Von-Mises) contour of the e-POP scientific instrument is shown in Fig. 6. The stress wave traveled to the MCPs through two paths:

a) e-POP scientific instrument's boom adapter plate, the lower skin and MCPs housing
b) printed circuit board (PCB) stand-offs, anode PCB and MCPs housing

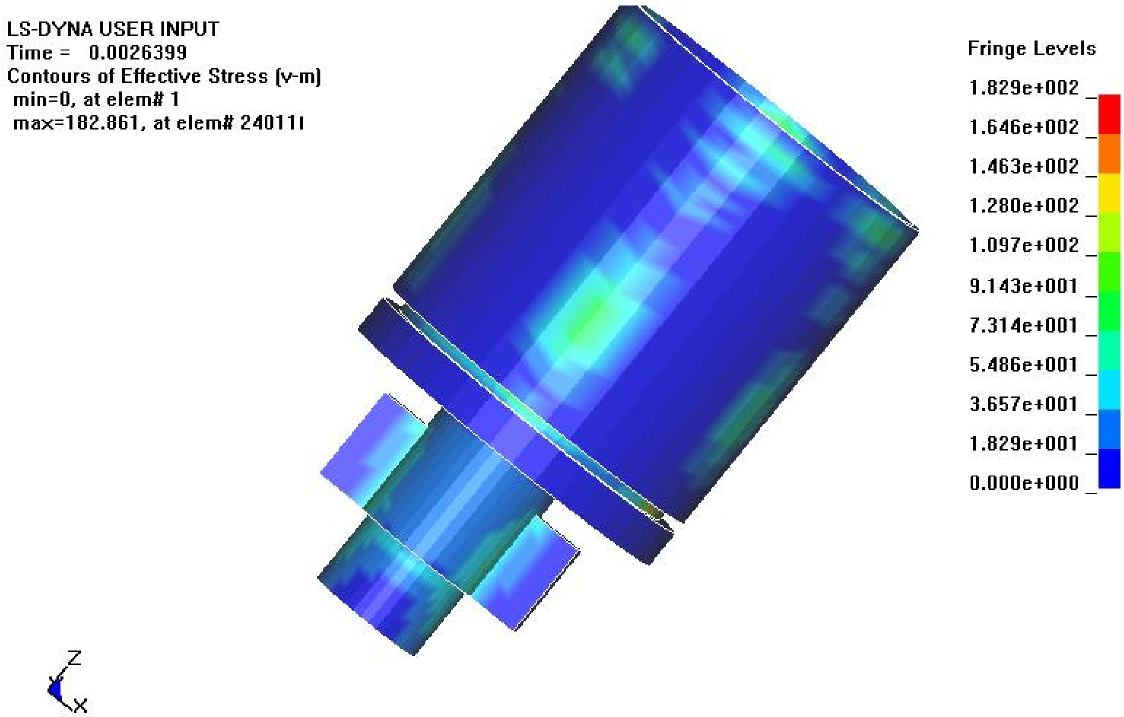

Fig. 6: Effective Stress (Von-Mises) Contour (psi) of scientific instrument of e-POP.

The maximum effective stress contour of the adapter plate and PCBs of the e-POP scientific instrument at t=0.96 ms is shown in Fig. 7.

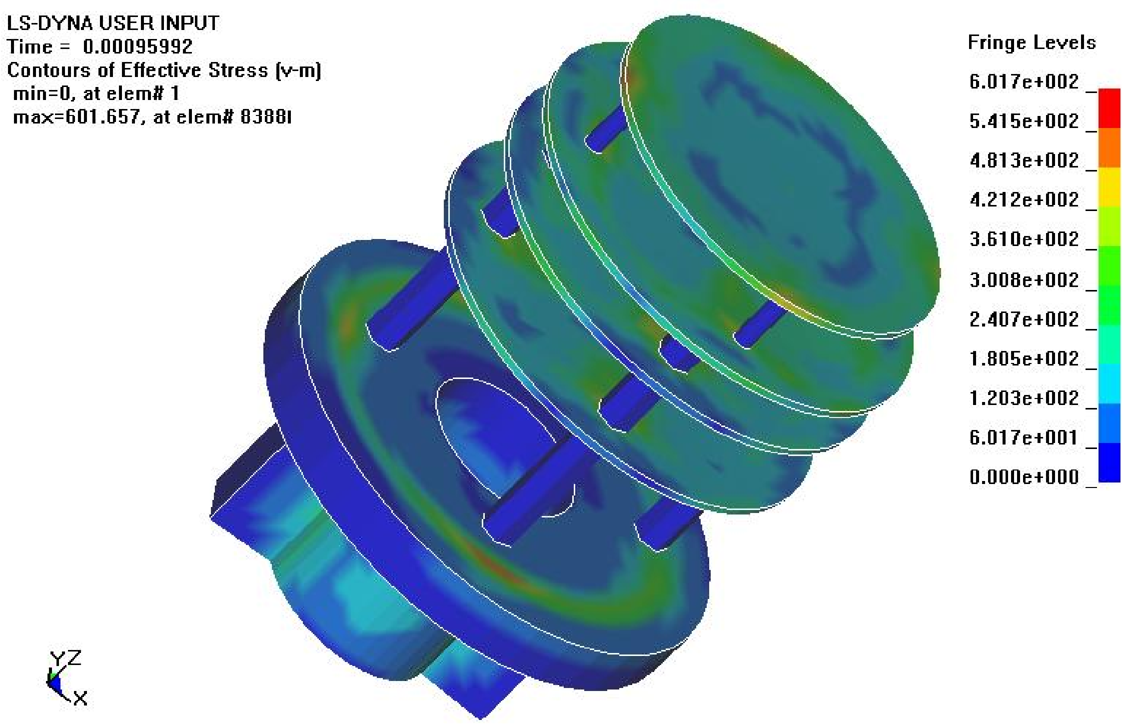

Fig. 7: Effective Stress (Von-Mises) Contour (psi) of sensor adapter plate and PCBs of scientific instrument of e-POP at t=0.96 ms.

The maximum effective stress distribution in the Microchannel Plate (MCP) is depicted in Fig. 8. As shown, the maximum stress was 186.7 psi in the top MCP and occurred at 0.52 ms after the start of impact. The numerical results indicated a positive margin of safety of over 8.5% at the MCP based on the ultimate stress allowable shown in Table 1. It can be observed from Fig. 8 that the areas where high stress prevailed after impact was located at the middle of the MCP. This predicted peak stress is consistent with a modal shape that one might expect for a circular disk clamped at the edge to reflect the wavy washer. Recall that this stress results from an input pulse that had slightly higher amplitude than required by the SRS for the e-POP scientific instrument. Accordingly, the margin in the predicted results is mildly conservative.

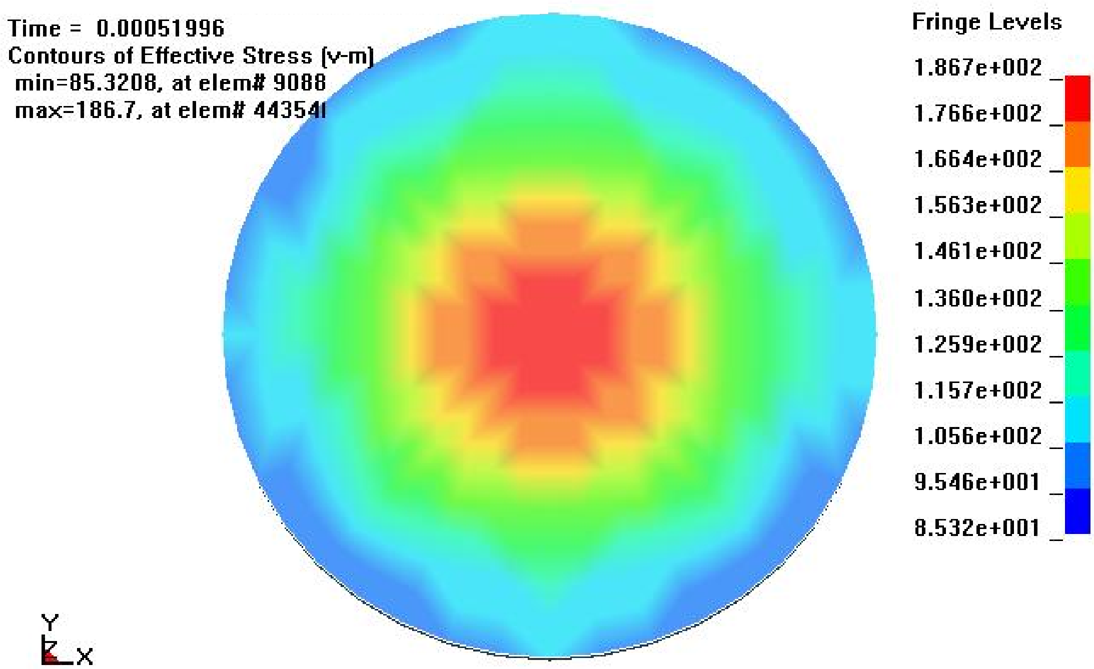

Fig. 8: Maximum Effective Stress (Von-Mises) Contour (psi) for MCP1 (top).

Conclusions

Analysis of components under the short duration dynamic loading to assess component structural integrity and to demonstrate design robustness has been presented. This method has been applied to a sensitive scientific instrument of the e-POP payload on the CASSIOPE spacecraft to simulate transient dynamic analysis of the shock. The shock response spectrum along the interfaces have been discussed in detail in this finite element simulation to provide data on the overall shock event. Evaluation of the model showed that it was computationally stable, reliable and repeatable. The method is shown to provide a more accurate simulation of the shock event compared to traditional stationary random equivalent finite element models. The methodology reduces the unknown conservatism of the traditional analysis approach by providing accurate simulation of the transient dynamic response of the structure.

Acknowledgement

This work was completed under a collaborative program with the Canadian Space Agency. The authors would like to thank Professor A. Yau of the University of Calgary for providing detailed information on the structure of the e-POP scientific instrument.

References

[1] Goldsmith W., "Impact, the Theory and Physical Behaviour of Colliding Solids", Mineola, NY: Dover Publications, 2001.

[2] Jones, N., "Structural Impact", Cambridge University Press, 1997.

[3] Zukas, J.A., "High Velocity Impact Dynamics", John Willey & Sons, 1990.

[4] Chang C, Engblom J., "Nonlinear dynamic response of impulsively loaded structure: a reduced bias approach", AIAA J., 29(4), PAGE 613-8, 1991.

[5] Belytschko, T., Liu, W.K., and Moran, B., "Nonlinear Finite Elements for Continua and Structures", John Willey & Sons, 2004.

[6] LS-DYNA Keyword User's Manual", Version 960, Livermore Software Technology Corporation, Livermore, CA, March 2001.

[7] A. Yau, G. James, G. Enno, R. Hum, P. Duggan, M. Senez, Z. Ali, G. Brassad, B. Desjardins, L. Dube, R. Giroux, D. Beattie, I. Walkty, "Multidimensional Challenges and Benefits of the CASSIOPE mission", ESA/ the 4S Symposium (Small Satellites Systems and Services), Rhodes, Greece, 26-30 May 2008.