Design and Validation of Notch based Separation System using ABAQUS Software

Sonali Gyan[#], N. Murali[+], S. Sirajudeen Ahamed[++]

[#]Scientist Engineer SD, Vikram Sarabhai Space Centre

[+]Scientist Engineer SG, Vikram Sarabhai Space Centre

[++]Scientist Engineer G, Vikram Sarabhai Space Centre

Abstract:

A pyro based separation system is widely used in the launch vehicle stage separation system. Expandable tube assembly (XTA) is one such system which comprises of a frangible ring with a notch to initiate the fracture and facilitate the separation. XTA separation system also has a cut-out to house the charge initiator. In view of stress raisers such as a notch, cut-out and load path eccentricities in the structure, the design of expandable tube assembly is quite involved. Design verification of the notch based separation system is carried out through finite element analysis. The structure is tested for the expected flight loads to validate the design. This paper presents the complexities of the notch based separation system design and design verification through FEM using ABAQUS software. A material nonlinear analysis of a 3D model of the separation system is essential on account of the stresses near notch going beyond yield strength of the material to withstand flight loads. ABAQUS software is extensively made use of to carry out the 3D nonlinear analysis to confirm the design adequacy of the separation system. FE analysis results and test validation depict that ABAQUS software is able to capture the plastic region in a complex system with notch, cut out and eccentricities.

Keywords: Expandable tube assembly, Separation system, Notch, Cut-out, Strains, Material nonlinearity, Digital image correlation.

1. Introduction

A separation system is an indispensable requirement to jettison the spent stages, payload fairing, strapons, retro rockets and satellites during the flight of a launch vehicle. A typical separation system comprise of an actuator, a release mechanism and a jettisoning system. An actuator can be electric or pyrotechnics. A release system can be mechanical such as merman band, collets mechanism or ball & lock systems. It can also be pyrotechnics such Flexible linear shaped charge (FLSC) or expandable tube assembly (XTA). Jettisoning systems in the form of thrusters such as spring are used to detach the two separating bodies with a relative velocity. NASA space vehicle design criterion provides a comparative study of various subsystems of the separation systems [NASA Space vehicle design criterion, 1970].

There are two vital aspects in the design of a stage separation system. The first one is to ensure that no contact happens between the separating bodies after separation. The second aspect deals with the structural adequacy of the separation system which has to withstand the flight loads with least joint rotation.

Expandable tube assembly (XTA) is a release mechanism that uses an explosive charge to fracture a ring with a notch to initiate the failure. The configuration and the cross section of XTA are shown in Figure 1.

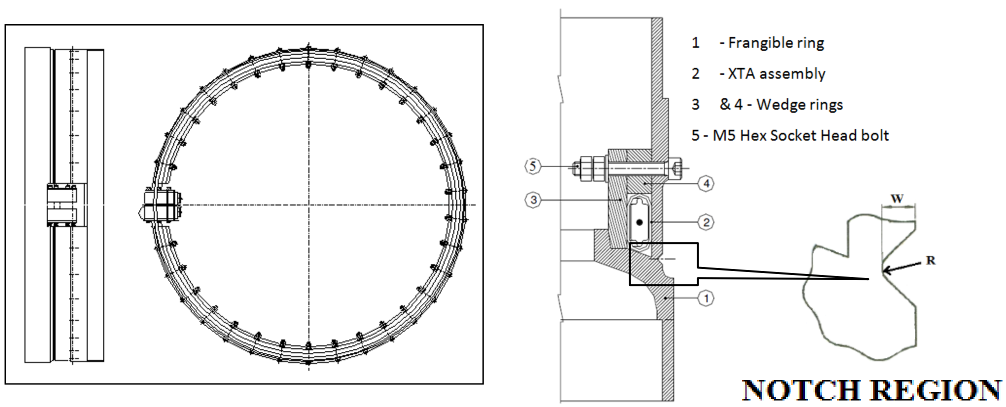

Figure 1. Configuration and cross section of expandable tube assembly.

As the name implies expandable tube assembly is an elliptical tube filled with explosives housed in the separation system. As the charge is initiated the tube expands due to the pressure and develops a fracture at the notch within 30μs and separation takes place subsequently within 40μs. The functioning of XTA at separation is shown in Figure 2.

The design philosophy of XTA lies in the design of the notch that initiates the fracture, enabling an effective separation with the contradictory requirement of the notch withstanding the expected flight loads successfully without any failure prior to separation. The design and analysis of XTA based separation system is carried out by parametric studies for various notch parameters such as notch radius (R) and severance distance (w) meeting the design and functional constraints. The design verification is carried out through FEM using ABAQUS software. The test results for design validation of notch are also carried out. This paper present the structural adequacy of an XTA based separation system and its validation through structural test.

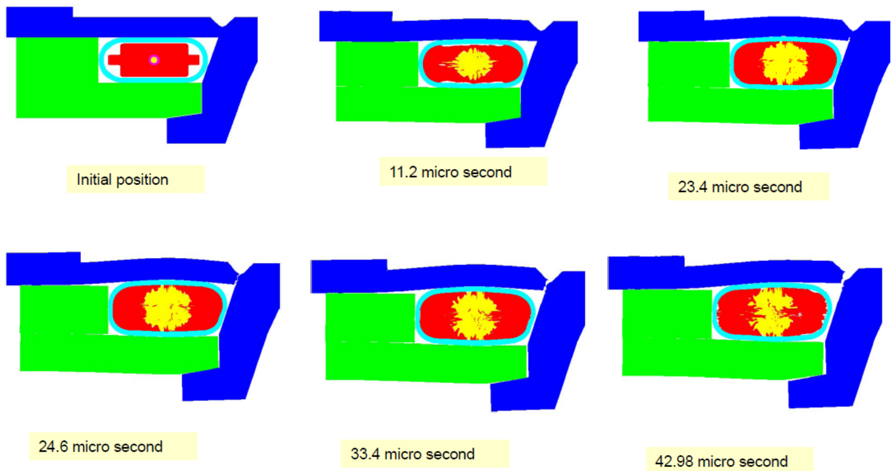

Figure 2. Functioning of expandable tube assembly.

1.1 Structural design of XTA separation system

The structural design of XTA based separation system calls for an estimation of stresses under expected flight loads in the frangible ring comprising of a notch and a cut out. The presence of a rectangular cut out in the ring enhances the load at the notch. Stress at the notch location also increases due to the higher bending from the diametrical offset in the load path from the upper and lower structures. The presence of a notch for initiating rupture along with the stress raisers such as cut out and eccentricities in the load path in XTA system makes its design challenging and calls for parametric studies. The material for the frangible ring is AA 7075 and is designed to withstand an equivalent axial compressive load of the order of 235 kN.

The notch radius is finalised based on parametric studies. As the stress at the notch is expected to be higher than yield stress, elasto-plastic analysis is essential to confirm the design.

1.2 Finite element analysis using ABAQUS software

1.2.1 Finite element model

The notch at the cut-out location of the separation system is the critical area and analysis is carried out to study this area more closely. A 3D model of the separation system with 45° sector from the centre of the cut-out along with the interfacing structure is modeled. A 16° cut-out is also modeled along with the notch. The eccentricities due to the top and bottom interfacing structures are also simulated. FE model generation and analysis is carried out using ABAQUS software. The FE model of structure analyzed is shown in Figure 3. A close up view of the notch is also shown in Figure 3. A top down mesh generation is adapted. Structured as well as swept meshing has been used for meshing. Linear hexahedron type element C3D8R is used for the analysis. FE data is as follows:

Number of nodes = 103827

Number of elements = 92299

ABAQUS provides the color coding of the mesh module and is advantageous because it is easy to identify the regions which cannot be meshed with the assigned elements. If a region cannot be meshed with the assigned elements then we can change the order of element or partition the region. ABAQUS provides a wide range of tools to refine the mesh such as the partition tool, combining small edges and faces with the adjacent faces and edges, edit mesh tool, re-meshing and mesh optimization based on the analysis results.

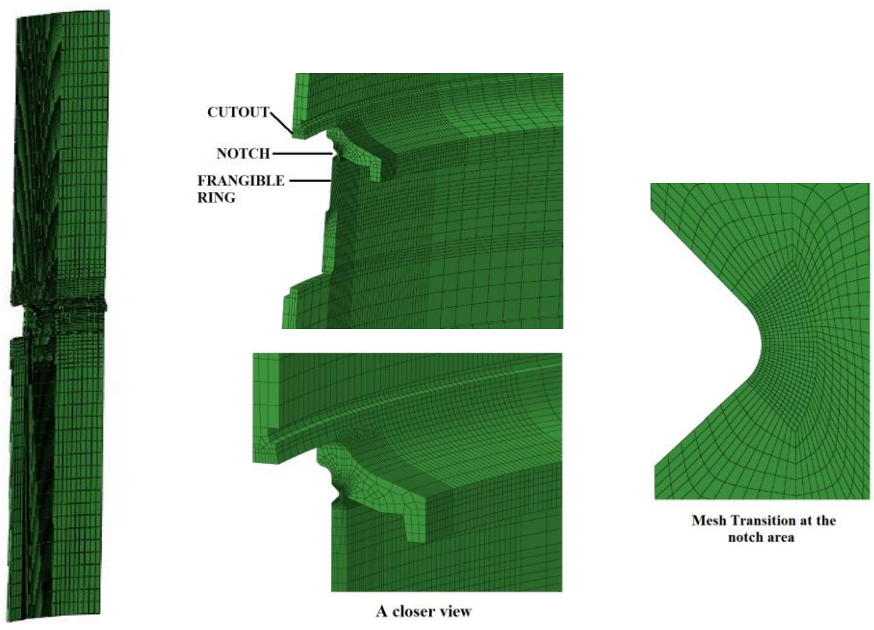

Figure 3. Details of Finite element model.

Mesh control provides a mesh transition option to mesh from coarse to fine elements. Verification of mesh is a feature in ABAQUS which allows the user to verify the quality of each element or the part as a whole. It provides the information regarding aspect ratio, minimum and maximum face corner angles, shortest edge and longest edge, geometric deviation factor and many more. This feature is available in almost all the software but it is more user friendly in ABAQUS.

All nodal connectivity is checked through query options available in ABAQUS. Query for mass properties is a superior feature for calculating the mass based on the density data mentioned in the material properties. Features such as mesh control, verification of mesh, free edges option etc is available in ABAQUS ensure the finite element model adequacy.

1.2.2 Loads and boundary conditions

Compressive axial load of 235 kN is applied at the fore end and the aft end is simply supported. Symmetric boundary conditions are provided at the edges of the structure and are shown in Figure 4.

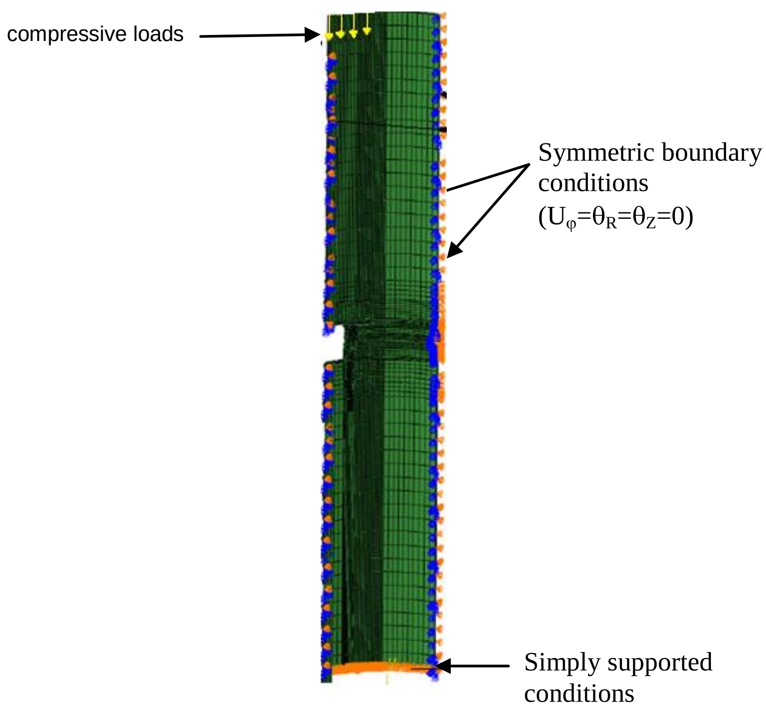

Figure 4. Boundary conditions applied on the FE model.

1.2.3 Material models

Material non-linear analysis has been carried out in ABAQUS software. The stress-strain curve for the material of frangible ring for the minimum guaranteed properties is shown in Figure 5.

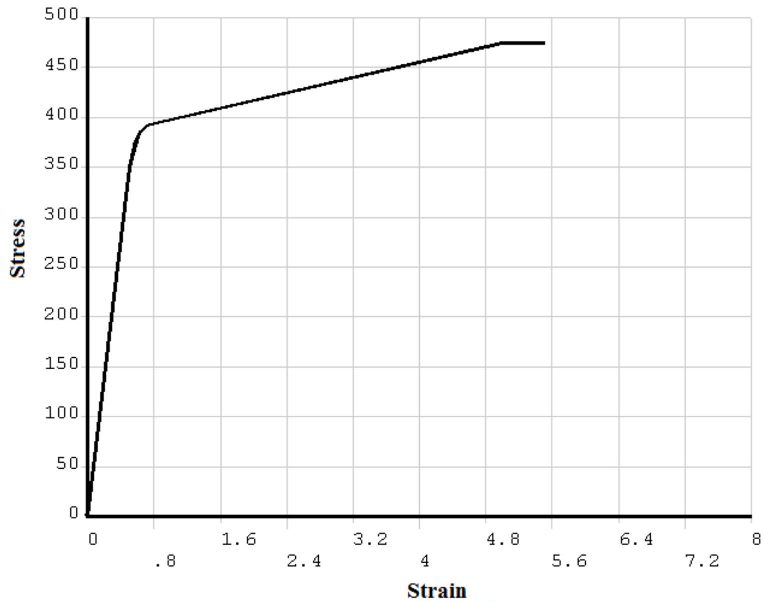

Figure 5. Stress-strain curve for minimum assured properties.

ABAQUS provides a wide variety of material models viz. elasticity, plasticity, damage etc. In elasticity, software defines linear elasticity, porous elasticity, hypo elasticity, hyper elasticity, viscoelasticity etc. In plasticity, software defines classical metal plasticity, creep, viscous and many more material models [Abaqus user manual].

The linear elastic model can define isotropic, orthotropic, or anisotropic material behavior. Stress and strain based failure theories are also provided with linear elasticity. However, the material model followed in this work is that of elasto plastic. The values of stress and strain are taken from Figure 5 and feed in the software under isotropic plastic material model. The isotropic hardening model is used because it involves the overall plastic straining.

1.2.4 FE analysis results

Maximum axial stress is -404 MPa which is at the notch location and is beyond the yield strength of the material. This is shown in Figure 6. The variation of axial stress with the distance away from the notch tip is shown in Figure 7.

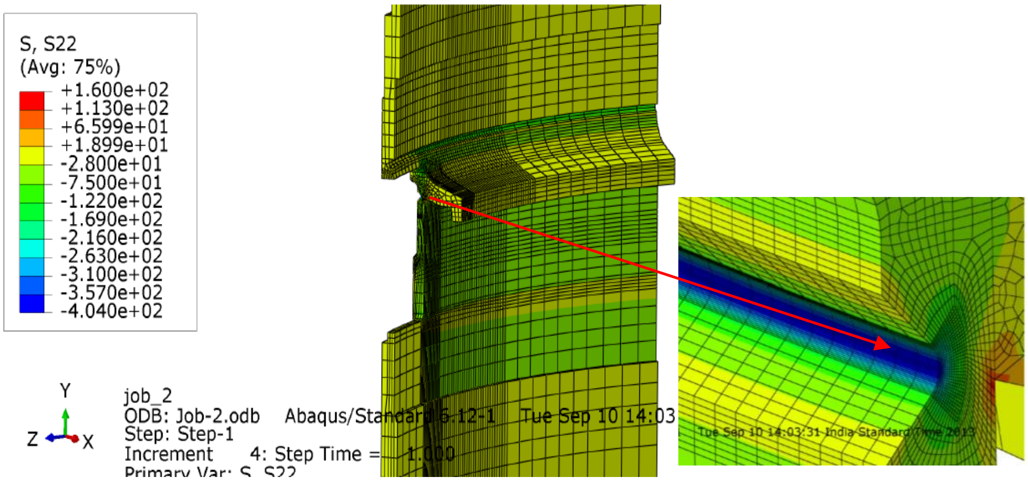

Figure 6. Axial stress in the structure.

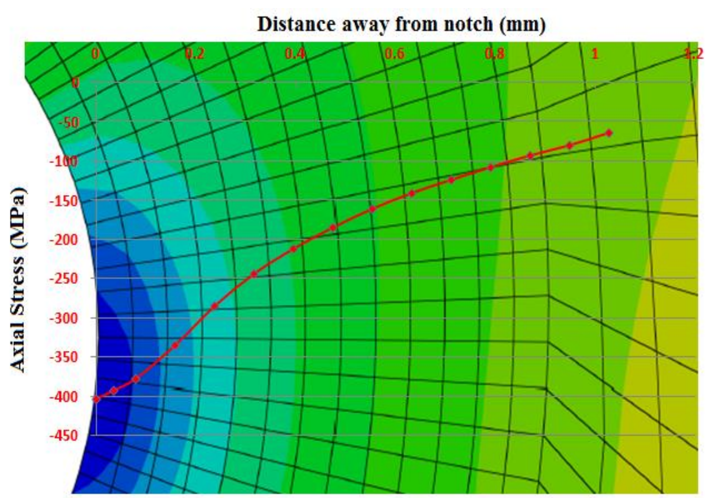

Figure 7.Variation of axial stress w.r.t distance from the notch location.

Maximum total strain in the structure is -6598 με at the notch location. The value of total stain at the notch location away from the free edge is -5118 με. This is shown in Figure 8. The variation of axial total strain with distance away for the notch tip is shown in Figure 9.

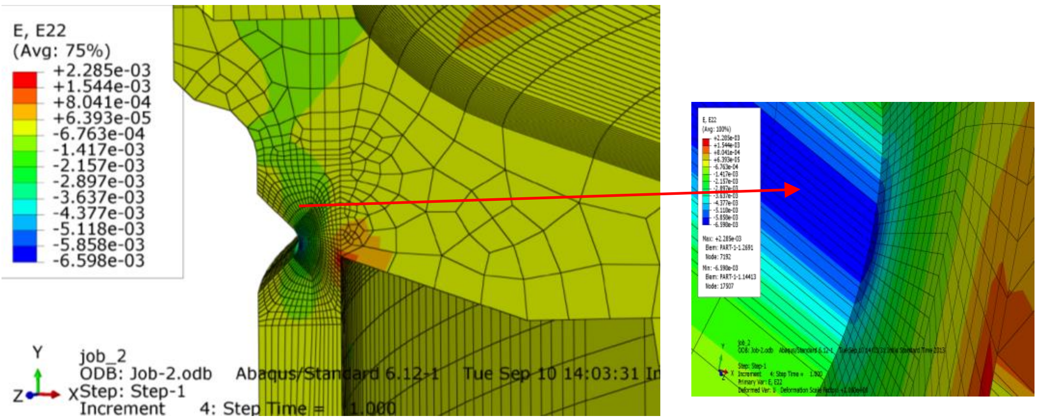

Figure 8. Total strain at the notch location.

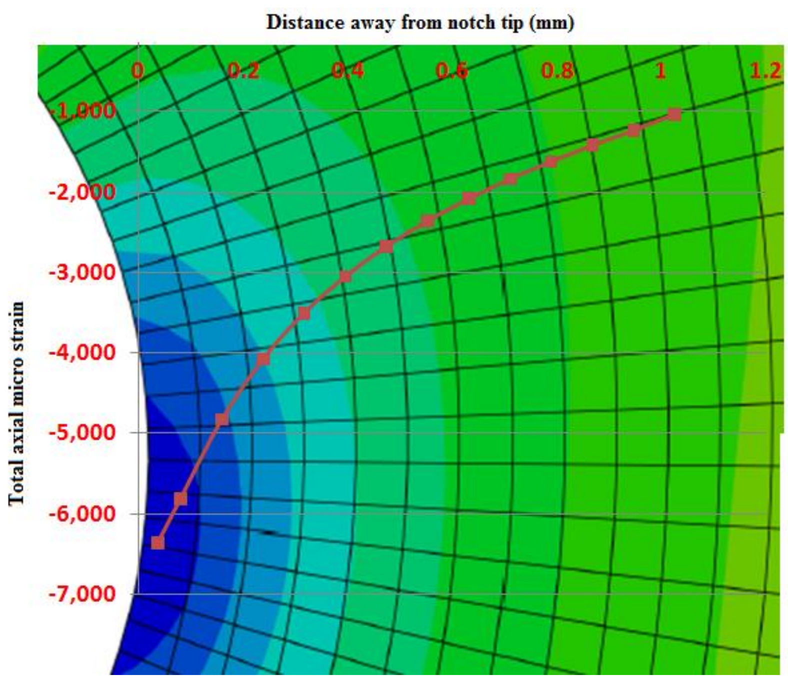

Figure 9. Variation of total axial strain w.r.t. distance from the notch location

Maximum value of plastic strain is -850 με. Plastic strain at the notch location away from the edge is -566 μ. Plastic strain at the notch location is shown in Figure 10. The variation of axial plastic strain with distance away for the notch tip is shown in Figure 11.

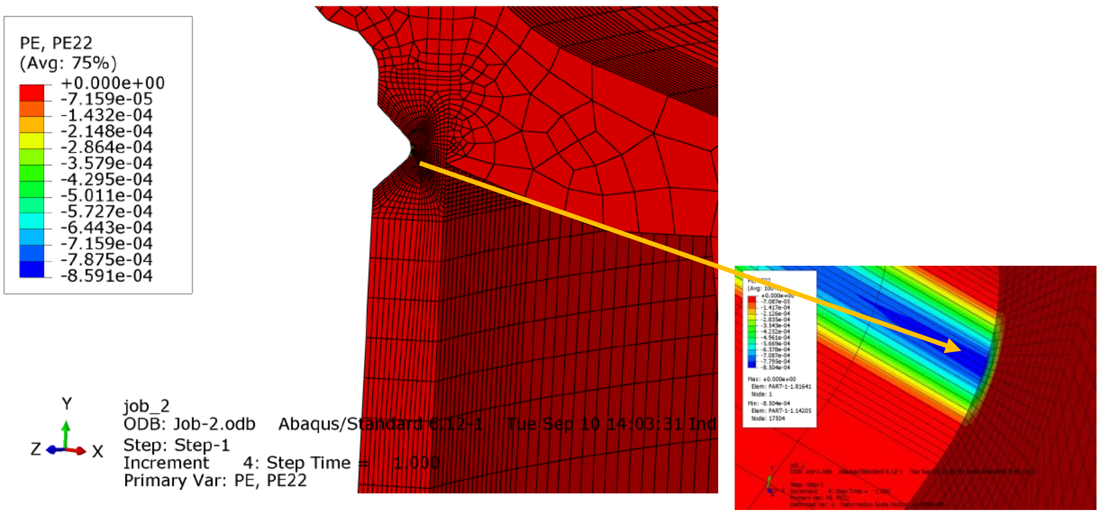

Figure 10. Plastic strain at the notch location.

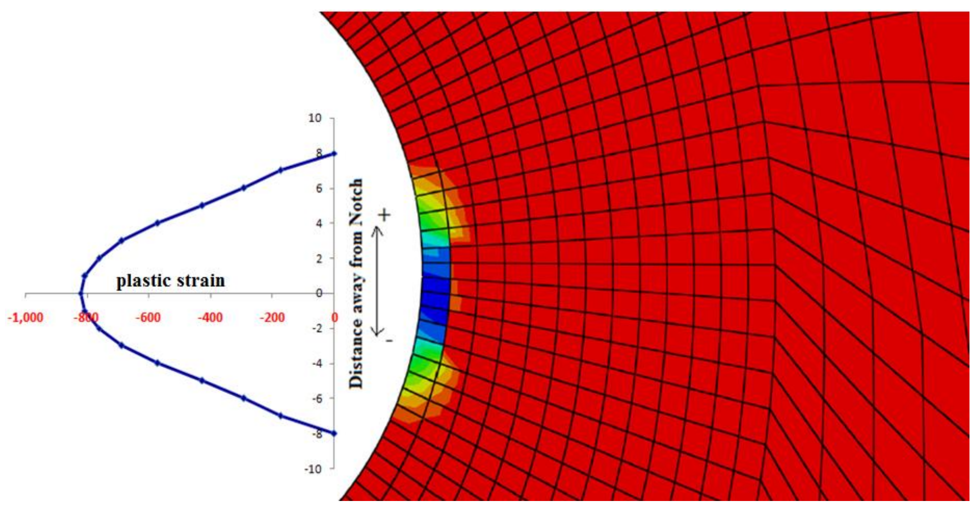

Figure 11. Variation of plastic axial strain at the notch location.

We observe that the general total strain is 0.13 ε_u and concentrated plastic strain is 0.02 ε_u which is in agreement with the elasto plastic strain acceptance criterion. This shows that the design is adequate to withstand the flight loads.

2. Design validation

Structural test of expandable tube assembly is carried out for an equivalent axial compressive load of 235 kN and the test setup is shown in Figure 12. Digital image correlation is a non contact optical technique for obtaining full field deformation by recording deformation and motion of speckle patterns on a specimen surface before and after deformation of the body. It is based on the notion that the applied stresses change the thickness as well as the optical properties of materials. Digital image correlation technique is used to measure the strains for the XTA separation system.

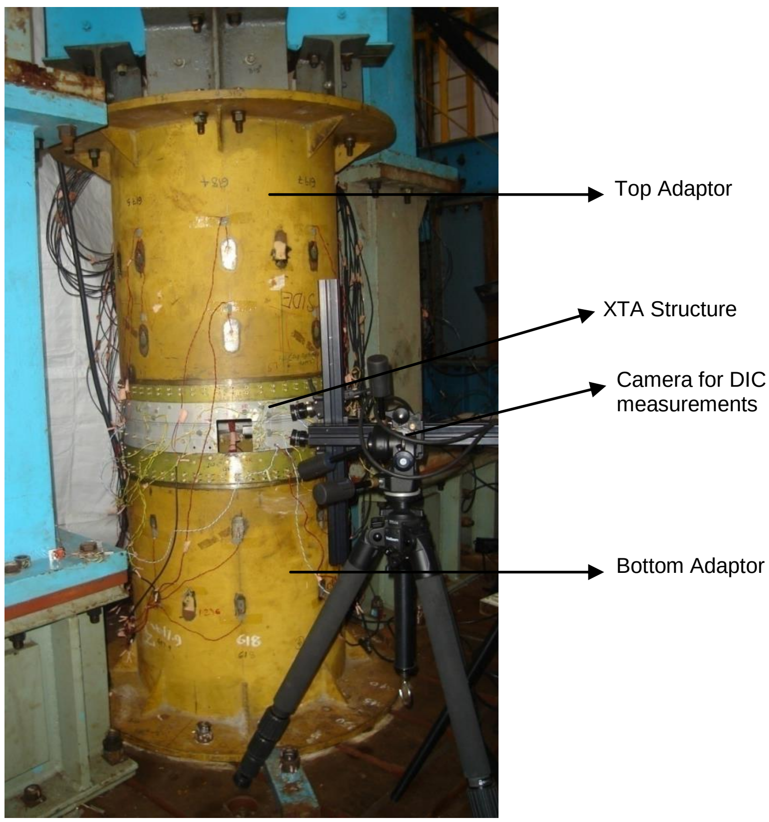

Figure 12. Structural test setup of XTA separation system.

Total strain at the notch location is shown in Figure 13 based on DIC measurements. The value of total stain at the notch location away from the free edge is -4924 με. Plastic strain at the notch location is shown in Figure 14. Plastic strain at the notch location away from the edge is -556 με.

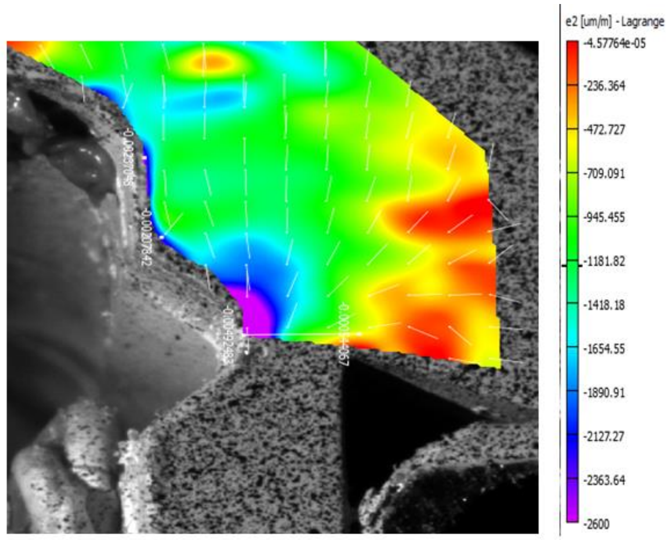

Figure 13. Total strain at the notch location.

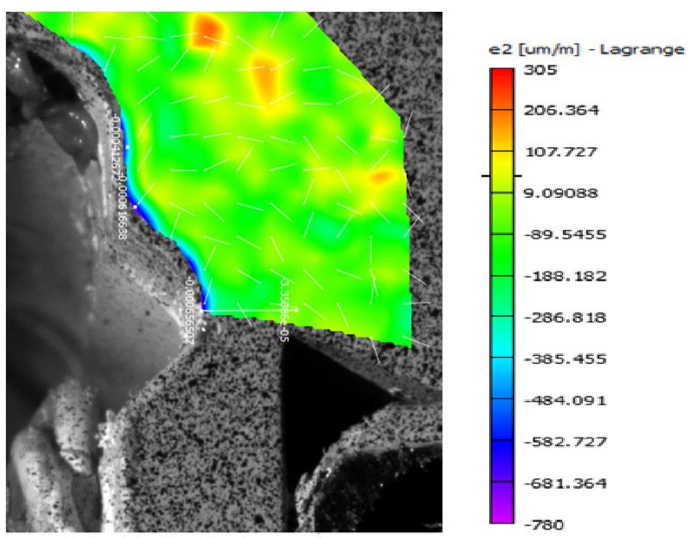

Figure 14. Plastic strain at the notch location.

It is observed that there is 4% and 1.7% difference in the total and plastic strain respectively between the analysis results and measured values. Hence there is a good match in the total and plastic strain values through ABAQUS software and test results.

3. Conclusion

Expandable tube assembly (XTA) is a separation system that uses an explosive charge to fracture a ring with a notch to initiate the failure. FE analysis through ABAQUS software and validation through structural test has been carried out and is presented in this paper. The notch provided to initiate separation, results in complexities of high stress concentration and plastic zones which demands a material non linear analysis of the separation system for the flight loads.

A 3D model with material nonlinear analysis using ABAQUS software has been carried out and the capability of the software for meshing, job and visualizations are explored. The results of high stresses near the notch in the zone of yielding have been captured through a detailed FE analysis. It is found that the plastic strain at the notch location away from the free edge from FE analysis and test is -566 με and -556 με respectively. This is a good match of the plastic strain in notch based separation system with stress raisers such as cut-out, notch and eccentricities. FE analysis using ABAQUS software and test validation clearly indicate that ABAQUS software is able to capture the plastic region in a complex system with notch, cut out and eccentricities.

4. Acknowledgements

Structural qualification test of the XTA separation system by Integrated Structural Test Facility (INSTEF) and post test data by Experimental Mechanics Division (EXMD) are highly acknowledged.

5. References

1. NASA Space vehicle design criterion (Structure) NASA SP 8056, Oct 1970.
2. ABAQUS/CAE user’s manual.