1. Abstract

This Finite Element (FE) analysis of Hemispherical Gyroscope is required to evaluate the behavior of the component in actual working conditions. The experimental study together with FE modelling to capture the delamination failure is presented.

Keywords**: Shock Analysis, Piezoelectric Analysis, Orthotropic Material, Hypermesh, Abaqus**

2. Introduction

The hemispherical gyroscope is required to evaluate in the three main different loading/working conditions like shock, thermal and piezoelectric loads. In the shock analysis, the assembly will be subjected to sudden shock of 60g at 11 millisec. The behavior is upon the sudden shock is observed at every component within the assembly in terms of displacement and acceleration w.r.t time.

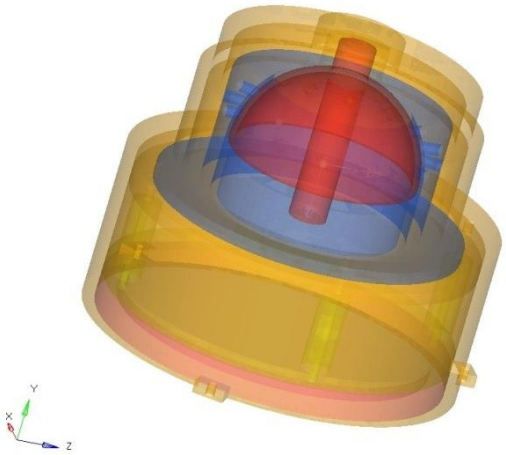

Figure 1: Hemispherical Gyroscope

The outer component is fitted at the constraints, internal components within the outer component are tight fitted. To identify the behavior of the assembly in extreme weather conditions like +70ºc to -30ºc with and interval of 10ºc applied as a thermal loading (convection). A temperature distribution in the assembly has been observed from the analysis. This gyroscope in its working conditions is rotating in its own axis. The internal component will then be subjected to deformation of 6th mode from the modal analysis, as it is operating with that frequency.

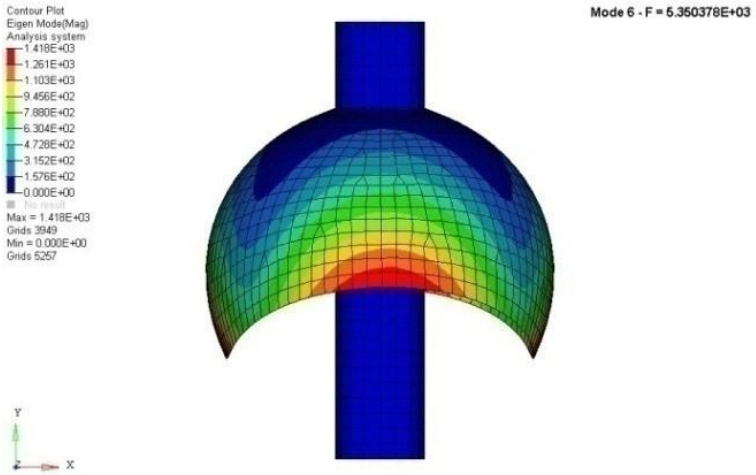

Figure 2: 6th Mode from the modal analysis

So to counter act this behavior an electricity is passed into the component. This phenomenon is called piezoelectric analysis. A very thin piezoelectric strips will be attached to the outer component. Finally the force will be generated as an outcome from this analysis. From that point a stress will be calculated to observe that it is within the yield limit.

3. Discussion of FEA approaches

The internal three components are made quartz material (orthotropic) and rest of the components including main outer surface is made of aluminum. An FE model for all the component within the assembly are generated with the help of CHEXA8, CTETRA4 element types. A shock load (acceleration vs time) of 6g at 11 milli sec has been shown below.

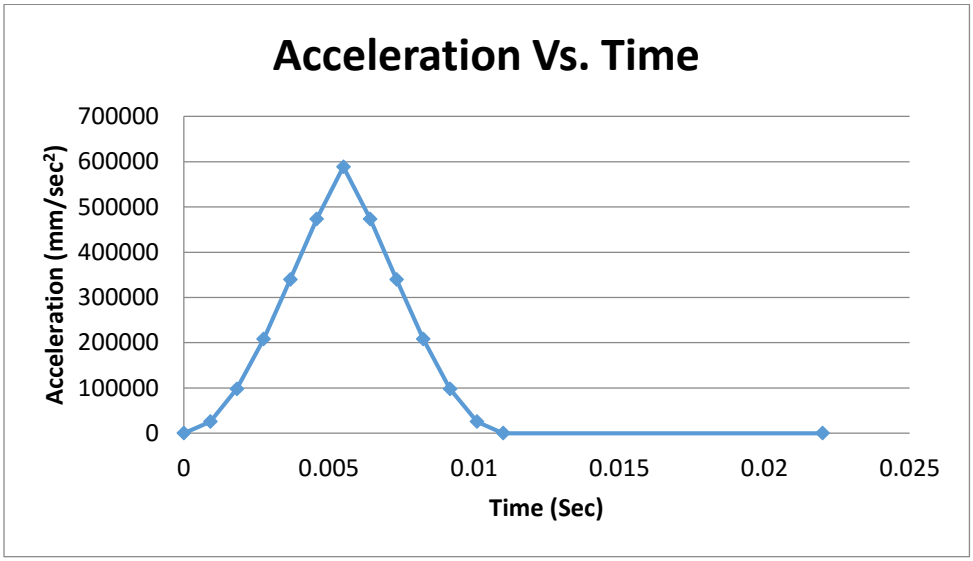

Figure 3: Shock Load: Acceleration vs. Time

Table 1: Piezoelectric analysis loading cases

Sub-Case	Piezoelectric StripsThickness (mm)	Voltage (Volts)
1	0.01	90
2	0.03	90
3	0.05	90
4	0.1	90
5	0.01	900
6	0.01	9000

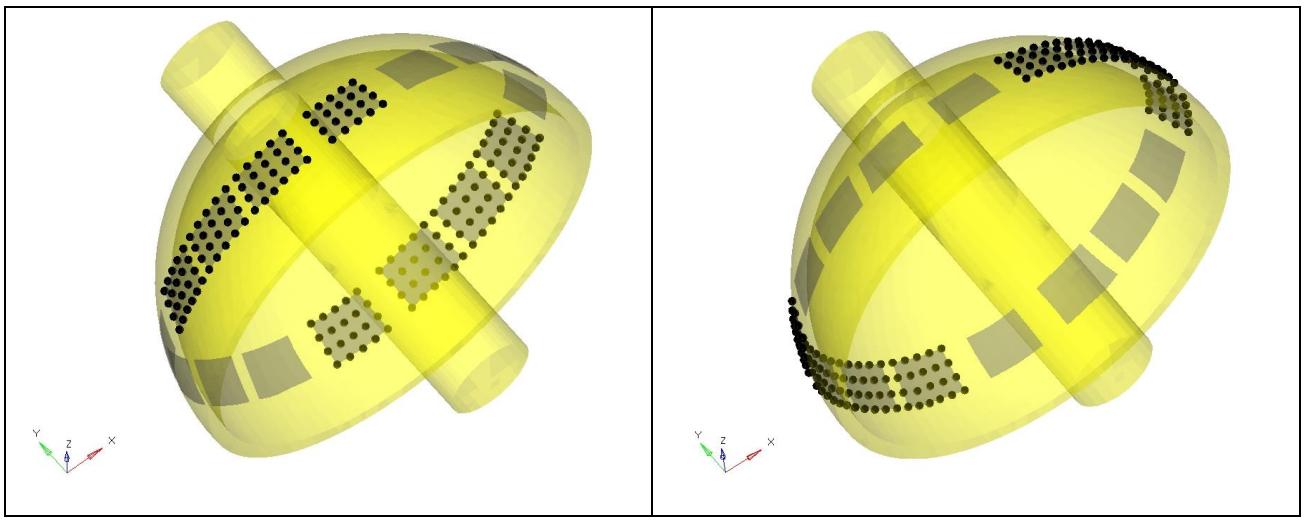

Figure 4: Voltage applied on the internal component within the assembly

4. Numerical Simulation

The piezoelectric analysis, shock analysis and thermal analysis is done using commercially available software like Abaqus CAE 6.10 and Radioss 12.0 (Hyperworks).

5. FEA Results

The numerical analysis was conducted using transient analysis, thermal analysis were modelled with quartz orthotropic material behavior. Displacement & Stress distribution at the constraint location in the below figure 5 can be observed.

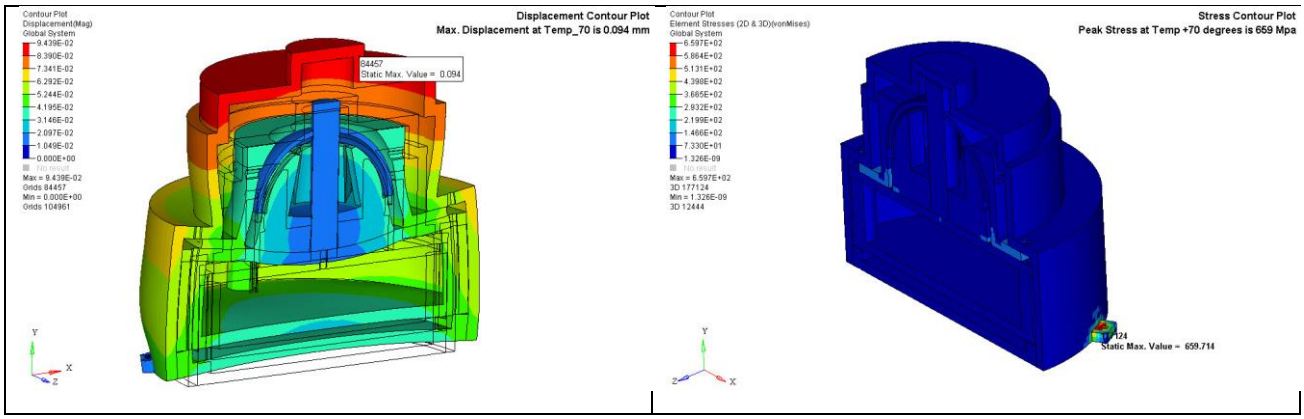

Figure 5: Displacement and Stress Contour Plots of assembly in the +70ºc thermal loading

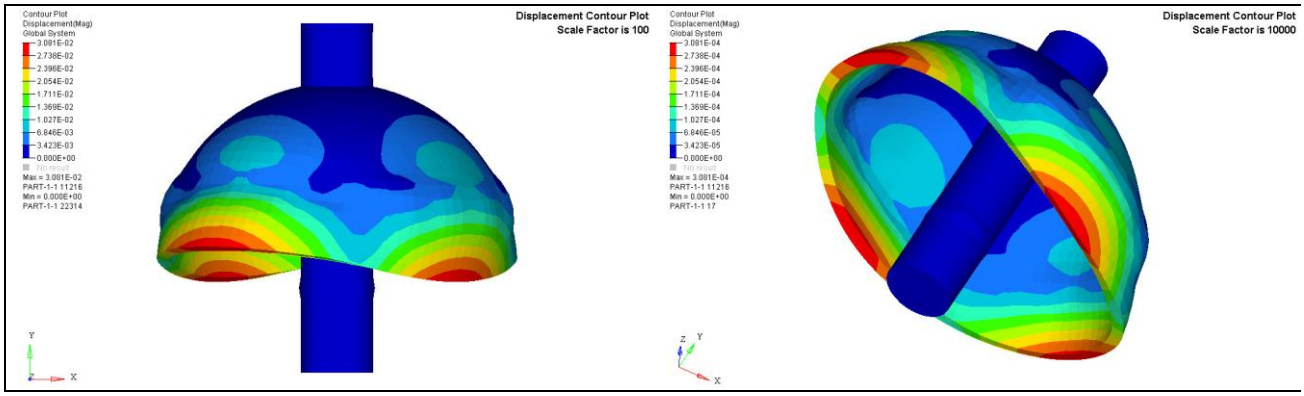

Figure 6: Piezoelectric Analysis: 9000v with 0.01 mm (left) and 90 v with 0.01 mm (piezoelectric strips thickness) (right)

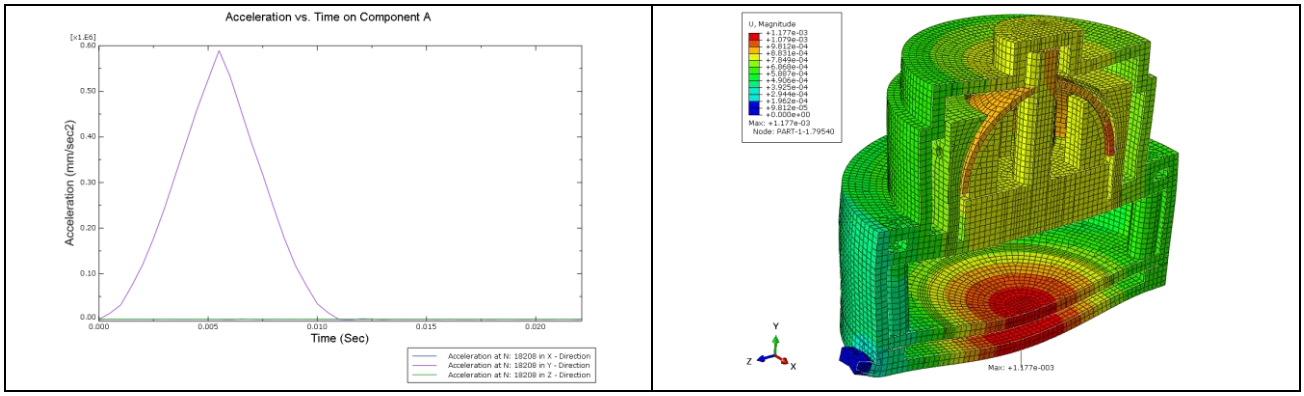

Figure 7: Shock Analysis: Acceleration vs time response on the component shown in Fig 6 (Left) and Stress contour of the assembly (Section-cut)

6. Conclusion

The behavior of the assembly in piezoelectric analysis was observed and a force has been calculated from it with different voltages and piezo strips thickness.

Table 2: Force extracted according to the Thickness and Voltage applied

SUB CASE	Thickness (mm)	Voltage (Volts)	Max Force exerted (N)		
			X	Y	Z
1	0.01	90	0.5078	0.8540	0.5453
5	0.01	900	5.078	8.54	5.453
6	0.01	9000	50.78	85.4	54.53