Modal Analysis of PVC Foam for Automotive Applications using AUTODESK INVENTOR

APRIL [08-04-2008]

Ramesh S Sharma
Ramesh.sharma@augi.com

In this session we shall focus on the Modal Analysis of PVC foam for automotive applications using Autodesk Inventor 2008 – Modal Analysis.

Introduction

PVC foam products have many uses. Over three quarters of the global consumption of PVC products is in the form of foams, with flexible and rigid types being roughly equal in market size. Flexible foams are used in upholstery fabrics in commercial and domestic furniture; rigid foams are inside the metal and plastic walls of most refrigerators and freezers, or behind paper, metals and other surface materials in the case of thermal insulation panels in the construction sector. PVC is also used for moldings which include door frames, columns, window headers, pediments, medallions and rosettes. Found in more than 40 automotive parts, PVC provides acoustical insulation, air filtration, thermal insulation, reinforcement and protection, as well as increased comfort and reliability for vehicle owners. In the light of the above applications of PVC especially when they are subjected to dynamic loading conditions are required to understand the basic dynamic behavior by carrying out modal analysis of PVC using AUTODESK INVENTOR 2008 SERIES with ANSYS built-in.

Problem: To investigate the six natural frequencies of PVC foam with dimensions of 100 mm x 100mm x 20 mm and obtain the corresponding mode shapes.

Let's commence.

Step 1: Create a simple square of 100mm x 100mm as shown in fig 1

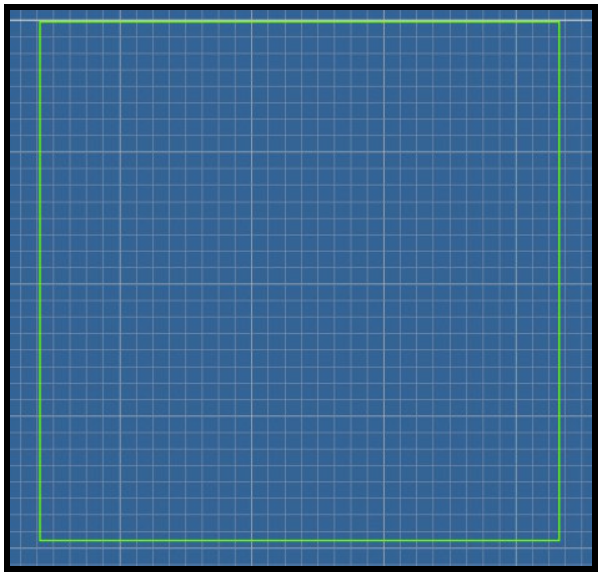


Figure 1

Step 2: Extrude the square to a distance of 20 mm

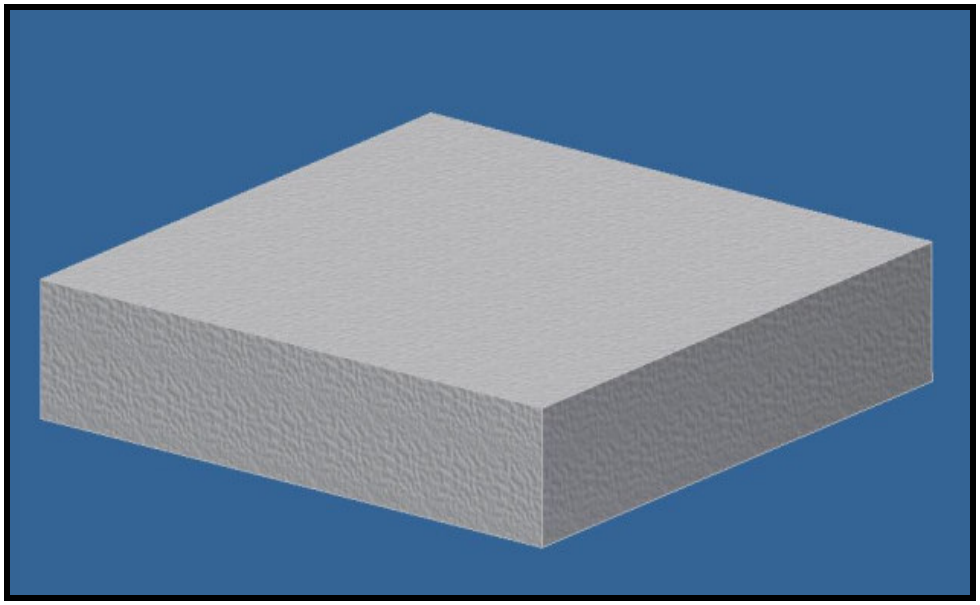


Figure 2

Step 3: Once the modeling of PVC plate is done, Open Autodesk Inventor stress Analysis module as shown in Fig 3.

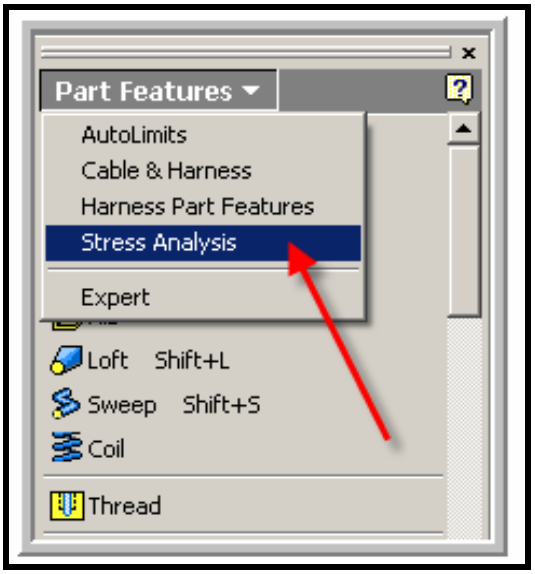


Figure 3

Step 4: Select the Material as PVC (U) to carryout analysis shown in fig below and press 'OK' button

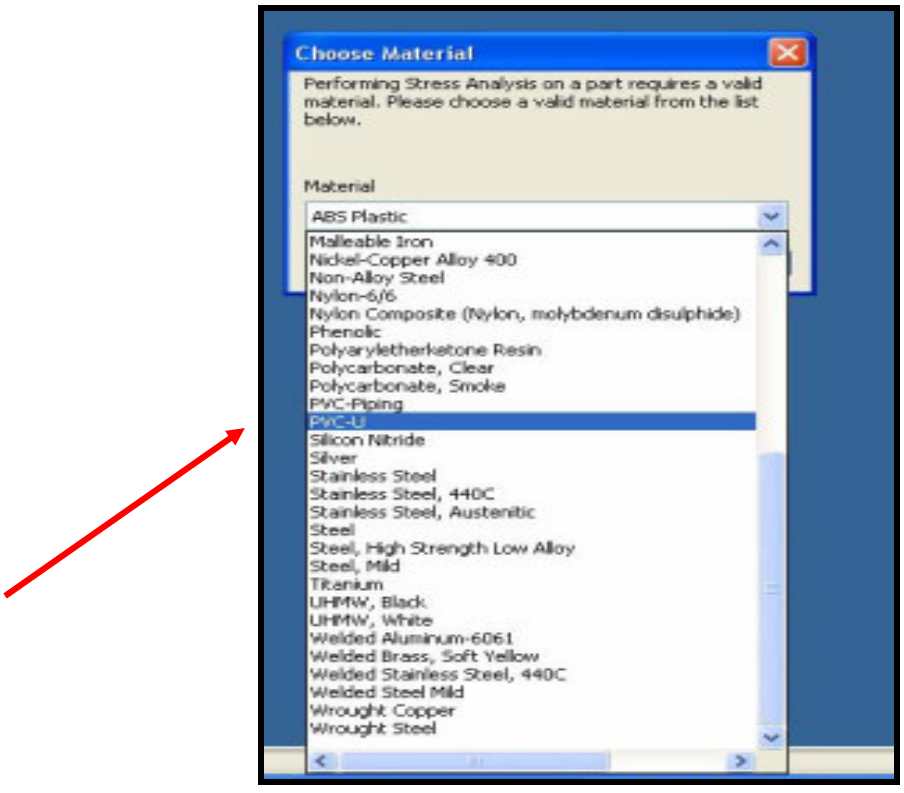
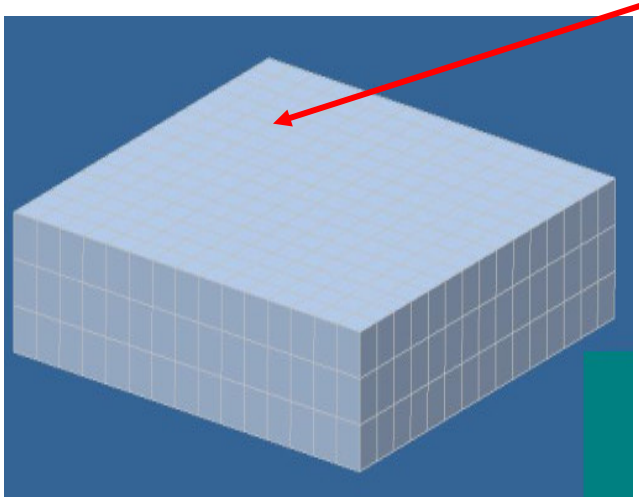


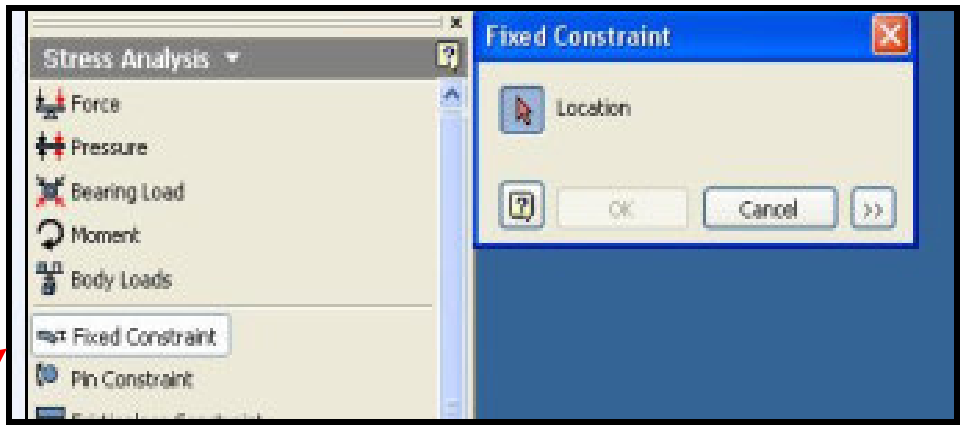
Figure 4

Step 5: Meshing of Model is the next step in any analysis. Select the model and apply mesh as shown in the fig below. Select Preview Mesh to look at Mapped



Finite Elements (Mapped Meshed Model)

Step 6: Application of Boundary Condition is one of the essential part of Analysis. Follow the steps shown in fig 6 to apply boundary condition.



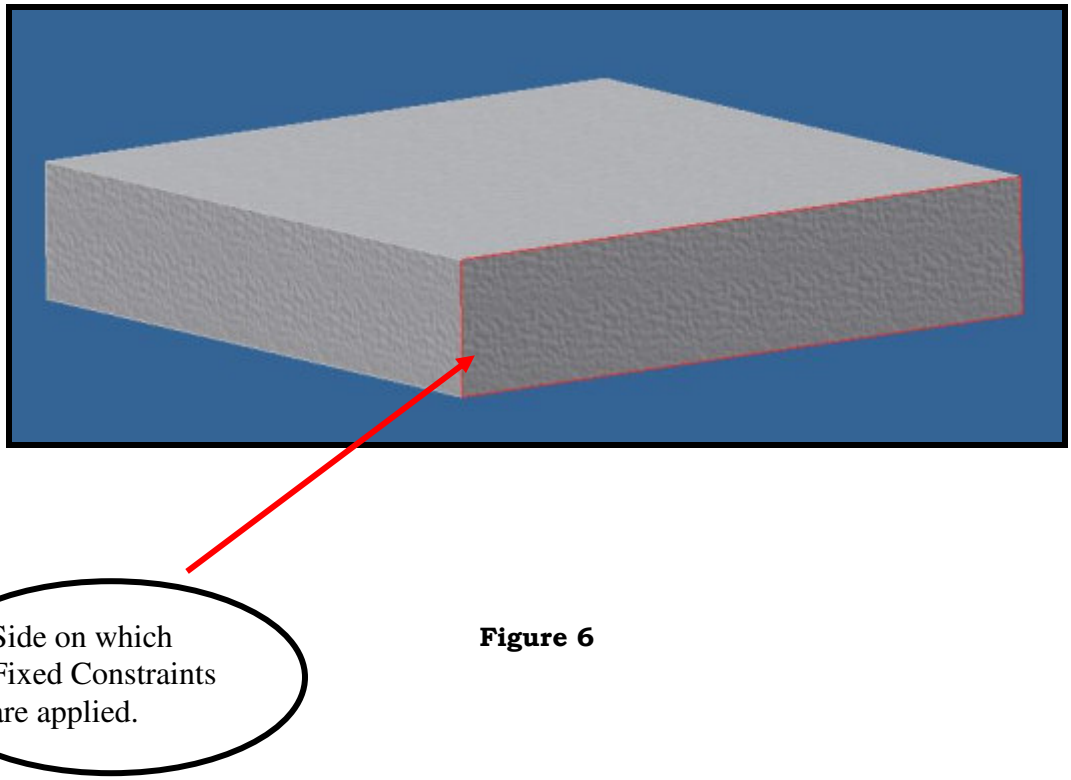


Figure 6

Step7: Click on Stress Analysis Update to solve for Modal Analysis solution as shown in Fig 7.

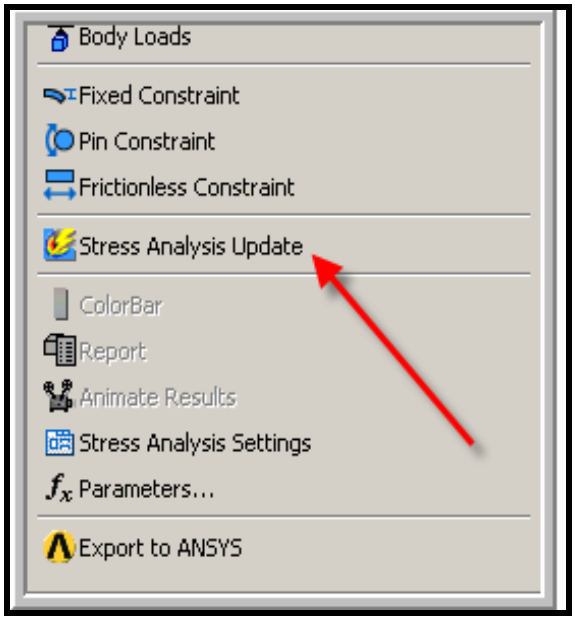


Figure 7

Step 8: Once the solution is done the browser bar indicates all the six mode shapes shown in fig 8.

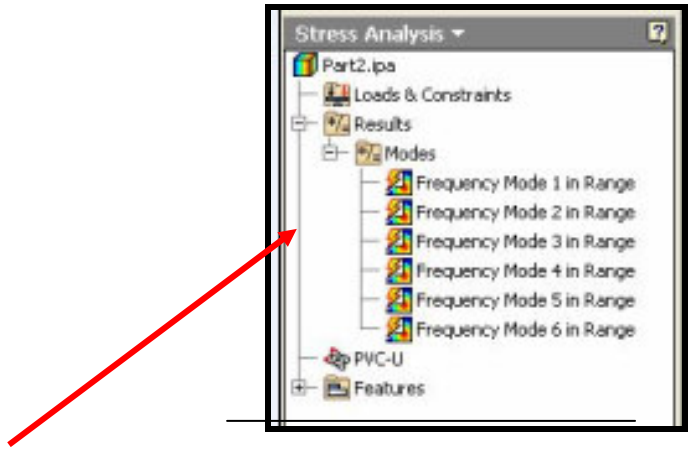


Figure 8

Step 9: Click on the First Mode Frequency shown in Fig 9.

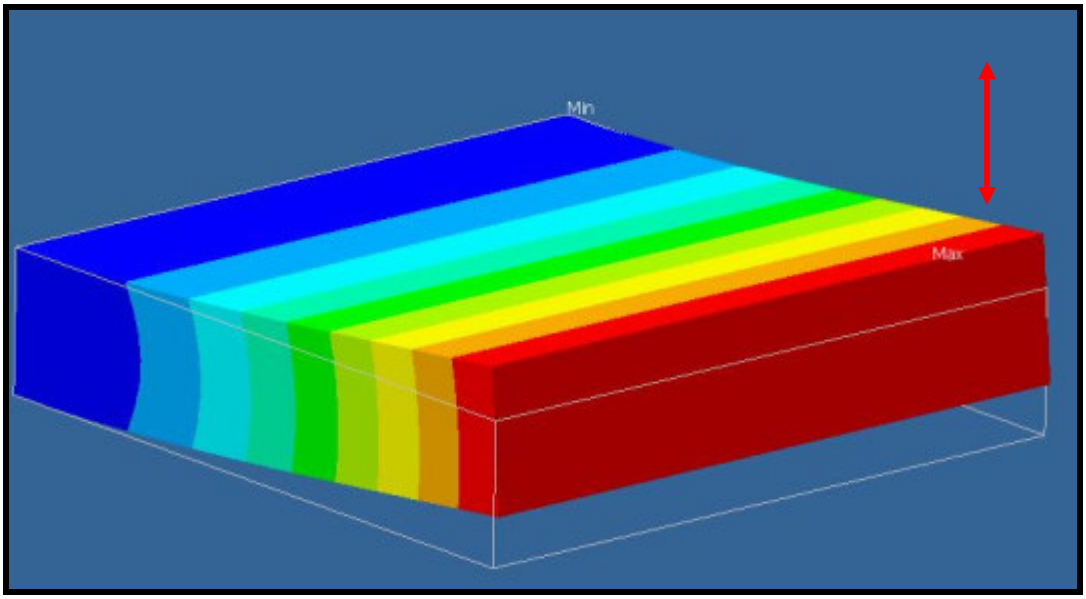


Figure 9- First Frequency Mode – First Bending

Step 10: Similarly, Click on the other Frequencies shown in Fig below

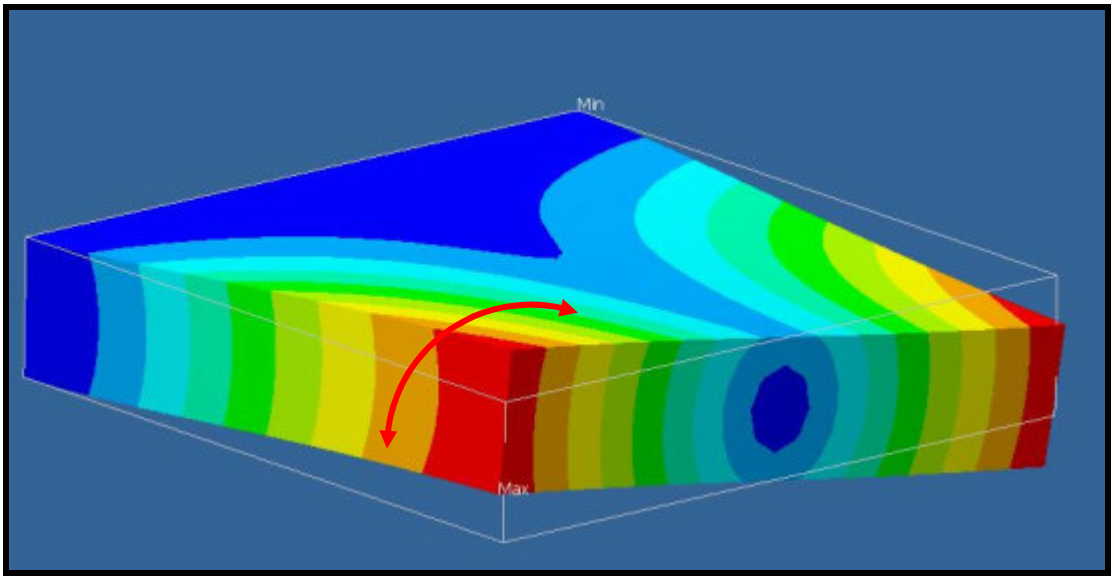


Figure 10 - Second Frequency Mode – First Torsion

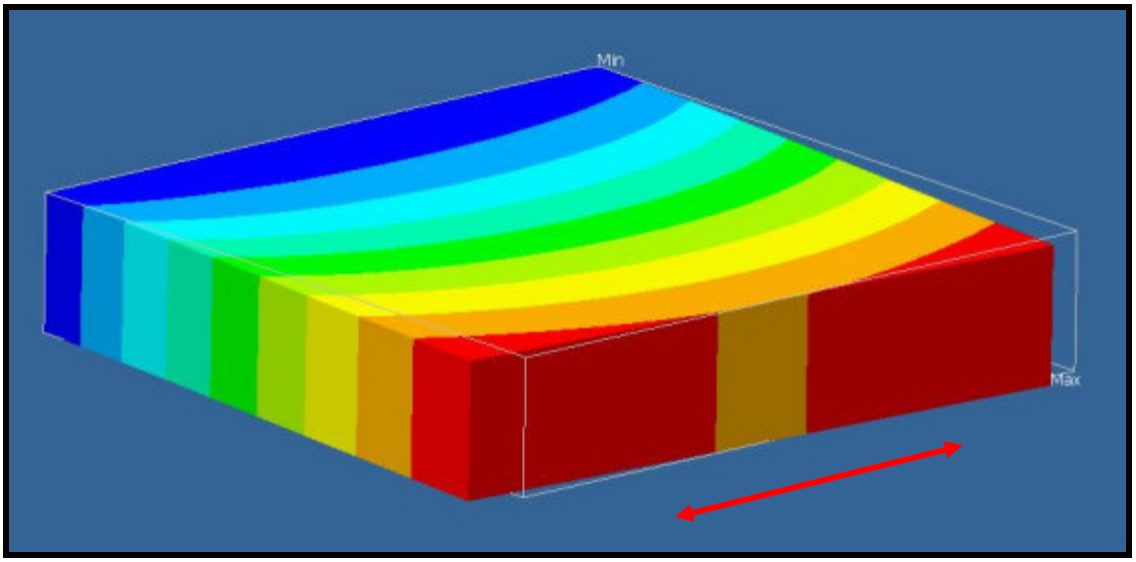


Figure 11 - Third Frequency Mode – Lateral Vibration

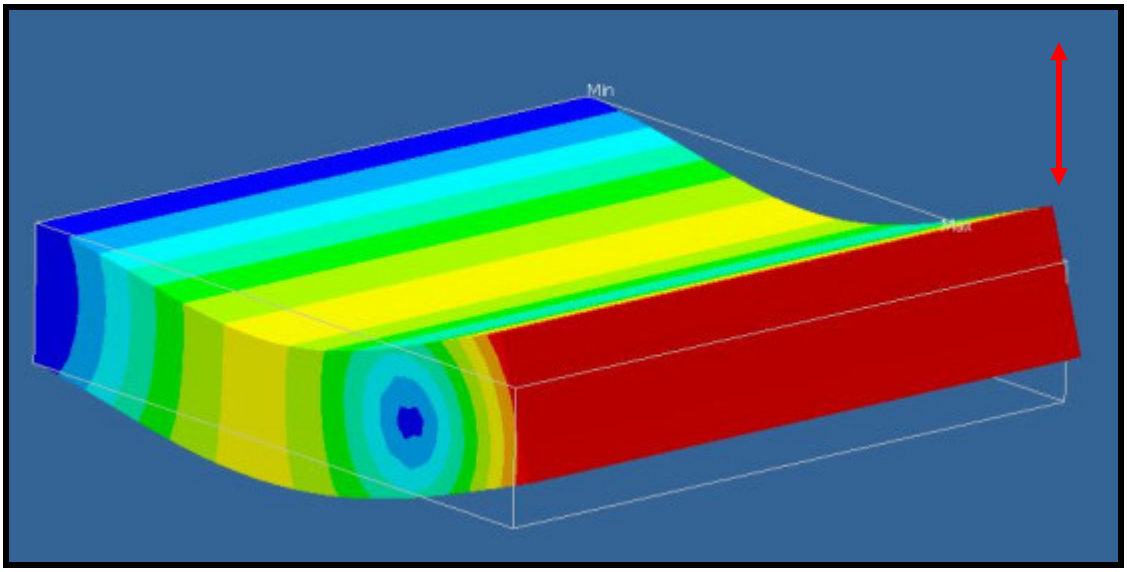


Figure 12 - Fourth Frequency Mode – Second Bending

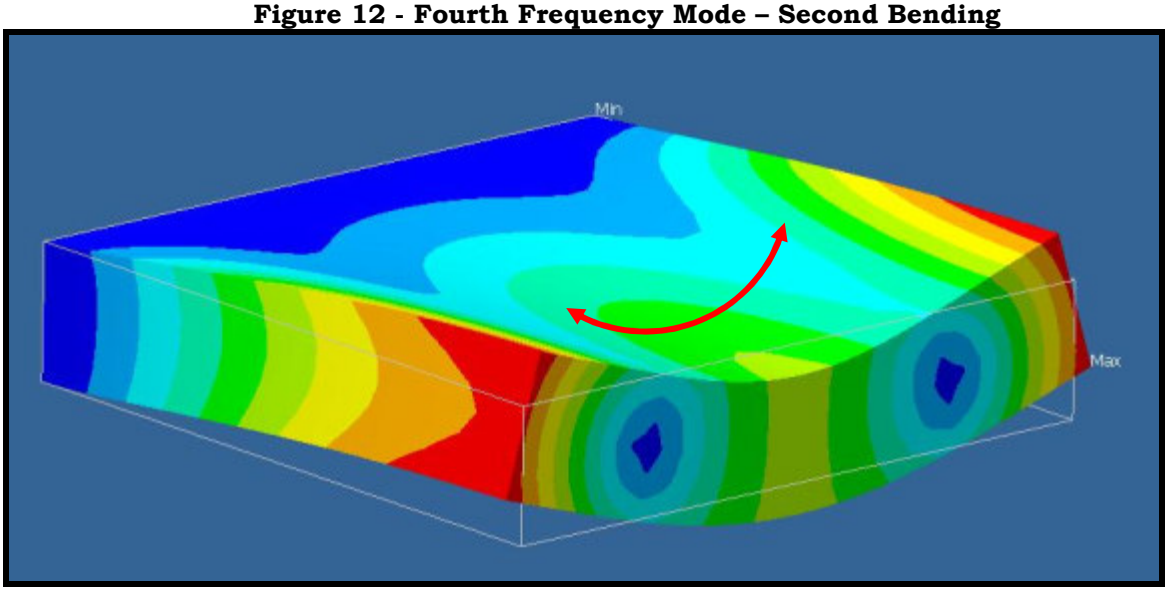


Figure 13 - Fifth Frequency Mode – Second Torsion

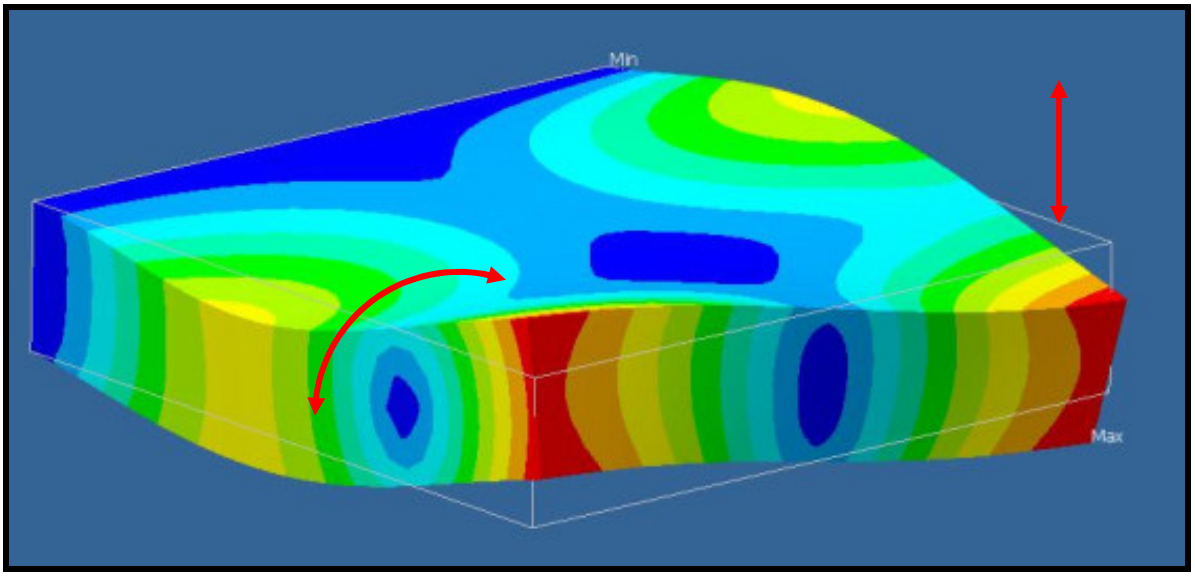


Figure 14 - Sixth Frequency Mode – Combined Bending and Torsion

Step 11: The frequency Range is provide in the table below

Frequency Mode 1 in Range	1	420.94 Hz
Frequency Mode 2 in Range	2	1044.5 Hz
Frequency Mode 3 in Range	3	1469.9 Hz
Frequency Mode 4 in Range	4	2320.8 Hz
Frequency Mode 5 in Range	5	2922.2 Hz
Frequency Mode 6 in Range	6	3327.1 Hz

Step 12: The Mode Shapes mentioned above can be animated by selecting the Animate Results.

Article submitted by **Ramesh S Sharma**, Board of Director, AUGI India. Currently, he is working as Assistant Professor in the Department of Mechanical Engineering at P E S Institute of Technology, Bangalore, INDIA, a leading technical educational institution in INDIA. He has been working on Autodesk products and training students in the areas of Modeling of automotive components and analysis. He is actively involved in the CAD/CAE projects in the department. *rssharma25@yahoo.com*