



ABOUT US

Qantur Technologies is the authorized distributors of ANSYS engineering simulation products and services in India, Srilanka & Bangladesh. We also partner with AlphaSTAR for advance composite & polymer/metal additive solutions within India and United Arab Emirates.

Qantur Technologies have used ANSYS Mechanical and ANSYS CFD with system coupling feature for solving multiphysics interaction between solid and fluid for the analysis of shock absorber.



CONTACT US

830, JMD Megapolis,
Sohna Road, Gurgaon
Haryana – 122018

Phone: +91 124 497 3337
Email: info@qantur.com
URL: www.qantur.com



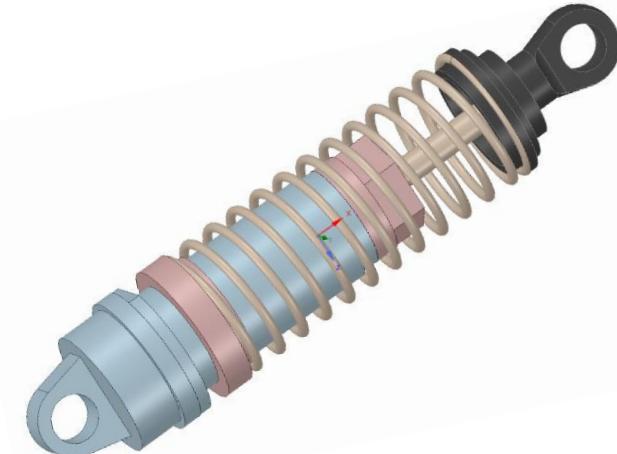
DESIGN AND ANALYSIS OF SHOCK ABSORBER IN ANSYS



PROBLEM DESCRIPTION

A leading antivibration component manufacturer had to develop a new product i.e. shock absorber to improve NVH characteristics of their components. However, manufacturer is not able to predict the performance parameters of shock absorber since it includes an interaction between fluid and solid. User wanted to predict stresses, deformation, pressure distribution, velocity distribution under different loading conditions so that they can design efficient shock absorber at minimum cost.

Shock absorbers do two things apart from controlling the movement of springs and suspension, shock absorbers also keep your tires in contact with the ground at all times. At rest or in motion, the bottom surface of your tires is the only part of your vehicle in contact with the road.



CAD Model of Shock Absorber



UNDERSTANDING

Shock absorber performance prediction is difficult since fluid and solid interacts with each other simultaneously. The correct stiffness is essential to maximize vehicle road holding and they must be able to function in harmony with the complete vehicle system. Therefore, the properties that are required by manufacturer cannot be obtained with physical testing directly. Hence, ANSYS Simulation is required to capture Multiphysics (fluid & solid) of shock absorber simultaneously for providing the required properties to the manufacturer.



SOLUTION

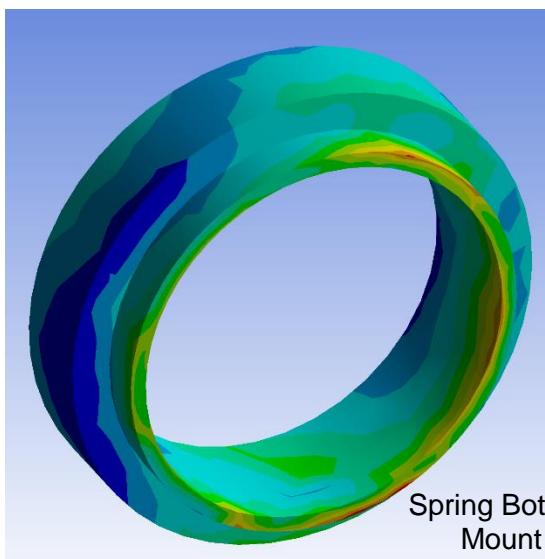
CAD model is created for shock absorber as shown in Figure. Displacement of 15 mm is applied to structural component. Subsequently, ANSYS Mechanical & ANSYS CFD with system coupling feature of ANSYS is used for solving Multiphysics interaction between solid and fluid. ANSYS successfully provided the stresses, deformation, pressure, velocity distribution and reaction force from the fluid in the shock absorber.



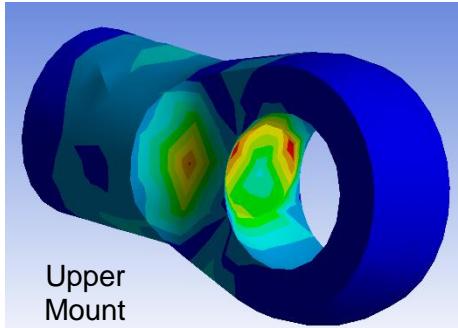
RESULTS

STRESS INDUCED IN DIFFERENT PARTS

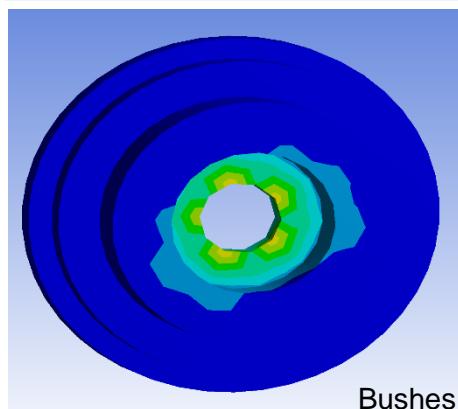
The red coloured highlighted zones depict the maximum stress region which is due to the pressure generated by the fluid on the surfaces. The maximum stress is less than the yield stress of material.



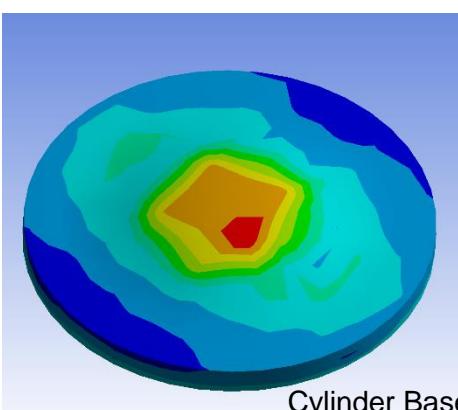
Spring Bottom Mount



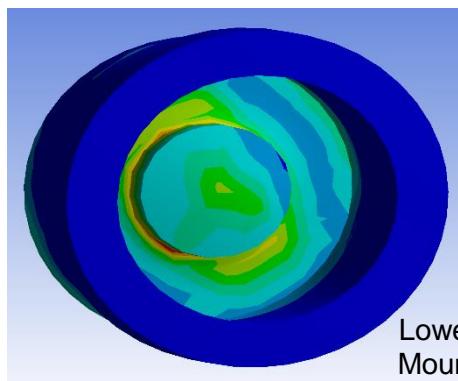
Upper Mount



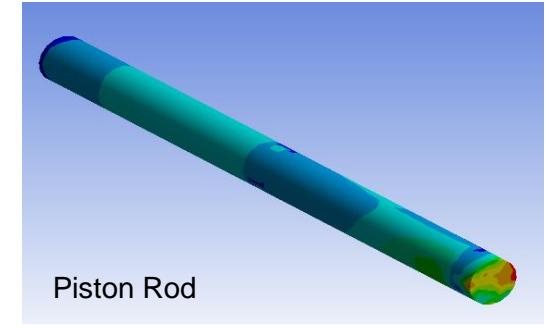
Bushes



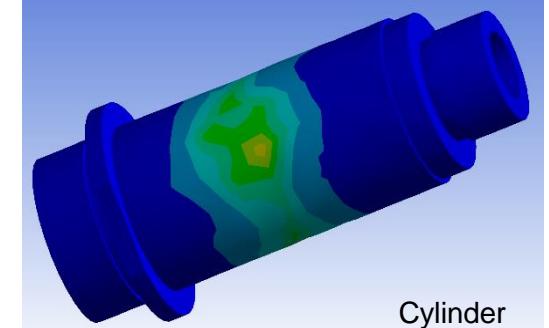
Cylinder Base



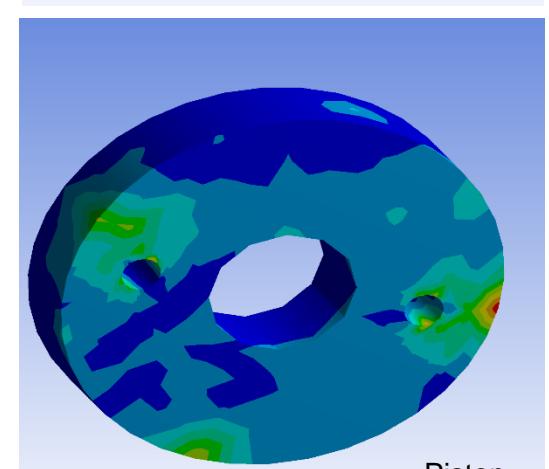
Lower Mount



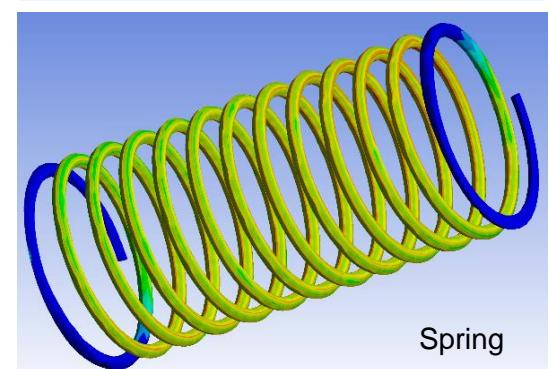
Piston Rod



Cylinder



Piston

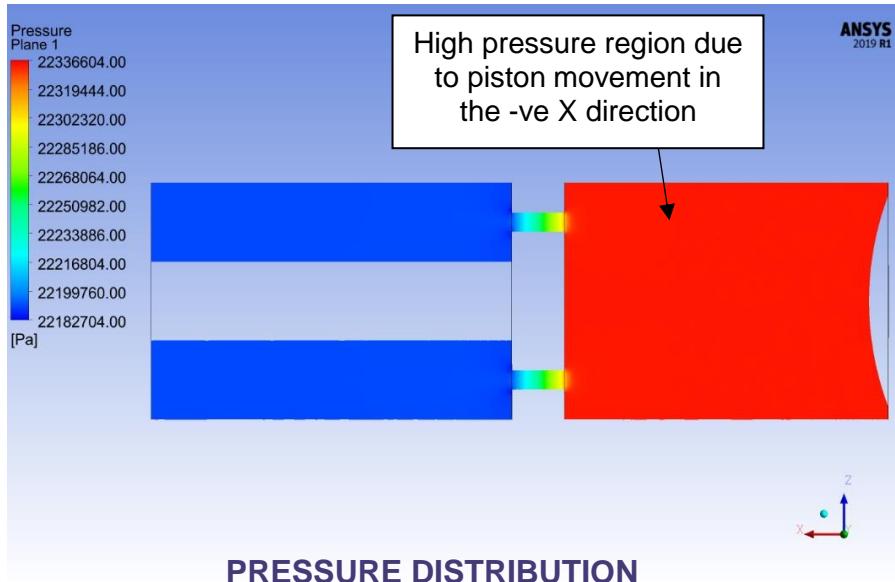


Spring

RESULTS CONTD.

CFD RESULTS FOR THE FLUID BEHAVIOUR WITHIN THE SHOCK ABSORBER

The pressure generated due to the movement of the piston increases as the piston keeps moving towards the bottom as there is less space for the fluid to displace abruptly.



The velocity at the passage region is high, as the piston movement forces the fluid to move through the passages.

