DYNAMIC ANALYSIS OF A REACTOR CORE SHROUD

Dr. K Suresh, M/s Transoft International Pvt. Ltd., Bangalore, INDIA.
Phone: 080-552 6507. Email: support@fluidyn.com
Dr. H.S.Kushwaha, Reactor Safety Division, BARC, Mumbai, INDIA.
Phone: 022-559 3743.
Dr. S.K.Gupta, Reactor Safety Division, BARC, Mumbai, INDIA.
Phone: 022-559 3776. Email: guptask@apsara.barc.ernet.in

ABSTRACT

The objective of this study is to find the acoustic loads on the reactor core shroud and dynamic analysis of the shroud following instantaneous guillotine rupture of the recirculation pipe. Analysis was done using a Finite Volume/Finite Element coupled code, ***fluidyn*-FSI**, for solving fluid-structure mechanical interaction problems. High -speed interaction between a structure and fluid requires simultaneous modeling of large deformations of structures and shock propagation in fluids. Finite Volume procedure is better suited to modeling varying flow regime and boundary condition in the fluid, whereas, Finite Element procedure is suited for nonlinear transient stress analysis in the structure. Interaction also requires remeshing of the fluid domain to ensure that the fluid domain conforms to the deforming structure and that the effects of the structure deformation are imposed on the fluid. ***fluidyn* -FSI** code performs this remeshing automatically.

Hexahedral elements and triangular shell elements are used to model the fluid and structural domains. The fluid is resolved with an expansion equation of state [1], which is a critical aspect of accurate hydrodynamic solution. A robust solver using MUSCL [2] approach is employed for the study. Depending on the propagation of acoustic wave, it employs effective fluxing at each volume interface. Explicit multistage Runge-Kutta time stepping [3] helps resolve time accurate behavior of flow field and update flow variables in time marching calculation. Third order accuracy ensures accurate resolution of spatial gradient and controls solution oscillation. Time step is calculated by considering CFL and von Neumann stability criterion to ensure stability of hydrodynamic flow field.

Stress calculation in the structure is carried out using finite element approach. Explicit time integration scheme is used for transient analysis. After every fluid cycle, the resulting load on the structure due to fluid pressure is used to compute the deformation of the structure. These deformations are used in an automatic remeshing algorithm, which moves the fluid mesh so as to conform to the deformed structure. Such a remeshing imparts velocity to the fluid mesh. The new state of the fluid is then used for the next fluid cycle.

The results demonstrate the applicability of the Fluidyn code for this class of problems.

1. PROBLEM DEFINITION

The objective of this study is to determine the acoustic load on a reactor core shroud following the instantaneous guillotine rupture of the recirculation pipe and carry out the dynamic analysis using numerical tools. The present study focuses on the dynamic loads acting on the core shroud and its effect on the deformation and stresses. The reactor is initially filled with superheated water and steam. Following an instantaneous rupture of the recirculation pipe, acoustic loads act on the shroud and deform it.

Variations of pressure at various points on the shroud, the resultant displacement of the shroud and stresses in the shroud are presented, as also the pressure, velocity and vapor fraction in the fluid region.

2. Methodology

The tool used for this analysis is ***fluidyn* - FSI**, a Finite Volume/Finite Element coupled code for solving fluid-structure mechanical interaction problems. The procedure has been developed for the solution of complex fluid and structure interaction problems involving fluid flow, and structure deformation that influence one another. The fluid solver solves the Navier-Stokes equations for compressible or incompressible flows based on finite volume method. The code is capable of simulating viscous, laminar or turbulent flows. It can also handle 2D and 3D structured, unstructured or hybrid, stationary or moving meshes. The structural solver is based on a finite element formulation for solving transient problems involving large deformation and material nonlinearity.

Automatic remeshing of the fluid domain is done to ensure that the fluid domain conforms to the deforming structure and that the effects of the structure deformation are imposed on the fluid. These features are built into the numerical code – ***fluidyn* -FSI.**

In the present study, the fluid flow is time dependent (transient). Simulation has been carried out with transient behavior of flow for accurate prediction of the flow field and its effect on the structure (shroud). Viscous effect plays a significant role in hydrodynamic and thermal diffusion. As the fluid is at a very high pressure and high specific energy content, there is a chance of breakdown of the system that exposes fluid to ambient condition.

State of the fluid is resolved with an expansion equation of state [1], which is a critical aspect of accurate hydrodynamic solution. MUSCL [2] approach, a robust solver, has been used. Depending on the propagation of acoustic wave, it employs effective fluxing at each volume interface. Explicit multistage Runge-Kutta time stepping [3] helps resolve time accurate behavior of flow field and update flow variables in time marching calculation.

Third order accuracy ensures accurate resolution of spatial gradient and controls solution oscillation. Time step is calculated by considering Courant, Fredrich and Lewy criterion and von Neumann stability criterion to ensure stability of hydrodynamic flow field.

The stress calculation in the structure is carried out using finite element approach. Explicit time integration scheme based on convected coordinates is used for transient analysis. After every fluid cycle, the resulting load on the structure due to fluid pressure is used to compute the deformation and stresses in the structure. These deformations are used in an automatic remeshing algorithm, which adapts the fluid mesh so as to conform to the deformed structure. Such a remeshing imparts velocity to the fluid mesh. The new state of the fluid is then used for the next fluid cycle.

3. Computational grid

Only one half of the structure (reactor + shroud) is modeled, as the problem is symmetric about the vertical plane (XY plane) passing through the center of the core shroud. The nozzle lies parallel to X-axis. The axis of the core shroud is along Y-axis. The pressure vessel wall is considered rigid.

The fluid is modeled using 8-noded hexahedral and 6-noded wedge elements. Figure-1 shows the finite volume mesh used for the computation. The mesh has 2512 cells (3091 nodes).

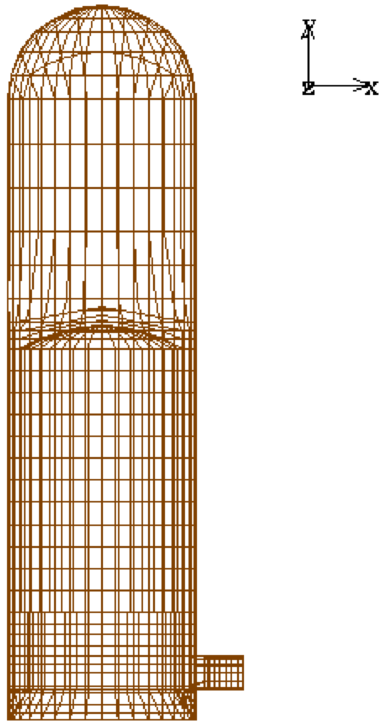

Figure-1

The core shroud is modeled using 3-noded linear triangular shell elements. Figure 2 shows the finite element mesh. A total of 1222 elements (644 nodes) have been used. Mesh includes top and bottom grid plates that reinforce the core shroud.

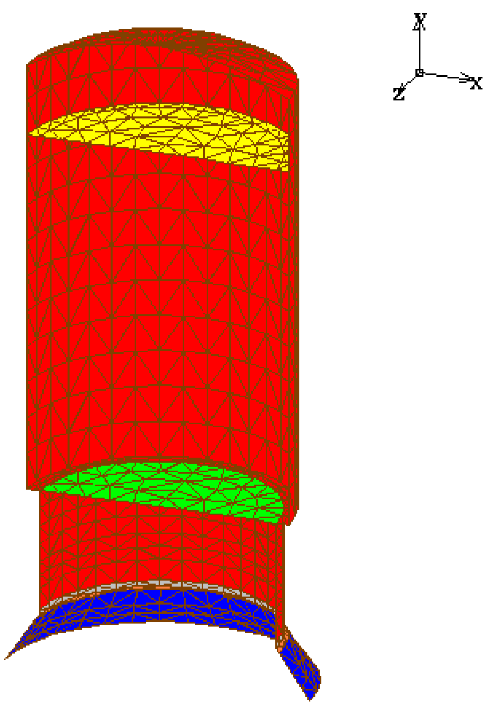

Figure-2

4. Initial conditions

The height of water in the reactor is 9.45m from the base. All the cells up to this height are initialized with initial conditions that correspond to water and cells above this height are initialized with conditions that correspond to steam.

5. Boundary conditions

Fluid:

a) Symmetry boundary condition on the symmetry plane ($z=z_{max}$)
b) Pressure outflow with 1.0e5 Pa at outlet (where rupture is postulated to occur)
c) Wall boundary conditions along all other boundaries.

Structure:

a) All the 6 degrees of freedom (3 translations and 3 rotations) are arrested at base of skirt
b) Along the symmetry plane ($z=z_{max}$), displacement along z-axis, rotation about x and y-axes arrested
c) At location of thermal stabilizers, radial displacement is constrained, i.e., displacement is permitted only along the y-axis (vertical).
d) A uniform pressure of 69.0331e5 Pa is assumed to act on the inside of the shroud throughout the analysis. This simulates the fluid inside the core shroud, which is not modeled.

6. Results and discussion

Simulation is carried out for 500ms and the results are presented in the form of time history as well as contour plots of various quantities in the fluid and the structure.

Fluid

Figure 3 shows some selected fluid nodes in the mesh where results are traced.

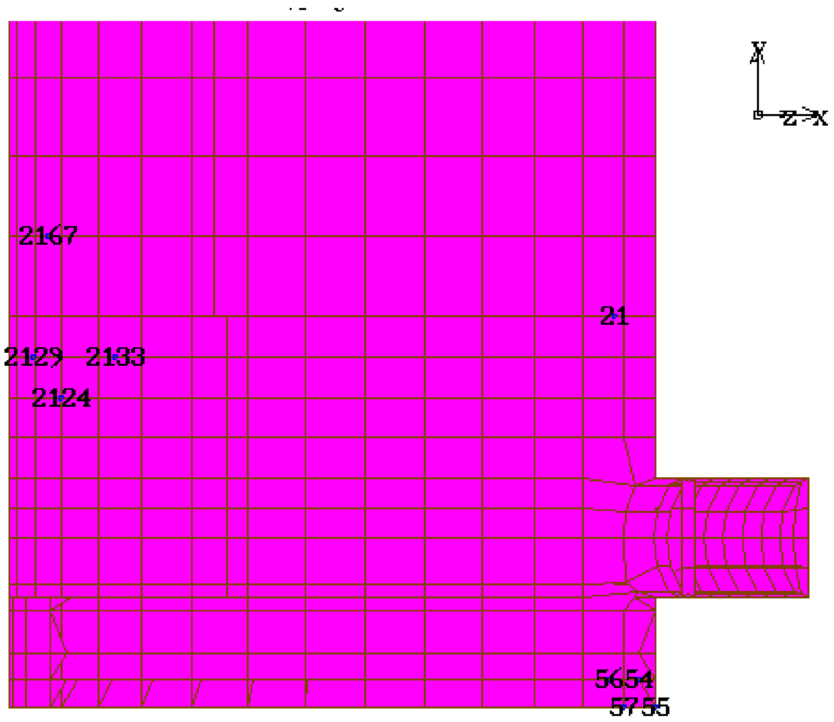

Figure-3

Figure 4 shows the variation of pressure with time at selected points P_57, P_2133, P_56, P_2129, P_55, P_2124, P_54, P_2167, P_21) along the axis of the nozzle. The maximum drop in pressure occurs at the node at the outlet and a steady value of about 18e5 Pa is reached. Magnitudes of steady pressures are higher as one moves along the axis towards the shroud.

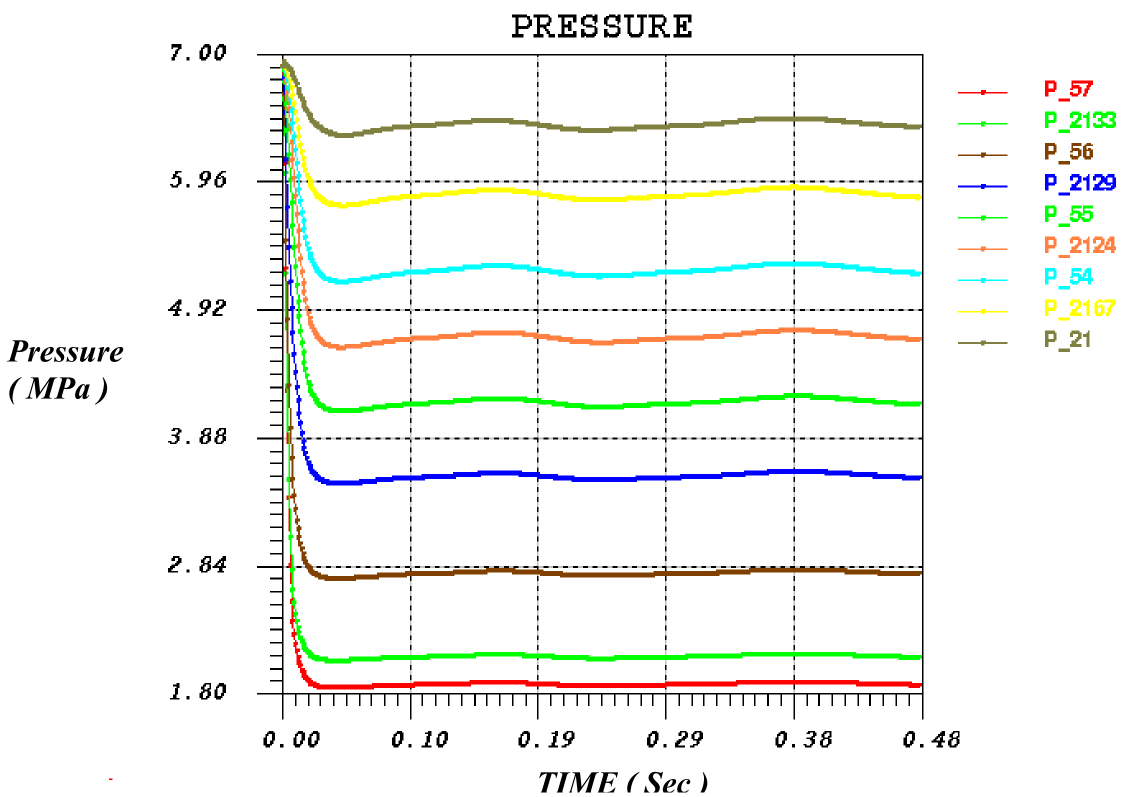

Figure-4

At the corresponding points, the velocities are seen to attain maximum values at the outlet and the magnitude of maximum velocity attained drops as one moves towards the inside. This is shown in Figure 5. The pressures and velocities attain the maximum values within the first 50ms.

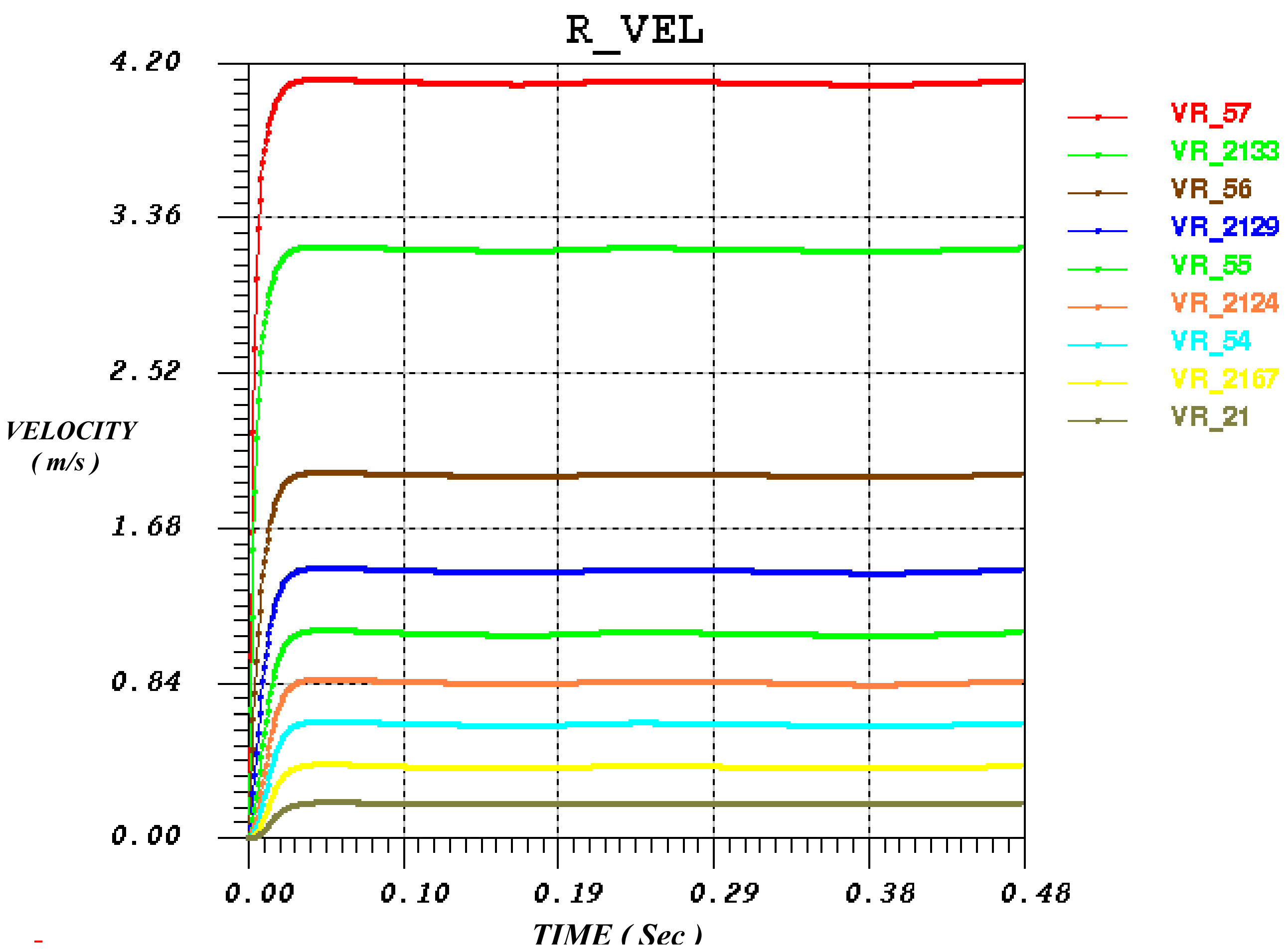

Figure-5

Figure 6 shows the variation of pressures at points on the symmetry plane lying on the core shroud at two points (one nearer to the pipe and the other being diametrically opposite).

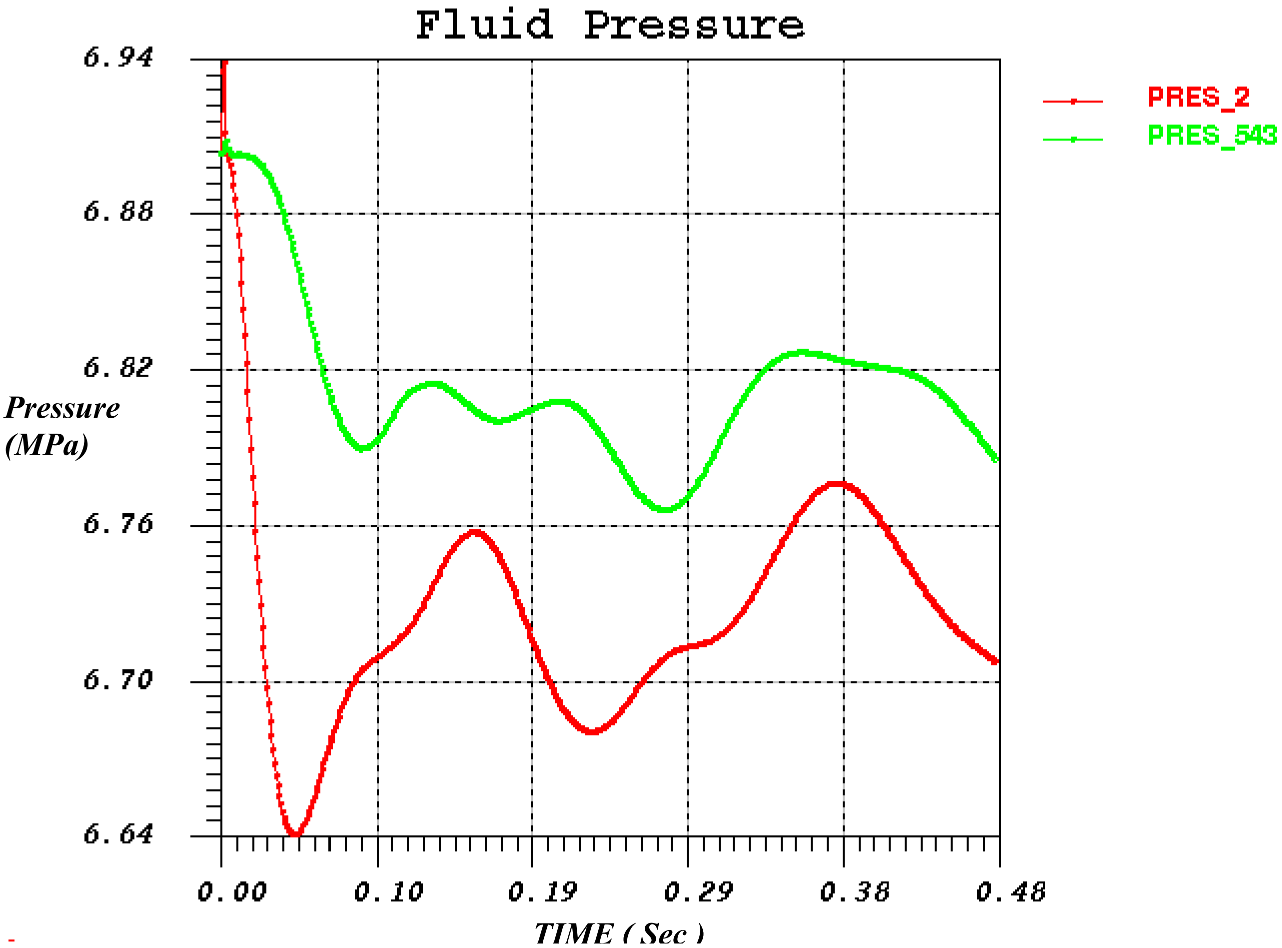

Figure-6

Figure 7 shows the contour plot of the pressures in the symmetry plane at the end of simulation. The maximum drop in pressure occurs in the nozzle region. A zoomed in view around the nozzle area (for better visualization, the nozzle portion is erased) is shown in Figure 8.

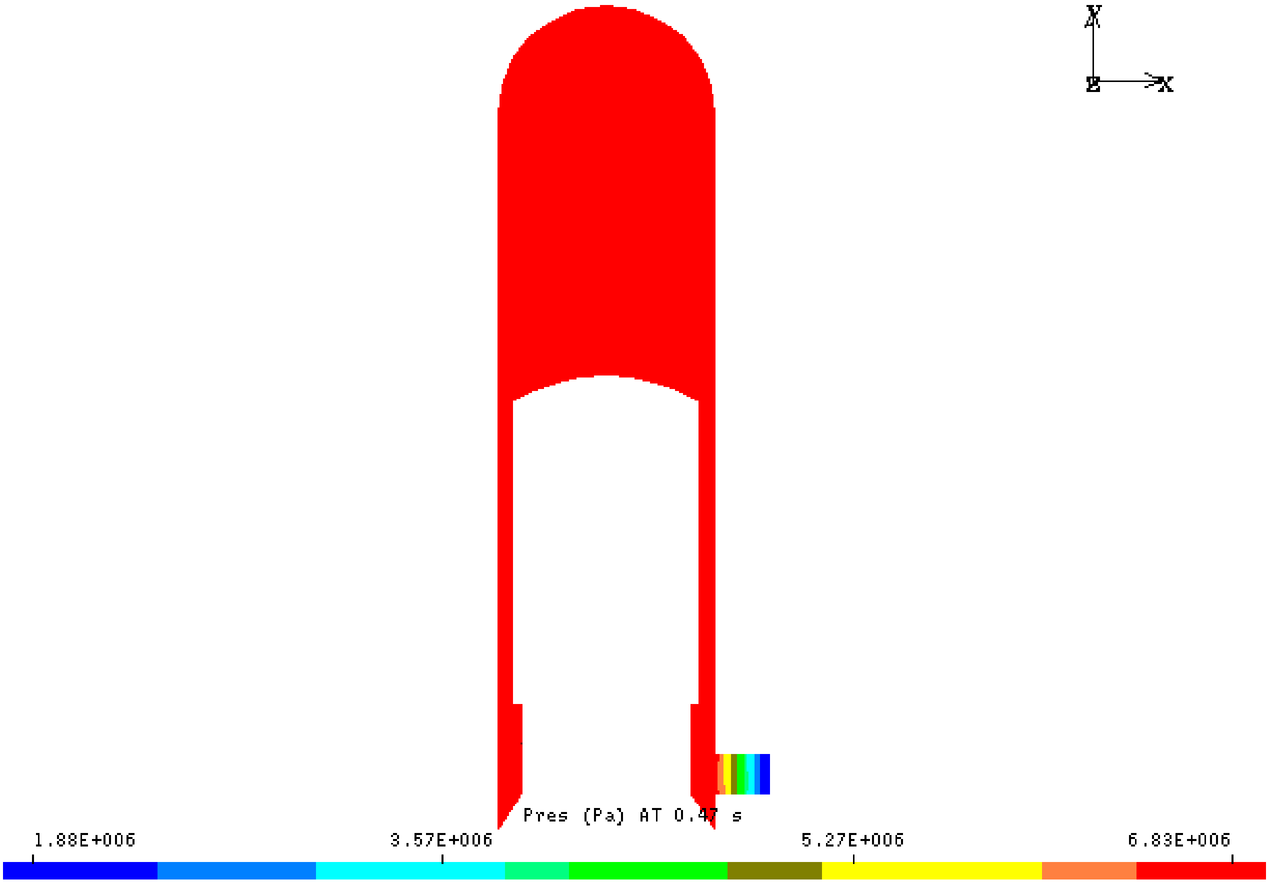

Figure-7

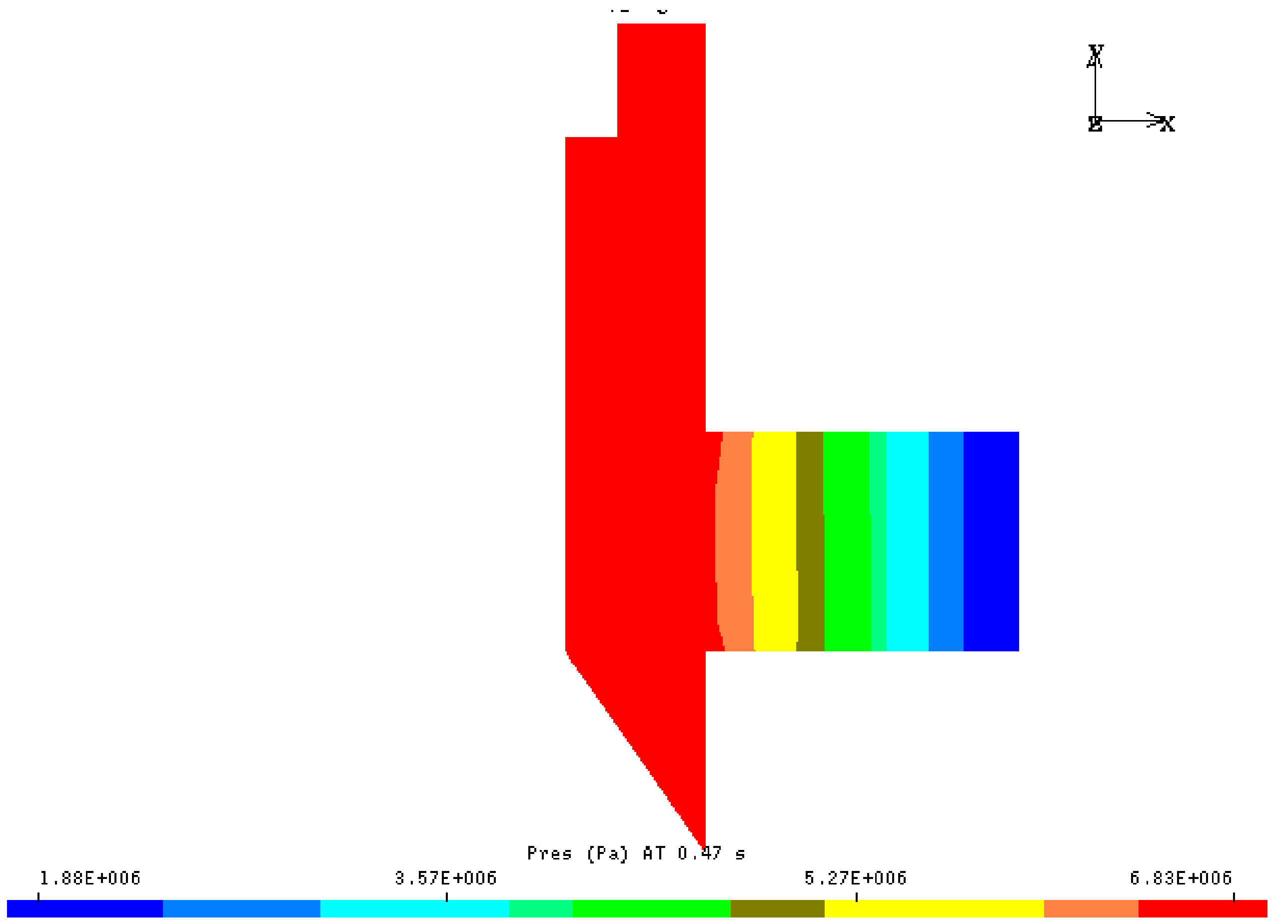

Figure-8

As this calculation is a two-phase calculation, the vapor fraction is of importance.

Figure 9 shows the contour plot of vapor fraction at 470ms.

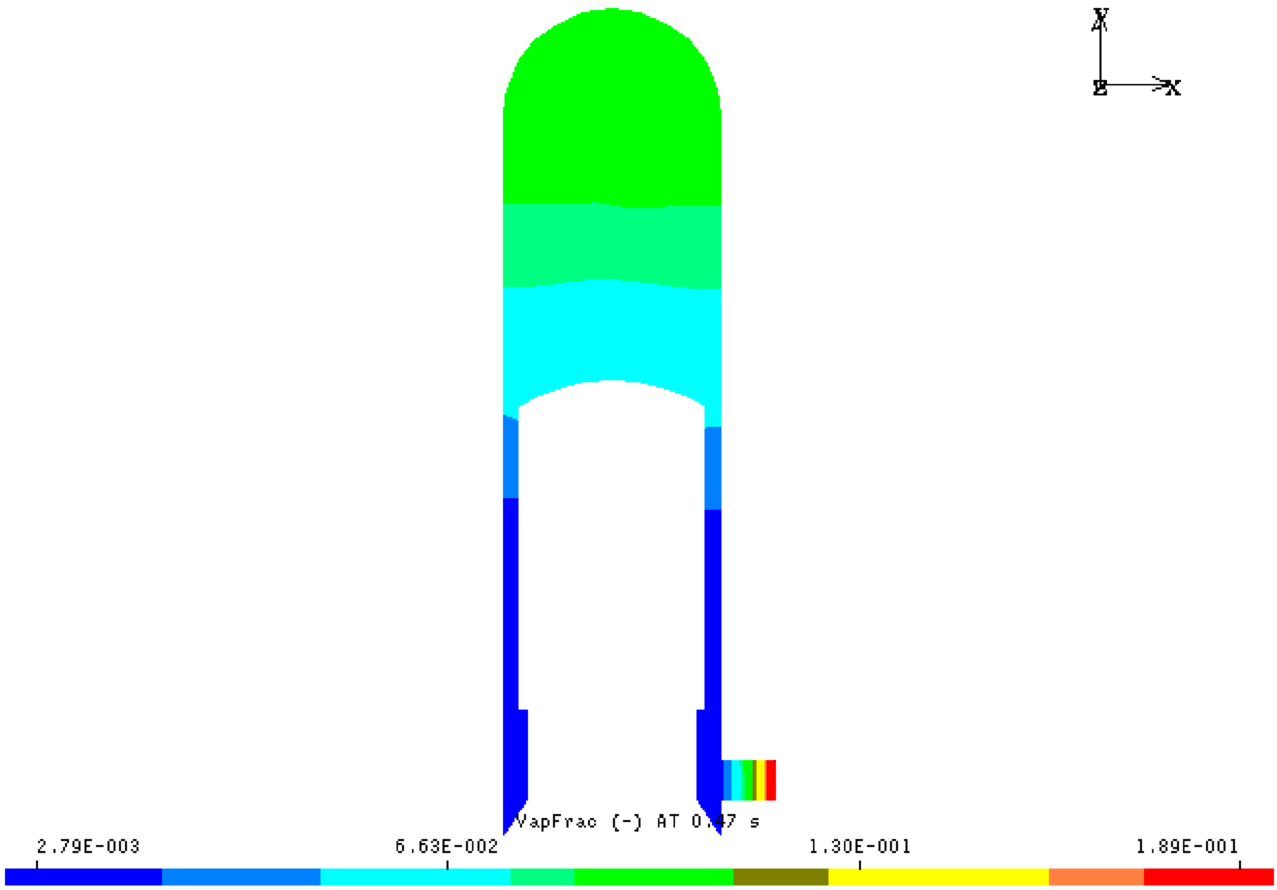

Figure-9

Figure 10 shows the velocity vectors at 470ms in the entire fluid region. Maximum velocity is observed at the outlet, as expected.

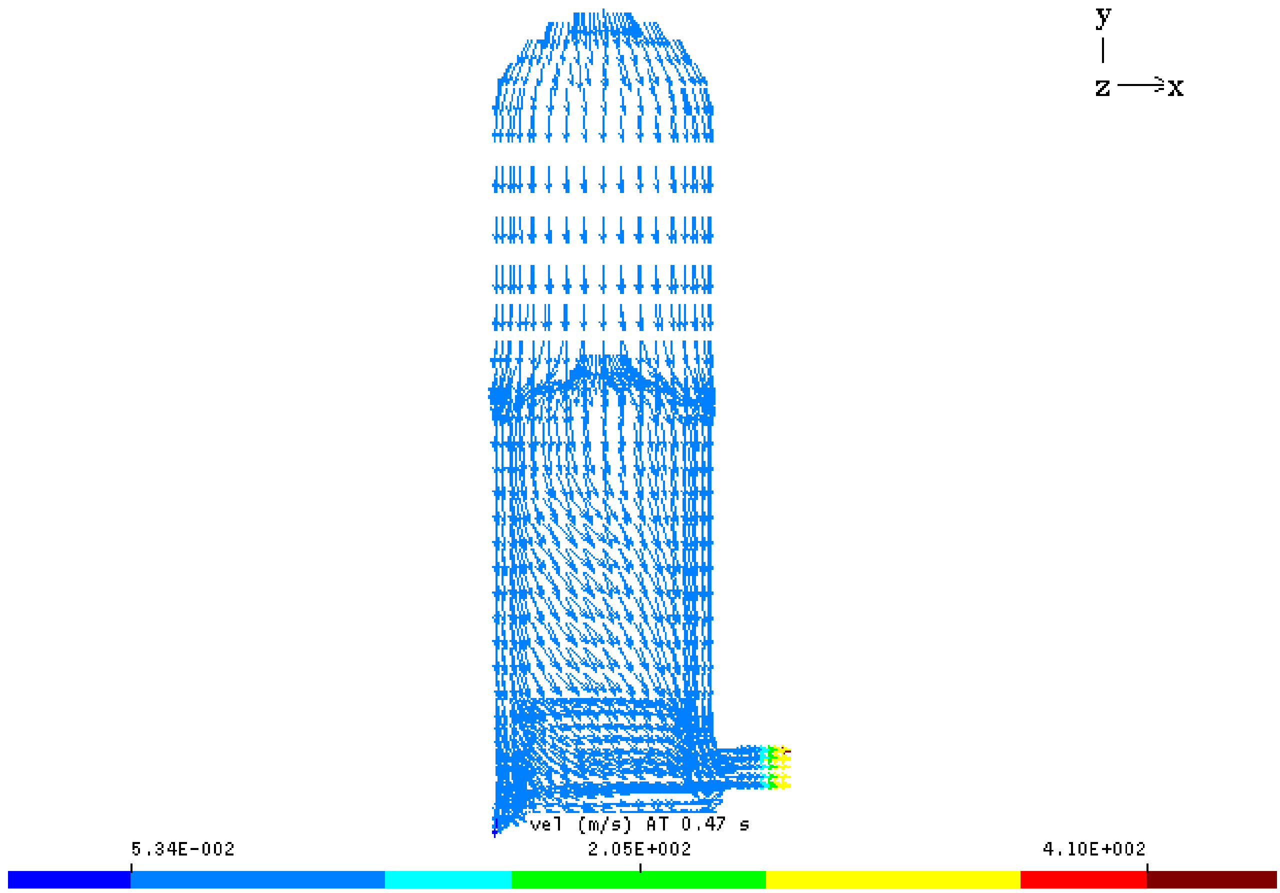

Figure-10

.

Structure

Figure 11 shows some selected structural nodes in the mesh where results are traced

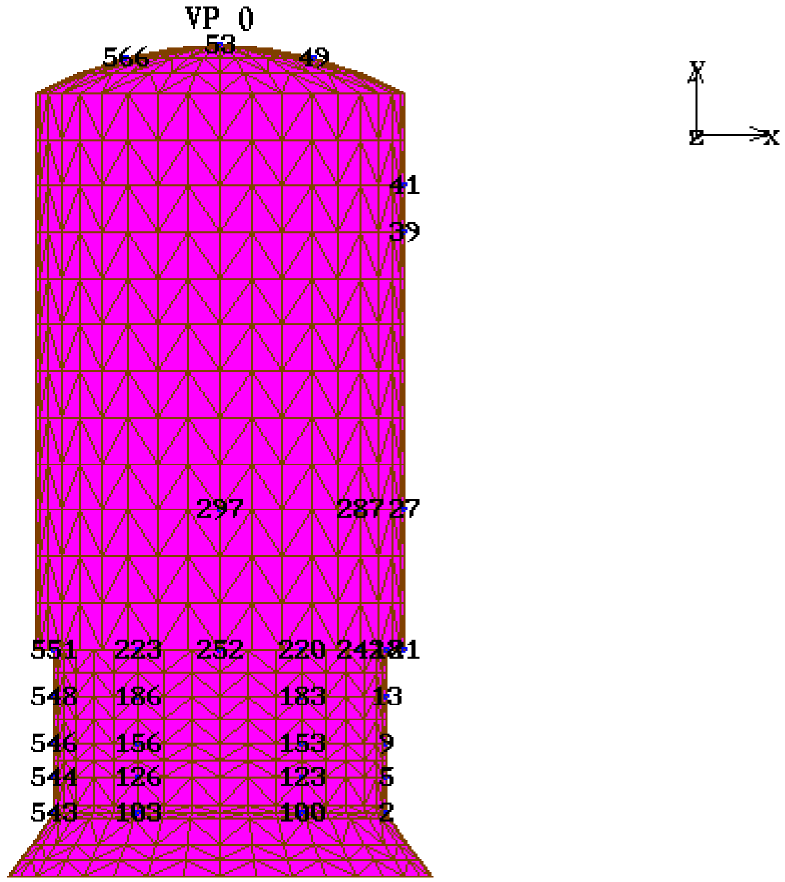

Figure-11

Figure 12 shows the variation of the u_x (displacement along x-axis) with time at points on the shroud around the axis of the nozzle.

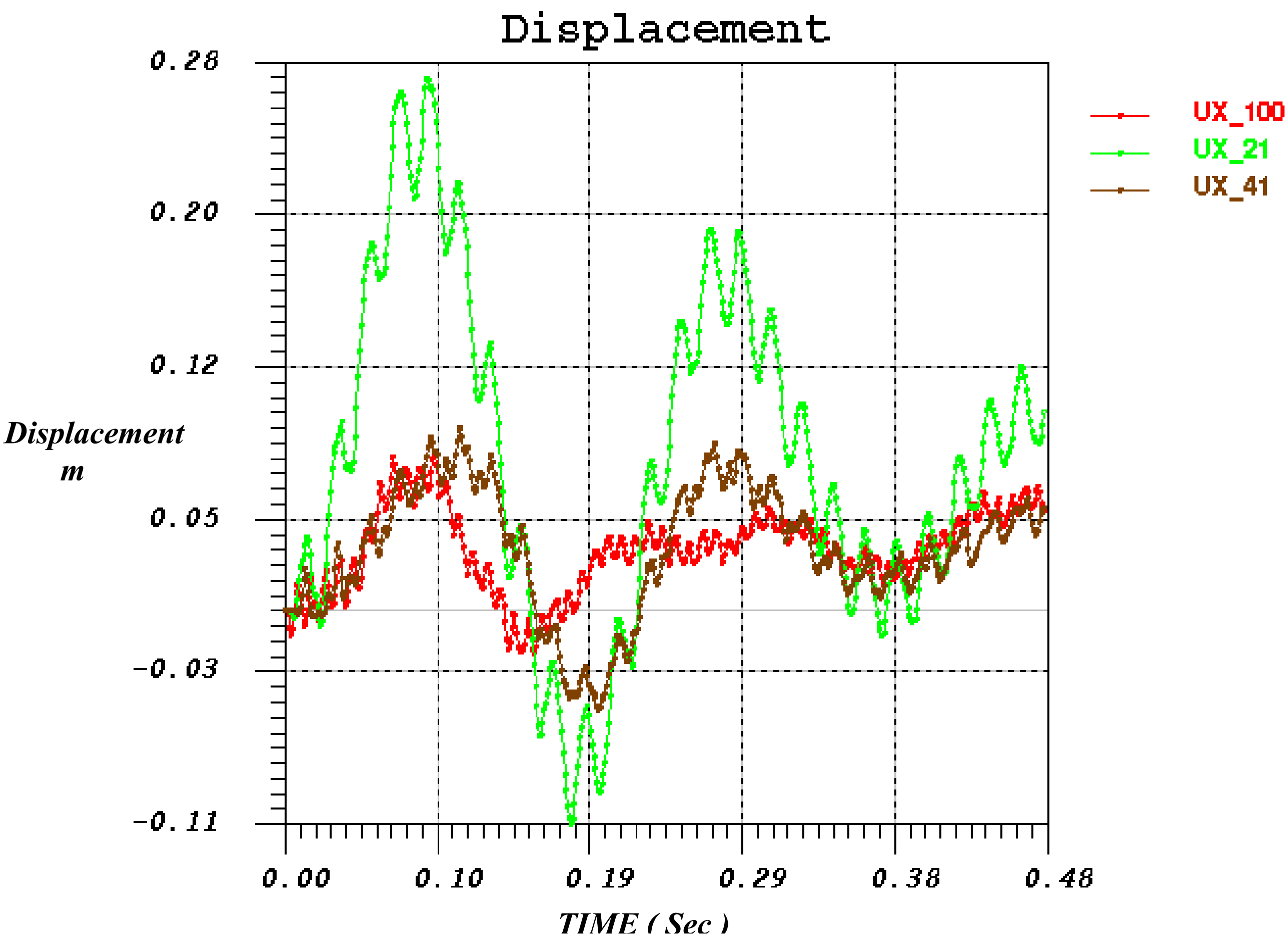

Figure-12

Figure 13 shows the superimposed views of the deformations of the shroud at 500ms. The bulging is due to the internal pressure of 69 bar. In this figure, the deformations have been magnified 500 times for the sake of clarity. Zoomed in views around the nozzle and at the top of the shroud are shown in Figure 14.

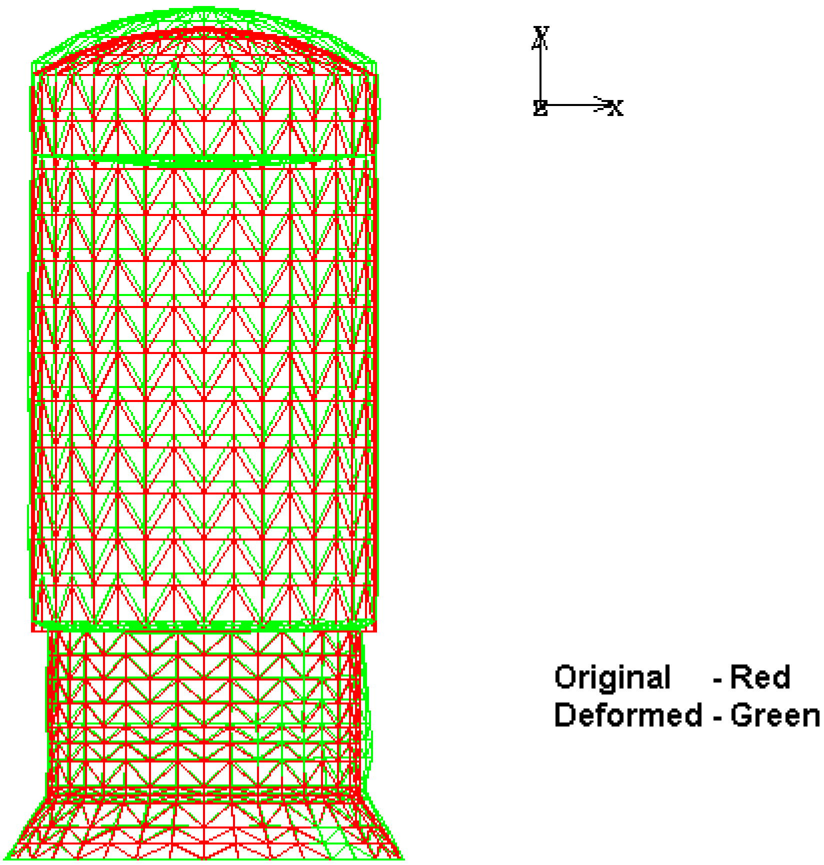

Figure-13

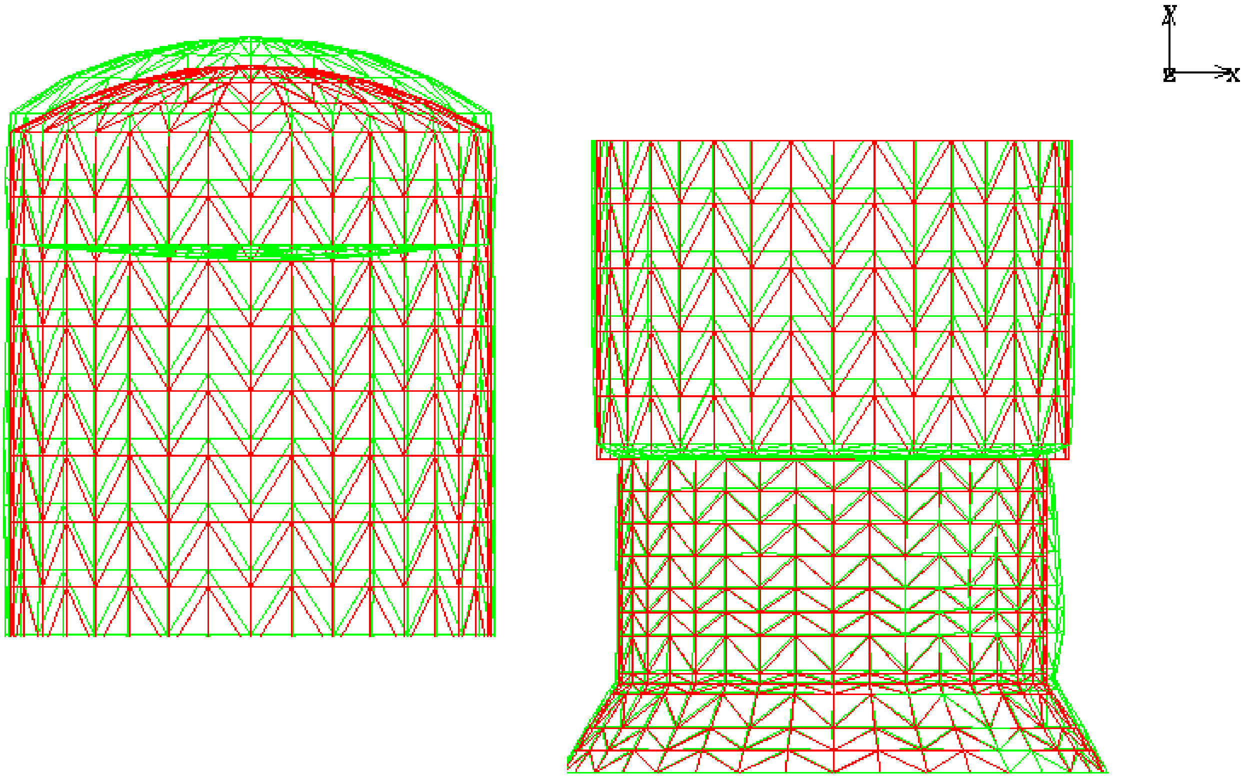

Figure-14

Figure 15 shows the contour plot of Von Mises stress in the shroud at the end of simulation. Peak value of 10MPa is observed near the shroud base.

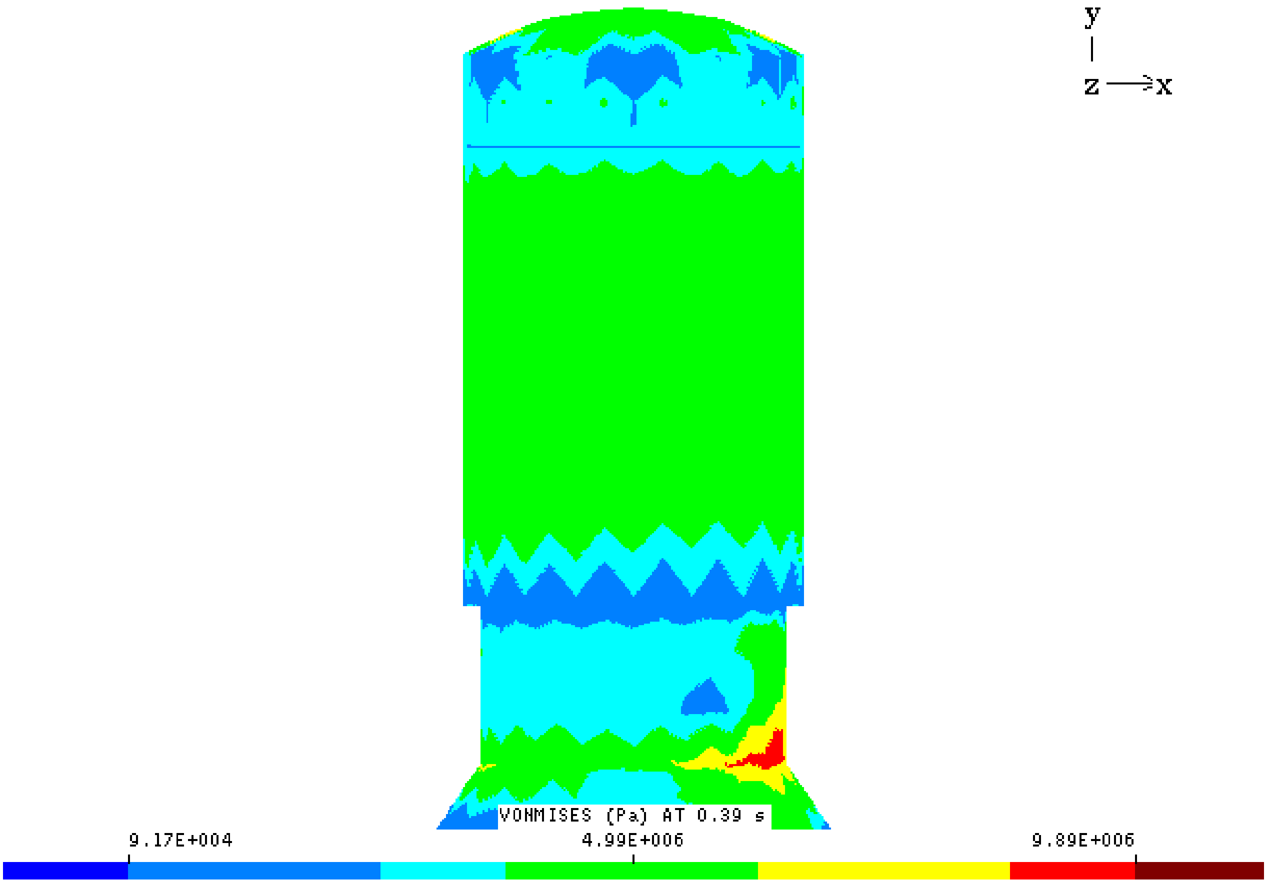

Figure-15

7. Conclusions

Results presented show the capability of the ***fluidyn*-FSI** for analyzing high-speed fluid-structure interaction phenomenon. The code also has the capability to model material nonlinearity (elasto-plastic analysis) of the structure. Though implicit scheme is available in the code, explicit time integration has been used for stress analysis since the present case is a high-speed phenomenon and small time steps are required. ***fluidyn*-FSI** has been found to be an effective tool for numerical simulation of rapidly varying fluid-structure interaction phenomenon (such as that discussed in this paper).Currently, the code does not include time-rate dependent material models.

References:

K.Morgan, "An Expansion Equation of State Subroutine," Computer Physics Communication, pp 64-68, vol.5, 1973.

M.S.Liou, B.van Leer and J.S.Shuen, "Splitting for inviscid fluxes for real gases," Journal of Computational Physics, pp 1-24, vol.87, 1990.

A.Jameson, W.Schmidt and E.Turkel, "Numerical solution of the Euler equations by Finite Volume methods using Runge-Kutta time stepping scheme," AIAA Paper 81-1259, June 1981.