42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit
9 - 12 July 2006, Sacramento, CA
AIAA-2006-4758

Simulations of Instabilities in Complex Valve and Feed Systems

Vineet Ahuja,* Ashvin Hosangadi † Jeremy Shipman‡ and Peter A. Cavallo*
Combustion Research and Flow Technology, Inc. (CRAFT Tech), Pipersville, PA 18947

Abstract

CFD analyses are playing an increasingly important role in identifying and characterizing flow induced instabilities in rocket engine test facilities and flight systems. In this paper, we analyze instability mechanisms that range from turbulent pressure fluctuations due to vortex shedding in structurally complex valve systems to flow resonance in plug cavities to large scale pressure fluctuations due to collapse of cavitation induced vapor clouds. Furthermore, we discuss simulations of transient behavior related to valve motion that can serve as guidelines for valve scheduling. Such predictions of valve response to varying flow conditions is of crucial importance to engine operation and testing.

I. Introduction

Rocket engine and component testing facilities usually have a network of control elements such as cavitating venturis, orifices, and control valves that provide test engineers with the flexibility of exerting control in terms of mass flow rates and pressures in the system. Introduction of such control elements in the test loop, however, introduces complexity in the flow path with the potential of flow induced instabilities due to vortex shedding, cavitation, turbulence and large scale flow separation. Such flow phenomena are usually accompanied by pressure fluctuations that can interact with test system components as well as test articles such as pumps and combustors. Alternately, the pressure fluctuations can excite certain structural modes and lead to system wide resonance, thereby compromising the safety of the test stands and leading to premature shutdown of tests. Furthermore, substantial thermal effects can couple with instability modes (such as seen with cavitation) when operating in cryogenic regimes that is typical for rocket engine testing.

From an analysis perspective, the instability mechanisms seen in testing facilities can be classified into three distinct classes: (a) hydrodynamic/fluid dynamic instabilities[1] that are predominantly attributed to fundamental flow physical mechanisms such as vortex shedding, turbulence, etc. largely associated with complex structural configurations such as valve housing, plug shapes, manifolds and bends in the piping system. (b) flow transients associated with valve timing and valve scheduling [2]. Such transients can play an important role especially during startup\shutdown and valve response can be critical for safe and reliable operation. Furthermore, estimation of flow response to valve timing can provide tremendous savings by reducing test runs needed to characterize the system. (c) multi-phase instabilities such as cavitation related instability mechanisms[3] that become especially important with cryogenic working fluids due to reduced liquid to vapor density ratios and strong evaporative cooling effects resulting in local temperature fluctuations that can couple with the primary cavitation instability.

Computational simulations can play an integral role in supporting testing and developmental activities by identifying and characterizing the above-mentioned instabilities. However, the diversity of flow regimes and the types of instabilities that can range from turbulent pressure fluctuations due to vortex shedding from bends and elbows to dynamic events due to valve scheduling, to large scale pressure fluctuations due to collapse of vapor cavities in venturis, place very stringent requirements on any computational framework. For example, the identification of dominant frequencies associated with flow instabilities in such systems requires high order numerics, advanced turbulence modeling capabilities, sophisticated grid topologies to resolve local physics in complex geometries, embedded models for unsteady cavitation, capture thermal effects in cryogenic fluids, and

* Senior Research Scientist, 6210 Keller's Church Road, Pipersville, PA 18947, Senior AIAA Member (for correspondence email: vineet@craft-tech.com).
† Principal Research Scientist, 6210 Keller's Church Road, Pipersville, PA 18947, AIAA Member.
‡ Research Scientist, 6210 Keller's Church Road, Pipersville, PA 18947, AIAA Member.

dynamic handling of valve motion. In this paper, we utilize our multi-element unstructured framework that comprises the CRUNCH CFD® code with advanced cavitation, turbulence modeling and grid motion capabilities to simulate the different types of instabilities seen in valve/feed systems.

In the past, we have tailored our multi-element unstructured CFD solver to carry out simulations of complex valve and feed system components [4,5]. Computational analyses evaluating flow coefficient curves and other performance metrics have been carried out for a variety of structurally complex valve systems such as the pressure regulator valve, split body valve and cavitating flow control elements. We have also extended our multi-element unstructured framework to include timing studies for transient valve operation by including valve movement and capturing the flow instabilities associated with it. The distinguishing feature of grid motion in internal flows such as valves from that seen in external aerodynamic type flows is that the topological and physical requirements of the grid can change substantially in localized regions of the flow domain as a result of the grid/valve motion. A valve operating at a 10% open setting has very different flow physics from one operating at 80% open position. Our approach accounts for this change in grid topology through a unique methodology of mesh movement to accommodate the translating valve.

Cavitation related instabilities are endemic in liquid rocket feed systems and test facilities. Since these flow regimes primarily consist of cryogenic fluids that operate in close proximity to the critical temperature, there are substantial thermal effects and property variations associated with such flows. Our cavitation models [6,7] account for the coupling of thermodynamic processes with the cavitation processes and have accurately captured such effects as leading edge pressure and temperature depression due to evaporative cooling, and frothy cavitation zones that are the hallmark of cavitated regions in cryogenic fluids. Furthermore, with the incorporation of an unsteady cavitation model we are able to predict amplitudes and frequencies of dynamic pressure loads and track bubble clouds sheared off cavities due to the interaction of reentrant jets in the cavity closure region.

In this paper, we provide an overview of the problems related to the different classes of instability mechanisms discussed above. In the next section we discuss two problems related to fluid dynamic instabilities in control valves. The first problem discusses a system instability associated with chatter in a pressure regulator valve. The second problem relates to valve sticking as a consequence of resonance in the valve plug cavity. In Section III, we discuss dynamic control valve simulations where valve response is a critical function of the operating conditions. In Section IV we show results for a cavitating instability for a step down orifice used in the test facility at NASA SSC.

II. SIMULATIONS OF PRESSURE REGULATOR CONTROL VALVE

CFD simulations utilizing CRUNCH CFD® were performed to carry out detailed analysis of a pressure regulator valve. The flowfields, associated with such a structurally complex valve, are replete with a rich variety of length and time scales and diverse flow regimes from low Mach number flow regions to supersonic flow in the seat region. Moreover, the multiple corners and edges have secondary flow structures and transient phenomena such as shedding vortex structures. Associated with these unsteady phenomena is a dominant chattering-like behavior that has been observed when the valve is operational under certain conditions. A multi-element grid comprising of tetrahedral, prismatic, hexahedral and pyramidal cells was constructed for the dome pressure regulator geometry.

Unsteady simulations were performed with the multi-element grid. An ideal gas flow was assumed in the simulations. Inflow boundary conditions based on an inlet total pressure of 4400 psi and a total temperature of 540 degrees R were specified at the inlet plane to the feed duct. At the outflow plane of the discharge duct a back pressure of 800 psi was maintained.

Figure 1(a) shows the time-averaged Mach number distribution along the plane of symmetry – it can be seen that the flow mildly accelerates as it transitions from the intake pipe to the feed ports leading into the valve housing. The flow turns supersonic as it rapidly accelerates through the valve seat region, which has a very narrow clearance. The flow emerges from the valve seat region like a jet into the upper housing where it turns into a feed channel that exits into the discharge pipe. The Mach number distribution also reveals flow expansion in both the discharge feed channel and the discharge duct. These observations are corroborated by the pressure distribution along the plane of symmetry in Figure 1(b). It is clear from the pressure distribution that most of the pressure losses accrue in the seat/throat region of the dome pressure regulator where the flow experiences a sharp acceleration. Furthermore, the pressure distribution also shows a pressure gradient in the upper housing indicating that flow is being forced tangentially towards the discharge duct. The pressure distribution on the poppet Figure 1(b) also shows a region of high pressure on the shaft just downstream of the throat region. This is primarily from flow mixing due to the fact that multiple feed channels located at different azimuthal locations introduce flow into the lower housing. Furthermore the stream traces reveal a complex flow pattern with strong recirculation in the upper housing indicating that the flow does not transition smoothly from the valve assembly into the discharge piping Figure 1(c).

The unsteady modeling effort was geared towards understanding an observed "chatter" of the poppet-shaft assembly during operation of the valve. A transient analysis with a fixed poppet setting was performed with a time step of 3x10-7 seconds and captured a dominant oscillation of the flow with a frequency of about 4 KHz. This frequency was detected in pressure fluctuations across seven probe points located on the poppet surface. Through flow visualization of the transient simulations, an axial mode was identified as the source of the instability, which manifested itself as a periodic pulsation of a jet like structure through the throat coupled with tangential modes in the discharge portion of the flow. Pressure histories were recorded at seven different points on the poppet close to the throat (Figure 2(a)). The recording tabs were mostly distributed along the base of the throat on the poppet. Three pairs of tabs were located on the base (on the periphery and midway on radial lines running from the shaft to the outer edges) aligned with the direction of the feeding channels leading into the inner housing. One of the taps was located on the shaft at the location where the steady state simulations indicated a localized region of high pressure. All seven probes show an almost identical periodic variation in pressure. The variations are strongest at the three points on the periphery of the base region (1, 2 and 3) that also coincides with the throat region of the valve. Fourier decomposition of the histories indicates a fundamental mode corresponding to a frequency of 4043 Hz (see Figure 2 (b,c)). Furthermore, Figure 2 (c) shows significant energy associated with the first harmonic. More importantly, the Fourier decomposition also revealed an active low frequency mode of approximately 250 Hz. This mode could be structurally significant since it could excite poppet vibration modes leading to significant noise and couple with structural modes leading to potential structural failure. Further structural analysis of this valve would be required to be determined if this frequency might pose a structural coupling risk.

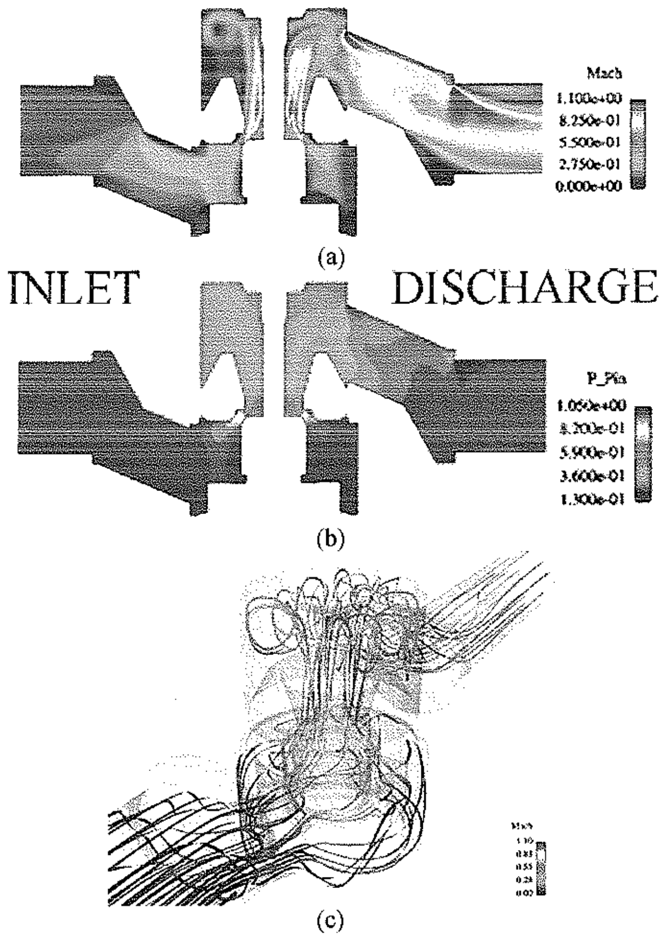


Figure 1. Plots of (*a*) streamlines through the valve housing colored with Mach number illustrating the large recirculation region, (*b*) pressure contours, and (*c*) Mach number contours on the symmetry plane.

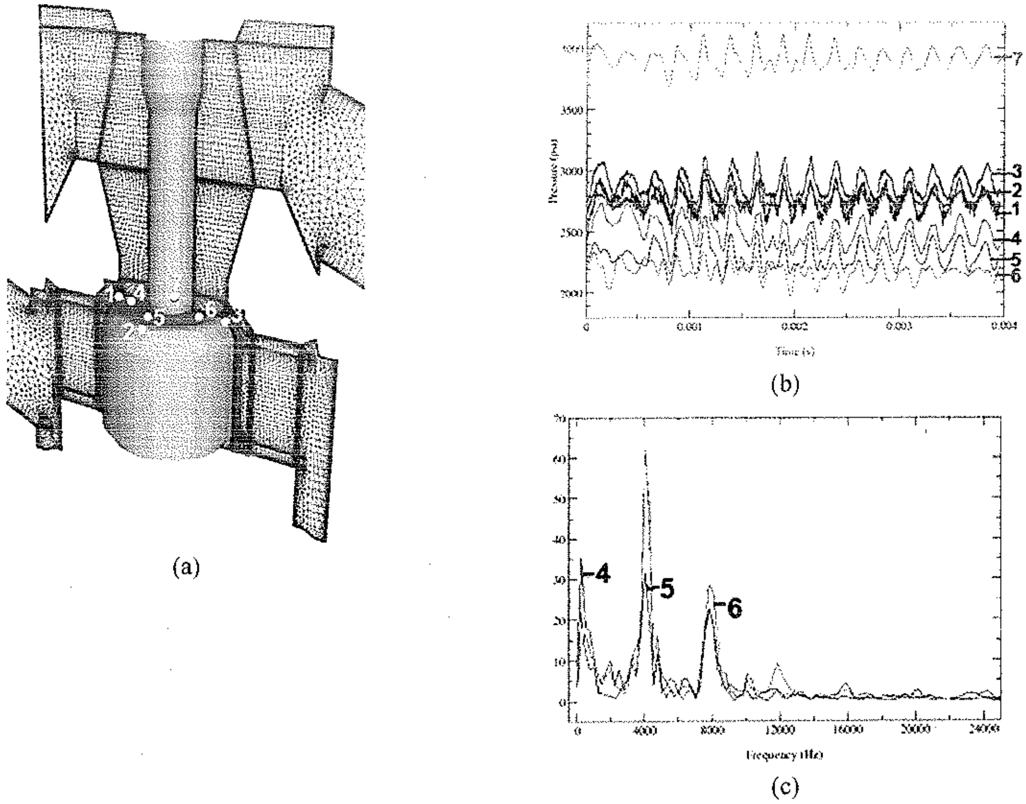


Figure 2. Plots of the transient solution showing (*a*) location of probe points on the poppet, (*b*) pressure history for the seven points, and (*c*) frequency spectrum indicating the dominant instability modes.

III. Unsteady Simulations of Valve Stall Problem

In this section, we discuss analysis work done on the 6-inch gaseous hydrogen valve (see Figure 3) currently in operation. This particular system is interesting, both, for the structural complexity in the valve geometry, particularly in the seat region and wide array of physical flow phenomena and flow regimes involved.

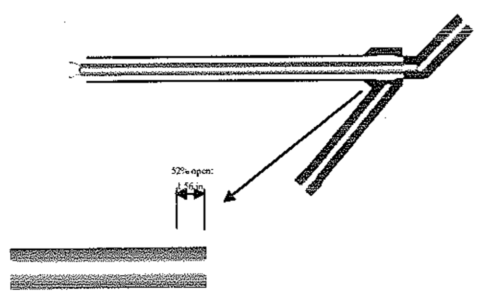


Figure 3. Schematic of the 6 inch Gaseous Hydrogen Valve.

Furthermore, at testing the valve stalled at a stroke setting close to 52%. CFD Analyses of the flow path at the constant valve setting position helped reveal details of the flow physics and the resultant integrated force balance on the plug. Simulations of the 6-inch gaseous hydrogen valve were performed with an inlet pressure of 5710 psi and inlet stagnation temperature of 530 R. Our simulations indicate an inlet Mach number of 0.11 with a peak Mach number close to 2.8 in the valve seat. The Mach number distribution is shown in Figure 4 indicating flow expansion with the sudden change in area in the valve seat region. The pressure distribution (shown in Figure 5) also shows the formation of alternate bands of expansion and compression waves. The instantaneous Mach number and pressure snapshots shown in Figures 4 and 5 are 1.5e-4 seconds apart and indicate that the cavity in the plug does not represent a benign flow region as is generally the case in such valve configurations. In this case, the cavity acts as a resonance tube with pressure waves from the cavity periodically interacting with the flow in the seat region. This in turn, breaks the symmetry of the flow near the seat region and leads to variable loads on the plug (Figure 6).

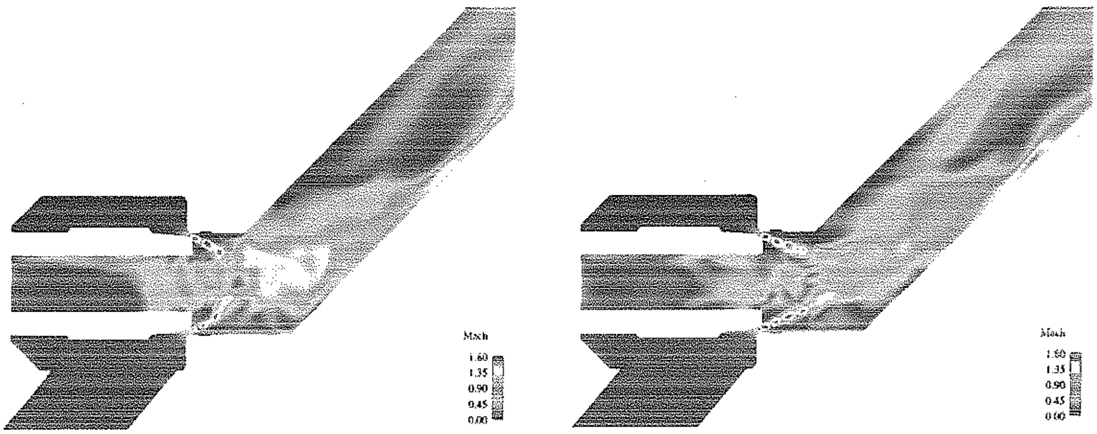


Figure 4. Instantaneous Mach Number Distribution in 6-inch gaseous Hydrogen Valve.

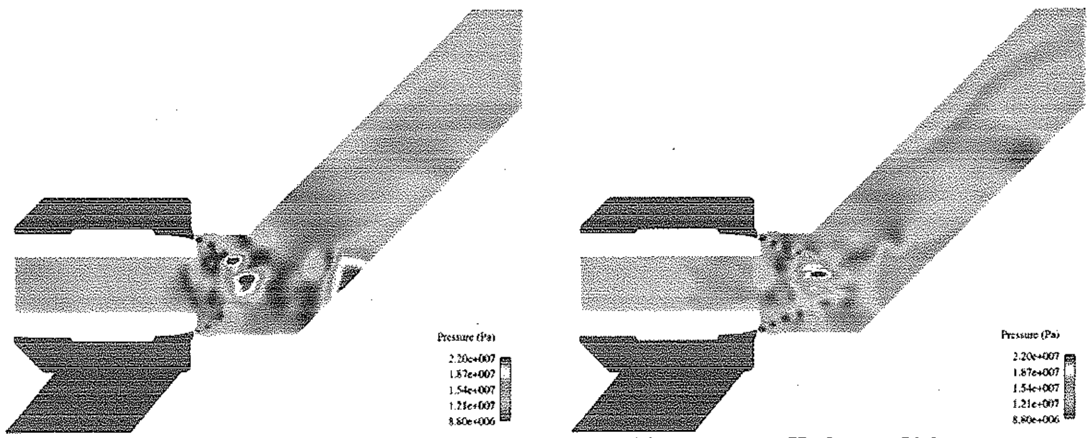


Figure 5. Instantaneous Pressure Distribution in 6-inch gaseous Hydrogen Valve.

Numerical pressure probes were inserted in different parts of the flow domain and the pressure traces were recorded. A Fourier analyses of the traces revealed that globally the fundamental tone of 257 Hz associated with the cavity is seen to play a significant role in exciting the flow in the valve housing. However, there are other significant high-frequency tones associated with the dynamics of the jet through the valve seat that are excited in the valve housing. The unsteadiness in the valve housing is seen to dynamically influence the load on the plug. The time history of the axial force on the plug is plotted in Figure 6. Each count on the x-axis is 10 time iterations or $2x10^{-6}$ seconds and portions above the 12,000 lbf on the y-axis are colored indicating that the force on the valve plug has exceeded the force rating on the actuator, which would cause the valve to stall. The figure shows dramatic fluctuation in the force of more than 3000 lbs and explains the valve stall seen during testing.

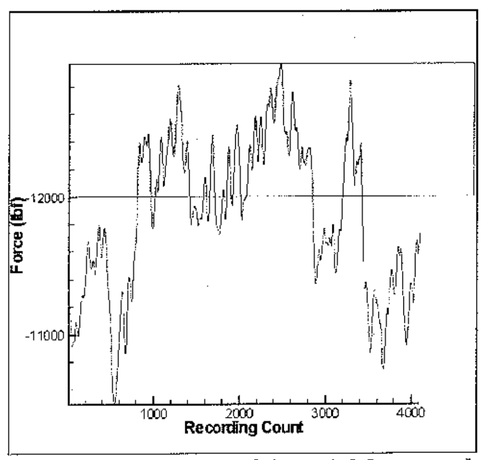
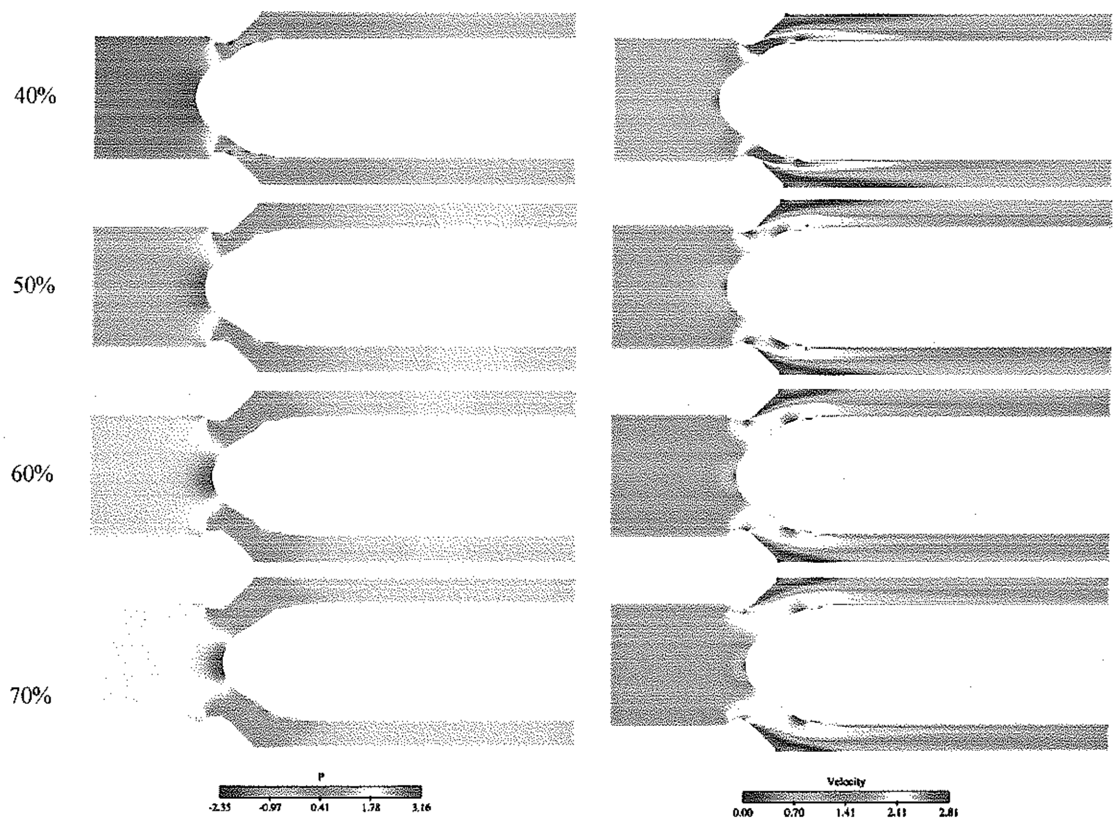


Figure 6. History of the axial force on the plug (1 unit on the x-axis is 10 time iterations or 2×10^{-6} seconds).

IV. Transient Studies Related to Valve Timing

In the previous sections, we discussed unsteady simulations of valves at constant plug positions. In this section we present moving valve simulations where the rate of change of mass flow varies with the speed of operation (plug speed). Our strategy for valve motion entails making a library of grids for every 10% of change in plug setting and use a generalized mesh motion solver to translate between successive recorded plug setting meshes in the library. Our scheme models the computational mesh as an elastic solid subject to the equations of elasticity. Any movement of the mesh boundary results in the propagation of a stress through the domain, producing motion of the interior grid points [8]. The current formulation is edge-based and therefore applicable to arbitrary unstructured meshes composed of any combination of tetrahedra, pyramids, prisms, or hexahedra. Details of the grid movement procedure such as prescription of the valve displacement curves, mesh movement scheme, creation of mesh libraries are not dealt here and the reader is referred to an associated publication [8].

In this section we discuss the axisymmetric split-body valve simulations using our mesh movement approach. To accurately model the moving valve problem, the upstream boundary condition had to be modified for incorporation of a variable mass inflow boundary condition. Therefore a stagnation pressure boundary condition is imposed at the inflow. Furthermore, we track critical parameters such as the valve flow coefficient, upstream and downstream pressures and mass flow rates to ensure that the simulation closely mimics the valve behavior. We operate the valve by opening it from a plug position of 40% to a position of 80%. The valve is opened at a constant velocity of 10 inches/sec and an initial mass flow rate of 60lbm/sec. Figure 7 shows a series of instantaneous pressure and velocity distributions in approximately 10% increments. As expected the pressure gradient in the seat relieves itself as the valve progressively opens. Interestingly, the sequence of velocity distributions show the change in flow patterns in the seat region – the intensity of the jet that emanates from the inflow duct decreases as the valve opens. As a consequence, the jet spreads out further (in the radial direction) in the discharge duct and the reattachment length of the corner re-circulation zone in the discharge duct shortens.



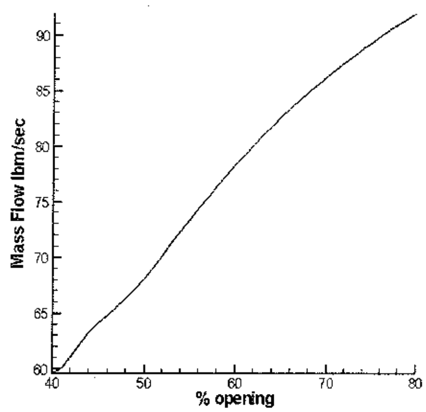
a) Pressure contours　　b) Velocity contours

Figure 7. Split-Body Valve results at 10% intervals using the library approach.

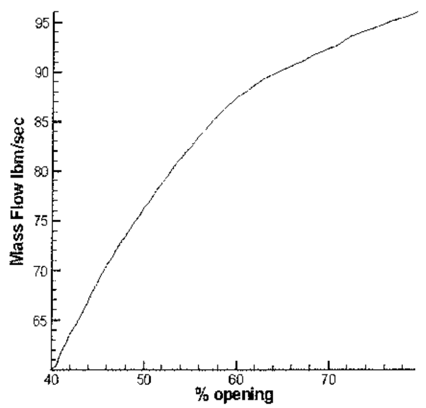
The mass flow variation as a function of plug opening for two different plug speeds is plotted in Figure 8(a-b). Figure 8(a) shows the change in mass flow for a plug speed of 10 inches/sec whereas Figure 8(b) corresponds to a slower plug speed of 1 inch/sec. The mass flow variation in Figure 8(a) shows a linear curve in contrast to the mass flow variation in Figure 8(b) where the increase in mass flow closely approximates the variation in plug profile which in turn directly affects the area increase in the seat region. Although both the curves in Figure 8 show a monotonic increase in mass flow, Figure 8(b) indicates that the rate of mass flow increases substantially faster in the case of slower plug velocity. For example, at 45% open, the mass flow through the valve is already at 70 lbs/sec in the case of plug moving at 1 inch/sec whereas the faster plug traveling at 10 inches/sec indicates a mass flow rate of 65 lbs/sec at the same plug setting.

The difference in valve performance can be better understood by comparing the flow coefficient curves (C_v) for the two cases. Figure 9 shows a comparison of the variation of C_v with valve opening for the two cases along with C_v variation for quasi-steady calculations performed at discrete plug settings that are 10% apart.

It is seen here that the dynamic C_v curve for the 1 inch/sec plug velocity simulation closely matches the data for the quasi-steady simulations, whereas the C_v curve for the faster plug velocity lags the quasi-steady data indicating the mass flow rate does not respond fast enough to valve opening at higher plug speeds. The effect of valve speed on system response is critical in valve scheduling for efficient engine operation and controlling deleterious fluctuations from impacting component performance.



a) Mass flow variation with valve opening at plug speed of 10 inches/sec



b) Mass flow variation with valve opening at plug speed of 1 inch/sec

Figure 8. Performance of Split-Body Valve during opening.

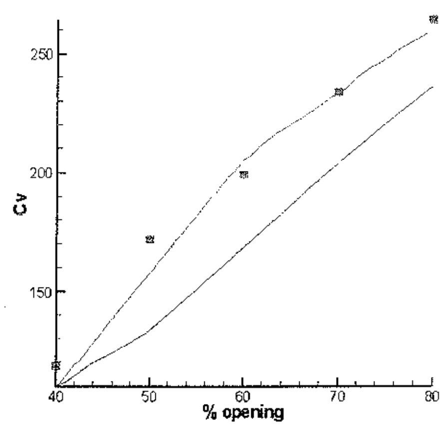


Figure 9. Performance of Split-Body Valve showing flow coefficient as function of valve opening.

V. Cavitating Instabilities in an Orifice

An orifice is routinely used in testing facilities to step down the pressure and in this section we analyze the cavitating instability that sets up in the testing facility(NASA SSC) due to a pump discharge orifice. The flow rate of liquid hydrogen through the orifice is 130 lbs/sec at an operating temperature at 21.7 K. The inlet pipe has a diameter of 6 inches and the orifice throat diameter is 3.26 inches (with an inlet radius of 0.75 inches). A back pressure corresponding to 65 psia is maintained on the outlet end of the configuration and the corresponding vapor pressure of liquid hydrogen at the operating temperature is 21.755 psia. A snapshot of the instantaneous axial velocity distribution is shown in Figure 10(a), which indicates the formation of a primary jet as flow accelerates to negotiate the orifice. It should be noted that this jet is representative of a very high Reynolds Number flow since cryogens such as liquid hydrogen typically have very low viscosity. Figure 10 (b) shows the vorticity associated with the fringes of this jet and Figure 10 (c) depicts a snapshot of the pressure distribution. Vorticity production at the lip of the orifice leads to unsteady shedding and the periodic formation of pockets of low pressure. When the pressure in these pockets falls below the vapor pressure cavitation sets in leading to the formation of a vapor cloud that grows and convects downstream(Figure 11).

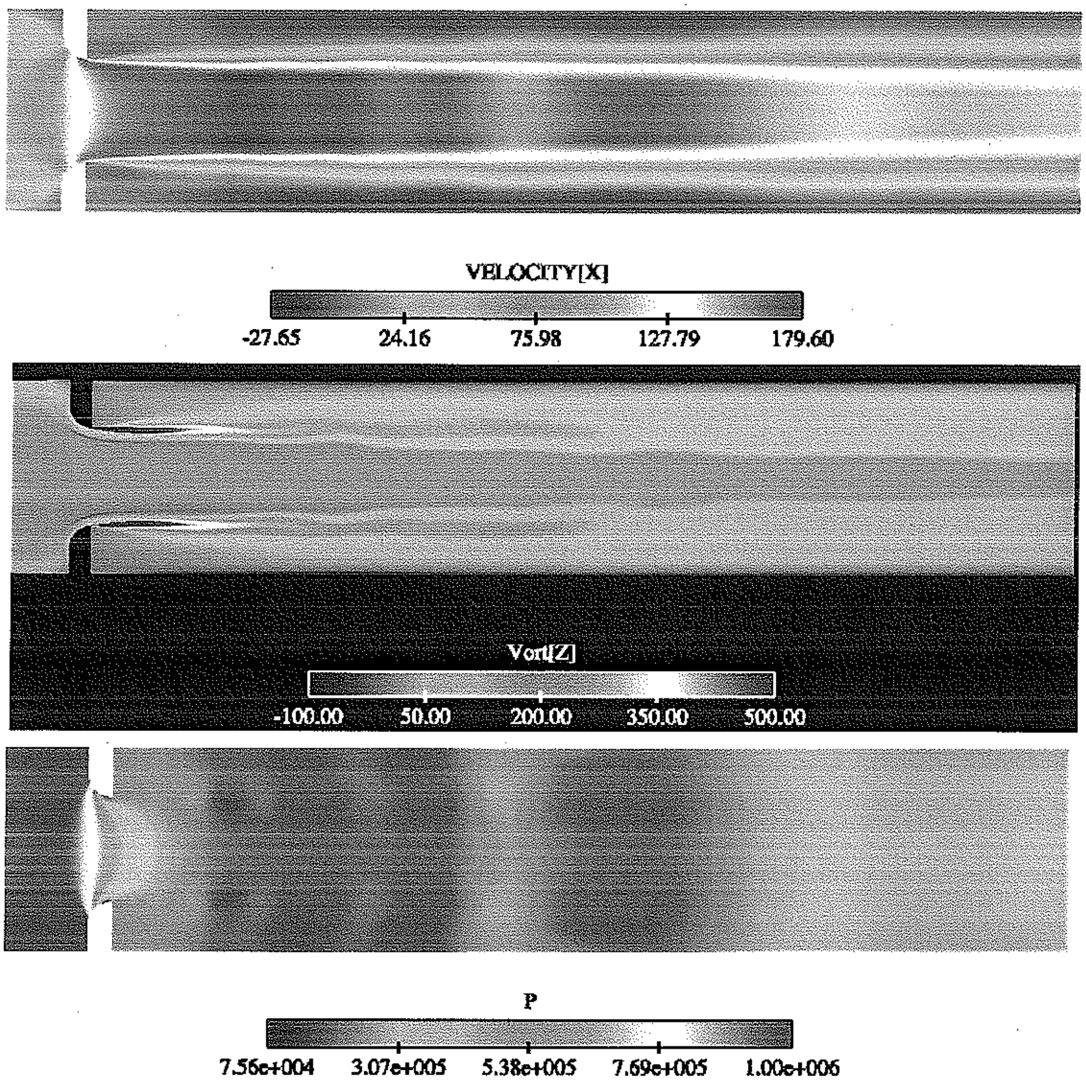


Figure 10. Instantaneous snapshots of (a) axial velocity distribution (b) vorticity distribution and (c) pressure distribution for flow through an orifice.

The shedding of the vapor clouds is a fairly periodic phenomena as evidenced by the instantaneous void fraction distribution seen in Figure 11. Furthermore, these clouds are formed in regions of high vorticity in the shear layer associated with the primary jet. The growth and development of the vapor clouds be better explained by the sequence of instantaneous void fraction contour plots in a blown up region around the orifice (Figure 12). Here, we see a small cavity formed at the lip of the orifice with a well-defined gas-liquid bubbly wake. As the wake encounters a region of low pressure downstream of the orifice, it leads to sudden expansion and growth of the vapor cloud. Pressure traces of two numerical probes located 5 inches and 10 inches downstream of the orifice along the piping wall show large scale pressure fluctuations attributed to the highly dynamic processes of formation and collapse of the vapor clouds. The frequency spectrum of the pressure oscillations indicates a cavitation shedding frequency of 80 Hz is excited in both probes with significant energy in a higher overtone of 320 Hz.

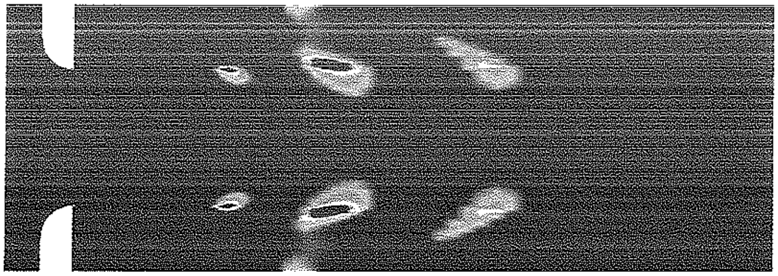


Figure 11. Instantaneous snapshot of vapor void fraction showing shedding of vapor clouds

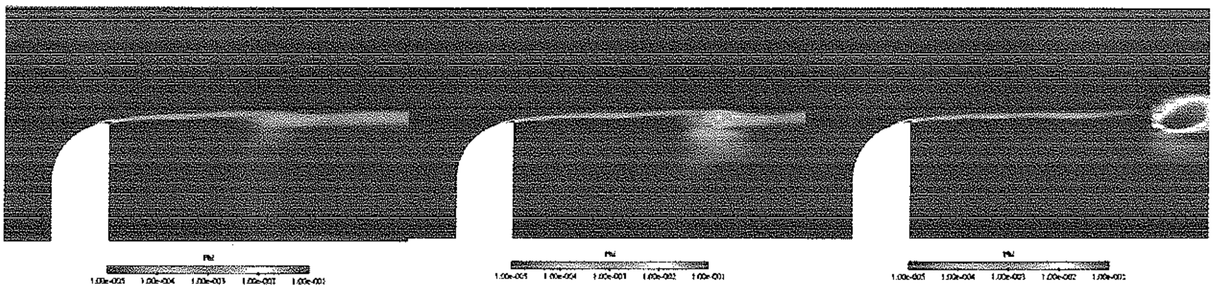


Figure 12. Instantaneous snapshot of vapor void fraction showing formation of cavitation clouds in a blown up region near the orifice.

VI. Conclusion

High fidelity unsteady analyses of valve and feed system components are carried out with our multi-element unstructured numerical framework. The framework is comprehensive and includes sub-models for cavitation, cryogenic fluid handling, as well as generalized mesh motion. Problems from three different classes of instabilities were analyzed and ranged from hydrodynamic instabilities due to structural complexities and transients due to valve motion to cavitation related instabilities in cryogenic regimes with significant thermodynamic effects. In the first sub-class of flow induced instabilities we analyzed a system instability related to chatter in a pressure regulator valve and a valve stall problem in a cryogenic control valve, followed by a valve scheduling problem of timing studies related to the split body valve. The final sub-class of problems included a cavitating instability for a step down orifice used in the E-facilities at NASA SSC.

Unsteady simulations of the pressure regulator valve were performed on the multi-element grid. The simulations revealed a rich variety of flow phenomena over a multitude of length scales. This included a jet like penetration of the flow through the throat region, secondary flow patterns in the feed channels and corner regions of the valve housing, and significant flow expansion in the buffer channels feeding the discharge duct and in the discharge duct. Furthermore, flow from the throat region of the valve impinges on the upper wall of the valve assembly and forms a dominant re-circulation pattern before discharging through the exhaust ducting. Our transient analyses were performed with a fixed poppet setting and captured a dominant chatter with a 4 KHz frequency. This frequency was detected across seven different probe points located at different parts of the poppet. Through flow visualization of the simulations we were able to identify an axial mode as the source of the instability. The instability was generated due to tangential discharge of flow from the far end of the upper housing that was periodically cutting off the axial flow through the throat region. A closer investigation of the frequency content of the instability modes revealed most of the energy is associated with the fundamental frequency 4043 Hz and its first harmonic. Furthermore, there was significant energy associated with a low frequency mode (around 250 Hz). This was an important finding since this mode can potentially induce structural vibrations.

Numerical simulations of the cryogenic control valve revealed the physics behind the valve stall related directly to flow resonance in the plug cavity. Large flow gradients that set up in the seat region were accurately captured and the unsteady interaction between the between flow in the plug cavity and the jet structure in the seat region creates unsteady loads that were shown to be greater than the actuator force rating on the plug leading to valve stall.

Unsteady simulations of valve timing for the split body valve were carried out utilizing a novel moving grid approach that permits the use of variable grid topology by maintaining a library of high-fidelity meshes and dynamic mesh motion via mesh deformation between the adjacent library meshes. The split body valve simulation undergoes valve motion from a 40% open plug position to a 80%open position through a succession of four library grids. The solutions evolve smoothly through the simulation process and the valve coefficient at lower plug speeds compares favorably with quasi-steady simulation data. Furthermore, we show that at higher plug speeds the valve does not respond fast enough to plug motion. Dynamic flow coefficient curves such as the ones obtained during our transient simulations are very valuable for testing purposes because it can help in improving valve scheduling on test stands and predicting unsteady phenomena such as valve stall.

Unsteady cavitating simulations were also performed for a step down orifice used in test facilities. The simulations were performed for cryogenic fluid (liquid hydrogen) in flow regimes where thermodynamic effects are significant. Simulations of the orifice show a periodic shedding of vapor clouds from the lip of the orifice and the formation of these clouds is directly linked to vorticity production from the orifice. Furthermore, fundamental frequencies associated with the cavitation shedding process were captured as part of the simulation process and could provide valuable insight to testing facilities.

Acknowledgments

We acknowledge funding for this work through a NASA SBIR, Contract Numbers NNS04AA08C and NNS05AA36C funded by NASA Stennis Space Center. Mr.Robert Fields is the technical monitor and his input along with those of Dr Harry Ryan and Dr Russell Daines is gratefully acknowledged.

References.

[1] Shipman, J., Hosangadi, A., and Ahuja, V., "Unsteady Analyses of Valve Systems in Rocket Engine Testing Environments," Paper No. AIAA-2004-3663, 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Fort Lauderdale, FL, July 11-14, 2004.

[2] Cavallo, P.A., Hosangadi, A., and Ahuja, V., "Transient Simulations of Valve Motion in Cryogenic Systems", AIAA Paper No. 2005-5152, 35th Fluid Dynamics Conference and Exhibit, Toronto, Canada, June 2005.

[3] Ahuja, V., Hosangadi, A. and Shipman, J., "Computational Analyses Of Cavitating Control Elements In Cryogenic Environments," Paper No. HT-FED2004-56377, The 2004 Joint ASME-JSME Fluids Engineering Summer Conference, Charlotte, N.C., July 11-15, 2004.

[4] Ahuja, V., Hosangadi, A. and Shipman, J., "Multi-Element Unstructured Analyses of Complex Valve Systems," 52nd JPM/1st LPS Meeting, Las Vegas, NV, May 10-14, 2004.

[5] Ahuja, V., Hosangadi, A., Shipman, J., Daines, R., and Woods, J., "Multi-Element Unstructured Analyses of Complex Valve Systems, *Journal of Fluids Engineering*, to be published July 2006.

[6] Hosangadi, A. and Ahuja, V., "Numerical Study Of Cavitation In Cryogenic Fluids," *Journal of Fluids Engineering*, Vol. 127, pp. 267-281, March 2005.

[7] Hosangadi, A., and Ahuja, V., "A New Unsteady Model for Dense, Cloud Cavitation in Cryogenic Fluids," Paper No. AIAA-2005-5347, 17th Computational Fluids Dynamic Conference, Toronto, Ontario, CA, Jun. 6-9, 2005.

[8] Cavallo, P.A., Sinha, N., and Feldman, G.M., "Parallel Unstructured Mesh Adaptation for Transient Moving Body and Aeropropulsive Applications", AIAA Paper 2004-1057, 42nd Aerospace Sciences Meeting and Exhibit, Reno, NV, January 5-8, 2004.