2010 NDIA GROUND VEHICLE SYSTEMS ENGINEERING AND TECHNOLOGY SYMPOSIUM
MODELING & SIMULATION, TESTING AND VALIDATION (MSTV) MINI-SYMPOSIUM
AUGUST 17-19 DEARBORN, MICHIGAN

Analysis Comparison between CFD and FEA of an Idealized Concept V- Hull Floor Configuration in Two Dimensions

Dr. Bijan Khatib-Shahidi
Global Technology Associates

Rob E. Smith
TARDEC
Warren, MI

Abstract:

An idealized concept of a v-hull vehicle design for blast analysis has been studied in two different commercial software packages and results are compared to one another. The two software packages are different in nature: one code is an Eulerian Computational Fluid Dynamics (CFD) Finite Volume Solver while the other code is a Lagrangian Finite Element Analysis (FEA) Solver with the ability to couple structures to fluids through a special technique called Arbitrary Lagrangian Eulerian (ALE).

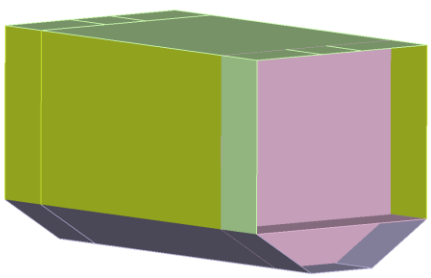
The simulation models in this paper have been set up for both CFD and FEA using a commercial pre-processing tool to study the effect of an idealized blast on the vehicle configuration: A pressure blast charge has been placed under the center of the vehicle at the symmetry line. The charge is composed of a prescribed pressure and a temperature pulse in a medium with the properties of air. In the CFD solver, an explicit unsteady solver has been chosen for analysis purposes. This was done because this type of solver is also available in the explicit non-linear finite element code.

This paper will compare the analysis results for the two different software packages paying particular attention to mesh density and the Courant number. The metrics to be assessed include the supersonic wave propagation, Mach number, velocity, pressure fluctuations and distribution that propogate from the explosive device toward the vehicle. Additionally, the floor and roof line forces are captured as another metric.

Introduction:

A fully detailed vehicle CAE / CFD model can be computationally expensive for analysis. This is especially true if one wants to consider several alternatives in the early stages of design when many parameters are still fluid in the design space. It is also prohibitive to investigate certain unknown or less known physics when one considers a 'detailed' representative of a design. In this paper, the authors suggest one way to get around the aforementioned bottle necks at early stages of design development is to consider a 2-D cut analysis instead of a 3-D representation of it. This approach would shed light to phenomenological aspect of a certain design consideration or physics phenomena associated with a particular design. The illustration below shows that one can cut a section either horizontally or vertically depending on what interest one would have for analysis purposes.

In this paper, the study has been conducted for a vertical cut illustrated in pink.



The 2-D approach is an ideal case for concept design consideration or the underlying physics of a certain phenomena that can be looked into from either an Eulerian approach or that of a Lagrangian one. For a fluid in motion, the Eulerian approach is commonly employed for a numerical study. This fluid centric approach is often known in the literature as Computational Fluid Dynamics or CFD modeling. The most common numerical method utilized in CFD modeling is that of the Finite Volume Method. Another, but less common, method is the classical Finite Element Analysis approach referred in the literature as FEA. It is still Eulerian, but the formulation follows the Galerkin Finite Element Approach that is common in the structural based codes. This paper will utilize the commercial software codes StarCCM+ (version 5.02) as the CFD solver and LS-Dyna (version 971) as the FEA solver.

It is particularly useful to understand the possibilities and the limits of the different modeling methods. It is also interesting to realize, that for certain instances there is a need to combine the two techniques. An example of this need is when there is an interaction between the fluids and structures. One can investigate a combination of analyses where a structure can be subjected to a fluid loading which causes a structural deformation. This type of analysis can be cost prohibitive if it is done in a full 3-D, while one can conduct repetitive 2-D analysis to 'perfect' the certain elements of the design before jumping to a full 3-D analysis.

In short, this paper attempts to explore the following; and it is hoped that one can learn from the presented technique for other scenarios based on the users individual needs and creativity. The following is what can be realized, but not limited to from this type of procedure:

A) Utilization of a 2-D model instead of a complex 3-D analysis at the early stages of design so the analyst can impact the design.
B) Investigation of some unknown physics that is not easily understood in a complex 3-D multi variable geometric and material environment.
C) Understanding differences between the modeling in a CFD Finite Volume vs. a Finite Element Analysis applied to the Eulerian Fluid medium.
D) The learning point for the 'FEA engineer' / 'CFD engineer' to cross the divide from one discipline to another.

There can be multiple objectives defined for an analysis project, depending on the focus of the researcher. For example, the design engineer desires to know what the optimum v-hull angle is for blast mitigation. The analyst/physicist would inquire about the location, size, and/or type of an explosive charge and the resulting effect on the v-hull structure. Finally, a research analyst would want to investigate the accuracy of the computational analysis capability given by different fundamental approaches.

The authors in this paper explore the problem from the differing numerical approaches. The first approach being the finite volume representation of a fluid interacting with a surface through the classical CFD approach. The second approach is the finite element representation of 2-D structure with a boundary condition of blast pressure, i.e. the classical FEA. The resultant forces on the structure, as well as, the pressure distribution and velocities are compared against one another in this study.

It is important to note that the example geometry is not derived from any production vehicle. Furthermore, that the geometry used for modeling purposes does not represent or suggest, in any way, a 'preferred' section design for a v-hull configuration. Thus, the example is merely used herein for illustration purposes as described above.

A final clarification is the use of a 2-D phenomenological model is not meant to replace a 3-D model for actual design analysis. The 2-D model used herein is for computational simplification only. A 2-D model approach should NOT be substituted for the more physically representative 3-D model in production design cases.

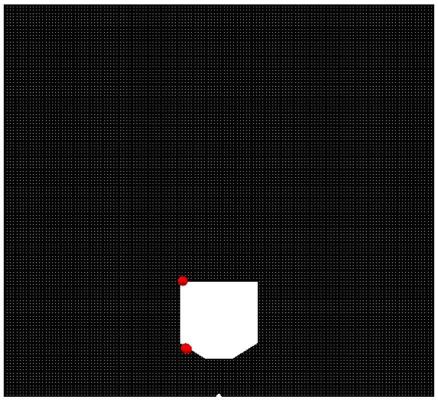
Model:

A 2-D cut of a structural model was made from a section of an idealized vehicle. This cut can be made at any area of interest in the geometry. It can be cut vertically or horizontally, depending on the objective of the analysis. In this study, a section has been taken out similar to that of the beam analysis. Think of a vehicle as a long slender beam. Then, consider the internal shear and moments in the beam section that is subjected to an external load. The load, in this case, is that of the blast load and is applied as a pressure impulse to the air. For that reason, the section cut has to embody the air or (soil) of interest as well.

In the finite volume CFD approach, the structure is fixed and represented as a rigid body, while the fluid surrounding applies pressure to it. For comparative purposes, the FEA model has also been fixed. This will enable a one-to-one comparison between the forces generated by the pressure of the fluid to that of the structure. This was necessary to build confidence and establish the basis for the section force comparisons between the two methods as much as possible from the user perspective. It is noted that the internal workings of the algorithms are completely different between the two methodologies. External users, such as the authors, do not have any control over convergence criteria but the input to the models will be kept identical as much as possible.

Mesh Considerations:

Three different sets of meshes were created for both solvers having differing levels of mesh density. The meshes are primarily composed of quad type of elements. The first level is a coarse mesh that was built to validate the viability of the comparisons and is about 45,000 elements, as shown in Figure 1.

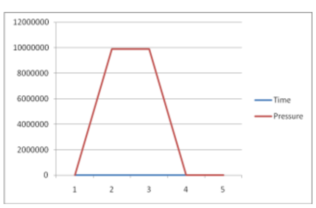


Coarse Mesh of 45,000 elements
Figure (1)

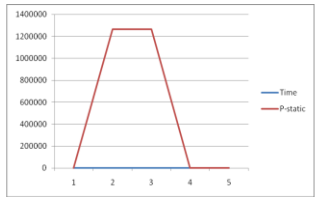
Once the technique for the comparisons was established, a second and third level mesh was created to assess: 1) the consistency of the two comparisons and 2) the dependence of the force levels on the mesh density. The medium level mesh density was created with around 145,000 elements. Finally, a fine mesh with approximately 1,000,000 elements was considered.

Analysis:

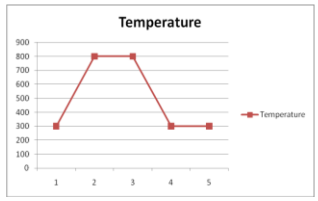
To begin with, a CFD model was created using StarCCM+. A 45,000 element mesh was generated for the 2-D case. The model is subjected to a supersonic pressure impulse with a value of 99 Bar for a duration of 10 milliseconds. The supersonic static pressure of one bar is needed for the analysis as well. For the same time frame, the temperature has been considered to rise from an initial 300K to 800K. Boundary conditions used for the simulation are shown in Figure 2, 3, and 4.



Pressure Pulse
Figure (2)



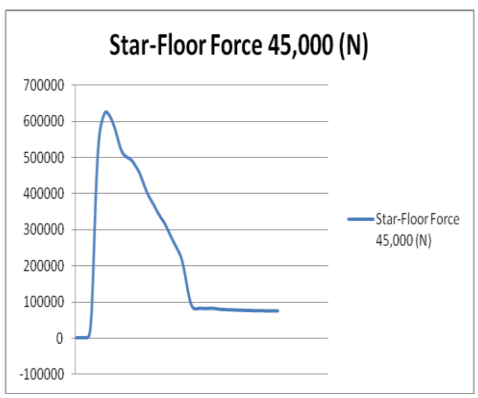
Static Supersonic Pressure Pulse
Figure (3)



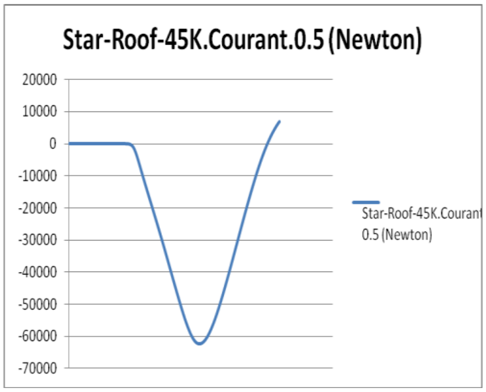
Temperature Pulse
Figure (4)

In this case, the explicit-unsteady numerical solver with no turbulence was used. The choice of no turbulence option was required, in order to compare with the analysis done using LS-Dyna, which does not have a turbulence model. It was assumed that for a short duration of blast time, in this case 20 milliseconds of total analysis time, the influence of turbulence would be insignificant.

The result of the first CFD analysis is shown in Figures 5 and 6, where the reaction forces from the flow are computed on the floor and roof respectively.



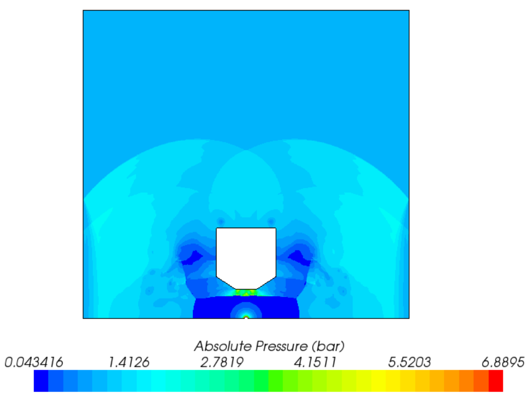
Floor Force by StarCCM+
Figure (5)



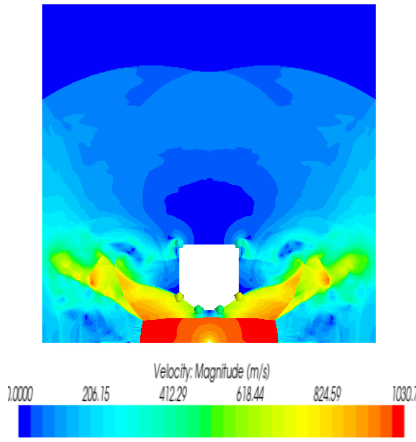
Roof Force by StarCCM+
Figure (6)

These forces are considered to be the 'lift' or 'down force' that is normally of interest to an aerodynamicists- if the body that is considered herein was an 'airfoil'.

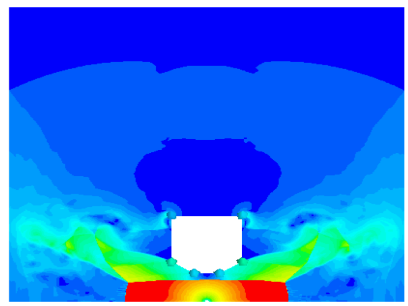
The analysis shows the force on the floor as always positive, while the force on the roof indicates a negative value. The pressure, temperature, Mach number and velocity calculations are shown in Figures 7 through 10 respectively.



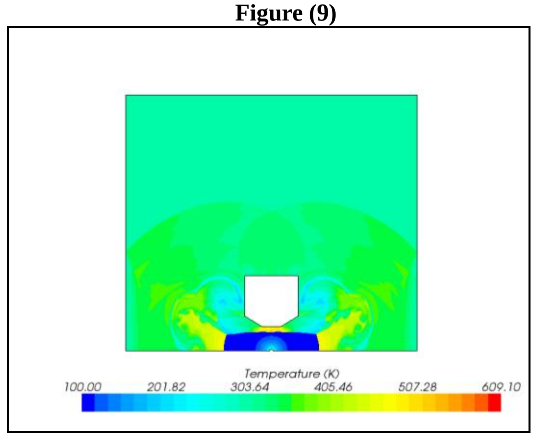
Pressure Distribution StarCCM+
Figure (7)



Velocity Distribution StarCCM+
Figure (8)



Mach No. StarCCM+
Figure (9)



Temperature Distribution StarCCM+
Figure (10)

In LS-Dyna, the same size mesh is considered for the analysis with the same boundary conditions as StarCCM+, but the load was transformed from total pressure to relative volume using the ideal gas law (see below):

V = Initial density / density.

And

Density = Pressure / (Cp-Cv) * T

Where

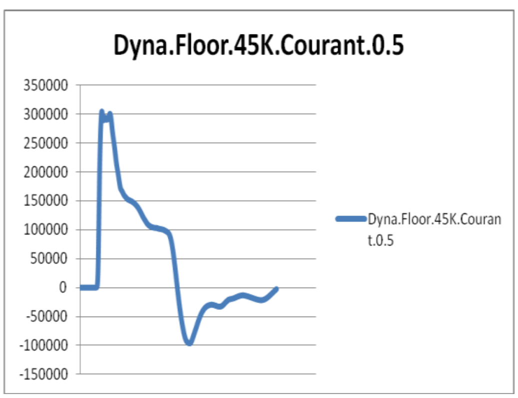
Cp = Specific heat under constant pressure
Cv = Specific heat under constant Volume
V = relative Volume
T = Temperature

For Air, at 1 ms to 10 ms, the value is:

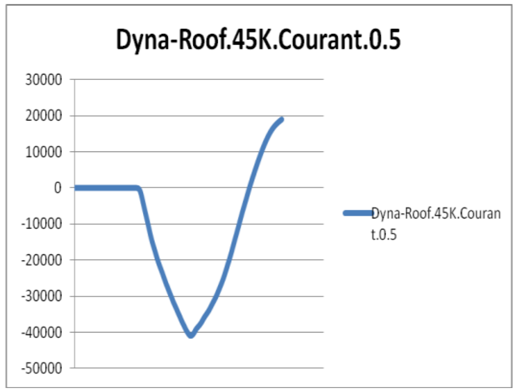
Density = 100 Bar / 286 * 800 Kelvin

And the rest of the time, it will be initial density. This is due to the way LS-Dyna takes the material pressure of a fluid into account as '*EOS', which is the equation of state. One can represent different types of pressure equations as a 'material characteristic'. In this case, the EOS was chosen to be Ideal Gas Law type.

Results for floor and the roof line forces derived from the LS-Dyna analysis are shown in Figures 11-12.



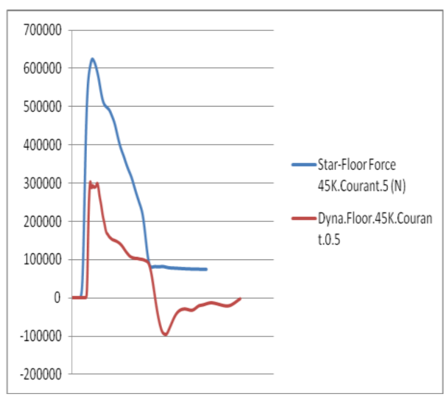
Floor Force by LS-Dyna (Newton)
Figure (11)



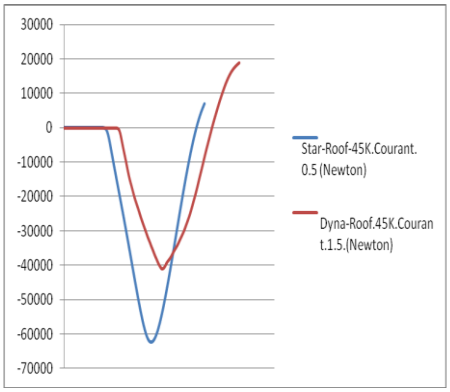
Roof Force by LS-Dyna (Newton)
Figure (12)

In looking at the plots in Figures 13 and 14, one can notice that although qualitatively the two plots look similar, the peak and residual forces are disparate from a quantitative point of view. StarCCM+ calculates the Floor peak force of about 600 kN, while LS-Dyna calculates a peak level of about 300 kN. This was unexpected and somewhat of a surprise.

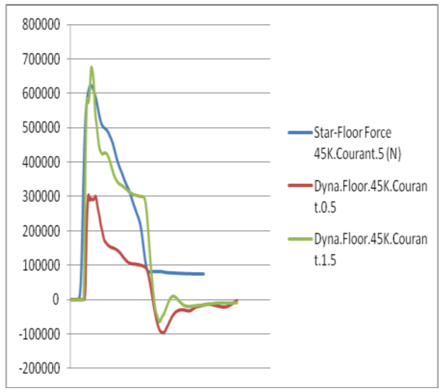
A further investigation revealed that in order to get a similar peak performance, the Courant number needed to be adjusted.



Floor Force
StarCCM+ & LS-Dyna
Figure (13)



Roof Force
StarCCM+ & LS-Dyna
Figure (14)

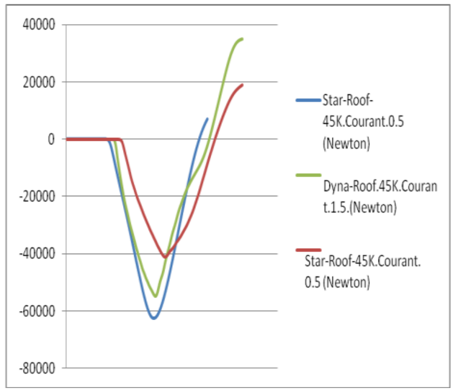


Floor Force
StarCCM+ & LS-Dyna
Courant setting 0.5-1.0
Figure (15)

The Effect of Courant Number:

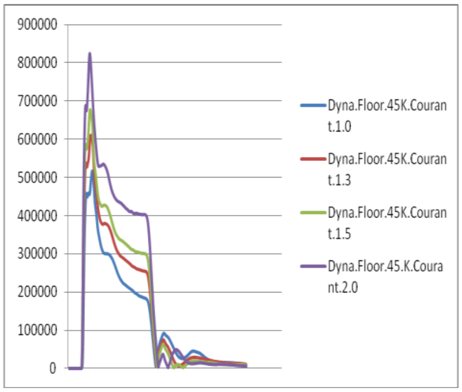
In the finite volume CFD analysis, the Courant number was set to be 0.5 in the analysis procedure. The results of the peak and the residual force are reported in Figures 5 and 6, for the floor and roof force respectively. The same Courant number was set in the input deck for LS-Dyna as well, so a direct comparison between the two could be made. However, the results were very different as shown in Figures 13 and 14. As a result, a series of subsequent runs with different Courant numbers were established to attempt a better match in results. Figures 15 and 16 show, that by setting the courant number in LS-Dyna to 1.5 instead of 0.5, the peak values and the shape of the two curves now match quite well between Star-CCM+ and LS-Dyna.

It is however noticed, that the residual values between the LS-Dyna and Star-CCM+ remain different. This could not be explained at the time of the analysis (Figures 15-16). To explain the discrepancy, a further investigation is required.



Roof Force
StarCCM+ & LS-Dyna
Courant 0.5-1.0
Figure (16)

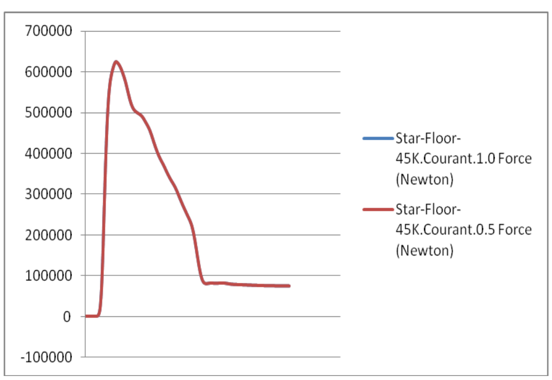
In Figure 17, one can observe that peak floor force values vary with Courant number in LS-Dyna. The same observation can be seen for the roof forces. The plot below shows the results from the different Courant settings and how it affects the total force levels.



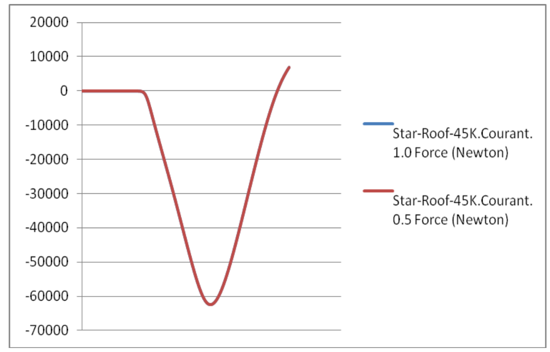
Total Floor Force (N) comparison
LS-Dyna variation
Courant setting 1.0-2.0
Figure (17)

Overall, the results should caution the analyst that one needs to take care when comparing the analysis iterations for a given design consideration; one needs to keep the Courant stability criteria the same for all comparisons. It is also advisable to have a validation plan with a test data to determine a suitable Courant parameter correctly for future work.

A further investigation of Courant number sensitivity was made with StarCCM+. These results showed, that unlike LS-Dyna, a Courant number change from 0.5 to 1.0 yields no noticeable difference in the floor and the roof results as shown in Figures 18 & 19. In fact, the two plots (below) are on top of one another, and no difference can be seen.



Total Floor Force comparison
StarCCM+ variation
Courant setting 0.5-1.0
(Note: lines are the same)
Figure (18)

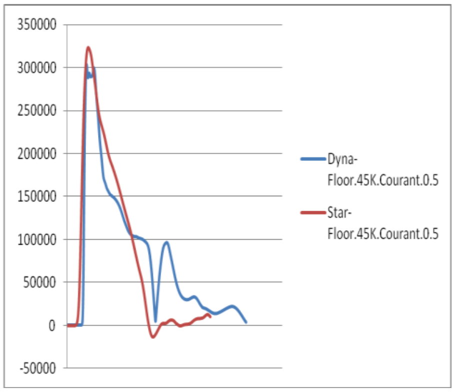


Total Roof Force comparison
StarCCM+ variation
Courant setting 0.5-1.0
(Note: lines are the same)
Figure (19)

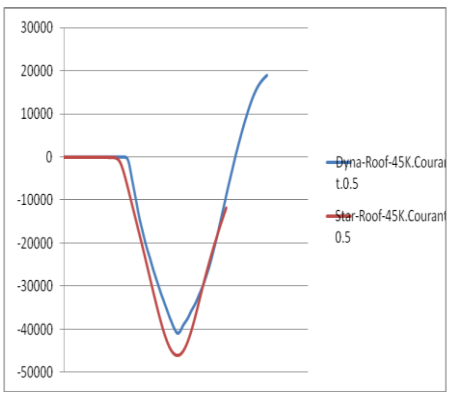
Another attempt was made to match the results between StarCCM+ and LS-Dyna while keeping the Courant number the same as the nominal case of 0.5. In this approach, the LS-Dyna input was left to be as total pressure of 100 bars, while the input for the inlet supersonic static pressure in StarCCM+ was changed from 12.6 bars to only 1.0 bar and the inlet total gage pressure remained unchanged at 99 bars. The inlet static pressure was fixed, instead of allowing it to be calculated from the supersonic input velocity of Mach 2. In other words, the 100 bar pressure can be applied without any velocity considerations. Effectively, the absolute pressure in both software setups remained at 100 bars.

At this point, the results from the two approaches matched quite well. A comparison of the forces can be seen in Figure 20 & 21.

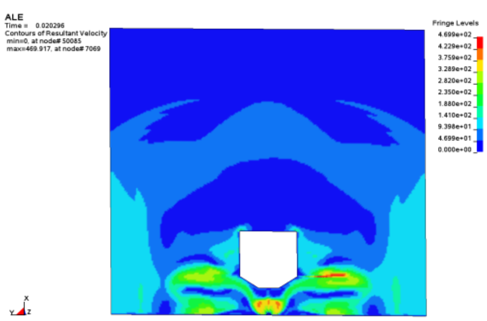
With the adjusted supersonic input, the peak floor pressure dropped in StarCCM+ from what was at 620 kN to about the same as the LS-Dyna peak of 320 kN. This is due to drop in the supersonic static pressure that is changed from 12.6 bars to 1. This is a significant drop due only to the static pressure input, while the total pressure remained the same.



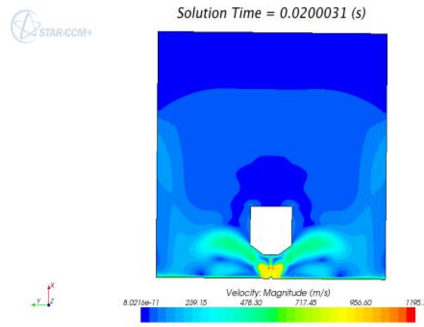
Floor Force (N)
StarCCM+ & LS-Dyna
StarCCM+ supersonic pressure = 1.0 bar
Figure (20)



Roof Force (N)
StarCCM+ & LS-Dyna
StarCCM+ supersonic pressure = 1.0 bar
Figure (21)



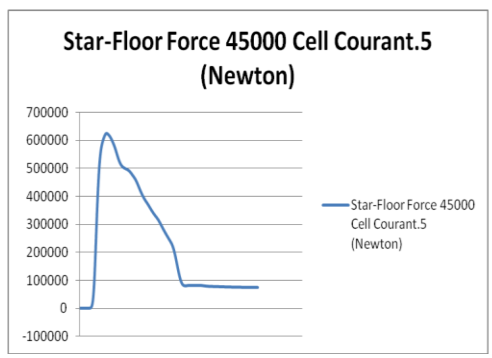
LS-Dyna Velocity fluctuations
Figure 22



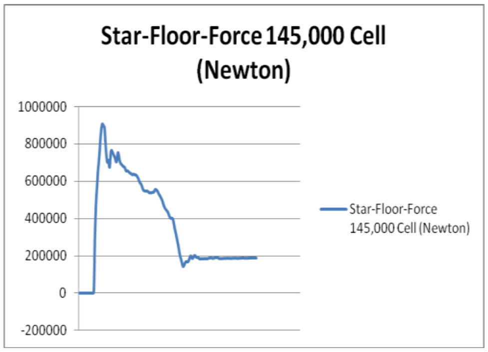
Star-CCM+ Velocity fluctuations
Figure 23

Mesh Refinement:

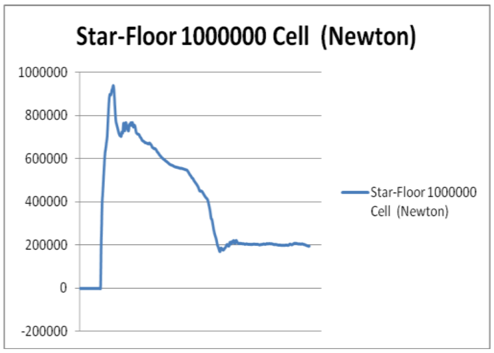
To see if the baseline mesh was sized to capture the blast accurately, a mesh dependency study was performed. The StarCCM+ results with different mesh densities were compared to one another in the following plots (24-26). The 45,000 element model would serve as a basis to compare the subsequent results to the more refined 145,000 element model, and finally the 1,000,000 element model.



StarCCM+ Floor Force
45,000 Mesh Density
Figure(24)



StarCCM+ Floor force
145,000 Mesh Density
Figure (25)

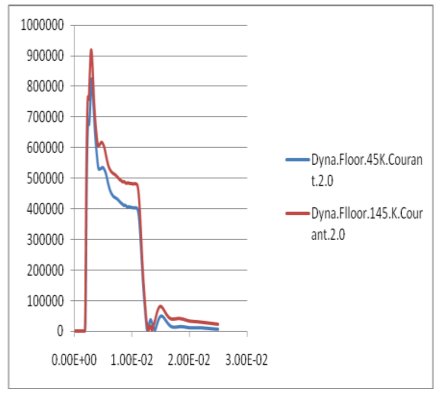


StarCCM+ Floor force
1,000,000 Mesh Density
Figure (26)

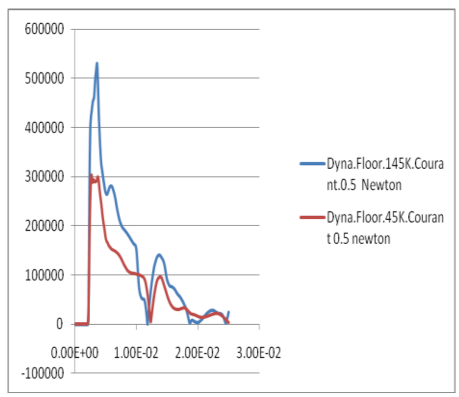
The plots 24-26 reveal that the 45,000 element model has a peak value of about 640 kN, while the residual value is 80 kN. As the mesh is refined to 145,000 cells, The peak rises to about 900 kN and the residual force settles at 200 kN. Finally the 1,000,000 cell model reveals a rise in the peak to 950 kN while the resedual force stays at 200 kN.

It can be concluded that as the mesh density of the model increases, the peak and residual forces also increase. Mesh dependency does not occur between 45,000 and 100,000 elements, as there is very little difference in the force plots between those two mesh densities.

The LS-Dyna results for different mesh densities are now compared to one another in the following Figures 27-28. The 45,000 element size model would serve as a basis to compare the subsequent results to the more refined 145,000 size element model. A similar trend to the StarCCM+ results is noted in the figures. As the model mesh density increases, the peak and residual force also increase. This is a far more noticeable effect with the Courant value of 0.5 than it is with 2.0.

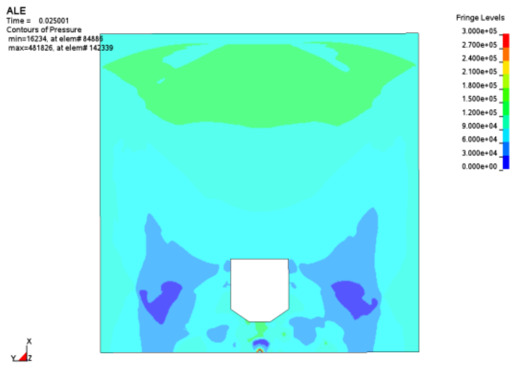


LS-Dyna 45K and 145K Mesh Density
Floor Force (N)
Comparison for Courant-2.0
Figure (27)

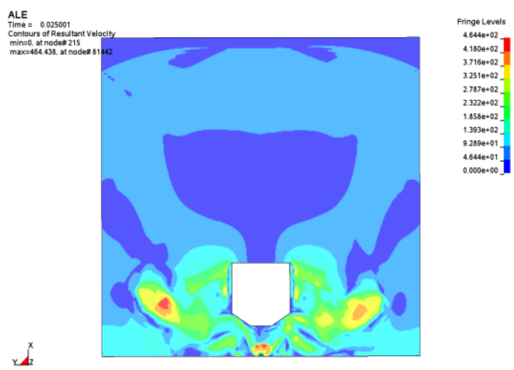


LS-Dyna 45K and 145K Mesh Density
Floor Force (N)
Comparison for Courant-0.5
Figure (28)

In figures 29-30, one can observe the pressure and velocity fluctuations for the 145,000 element model, computed by LS-Dyna.



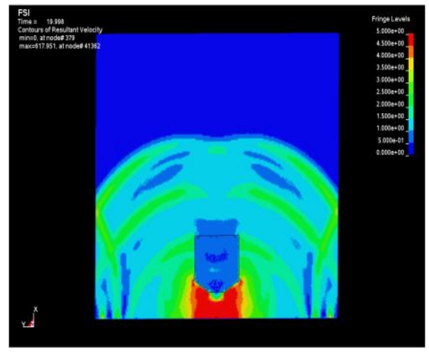
LS-Dyna Pressure fluctuations
Figure (29)



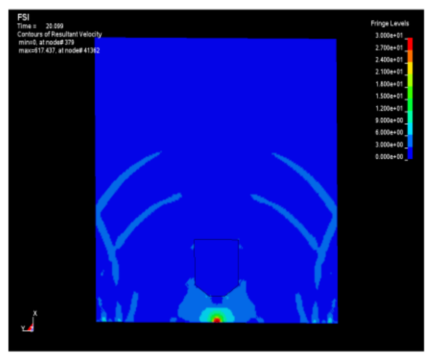
LS-Dyna Velocity fluctuations
Figure (30)

FSI Analysis:

Finally, an FSI analysis was conducted with LS-Dyna. This type of analysis requires a simultaneous solving of a solid and fluid domain which allows for solid deformation. The solid domain side is handled in the usual way by the Finite Element based software. Additionally, the fluid mesh is represented by an Eulerian domain. The two domains are coupled with the ALE (Arbitrary Eulerian Lagrangian) technique that is available in LS-Dyna. This capability is particularly useful to predict the reaction of the fluid induced pressure on the 'deformed' structure. For this case the 'cavity', that was considered as the bluff body in the previous cases, is now replaced by a 'deformed structure' imbedded in the fluid domain. The reaction forces are then computed between the domains. The explosive pressure pulse causes the cavity to be deformed or moved relative to the fluid domain. At 20 ms, the results showed a 12 mm displacement of the vehicle. Results are shown in Figures 31-36.

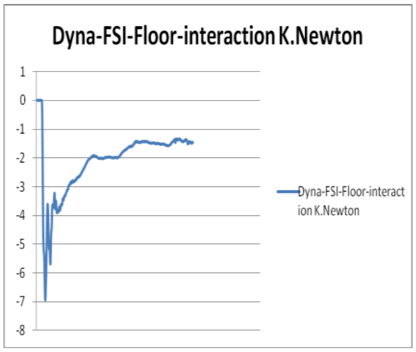


LS-Dyna FSI Pressure fluctuations
Figure (31)

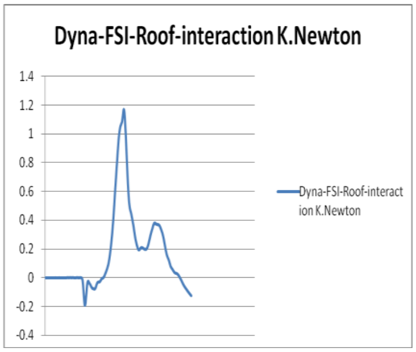


LS-Dyna FSI Pressure fluctuations
Figure (32)

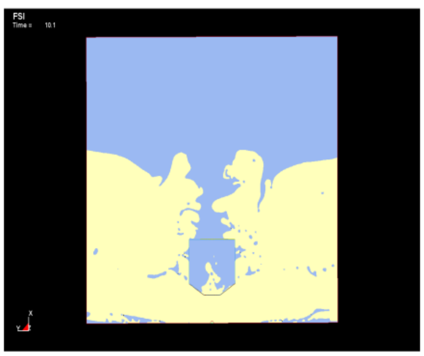
LS-Dyna FSI Vehicle Displacement
Figure (33)



LS-Dyna FSI Floor Interaction Force
Figure (34)



LS-Dyna FSI Roof Interaction Force
Figure (35)



LS-Dyna FSI Flow Interaction
With the Structure
Figure (36)

Conclusion:

The techniques discussed in this paper shed new light to possible ways of utilizing 2-D blast analysis procedures in both CFD and CAE. By paying very close attention to matching boundary conditions and solution options, the analyst can walk between the two disciplines and obtain comparable results.

The selection of Courant number was found to be extremely critical for LS-Dyna and directly influenced the reported peak forces. In comparison, the StarCCM+ results were not sensitive to the Courant number.

A conclusion particular to the CFD process was that the supersonic pulse at the inlet presents additional complexity to setup the boundary conditions. Both a total pressure and static pressure can be defined at the boundary. While holding the total pressure constant, decreasing the static pressure also decreases the peak and residual forces.

A final conclusion is that as the mesh density of the model increases, the peak and residual forces also increase up to a critical dimension.

Acknowledgements:

The Authors would like to thank the following individuals who have helped this effort throughout its course:

Mr. Sanjay Kankanalapalli and Dr. Ravi Thyagarajan who have conducted various 3-D analyses with LS-Dyna on full vehicle structures incorporating ALE, and assisted the authors to incorporate various setting and options in LS-Dyna. They also contributed to the better understanding of what comparisons can be drawn from the results of 3-D models to that of 2-D counterpart behavior.
Mr. Scott Shurin and Dr. Vamshi Korivi for their many valuable suggestions to try out various loading, boundary, physics models and analysis options available in Star-CCM+.

References:

LS-Dyna reference manual version 971R5
StarCCM+ V.5.02 online reference manual