A Coupled CFD-FEM Analysis to Simulate Blast Effects on High Security Vehicles Using Modern Hydrocodes

Arash Ramezani, Burghard Hillig, Hendrik Rothe
Chair of Measurement and Information Technology
University of the Federal Armed Forces
Hamburg, Germany
Email: ramezani@hsu-hh.de, burghard.hillig@hsu-hh.de, rothe@hsu-hh.de

***Abstract*—The present time is shaped by a variety of religious, political and military conflicts. In times of asymmetric warfare and constantly changing sources of danger from terrorist attacks and other violence based crimes, the personal need for protection continues to rise. Aside from military applications, there is a large area for the use of high security vehicles. Outwardly almost indistinguishable from the basic vehicles, security vehicles are used for protecting heads of state, as well as individuals. To remain state of the art it is necessary for security vehicles to permanently continue to develop protection against modern weapons and ammunition types. It is enormously cost intensive to check any new technology by firing or blasting of real vehicles. Therefore, more and more calculations of new security concepts and materials are carried out by numerical computer simulations. However, product simulation is often being performed by engineering groups using niche simulation tools from different vendors to simulate various design attributes. The use of multiple vendor software products creates inefficiencies and increases costs. This paper will present the analysis and development of an interface between the most common Computer Aided Engineering (CAE) applications ANSYS Autodyn and Abaqus to exploit the advantages of both systems for the simulation of blast effects.**

Keywords-CFD-FEM coupling methods; fully automatic structure analysis; high-performance computing techniques; blast loading; vehicle structures.

I. INTRODUCTION

Since the 1960’s, the simulation of physical processes has been a steadily growing and integral part of CAE. Especially, the Computational Fluid Dynamics (CFD) and the discretization of complex models using the Finite Element Method (FEM) have made an impressive development from individual highly specialized applications to the standard of industrial product development. This process was supported by the progressive development of increasingly powerful and less expensive computer hardware. Together with specialized software, a triumph of the simulation of physical processes in everyday technical work has emerged. Positive effects due to the use of simulation tools have been shorter development times, lower production costs, more innovative products, improved security and higher quality. The previous modelling of components and objects of the real world by Computer Aided Design (CAD) software is an important prerequisite for the efficient use of the simulation tools. This has been established as a standard in the automotive industry, so that almost every part of a vehicle can be constructed by using CAD. These complete and realistic vehicle models can be analyzed virtually by available simulation software.

The two leading software providers for CFD / FEM calculations are ANSYS (Canonsburg, USA) and Abaqus FEA from Dassault Systèmes (Vélizy-Villacoublay, France). Although both providers offer software with similar features available, their performance is characterized by different spreads and focuses. For example, Abaqus and CATIA, a CAD software, which is also distributed by Dassault Systèmes, is predominantly used by the automotive industry and provides excellent opportunities for the simulation and modelling of complete vehicles. This includes screwed and adhesive connections. On the other hand, ANSYS offers a wide range of sophisticated simulation capabilities in the field of CFD, which includes the modelling and simulation of explosive detonations and the subsequent propagation of shock waves. The different focuses of the performance of ANSYS and Abaqus yield to a mixed, demand-based use of the software in the research and development area, so that different software is used even within the same company on the same project in different areas of activity. This circumstance is amplified by the fact that product simulations are performed by engineering groups using niche simulation tools from different vendors to simulate various design attributes. Unfortunately the leading software providers avoid the effective interaction of their simulation tools due to mutual competition. This complicates the development effort and results in longer development times in research and industry.

Particularly, in the area of armored security vehicles, it is necessary to remain state of the art and to constantly consider the ongoing development of modern weapon and ammunition types. Experimental tests of the harmful effects of new technologies by blast or impact is associated with enormous time and financial costs.

In order to exploit the full potential of ANSYS and Abaqus, we have developed an interface between these two software platforms [1]. This interface allows an iterative transfer of the blast simulation of ANSYS to the structural mechanic solver of Abaqus, which simulates the effects on the vehicle model and vice versa.

This paper reports on the development of this interface between ANSYS and Abaqus, which will enable combining the strengths of the two leading software providers with the aim of generating synergies that result in short development times and lower costs.

After a brief introduction and description of the different methods of space discretization in Section III, there is a short section on ballistic trials where the experimental set-up is depicted, followed by Section V describing the analysis with numerical simulations. The paper ends with a concluding paragraph in Section VI.

II. STATE-OF-THE-ART

A first step in developing an interface between ANSYS and Abaqus has already been reported in [1]. The developed interface allows accessing a set of data and passing them to Abaqus. Python was used as a programming language. ANSYS provides the data records for the interface as .txt files. These files contain data points with Cartesian coordinates, which describe the propagation of shock waves after blasting. The interface takes this data and splits it into separate information. In a further step, the data is stored in a list, linked with the corresponding time points, pressure data and coordinates. It is also possible to use a set of data and to interpolate between the time points to produce a larger data set. After the data has been written and saved in a linked form, the interface retrieves the CAD model. Subsequently, the explosion data can be projected onto a selectable surface of the model. Then, an iterative loop realizes the coupling between CFD and FEM simulations. This approach for a coupled CFD-FEM analysis is called “strong coupling.” In another approach, the “semi-strong coupling,” a smaller amount of data is used and mathematically interpolated for a sufficient approximation. The third concept is a “weak coupling” solution. Here, neural networks and deep learning can be used to replicate blast effects on different vehicle structures. Until now, the basic functionality of the interface could be validated on different models, including the model of a safety vehicle.

III. FUNDAMENTALS OF SIMULATION

In the security sector, the partly insufficient safety of people and equipment due to failure of industrial components are ongoing problems that cause great concern. Since computers and software have spread into all fields of industry, extensive efforts are currently being made in order to improve the safety by applying certain computer-based solutions. To deal with problems involving the release of a large amount of energy over a very short period of time, e.g., explosions and impacts, there are three approaches, which are discussed in [2].

As the problems are highly non-linear and require information regarding material behavior at ultra-high loading rates, which are generally not available, most of the work is experimental and may cause tremendous expenses. Analytical approaches are possible if the geometries involved are relatively simple and if the loading can be described through boundary conditions, initial conditions, or a combination of the two. Numerical solutions are far more general in scope and remove any difficulties associated with geometry [3].

For structures under shock and impact loading, numerical simulations have proven to be extremely useful. They provide a rapid and less expensive way to evaluate new design ideas. Numerical simulations can supply quantitative and accurate details of stress, strain, and deformation fields that would be very expensive or difficult to reproduce experimentally. In these numerical simulations, the partial differential equations governing the basic physical principles of conservation of mass, momentum, and energy are employed. The equations to be solved are time-dependent and nonlinear in nature. These equations, together with constitutive models describing material behavior and a set of initial and boundary conditions, define the complete system for shock and impact simulations.

The governing partial differential equations need to be solved in both time and space domains. The solution over the time domain can be achieved by an explicit method. In the explicit method, the solution at a given point in time is expressed as a function of the system variables and parameters, with no requirements for stiffness and mass matrices. Thus, the computing time at each time step is short but may require numerous time steps for a complete solution. The solution for the space domain can be obtained utilizing different spatial discretization, such as Lagrange [4], Euler [5], Arbitrary Lagrange Euler (ALE) [6], or mesh free methods [7]. Each of these techniques has its unique capabilities, but also limitations. Usually, there is not a single technique that can cope with all the regimes of a problem [8]. The crucial factor is the grid that causes different outcomes. More details are discussed in Section IV.

Due to the fact that all engineering simulations are based on geometry to represent the design, the target and all its components are simulated as CAD models. Real-world engineering commonly involves the analysis and design of complicated geometry. These types of analysis depend critically on having a modeling tool with a robust geometry import capability in conjunction with advanced, easy-to-use mesh generation algorithms [9]. It often is necessary to combine different simulation and modeling techniques from various CAE applications. However, this fact can lead to major difficulties, especially in terms of data loss and computational effort. Particularly the leading software providers prevent an interaction of their tools with competing products. But to analyze blast loading and its effects on vehicle structures, different CAE tools are needed. Therefore, it is important that an interface is provided that allows a robust interaction between various applications. Using a CAD neutral environment that supports direct, bidirectional and associative interfaces with CAE systems, the geometry can be optimized successively and analysis can be performed without loss of data [10].

IV. MATERIALS AND METHODS

Various approaches are possible when it comes to solving problems that involve the release of large amounts of energy in very short periods of time, which then propagate as shock waves or act as impact on structures. Analytical solutions offer a very powerful way to describe such a process. Unfortunately, their applicability is restricted to problems with simple geometries and few boundary and initial conditions. In contrast, numerical simulations offer much more general applications with complex structures and feasible solutions.

The underlying physical model of numerical simulations is provided by physical conservation laws, the equation of state and the constitutive model. Partial differential equations for the conservation of energy, momentum, and mass form the physical conservation laws. Furthermore, the equation of state combines the internal energy or temperature and the density or volume of a material with the pressure. As a result, changes in the density and irreversible thermodynamic processes such as shock-like heating can be considered. In addition, the constitutive model includes the influence of the material to be simulated and describes the effect of deformation, i.e. changes in shape and material strength properties.

Together, these equations form a set of coupled, time- and location-dependent, highly non-linear equations, which can be solved by computer calculations. The governing partial differential equations need to be solved in both time and space domains. The solution over the time domain can be obtained by an explicit method, which is an iterative method and leads to a step by step solution in the time domain. Software for numerical simulation of shock and impact processes is called a hydrocode [11].

A. *Methods of Space Discretization*

All existing structural dynamics and wave propagation codes obtain solutions to the Differential Equations (DEs) governing the field by solving an analogous set of algebraic equations. The governing DEs are not solved directly, because currently only a handful of closed-form solutions for DEs are available. The equations of structural dynamics, being a coupled set of rate equations, which account for the effects of severe gradients in stress, strain and deformation, material behavior ranging from solid to fluid to gas, temperatures from room temperature to melt temperature are highly nonlinear and do not lend themselves to closed-form solutions in the general case.

To get a solution over the spatial domain a discretization of the material with a mesh is necessary. FEM uses such a discretization by dividing the problem space into separate elements. These elements can have different shapes: In two dimensions, the shape of quadrilaterals or triangles, in three dimensions hexahedrons and tetrahedrons are usually used. Even complicated geometries can be formed with these elements. Each FEM element has a certain number of nodes, which are located at its corners and have known spatial coordinates. The displacement of these nodes represents the unknowns of the partial differential equations to be solved. There are multiple, different spatial discretization methods related to FEM, such as Lagrange, Euler, Arbitrary Lagrange Euler (ALE) or mesh free methods. Each of these methods can be used independently, but some specific problems need a combination of different discretization methods.

1) Lagrange

The Lagrange method divides an object into a spatial grid where the grid is fixed to the object and moves with it. The material components within an element do not change. If forces are acting on a node, it is displaced, and thus the forces are transmitted to its neighboring nodes, similar to a spring-mass system. This results in deformations of the grid. The nodes of the edge elements of an object remain unchanged so that the boundary and interface conditions can be easily applied. Clear material boundaries are also available so that space outside the material does not require an extra grid and therefore the conservation of mass is automatically satisfied. Figure 1 shows two objects with its mesh as an example of the Lagrange method. Two objects consisting of different materials represented by the colors blue and green before (left side) and after impact (right side). The green object has an initial velocity in the direction of the blue object. The right side of the figure shows the discretization dependent deformation after the impact with the Lagrange solver. The mesh is bound to the objects and divides them into multiple elements. After an impact the objects deform due to the deformation of the elements. A weak point of the Lagrange method is a strong distortion of the mesh in heavily loaded regions, as shown in Figure 1. in the area adjacent to the green and blue object. In general, the Lagrange method is best suited for complex geometries and structures, projectiles and other solids. A disadvantage of Lagrange is the occurrence of strong distortions of mesh element at high loads. Such a distorted element can adversely affect the temporal solution of the simulation since the time step is proportional to the size of the smallest element.

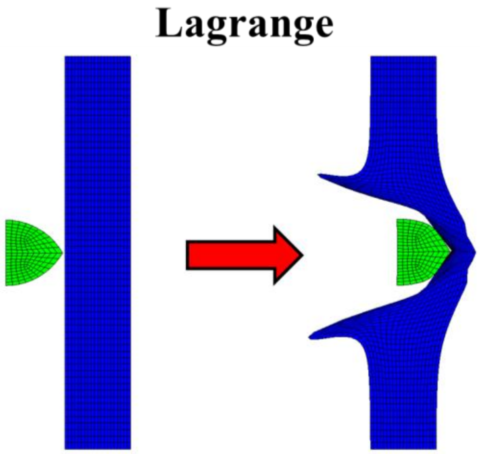


Figure 1. Lagrange method example.

In general the Lagrange method is best suited for complex geometries and structures, projectiles and other solids. A disadvantage of Lagrange is the occurrence of strong distortions of mesh element at high loads. Such a distorted element can adversely affect the temporal solution of the simulation since the time step is proportional to the size of the smallest element.

2) Euler

In the Euler method the coordinates of the nodes are fixed and form the entire mesh of the region to be solved. The material flows through the mesh as a function of time and changes the value of the element, while the spatial coordinates and the nodes remain fixed. This is the reason why no element distortion is possible in the Euler method. In contrast to Lagrange, boundary nodes do not necessarily coincide together with material boundary conditions. Thereby difficulties can arise with the application of boundary and interface conditions. Figure 2. shows two objects and the mesh as an example of the Euler method. Two objects consisting of different materials represented by the colors blue and green before (left side) and after impact (right side). The mesh fills the whole space. The green object has an initial velocity in the direction of the blue object. The right side of the figure shows the discretization dependent deformation after the impact with the Euler solver. The mesh is not bound to the objects like in the Lagrange frame. Instead the mesh fills the whole space with the objects and empty space between them. During the simulation the material of the objects is transported through the mesh of the space. After an impact the mesh stays clear but its content is partly deformed.

In general the Euler method is used to model the propagation of gases and fluids as a result of an explosion or impact. In the investigation of solids, the Euler method has a disadvantageous effect, since additional calculations are needed to transport the stress tensor and the history of the material through the lattice. In this case Euler needs more computing performance and smaller elements to resolve the occurring shock waves.

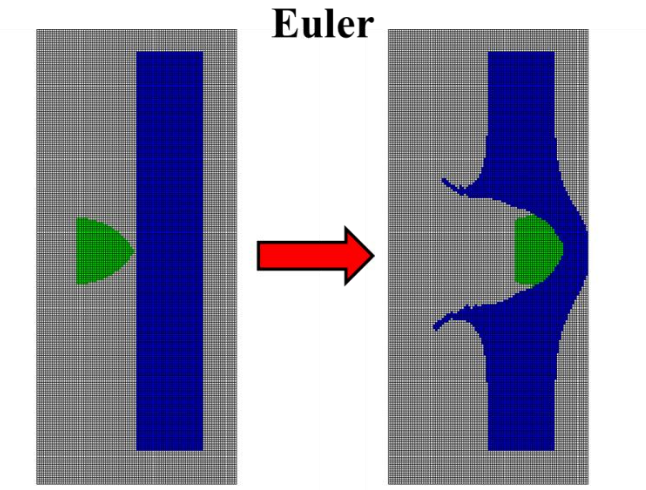


Figure 2. Euler method example.

3) ALE

The Arbitrary Lagrange Euler (ALE) method is a mix of Lagrange and Euler method. ALE allows an arbitrary redefinition of the mesh on each calculation step. Different predefined grid motions can be specified, such as free (Lagrange), fixed (Euler), equipotential, equal spacing and others. As an advantage distortions can be avoided. On the other hand, additional computation steps are necessary to move and to convert the grid. An example of ALE is shown in Figure 3. Two objects consisting of different materials represented by the colors blue and green before (left side) and after impact (right side). The blue object has an initial velocity in the direction of the green object. The right side of the figure shows the discretization dependent deformation after the impact with the ALE solver. In comparison to the pure Lagrange method (Figure 1.), no lattice distortions occur here.

4) SPH

Smoothed Particle Hydrodynamics is a method which is not based on a fixed topological lattice but on a finite set of particles. These particles are embedded to the material similar to the nodes of the Lagrange method, but their connections are not fixed. However, the particles represent not only mass points, but also interpolation points for the calculation of the physical variables. The calculations are based on the data of the neighboring particles and are scaled by a weighting function. Unlike Lagrange, no grid distortion can occur at SPH, since no grid exists. Related to the Euler method, SPH has the advantage that all material boundaries and interfaces are clearly defined. Figure 4 illustrates two objects consisting of different materials in the SPH frame represented colored particles before (left side) and after impact (right side). The green object has an initial velocity in the direction of the blue object. The right side of the figure shows the discretization dependent deformation after the impact with the SPH solver. As seen in Figure 4. two objects consist of small particles in the SPH frame. Their behavior before and after an impacts differs from the solutions in the Lagrange, Euler or ALE frame.

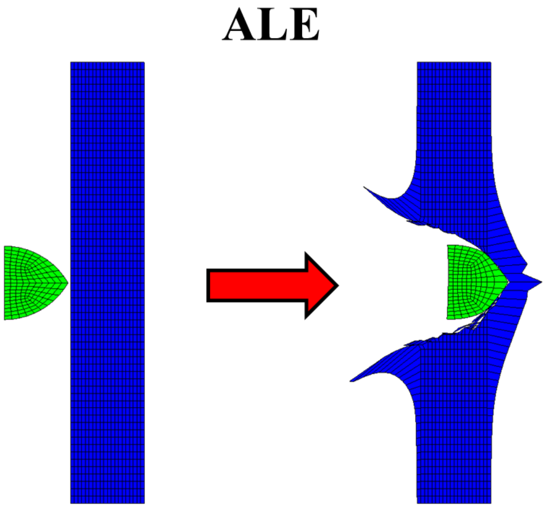


Figure 3. ALE method example.

Copyright (c) IARIA, 2017. ISBN: 978-1-61208-594-4

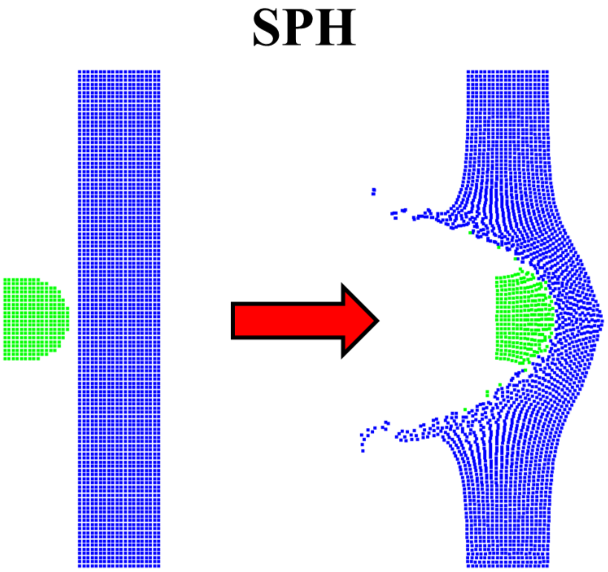


Figure 4. SPH method example.

The SPH method has proven especially useful in the simulation of impact processes on brittle materials [2]. It should be noted that the modelling of the material as particles leads to significantly higher computing effort per time step.

For problems of dynamic fluid-structure interaction and impact, there typically is no single best numerical method which is applicable to all parts of a problem. Techniques to couple types of numerical solvers in a single simulation can allow the use of the most appropriate solver for each domain of the problem.

One of the more important issues, which have to be carefully considered is the issue of mesh size. Different results are obtained if the number of cells per unit length is not adequate. For example, it was found that for penetration studies with eroding long rods, the number of cells on the rod’s radius should be at least eleven. The same density of cells should be kept in the target, at least for several projectile radii around its symmetry axis. In order to save computing time, the cell size at farther zones can be gradually increased according to their distance from the symmetry axis. The mesh cell size depends on the specific problem. As an example, a small cell size should be considered in cases where there is a fracture in the projectile or target. It is recommended that while preparing the code for its final runs, the numerical convergence with respect to mesh cell size should be checked. Another important issue, especially when material elements are expected to deform considerably, is the issue of erosion with Lagrangian codes. At large deformations the code may run into trouble when treating heavily deformed elements. The use of the erosion threshold condition is then necessary in order to eliminate elements at a predetermined value of the plastic or geometric deformation. The erosion should be monitored constantly, and when it is too high one should replace the Lagrangian with an Eulerian code.

The goal of this paper is to evaluate an interface between different hydrocodes, computational tools for modeling the behavior of continuous media. In its purest sense, a hydrocode is a computer code for modeling fluid flow at all speeds. For that reason, a structure will be split into a number of small elements. The elements are connected through their nodes (see Figure 5.). The mesh divides the object into small elements connected by its nodes.

The behavior (deflection) of the simple elements is well-known and may be calculated and analyzed using simple equations called shape functions. By applying coupling conditions between the elements at their nodes, the overall stiffness of the structure may be built up and the deflection/distortion of any node – and subsequently of the whole structure – can be calculated approximately [12]. Therefore, several runs are necessary: From modeling to calculation to the evaluation and subsequent improvement of the model.

Hydrocodes, or wave propagation codes, are a valuable adjunct to the study of the behavior of metals subjected to high-velocity impact or intense impulsive loading. The combined use of computations, experiments and high-strain-rate material characterization has, in many cases, supplemented the data achievable by experiments alone at considerable savings in both cost and engineering man-hours.

A large database exists of high-pressure Equation-Of-State (EOS) data. Considerable data on high rate deviatoric behavior exists as well although, unlike EOS data, it is not collected in a few compilations but scattered throughout a diverse literature. Experimental techniques exist for determining either EOS or strength data for materials not yet characterized under high-rate loading conditions.

By contrast, computations with non-metallic materials such as composites, concrete, rock, soil and a variety of geological materials are, in effect, research tasks. This is due to several reasons: lack of definitive computational models for high strain rate–temperature–pressure response; lack of a database for EOS and high rate strength data for such materials; lack of test methodologies for anisotropic materials subjected to high-rate loading.

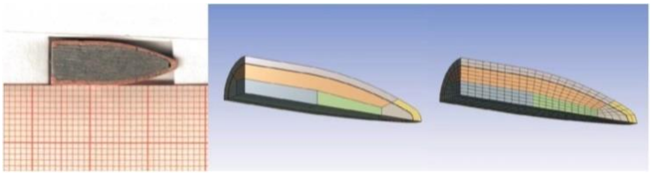


Figure 5. The left side shows the cross-section of a bullet over millimeter paper, which could theoretically divide the bullet into multiple elements. The right side shows an example of a mesh in the numerical simulation. Different parts are color coded in the representation of the bullet.

Copyright (c) IARIA, 2017. ISBN: 978-1-61208-594-4

A large number of ad hoc models exist for explosives, geological materials, concrete and other non-metallics. Many of these lack a firm theoretical foundation. This is an area where considerable research is required, both to devise appropriate test techniques to measure material response under high strain rates, elevated temperatures and high pressures as well as to develop appropriate constitutive models.

B. *Interface*

In general, an interface connects systems that have different properties with the purpose of exchanging information. For computers, this is mainly the case between software, hardware, peripheral devices and humans. Communication at the interface can be either in one direction, such as a remote control or keyboard, or in both directions, such as a touch screen or a network adapter [13].

In the context of numerical simulation of blast and impact processes, an interface is necessary to ensure an effective coupling of CFD / FEM simulations between the software Abaqus and ANSYS. For our research, ANSYS is to be used to provide data from simulated explosions using Euler-Lagrange coupling. On the other hand, the structure, which is affected by the blasting is simulated by Abaqus. The developed Interface has the task of conveying the data between ANSYS and Abaqus, so that the individual simulation steps can be performed successively with respect to the successive transfer of data.

V. EXPERIMENTAL SECTION

In computing, an interface is a shared boundary across two separate components of a computer system exchange information. The exchange can be between software, computer hardware, peripheral devices, humans and combinations of these. Some computer hardware devices such as a touchscreen can both send and receive data through the interface, while others such as a mouse, microphone or joystick operate one way only [13].

Coupled FEA/CFD analysis is an alternative technique, where separate FEA and CFD codes are used for solid and fluid regions, respectively, with a smooth exchange of information between the two codes to ensure continuity of blast loading data. The main merit of the approach is to enable users to take full advantages of both CFD and FEA capabilities.

The objective of this work is to develop an interface between ANSYS Autodyn and Abaqus. The software ANSYS is used to solve linear and non-linear problems of structural mechanics, computational fluid dynamics, acoustics and various other engineering sciences [14]. Here, ANSYS will provide data from the simulation of blast effects. The capability to couple Eulerian and Lagrangian frames in ANSYS is helpful in blast field modeling. The Eulerian frame is best suited for representing explosive detonations, because the material flows through a geometrically constant grid that can easily handle the large deformations associated with gas and fluid flow. The structure is modeled with the Lagrangian frame in Abaqus. Abaqus supports familiar interactive computer-aided engineering concepts such as feature-based, parametric modeling, interactive and scripted operation, and GUI customization [15].

First, every possibility of transferring the data from ANSYS outputs to Abaqus inputs has to be detected. A summary of this process is shown in Figure 6.

ANSYS will provide the data by generating a data set for the blast loading. Figure 7. shows the color coding of the shock wave goes from low pressures (0 hPa) in blue to high pressures in red (> 3500 hPa). The last picture shows the shock wave when the simulated vehicle is reached. This data set will include snapshots of given points in time. At this stage there is a data set of five points in time, between 0.0291s and 0.0475s (after detonation). Related to the points in time this data set includes the pressure values with Cartesian coordinates based on the simulation of the spread of explosive materials. A script is coded to read the blast loading data in Abaqus. This script, coded in Python, uses the line interface in Abaqus directly. First, a blast loading data is generated in ANSYS and saved as a normal text file in .txt format. The data set will be split to separate the different types of information. After that, a list will be created to save the data and connect the related time points to the coordinates and pressure values. At this point, there is a possibility to use linear interpolation between the five time points to generate a larger data base. After reading and saving the data set, the script will load the model used for impact tests in Abaqus. A surface of the model must be selected to project the blast data on it.

The goal is to investigate the impact of the blast data on a full vehicle model in Abaqus. This work (in progress) starts with a less complex model to validate the function of the script and the interface itself. The first model was a basic rectangle to be strained by the pressure data. Afterwards, two more complex models were tested successfully. This approach will lead to a surface similar to the silhouette of high security vehicles (see Figure 8.).

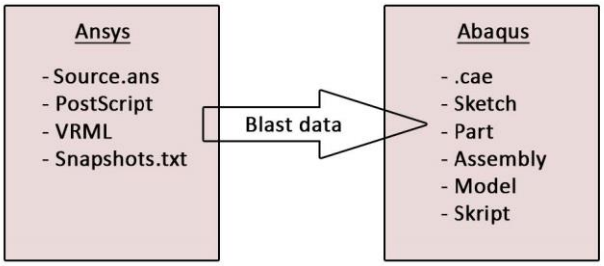
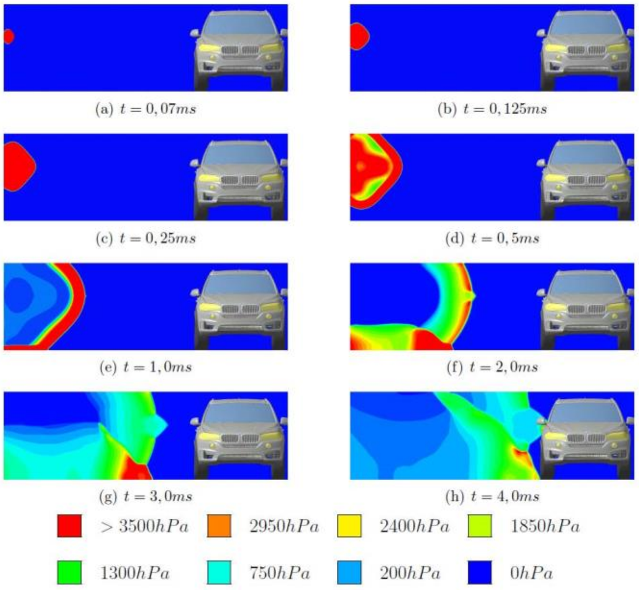


Figure 6. Inputs and outputs for an interface between Ansys and Abaqus. Blast data consisting of several files is exported by Ansys and read via a python script in Abaqus.



(a) $t = 0,07ms$ (b) $t = 0,125ms$

(c) $t = 0,25ms$ (d) $t = 0,5ms$

(e) $t = 1,0ms$ (f) $t = 2,0ms$

(g) $t = 3,0ms$ (h) $t = 4,0ms$

Figure 7. Progressive expansion of a blast in ANSYS Autodyn with a repr-esentation fo a vehicle on the rigth side.

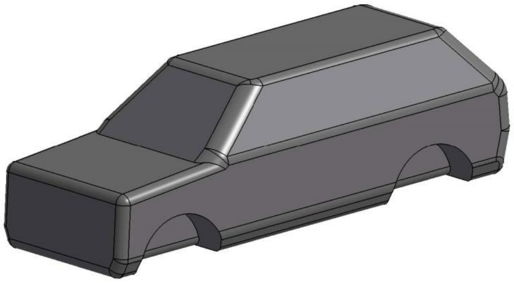


Figure 8. Testing structure in Abaqus with the coarse appearance of a vehicle.

The coupling is realized through an iterative loop between the FEA and CFD simulations, with communications ensuring continuity of shock compression data across the coupled boundaries between the FEA and CFD models. In the coupling process, intermediate individual FEA and CFD solutions are obtained in turn with dynamically updated boundary conditions.

To avoid exceptional deadlock of the individual CFD simulations, appropriate maximum numbers of iterations are assigned for each CFD model.

Testing means that the spatially discretized model is loaded with pressure. The change over time is decisive. An example is shown in Figure 9. . The unarmored SUV model was loaded with a typical explosive charge. The load on the vehicle is made visible by color coding from low strain (blue) to very high strain (red). The deformation on the sheet metal body parts is clearly shown. This data can be used to simply analyze vulnerabilities. The goal is, however, to use complete vehicle models and to carry out realistic investigations.

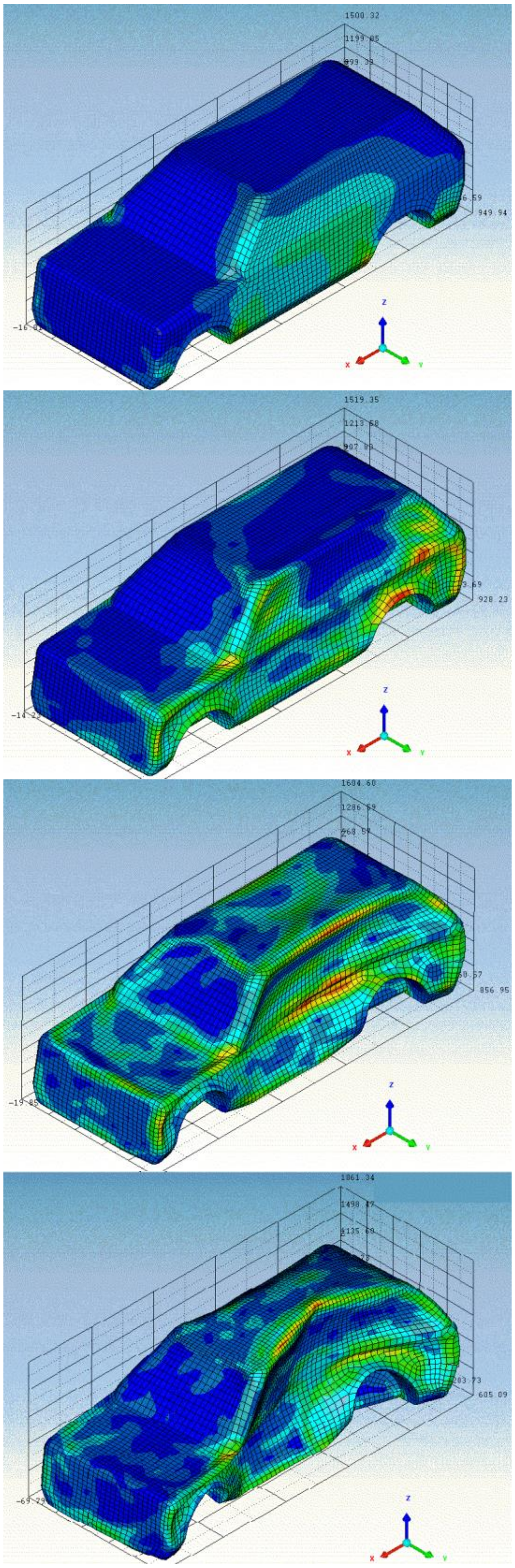


Figure 9. Simulated test structure (unarmed SUV) and deformation process after 3, 5, 10 and 20 ms (images arranged from top to bottom in order of increasing time).

VI. OUTLOOK

There are a variety of approaches in implementing the coupled FEA/CFD analysis. One is generally called “strong coupling,” where data have to be transferred between ANSYS Autodyn and ABAQUS in every single time step. A “semistrong coupling” can get along with a smaller set of date, using mathematical interpolation for a sufficient approximation. The third concept is a “weak coupling” solution. Here, neural networks and deep learning can be used to replicate blast effects on different vehicle structures. These approaches are going to be tested in a next step.

Furthermore, a larger blast loading data set has to be created in ANSYS. This will allow a more accurate illustration of blast effects on vehicle structures. Smaller time steps will enable a linear interpolation with a higher accuracy. Different explosives are going to be tested to expand the data base. The next step will be a model for the reflection of blast waves and dynamic changes of pressure values. Using a full vehicle model will provide important information about the behavior of armored structures under blast effects. But to validate the results of the simulation, more ballistic trials are needed. Based on the difficulties of full vehicle model simulations, the implementation of an automatic surface detection has to be taken into consideration. This could be helpful if a large number of different vehicles are investigated. In order to create a user-friendly interface it is possible to generate the script as a plug-in which can be started from the Abaqus user surface directly.

By using pre-defined blast data to create forces as vectors on our vehicle structures, the proposal can be generalized. Then, FEA analysis can be done with other software suites as well. Right now, the concept is not applicable to other systems. This is a major disadvantage and part of our future work. Furthermore, a parallelization of the problem should be considered.

VII. CONCLUSION AND FUTURE WORK

A technique for efficiently coupling FEA/CFD for the simulation of blast effects is described. An interface between ANSYS and Abaqus was created to provide blast data sets. The data sets from ANSYS include snapshots from the blast simulation saved at different points in time. The interface is coded in Python and also contains the possibility to use linear interpolation on the data sets.

A good agreement of blast load test data and simulation results was observed. Furthermore, it is shown that the coupled solutions can be obtained in sufficiently short turn-around times for use in design. These solutions can be used as the basis of an iterative optimization process. They are a valuable adjunct to the study of the behavior of vehicle structures subjected to high-velocity impact or intense impulsive loading. The combined use of computations, experiments and high-strain-rate material characterization has, in many cases, supplemented the data achievable by experiments alone at considerable savings in both cost and engineering man-hours.

REFERENCES

[1] E. Hansen, N. Ehlers, A. Ramezani, and H. Rothe, “Developing an Interface between ANSYS and Abaqus to Simulate Blast Effects on High Security Vehicles,” The Eighth International Conference on Advances in System Simulation (SIMUL 2016) IARIA, Aug. 2016, pp. 73-76, ISBN 978-1-61208-442-8.

[2] A. Ramezani and H. Rothe, “Investigation of Solver Technologies for the Simulation of Brittle Materials,” The Sixth International Conference on Advances in System Simulation (SIMUL 2014) IARIA, pp. 236-242, Oct. 2014, ISBN 978-61208-371-1.

[3] J. Zukas, “Introduction to Hydrocodes”. Elsevier Science, 2004.

[4] A. M. S. Hamouda and M. S. J. Hashmi, “Modelling the impact and penetration events of modern engineering materials: Characteristics of computer codes and material models,” Journal of Materials Processing Technology, vol. 56, pp. 847-862, Jan. 1996.

[5] D. J. Benson, “Computational methods in Lagrangian and Eulerian hydrocodes,” Computer Methods in Applied Mechanics and Engineering, vol. 99, pp. 235-394, Sep. 1992, doi: 10.1016/0045-7825(92)90042-I.

[6] M. Oevermann, S. Gerber, and F. Behrendt, “Euler-Lagrange/DEM simulation of wood gasification in a bubbling fluidized bed reactor,” Particuology, vol. 7, pp. 307-316, Aug. 2009, doi: 10.1016/j.partic.2009.04.004.

[7] D. L. Hicks and L. M. Liebrock, “SPH hydrocodes can be stabilized with shape-shifting,” Computers & Mathematics with Applications, vol. 38, pp. 1-16, Sep. 1999, doi: 10.1016/S0898-1221(99)00210-2.

[8] X. Quan, N. K. Birnbaum, M. S. Cowler, and B. I. Gerber, “Numerical Simulations of Structural Deformation under Shock and Impact Loads using a Coupled Multi-Solver Approach,” 5th Asia-Pacific Conference on Shock and Impact Loads on Structures, Hunan, China, Nov. 2003, pp. 152-161.

[9] N. V. Bermeo, M. G. Mendoza, and A. G. Castro, “Semantic Representation of CAD Models Based on the IGES Standard,” Computer Science, vol. 8265, pp. 157-168, Dec. 2001, doi: 10.1007/ 978-3-642-45114-013.

[10] J. Sarkar, “Computer Aided Design: A Conceptual Approach,” CRC Press, December 2014.

[11] C. E. Anderson, “An overview of the theory of hydrocodes,” International journal of impact engineering, 5.1-4, pp 33-59, 1987.

[12] G.-S. Collins, “An Introduction to Hydrocode Modeling,” Applied Modelling and Computation Group, Imperial College London, 2002.

[13] IEEE 100 – “The Authoritative Dictionary Of IEEE Standards Terms,” NYC, NY, USA: IEEE Press, 2000, pp. 574-575, ISBN 0-7381-2601-2.

[14] ANSYS. CAE. *Structures: FEA Simulation*. [Online]. Available from: http://www.ansys.com [retrieved: May, 2017].

[15] Abaqus *Complete Solutions for Realistic Simulation*. [Online]. Available from: http://www.3ds.com/products-services/simulia/products/abaqus/abaquscae/ [retrieved: June, 2017]