Developing an Interface between ANSYS and Abaqus to Simulate Blast Effects on High Security Vehicles

Enrico Hansen, Nicole Ehlers, Arash Ramezani, Hendrik Rothe

Helmut-Schmidt-University
Holstenhofweg 85
22043 Hamburg
Germany
Email: {e.hansen, ehlersn, ramezani, hr}@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 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 analyses; high-performance computing techniques; blast loading; vehicle structures.

I. INTRODUCTION

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 [1].

As the problems are highly non-linear and require information regarding material behavior at ultra-high loading rates, which is 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 [2].

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 costly or difficult to reproduce experimentally. In these numerical simulations, the partial differential equations governing the basic physic 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 low but may require numerous time steps for a complete solution. The solution for the space domain can be obtained utilizing different spatial discretisations, such as Lagrange [3], Euler [4], Arbitrary Lagrange Euler (ALE) [5], or mesh free methods [6]. 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 [7]. Fig. 1 gives a short overview of the solver technologies mentioned above. The crucial factor is the grid that causes different outcomes.

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 [8]. It is often 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

 ISBN: 978-1-61208-501-2

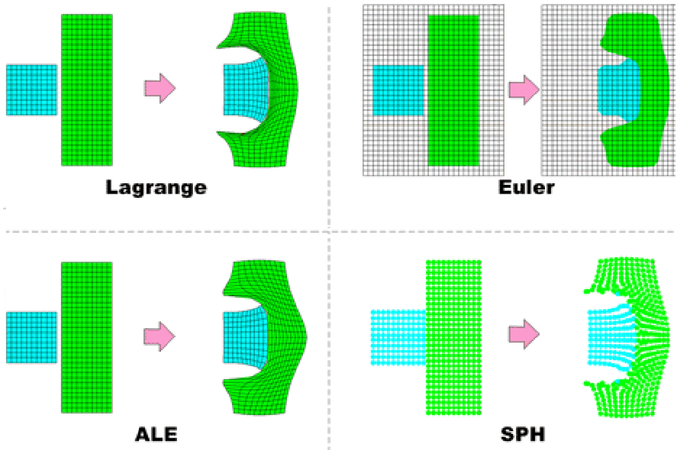

Figure 1. Examples of Lagrange, Euler, ALE, and SPH simulations on an impact problem [1].

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.

This work will present an interface between ANSYS and Abaqus, both leading software suites for finite element analysis and computer-aided engineering. The goal is to develop and demonstrate an efficient FEA/CFD coupling technique for vehicle structures under high-pressure shock compression. The coupling is achieved by an iterative procedure between FEA and CFD calculations using CATIA, ANSYS Autodyn, and Abaqus. ANSYS Autodyn provides shock compression data and the knowledge of shock-wave properties. Abaqus and CATIA (both developed by Dassault Systmes) implement the numerical models with all relevant information. Here, the major challenge is to establish a continuous and fully automatic transfer of blast loadings with high-variation rates from ANSYS Autodyn to Abaqus.

After a brief introduction with a description of the different methods of space discretization in Section I, there is a short section about the concept of this work. Section III describes the experimental set-up and the implementation of our interface. The paper ends with an outlook in Section IV and a concluding paragraph.

II. Concept

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 [8]. It is often 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 computer-aided design (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.

This work will present an interface between ANSYS and Abaqus, both leading software suites for finite element analysis and computer-aided engineering. The goal is to develop and demonstrate an efficient finite element analysis/computational fluid dynamics (FEA/CFD) coupling technique for vehicle structures under high-pressure shock compression. The coupling is achieved by an iterative procedure between FEA and CFD calculations using CATIA, ANSYS Autodyn, and Abaqus. ANSYS Autodyn provides shock compression data and the knowledge of shock-wave properties. Abaqus and CATIA (both developed by Dassault Systmes) implement the numerical models with all relevant information. Here, the major challenge is to establish a continuous and fully automatic transfer of blast loadings with high-variation rates from ANSYS Autodyn to Abaqus.

III. Experimental Section

In computing, an interface is a shared boundary across which 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 [9].

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 [10]. 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 [11]. First, every possibility of transferring the data from ANSYS outputs to Abaqus inputs has to be detected (see Fig. 2).

ANSYS will provide the data by generating a data set for the blast loading (see Figure 3). 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

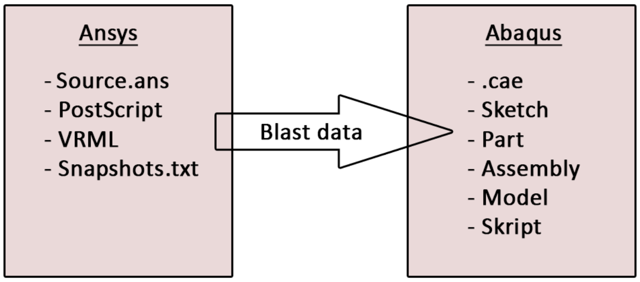

Figure 2. Inputs and outputs for an interface.

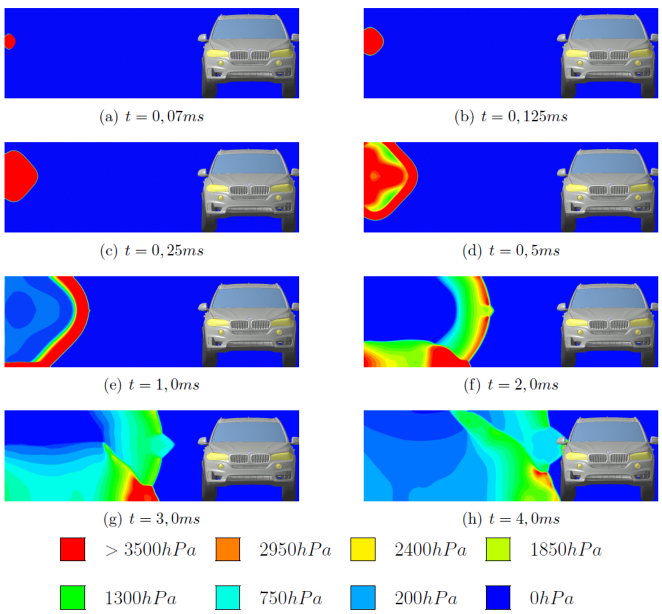

Figure 3. Expansion of blast in ANSYS Autodyn.

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 splitted 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 4). The coupling is realized through an iterative loop between the FEA and CFD simulations, with

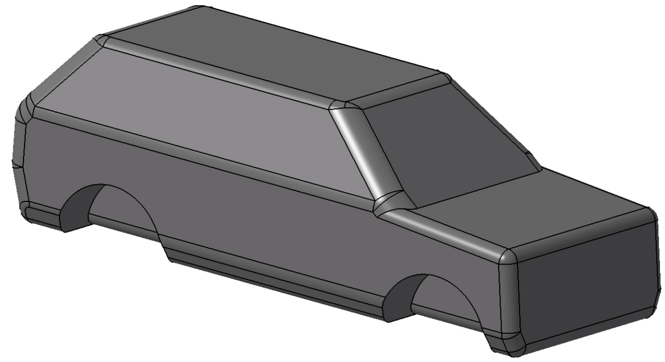

Figure 4. Testing structure in Abaqus.

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 dead lock of the individual CFD simulations, appropriate maximum numbers of iterations are assigned for each CFD model.

IV. 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 "'semi-strong 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.

 ISBN: 978-1-61208-501-2

V. 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] 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

[2] J. Zukas, 'Introduction to Hydrocodes,' Elsevier Science, February 2004.

[3] 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.

[4] 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.

[5] 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.

[6] 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.

[7] 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, pp. 152-161, Nov. 2003.

[8] 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

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

[10] "ANSYS", 2016, URL: http://www.ansys.com [accessed: 2016-04-01].

[11] "Abaqus CAE", 2016, URL: http://www.3ds.com/products-services/simulia/ products/abaqus/abaquscae/ [accessed: 2016-04-05]