

Side Crash Pressure Sensor Prediction: An Improved Corpuscular Particle Method

Tau Tyan, Ben McClain, Kirk Arthurs, Jeffrey Rupp, Mahmoud Ghannam, David Bauch and Todd Clark
Ford Motor Co.

Dilip Bhalsod and Jason Wang
LSTC

ABSTRACT

In an attempt to predict the responses of side crash pressure sensors, the Corpuscular Particle Method (CPM) was adopted and enhanced in this research. Acceleration-based crash sensors have traditionally been used extensively in automotive industry to determine the air bag firing time in the event of a vehicle accident. The prediction of crash pulses obtained from the acceleration-based crash sensors by using computer simulations has been very challenging due to the high frequency and noisy responses obtained from the sensors, especially those installed in crash zones. As a result, the sensor algorithm developments for acceleration-based sensors are largely based on prototype testing. With the latest advancement in the crash sensor technology, side crash pressure sensors have emerged recently and are gradually replacing acceleration-based sensor for side impact applications. Unlike the acceleration-based crash sensors, the data recorded by the side crash pressure sensors exhibits lower frequency and less noisy responses which is more conducive for CAE prediction.

In the attempt to predict the side crash pressure sensor responses, fourteen different benchmark tests were designed and conducted to provide data for model validations. The fourteen benchmark tests can be divided into three sets based on the structure designs. The first set of benchmark tests included a rectangular rigid container with one side being compressed while all other sides were fixed to simulate a piston compression condition. The second set of benchmark tests contained a rigid impactor or a deformable barrier hitting a rectangular steel box with and without a hole. Different speeds were chosen in the second set of benchmark tests to obtain the corresponding pressure responses. The third set of benchmark tests involved a rigid impactor or a deformable barrier hitting a real vehicle side door with different openings. In the baseline door test, the window weather strip and speaker were kept and all holes in door inner were closed to represent a production door. To ensure the robustness of CAE predictions for different door designs, the window weather strip was removed and some holes in the door inner were opened in some of the door benchmark tests. Computer models were created according to the corresponding test conditions.

The CPM method originally developed in LS-DYNA to simulate the deployments of side air bags and side air curtains was adopted and improved in this research to predict the responses of the side crash pressure sensors. One of the main purposes of adopting such method in this project is trying to expand the application of the CPM method to problems that do not involve inflators. With major improvements in the CPM method through this research in the past two years, not only the responses of side crash pressure sensor can be predicted but also the computation time required to complete such simulations has been shortened. The development of the modeling methodology to predict the responses of the side crash pressure sensors will also make it possible to use computer simulations as part of side crash sensor development and results in more robust sensor firing algorithm.

CITATION: Tyan, T., McClain, B., Arthurs, K., Rupp, J. et al., "Side Crash Pressure Sensor Prediction: An Improved Corpuscular Particle Method," *SAE Int. J. Mater. Manf.* 5(2):2012, doi:10.4271/2012-01-0043.

INTRODUCTION

Acceleration-based crash sensors have been used extensively in the automotive industry to detect crash severity and determine the air bag firing time in the event of a vehicle accident. The acceleration-based crash sensors record the deceleration pulses that were created by the crushes of vehicle components. Depending on the locations of the crash sensors and the structure designs, the deceleration pulse recorded by the acceleration-based crash sensors are usually noisy, especially those installed in crush zones and/or obtained from body-on-frame vehicles. The prediction of deceleration pulses by using computer simulations has been very challenging due to the high frequency and noisy responses obtained from the sensors. As a result, the sensor algorithm developments for acceleration-based sensors are largely based on prototype testing.

Due to the latest advancement in the crash sensor technology, pressure sensors have emerged recently and been adopted increasingly in the automotive industry for side crash sensing. The side crash pressure sensors detect the dynamic air pressure change in a door system with cavities. When a door is crushed, the change of door cavity volume creates pressure change which is recorded by the pressure sensors. Unlike the acceleration-based crash sensors, the pressure responses obtained from the side crash pressure sensors exhibit lower frequency and less noisy responses. In addition, the pressure can often produce faster firing times, more robust responses, and less sensitive to test variations. The lower frequency and less fluctuation characteristics have been viewed by CAE analysts as more conducive to CAE prediction. This paper documents the attempt to employ the CPM method in computer simulations to predict the pressure sensor responses in an effort to help reduce prototype testing and improve the robustness of sensor algorithm development.

Wang and Nefske [1, 2] developed a CV (control volume) method in the late 1980s to estimate the time-dependent behavior of the interior pressure of an airbag. The CV method was also called UP (uniform pressure) method in some literature due to the assumption that the pressure and temperature were uniform within a system at any specific time. Due to the assumption, discretization of the fluid field is not needed. Two inflow factors including mass flow rate and temperature over time values are used to model the inflating gas. The two inflow factors are usually obtained through tank tests. Assuming an ideal gas law and an adiabatic process, scalar thermodynamic equations were the main concept of this method. Due to the assumptions, the fluid response is not dependent of location. In the application of occupant protection, the method is usually used to simulate the interaction of a dummy or an occupant with a fully deployed airbag, called the "in-position" test condition. This method is also very suitable for simplified tire and fuel tank models that are needed for crash safety applications. In the condition that the dummy or occupant contacts the airbag in an earlier stage of the deployment process such as the case of OOP (out-of-

position) test condition, more accurate methods are needed to predict the airbag responses.

To solve structure-fluid interaction (SFI) problems, a more advanced approach called ALE method (Arbitrary Lagrangian Eulerian) was developed in the early 1980s. The Lagrangian-Eulerian description and general kinematics theory, developed by Hughes et al.[3] to solve the free surface flows for incompressible viscous fluids, have served as the foundation of the ALE method. Developed for structure-fluid interaction problems, the ALE formulation allows an arbitrary Lagrangian and Eulerian coordinate system to be used to describe the structure and fluid materials. In the ALE calculation, both the structures and the fluid (or gas) volumes need to be discretized so that their local responses can be predicted. As a result, the ALE method has been regarded as one of the most accurate methods for applications involving local predictions of structure-fluid interactions. More detailed information on the ALE theory and computation schemes can be found in Hughes et al. [3], Benson [4], Olovsson [5], and Hirth et al., [6]. With improvements through the years, the ALE method has been successfully applied to simulations of fuel tank sloshing [7, 8, 9], bird strike [10, 11, 12, 13], bulk metal forming [14, 15], air blast [16, 17], gun launch [18], and other scenarios. It also has been adopted to predict the deployments of air bags or air curtains [5, 19, 20, 21, 22, 23, 24], and pressure responses [25, 26, 27] recently.

The corpuscular particle method (CPM) in LS-DYNA [28] was developed in 2006 based on the kinetic molecular theory (KMT) to simulate inflator firing and airbag deployments in a more realistic manner. In this method, the gases are modeled as rigid particles obeying the Newton's laws of mechanics and the ideal gas law. Each particle represents multiple molecules depending on the numbers of particles provided in the system intended to simulate. A reasonable number of particles need to be provided in the system to obtain accurate prediction. A reasonable particle number can be obtained easily through sensitivity studies. The particles behave just like molecules that can store energy as they vibrate and spin. The total translational kinetic energy of the particles is the same as that of the molecules. The theory also ensures the correct ratio between C_p (heat capacity at constant pressure) and C_v (heat capacity at constant volume) as the gases get expanded or compressed. The same properties of the gases are assigned to the particles. As a result, the ideal gas law and kinetic molecular theory are seamlessly connected. The transient gas dynamics and thermodynamics of the system can be predicted. Uniform porous materials, treated in the UP or ALE methods, can be handled the same way in the CPM method. In addition, the vent holes can be modeled physically and allow the particles to leak through. This method will enable more realistic behavior to be predicted locally and globally in the system. Due to the small size of the spherical particle, the CPM method will allow the gas leakage through very small vent holes be captured which is its advantage over the UP and

ALE methods. More importantly, this approach does not have the numerical difficulties, such as hourglass mode, element distortion, complex contact, and small time step, associated with the continuum based approach.

As mentioned, the original objective for the development of the CPM method was to enable the applications related to the inflation of an airbag and its interaction with an occupant, especially the OOP test conditions. Following the initial development of the CPM method, Feng and Coleman [24] adopted both the ALE and CPM approaches to simulate the deployment of airbag through interior trims in 2008. In their study, both the ALE and CPM methods were found to simulate the deployment of curtain airbag through trims successfully. Potential failure mode was identified and fixed in early design stage. Additionally, the advantages and disadvantages of the ALE and the CPM methods were compared in the paper. Similar study was conducted by Lian et al. [29] to compare the ALE and CPM methods in OOP applications. In Lian's study, five benchmark problems were investigated to understand the performance difference between the ALE and the CPM methods. It was reported that the number of the particle will affect the accuracy of the results. Fewer numbers of particles will result in noisier responses. The CPM method consumes less computation time than that of the ALE method. In two of the benchmark problems, the venting flow of the CPM model was found to be slightly higher than that of the ALE method. The CPM method is more robust in terms of dealing with internal components. Overall, both methods predict very comparable results in the five benchmark problems.

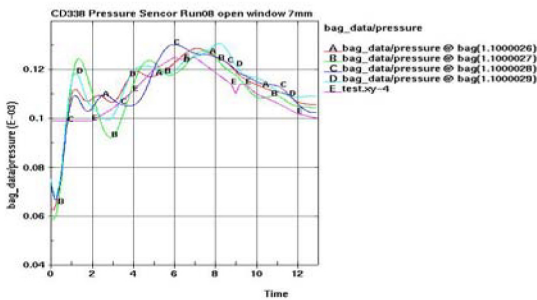
In this research, the authors made the first attempt to adopt the CPM method to predict the pressure responses for side crash sensors. The motivation was trying to expand the application potential of the CPM method to areas that involve in gases existing in the system initially. As a result additional features and enhancements of numerical techniques would need to be developed. To apply the CPM method to simulate problems with initial gases, the initial gases needs to be introduced into the system such as a vehicle side door for this research. The concept may sound simple but it actually created some major technical challenges. The introduction of initial air to the side door system caused fluctuation of system pressure in the initial stage of the door system. The initial pressure fluctuation in the system made the simulations un-robust. In addition, to reach the equilibrium stage of the initial system pressure, the initial air filling would take at least 15 ms (milli-second, 0.001 second) simulation time which became a disadvantage of this method from computation efficiency point of view. Also, due to representing multiple molecules by a single particle as the basic assumption of the CPM method, the simulation often led to dispersion and extremely noisy pressure responses at local pressure sensor locations. The objective of this research was to resolve these issues and make the CPM method more suitable for applications with initial gases in the system.

INITIAL AIR FILLING

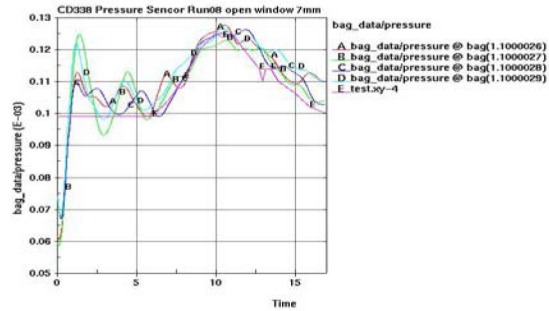
Two options were available in LS-DYNA to consider the air existing in a system initially. The two options used different approaches to model the initial air. The first option with IAIR equal to 1 is really a hybrid approach. In this approach, the initial air is actually modeled by using the CV formulation and only the inflating gas is modeled by using particles. This approach may induce pressure drop when the inflating gas enters the system and the volume of the system starts to increase. As a result, a new option, IAIR equal to 2, was induced in 2010 [30] to overcome the issue. In this approach the initial air is also modeled with particles which will enter the system from the inner surface of the system. By using particles to model both the initial air and the inflating gas, the pressure drop due to system expansion was fixed.

The option, IAIR equal to 2, was adopted in this research initially. As mentioned, this option actually treats the initial air the same as the inflating gas. As a result, it will require some time for the initial air to fill the door system. To achieve equilibrium state, different air filling durations were assumed and tested to understand their influences to the pressure responses. Through this research, a more robust feature was developed by LSTC with an intention to reach the equilibrium state within 2000 cycles of calculation. In normal conditions with reasonable time steps, it only takes several ms (milli-second) simulation time, depending on problems, to reach 2000 cycles. The new feature has reduced the fluctuation of initial system pressure significantly.

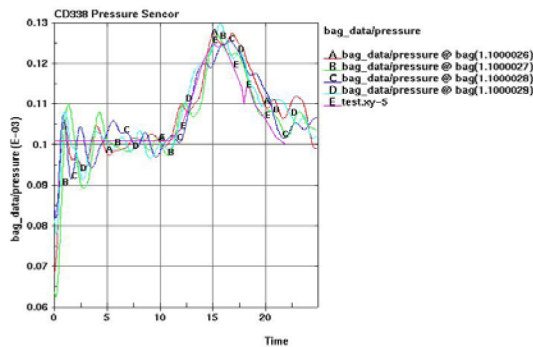
In the case illustrated below is a vehicle side door hit by a rigid impactor. The information regarding the side door impact test condition will be explained in details in latter sections. For illustration purpose, the pressure responses at four different sensor locations (A, B, C, and D in [Figure 1](#)) are being shown and discussed here. It can be seen that the initial system pressure response was very sensitive to the initial air filling duration as shown in [Figure 1](#) for 2 ms, 5 ms, 10 ms, and 15 ms of initial air filling durations. As can be imagined from [Figure 1](#), the initial air particles enter the door cavity from the inner surfaces of the door and are trying to reach an equilibrium state. Before reaching the equilibrium state, the pressure of the door fluctuates around 1 ATM (standard atmosphere, 101.3 KPa, or 1.013 bar). As the air filling duration increases the fluctuation of the pressure fluctuation reduces and the pressure gradually converges to 1ATM. The fluctuation of the system pressure, depending on where it is when the duration ends, determines the initial pressure state of the door. As a result, the initial system pressure can be higher or lower than 1 ATM. The initial pressure slopes are also affected by the air filling duration significantly. The longer the air filling duration is, the more consistent the initial slopes become and the more likely the 1ATM initial pressure can be reached. From many cases studied in this research, 15 ms seemed to be an optimal duration for the initial air filling.



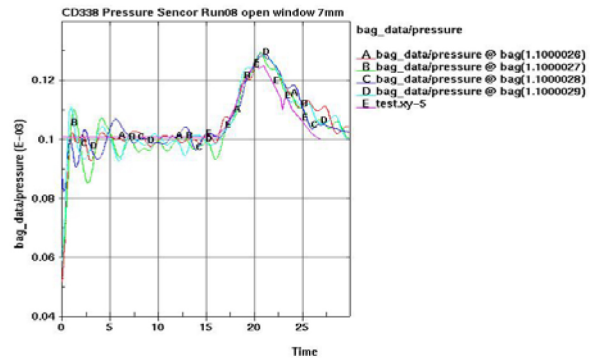
(a) 2 ms air filling duration.



(b) 5 ms air filling duration.



(c) 10 ms air filling duration.



(d) 15 ms air filling duration.

Figure 1. Pressure responses for different air filling durations.

NOISE REDUCTION

A new “CPM_SENSOR” was also developed in this research to reduce the noise of the local pressure response. One of the issues experienced in the early stage of this research was that the responses of the local pressure seemed to be extremely noisy. For illustration purpose, the noisy pressure responses obtained from the side door impact simulation mentioned previously are shown in [Figure 2a](#) (for front sensor location) and [2b](#) (for center sensor location). The plots shown in [Figure 2\(a, b, c, and d\)](#) are unfiltered and with enlarged sensor areas. Simulation results are shown in red solid lines and test in green solid lines. The noise actually muffled the real pressure response due to the large range that was covered by the predicted results. The noise is again caused by representing multiple molecules with only a single particle as mentioned previously. In addition, the pressure sensor is very tiny and much smaller than the contact area that would occur between an airbag and a dummy. As can be imagined, the noise of the pressure response obtained from the pressure sensor would be much more severe than that of the force response from an airbag. The old “CPM_SENSOR” calculated the pressure response based on the number of particles resting on a circular disk defined in the inputdeck (users define the offset and radius of the disk). Even with the radius of the disk being increased artificially still the particles

covered were not sufficient to provide smooth response. After trying other approaches unsuccessfully, a new “CPM_SENSOR” was developed to increase the number of particles covered in the pressure calculation. In the new “CPM_SENSOR”, the disk was extended to be a cylinder in which the thickness, in addition to offset and radius, of the cylinder can be defined. With the new cylindrical “CPM_SENSOR”, the particles covered increased dramatically and the noise was reduced significantly as can be seen in [Figure 2c](#) and [Figure 2d](#), also unfiltered and with enlarged sensor areas.

PARTICLE NUMBER

Similar to the finite element method that the predicted results are sensitive to the element size, the predicted results for the CPM method are sensitive to the particle number. For illustration purpose, the air was modeled with 200 thousand, 2 million, and 3 million particles to simulate the side door impact test mentioned previously. The pressure response predicted for the front sensor location is compared with that of the test and shown in [Figure 3](#) for the three cases. As can be seen, the number of particle affects not only the fluctuation but also the overall shape of the prediction. More particle number makes the predicted curve smoother and improves the accuracy of the prediction. In this case, both the

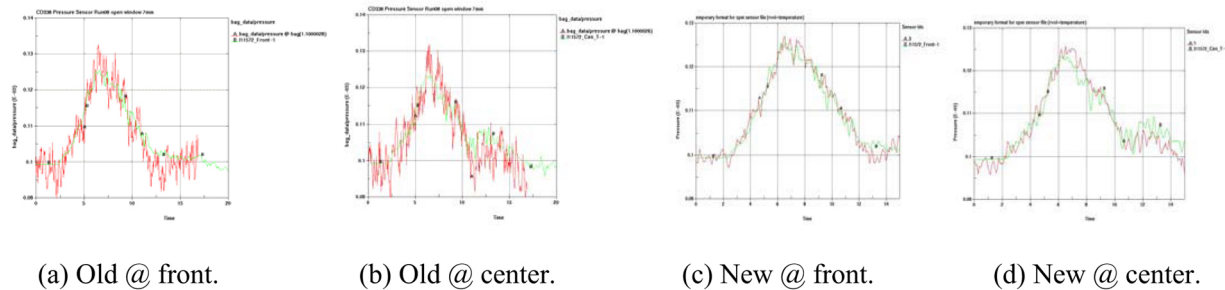


Figure 2. Pressure responses from old and new CPM_SENSOR.

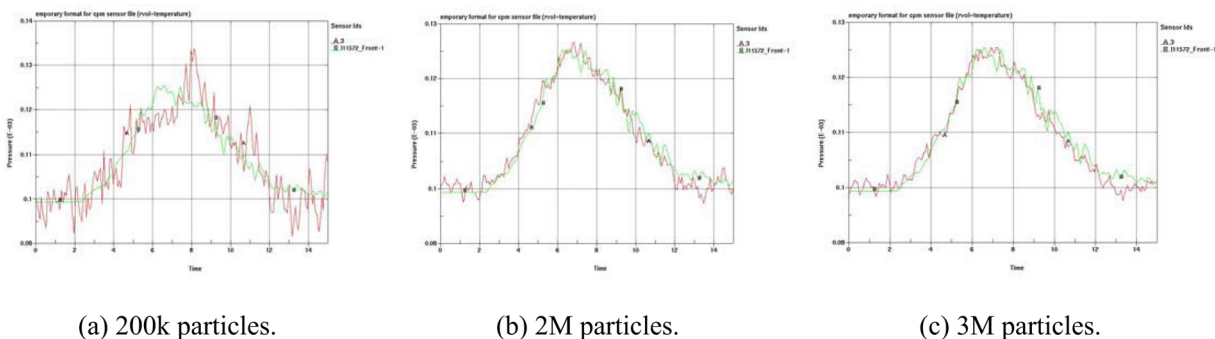


Figure 3. Pressure responses for different particle numbers.

2 million and 3 million particles seem to produce reasonable results with the 3 million particles being slightly better. The pressure response obtained by using 200 thousand particles not only over-predicts the peak value but also fluctuates significantly.

In the early research stage, it was found that when 4 million particles, and above, were used in simulation, leakage would occur and the pressure would not converge to 1 ATM in the initial air filling stage. The problem was found to associate with memory allocation and the bug had been fixed since. Test runs for up to 5.3 million particles had been conducted in this research to confirm the bug fix. In one of the cases studied, the results were also found to be inconsistent that different results were produced even with exactly the same input deck and the same number of processor assigned. The problem also have been fixed with an inclusion of a command card “*CONTROL_MPP_IO_LSTC_REDUCE”. When such incident is encountered, the command card can be used to assure consistent results.

COMPUTATION TIME

The computation time for the CPM method is roughly proportional to the number of particles used in the model. For the side door impact case mentioned above with 200 thousand particles would take one hour, 37 minutes, and 14 seconds (elapsed time) to complete. When the particle was increased to 2 million and 3 million, it took 16 hours, 15 minutes, and 4 seconds, and 26 hours, 36 minutes and 13

seconds to complete respectively. All the three benchmark cases were running with 16 MPP processors and finished in 66,667 cycles. For applications with full vehicle models, more computation time will be needed. Therefore, it is important to optimize against the numerical accuracy and the computation efficiency when choosing the number of particles for simulations.

BENCHMARK TESTS

To help understand the responses of pressure sensors and to obtain reliable data for model developments, a total of fourteen benchmark tests were designed and conducted in this research. The factors considered when designing the benchmark tests included different structures, hole sizes, hole locations, impactors, and impact speeds. The same test data was also used in a study employing the ALE method [27]. The fourteen benchmark tests can be divided into three different sets.

The first set, with only one hypothetical test, was designed to check the numerical accuracy of the ALE method. In this benchmark test, a simple rectangular container was designed to simulate a piston compression condition with which the theoretical solution could be derived. For modeling easiness, the piston and the container were assumed to be rectangular and rigid. The only gas inside the container was the air. No leakage between the piston and container was allowed in this case. For calculation simplicity, an isentropic process is assumed for the piston compression test. The compression speed was assumed to be a constant so

that the volume change of the rectangular container could be calculated and air pressure change relative to the piston displacement could be derived theoretically.

The second set of tests, eight test combinations, involved a rectangular steel box that was slightly larger than the container designed for the piston compression test. The boxes with and without a hole were designed in the second set of benchmark tests to understand the pressure loss due to the existence of a hole. The hole was created on the right and left hand side of the box to investigate the sensitivity of hole location. Two impactors, a rigid impactor and a deformable barrier, hit the rectangular steel box with two different impact speeds. The deformable barrier was obtained by using the center portion of a standard FMVSS (Federal Motor Vehicle Safety Standard) 214 moving deformable barrier (MDB) with 400 mm wide. The rigid impactor was designed with its dimension, 400mm wide, similar to that of the deformable barrier except it was made out of steel so that no plastic deformation would occur during the tests. The second set of tests were conducted by using a VIA dynamic test facility due to the heavy impactor and high impact speeds needed for the tests.

A production vehicle side door was adopted for the third set of tests, five different test conditions. The side doors with different openings were tested to simulate different door designs. The production door with its plastic trim and side mirror being removed was chosen as the baseline door. In the baseline door, the air could escape through the gap between the glass weather seal and the side mirror opening. Then the weather seal surrounding both sides of the glass along with some plastic fasteners on the door inner were also removed to simulation a different door design with more opening. In this case, more air was allowed to escape due to the additional openings created on the door. The same rigid and deformable impactors as those used in the second set of benchmark tests were used in this set of benchmark tests. Three different impact speeds were chosen in this set of tests. The third set of tests were also conducted by using the VIA dynamic test facility with which the side doors could be tested to approximate full vehicle test conditions.

SIMULATION RESULTS

Computer simulations for all fourteen benchmark tests were performed by employing the CPM method through the course of this research. Many development versions were tested in the last 2 years to overcome the different issues and improve the predicted results. The results shown below were obtained by employing the latest CPM method with the newest options and enhancements mentioned previously.

A. PISTON COMPRESSION TEST

As mentioned previously, the piston compression test was designed hypothetically so that the theoretical solution could be derived and no physical test was needed. The container being compressed is shown in [Figure 4](#) with dimensions of

1100×650×150 mm³. The top side of the container (1100×650 mm²) is being compressed downward, along the vertical axis, to simulate a piston moving toward the bottom of the container. The volume of the container is reduced and the gas pressure inside the container rises once the piston starts to move downward. An isentropic process is assumed so that no heat is added to the system and no energy transformations will occur during the entire piston compression process. Since no physical test is involved, this case will allow the numerical error of the CPM method to be estimated.

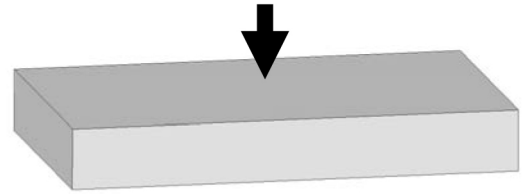


Figure 4. Illustration of a rectangular container being compressed downward by a piston.

Due to the simplicity of the test condition, a simple finite element model was created, also shown in [Figure 4](#), to represent the rectangular container. The finite element model is composed of only 6 shell elements with one shell element representing each side of the container. The total particle number used for this simulation is 200,000. A “boundary prescribed motion” with uniform compression speed was defined to control the motion of the top piston. During the compress process, the top side is moving downward while the bottom side is being fixed which was defined by using “boundary SPC set”. The four vertical sides will become shorter and shorter as the top side moves downward. Since we are unsure how much compression this method can sustain before diverging, the initial simulation was conducted with 96.7% volume change. Assuming air is the only gas inside the container, the air inside the container at different compression ratios is shown in [Figure 5](#). The prediction for the case is compared to that of the theoretical solution in [Figure 6](#) with simulation result plotted in red and theoretical solution in green. As can be seen, the two curves overlap with each other. This case not only proves the accuracy of the CPM method but also demonstrates the robustness of the contact algorithm between the particles and the shell elements.

With the results for 96.7% compression being obtained, the same simulation was extended to 99% to further investigate the limitation of the CPM method. Additional simulations with the particle number reduced to 20,000 and with a different gas (gamma or heat capacity ratio equal to 1.01) were also studied. The computations for all four cases did not exhibit any numerical problems under the extreme compression condition. The comparison of the simulation results, 200,000 (red) and 20,000 (green) particles, with the theoretical solution (navy) for the 99% compression