1 ABSTRACT

The deployment kinematics of an airbag depends significantly on the flow generated by the inflator module. Important design parameters are not only the flow characteristics of the gas generator, but also the geometry of the inflator housing. With the aim of improving understanding of the influence of these parameters on full deployment of the airbag, this study presents a detailed CFD analysis of an idealised inflator housing. The idealised configuration is used to investigate the deflection of the inflator gas by the airbag retainer ring. In particular, the dependence of jet deflection angle and wall pressure distribution on the relative positioning of the gas jet and retainer ring is quantified. A second focus is on the differences of single- and multi-species representations of inflator and ambient gases. All simulations are performed with a new Cartesian cut-cell solver based on Adaptive Mesh Refinement (AMR) and dynamic domain decomposition for Massive Parallel Processing (MPP). In a final step, early deployment of the airbag is simulated and analysed incorporating the results of the analysis of the idealised inflator housing.

2 INTRODUCTION

The aim of this study is to investigate the influence of the gas generator characteristics and the airbag retainer ring on the flow field in a driver airbag inflator housing. A unit with airbag removed and modified for testing purposes is depicted in Figure 1 which illustrates the integration of the unit into the steering wheel and its composition of cylindrical inflator cartridge and square-shaped retainer ring.
The cartridge and the retainer ring form a cavity. The gas generator is of pyrotechnical type, injecting hot gasses into the cavity through a series of radial holes in the cartridge. These gas jets hit the retainer ring and are consequently redirected into the deploying airbag. The flow deflection has a significant influence on the airbag deployment. In order to systematically investigate the influence of the inflator hole positioning on the deflection angle, an idealised flow problem is devised.

The inflator housing is approximated by planar boundaries, which are the result of unrolling the outer surface of the cartridge and introducing a constant width to the retainer ring. Since the focus is on the flow effects in the inflator housing during the first few milliseconds after firing the inflator, the partially deployed airbag is modelled accordingly by a simple rectangular volume with planar rigid boundaries and an outflow hole. The purpose of the outflow hole is to limit the pressure rise in the flow domain during simulation. Similar flow simulations of inflators in rigid, closed or open domains can be found in references [1] and [2], for example.

The hot gas provided by the pyrotechnical generator varies in composition and mass flow rate. In the single-species flow simulations, both the inflator gas and ambient gas are approximated as a “perfect gas” ($\gamma = c_p/c_v = \text{const.}$) with the thermo-physical properties of air. The purpose of the multi-species simulations is to assess the effect of gas species transport on the flow field. Since the computational analysis is limited to the first 2 ms of the deployment simulation, it is assumed that the gas composition and mass flow rate are constant within that timeframe. The inflator gas is composed of N$_2$, CO$_2$ and H$_2$O, whereas the air occupying the flow domain at $t = 0$ comprises N$_2$, O$_2$, CO$_2$ and Ar.

In short, this CFD study focuses on the following topics:

- The effect of the vertical jet position on the flow field inside the airbag and on airbag deployment
- The effect of the gas model (single- versus multi-species) on the flow field in the airbag
- How geometry-based Adaptive Mesh Refinement (AMR), dynamic domain decomposition and MPP can be employed to reduce the simulation runtime

The results include slices of the flow fields for the velocity magnitude. For the multi-species cases, the additional slices of the flow fields for mass fractions are also included. The variation of pressure along the tank walls is evaluated and discussed as this is an important variable of the fluid-structure interaction. Furthermore, the speed-up using Massive Parallel Processing (MPP) is also shown.
The new CFD module of MADYMO is based on a Cartesian cut-cell method [3], [4]. Block-structured Adaptive Mesh Refinement (AMR) is used in combination with Massive Parallel Processing (MPP) to improve the result quality and decrease the runtime of the simulations. Both concepts are illustrated in Figure 2 for a uniform mesh. Local refinement of the mesh is performed automatically and is controlled by the user by means of defining the lower and upper bound on the permissible cell size. The present implementation uses a geometry-based criterion resolving the flow region along the airbag boundary by means of fine cells while keeping coarser cells for the interior flow regions.

![Figure 2: Block-structured mesh (left) and dynamic domain decomposition (right) for a forward-facing step geometry](image)

To optimise the parallel performance of deployment simulations with rapidly changing airbag shapes, the domain decomposition process is complemented by a block-based load balancing strategy. For accuracy reasons, the cell-size ratio of cells belonging to two neighbouring blocks can be at most 2:1. Figure 3 illustrates a cut cell resulting from the intersection of triangular FE surface elements and a Cartesian cell.

![Figure 3: Cut-cell generation along the airbag boundary](image)

The employed Finite-Volume Method solves the unsteady Euler equations for compressible flow of a gas mixture. For each gas species, an individual transport equation for the partial density can be taken into account to model the effect of mixture non-homogeneities. These can have a significant influence, for example, on the early deployment of curtain airbags with elongated and slender flow geometries.
The temporal integration of the Euler equations is explicit and the flux evaluation implemented in the frame of the present study uses the Flux Vector Splitting method by van Leer [5]. Addition of more recent flux evaluation schemes is planned for the future.

4 IDEALISED INFLATOR HOUSING MODEL SETUP

In the following section, the setup of the idealised CFD model inflator housing is explained.

4.1 FLOW DOMAIN DEFINITION

The 3-D model used in the test is an idealised inflator housing. It is a part of the complete geometry of the inflator cartridge and the retainer ring as shown in Figure 4(a). The airbag is represented by rigid boundaries. To approximate the gas flow in a deploying airbag, a subsonic outflow (outlet) boundary is integrated in the top part of the right wall of the inflator housing from $y = 122$ mm to $y = 150$ mm. The dimensions are summarised in Figure 4(b).

![Figure 4](image)

Figure 4: Side view and basic dimensions (a) and perspective view of the idealised flow domain (b)

The inflator jet axis is situated 3 mm from the bottom wall of the cavity. The inflator cartridge penetrates 18 mm into the flow domain; the step formed by the retainer ring is 8 mm high. The distance between the retainer ring and the cartridge is 10 mm. Mesh refinement is applied along the boundaries.

In order to introduce any coarse cells in the interior of the flow domain, the quasi 2-D inflator housing geometry is duplicated 5 times, increasing the total thickness in z-direction to 100 mm. The flow simulation results presented later are based on the centre plane at $z = 50$ mm. The root cell size is set at $\Delta x^0 = 256$ mm. Division of the root cell in 2 halves 8 times in each direction results in a minimum cell size of $256/2^8 = 1$ mm. This corresponds to a level 8 mesh. Figure 5 shows the uniform mesh and the adaptive mesh at refinement level 8. The minimum cells for the adaptive mesh are located near the boundary (walls).
4.2 INITIAL AND BOUNDARY CONDITIONS FOR SINGLE-SPECIES SIMULATION

This section describes the initial and boundary conditions of the single-species test cases. The fluid used in the single-species simulations is a thermally perfect gas with constant specific heats and $\gamma = 1.4$.

The initial conditions in the flow domain are set to the following ambient values:

$\rho = 1.2252 \, \text{kg/m}^3$
$T = 288.15 \, \text{K}$
$u = v = w = 0 \, \text{m/s}$

The pressure is not directly set in the input, but rather indirectly by the density and the temperature. The pressure can be calculated from the perfect gas law with the aforementioned values resulting in

$p = \rho \times T \times \frac{R}{M} = 1.2252 \times 288.15 \times 8.314472 / 0.02897028 = 1013.2 \, \text{kPa},$

where $R$ and $M$ are the universal gas constant and Molar mass of the “perfect gas” air.

The pressure at the outflow boundary is set to an ambient pressure $p = 1013.2 \, \text{kPa}$. The inflator is idealised by using a constant mass flow rate. The inflator gas is the same as the gas in the inflator.
housing, only its temperature is higher. The inflator assumes a sonic velocity at the inflow face. The following values are used for the inflator:

\[ \rho = 35.474 \, \text{kg/m}^3 \]
\[ T = 695.302 \, \text{K} \]
\[ u = 528.55 \, \text{m/s} \]

The pressure at the inflow boundary is again calculated from the perfect gas law:

\[ p = \rho \times T \times \frac{R}{M} = 35.474 \times 695.302 \times 8.314472 / 0.02897028 = 7.1 \, \text{MPa} \]

This pressure is much higher than the initial pressure in the inflator housing, the jet is underexpanded. The pressure ratio is such that acceleration from sonic to supersonic flow can be expected in the cavity.

### 4.3 Initial and Boundary Conditions for Multi-Species Simulation

The purpose of the multi-species simulations is to assess the impact of multi-species gas composition. The multi-species model is based on a gas composition consisting of five species. Four gas species constitute the gas initially situated in the inflator housing and three species are part of the inflator gas. The composition of the multi-species fluids is approximately that of the atmosphere. The mass fractions of the gas composition are shown in Table 1 below.

<table>
<thead>
<tr>
<th>Gas</th>
<th>Ambient air mixture</th>
<th>Inflator gas mixture</th>
</tr>
</thead>
<tbody>
<tr>
<td>N(_2)</td>
<td>0.78084</td>
<td>0.4</td>
</tr>
<tr>
<td>O(_2)</td>
<td>0.20946</td>
<td>0.0</td>
</tr>
<tr>
<td>CO(_2)</td>
<td>0.00033</td>
<td>0.3</td>
</tr>
<tr>
<td>AR</td>
<td>0.00937</td>
<td>0.0</td>
</tr>
<tr>
<td>H(_2)O</td>
<td>0.0</td>
<td>0.3</td>
</tr>
</tbody>
</table>

The new input values are calculated by assuming a sonic flow in the inflator with the same mass flow rate and temperature as the single-species case. The new values for the multi-species are shown in Table 2.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Unit</th>
<th>Single-species gas</th>
<th>Multi-species gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Ambient air</td>
<td>(\rho) [kg/m(^3)]</td>
<td>1.225</td>
<td>1.220</td>
</tr>
<tr>
<td>- Inflator gas</td>
<td>(\rho) [kg/m^3]</td>
<td>35.474</td>
<td>35.366</td>
</tr>
<tr>
<td></td>
<td>T [K]</td>
<td>695.302</td>
<td>695.302</td>
</tr>
<tr>
<td></td>
<td>u [m/s]</td>
<td>528.550</td>
<td>530.159</td>
</tr>
</tbody>
</table>

### 4.4 Computational Parameters

The simulations were performed on a Linux cluster with 4 processors in MPP mode; for hardware specifications see Table 3. The maximum number of processors was utilised.

Start time is \(t = 0\) ms and the end time of the simulations is \(t = 2\) ms. The CFL number is set to 0.5. The output is limited to the end time; no intermediate time steps are recorded.
Table 3: Hardware characteristics
Number of nodes : 4
Processor type : Intel Itanium 2
RAM per node : 900 Mb
CPU speed : 1400 MHz
Operating system : Linux RHEL3/SGI Propack3

5 FLOW SIMULATION RESULTS IDEALISED INFLATOR HOUSING

The CFD simulation results visualised at the slice at the centre plane ($z = 50 \text{ mm}$) of the 3-D inflator housing. Due to a lack of experimental results for this test case, a solution was created on a uniform mesh. This solution will serve as a reference solution for the simulations on the adapted meshes. In this section we will present both the results on both types of meshes before crossing over to the parameter study.

5.1 SINGLE-SPECIES

Here the single-species fluid flow results are shown for different Euler meshes (uniform and adaptive) and for the variation of the inflator position.

5.1.1 Uniform Versus Adaptive Mesh

The contour plot results of the single-species uniform and adaptive flow fields for velocity magnitude are presented in Figure 7.
The pressure in the boundary cells adjacent to the wall is extracted along the surface co-ordinate and plotted in Figure 8 for uniform and adaptive level 8 simulation.

The surface co-ordinate \( s \) starts in the top left-hand corner of the inflator housing, proceeds to the top right-hand corner and then to the bottom right-hand corner (see Figure 4(b)). Figure 9 plots the wall pressure in the centre plane for the adaptive meshes at refinement levels 8, 9 and 10.

5.1.2 Parameter Study Jet Location Variation

The basic flow feature under investigation here is the deflection of the inflator jet by the retainer ring. Therefore, the inflator position in y-direction was varied from \( y = 3 \text{ mm} \) to \( y = 15 \text{ mm} \). The contour plots of the flow field results for velocity magnitude for the different positions of the inflator are shown in Figure 10.
Figure 10: Single-species simulation result for different vertical jet locations – adaptive mesh level 8

(a) Jet height 3 mm  (b) Jet height 6 mm  
(c) Jet height 8 mm  (d) Jet height 15 mm

Figure 11 shows the wall pressure. The pressure is extracted along the surface co-ordinate and plotted for the different inflator position simulation.

Figure 11: Pressure plot along the wall in the centre plane ($z = 50$ mm); adaptive mesh for the different jet heights

5.2 MULTI-SPECIES

The purpose of single- and multi-species simulations is to assess the impact of multi-species gas composition computed here on a uniform and a level-8 adaptive grid. The mass fraction distribution of the H$_2$O component and the velocity magnitude in the centre plane are shown in Figure 12.
The wall pressure in the centre plane is extracted along the surface co-ordinate and plotted in Figure 13 for adaptive single- and multi-species simulation.

![Wall Pressure Plot](image)

**Figure 13:** Wall Pressure plot along the wall in the centre plane \((z = 50 \text{ mm})\) on an adaptive mesh for single- and multi-species simulation

6 RESULTS OF DISCUSSION ON IDEALISED INFLATOR HOUSING

6.1 SINGLE-SPECIES
When the flow fields for the adaptive and uniform meshes are compared for refinement level 8 (Figure 7(a) and Figure 7(c)), it can be seen that the general flow features are the same for both mesh types. However there are some differences. It appears that the coarser blocks in the centre of the adaptive mesh have a numerical diffusion effect. The coarser mesh in the centre of the flow domain significantly increases the numerical diffusion. The mesh at the front and back wall is refined to $l_{\text{max}}$, resulting in an artificial asymmetry in the flow field. The gas plume that is clearly defined in the uniform meshes (Figure 7(a)) is broader and less defined in the adaptive mesh. This also has an effect on the pressure distribution along the wall (Figure 8). The pressure peak on the top wall of the adaptive mesh is broader and more smeared out than that of the uniform mesh. Furthermore, the gradient of the pressure of the adaptive mesh is smaller and changes in the pressure along the wall are more gradual. The pressure on the right wall is higher for the adaptive mesh. This is due to the lower speed of the flow there compared to the uniform mesh. The flow on the uniform mesh is more focused owing to less numerical diffusion. There are also similarities between the simulations on adaptive and uniform meshes. Basic features of the flow are the same. Both have a stagnation point at the top wall, high pressure, from which the flow accelerates towards the outflow boundary at $0.150 \leq s [m] \geq 0.172$, and the pressure drops as anticipated according to Bernoulli. The direction of flow and the stagnation point can be seen in Figure 14.

![Figure 14: Stream traces on a velocity magnitude contour plot of a uniform mesh, single-species](image)

The pressure at the outflow boundary approaches the pressure set at the wall, the difference is due to position of the pressure measurement, the values are cell-centred, cell centres of larger cells are further away from the wall. It is taken in the cell next to the wall and not at the wall itself. The flow accelerates to supersonic velocities in the cavity after being injected by the inflator (see Figure 15).
This is a gasdynamic phenomenon due to the geometry and pressure difference of the inflow boundary. The flow decelerates to subsonic velocities after leaving the cavity and then accelerates again as it approaches the outflow boundary, but stays subsonic, as can be seen in Figure 7. The comparison of the pressure along the wall of the adaptive meshes, see Figure 9, indicates that the pressure decreases, i.e. approaches the pressure of the uniform mesh, with increasing refinement. However the pressure peak is still smeared out. The differences in the pressure at the outflow boundary are due to the distance from the wall, on account of to cell-centred values. The lowest refinement level has the largest distance of the cell centre to the wall, and thus a higher pressure at the outflow region.

The runtime of the adaptive mesh with minimum cell size 0.001 m is approximately three times less than the runtime of the corresponding uniform mesh, while the number of active cells is approximately half. The reduction of the solution quality caused by the coarser mesh is compensated by a large reduction in calculation time. AMR would deliver far better results for this application solution driven. Application of AMR along the boundaries is sub-optimal since the coarser cells are at a position where smaller cells are needed, and vice versa.

6.2 **MULTI-SPECIES**

When the plots in Figure 12 are compared, it can be seen that the coarser blocks at the centre of the adaptive mesh generate a diffusive effect. This is best seen in the plot for the H₂O species, since it is only present in the inflator gas. The plume is less defined and broader for the simulation on the adaptive mesh. Due to the coarser blocks, some small flow features can not be seen, especially near the right wall. If the pressure along the wall of the single species is compared to the multi-species case, see Figure 13, it is apparent that there are both similarities and differences. The difference is the consistently higher pressure along the wall for the multi-species test. This is caused by the difference of the thermodynamic variables and the higher speed of the inflow boundary of the multi-species case. The similarity lies in the shape of the pressure distribution: it is almost the same. The pressure gradient along the wall is the same everywhere except close to the outflow region, where both the pressure of the single species and the pressure of the multi-species converge to the outflow pressure. The single-species approach can be valid if flow features are of interest; if absolute values of flow variables are of
interest, it might be beneficial to use the multi-species approach, although this involves extra calculation time. The multi-species test took approximately twice as long as the single-species test.

### 6.3 PARAMETER STUDY JET LOCATION VARIATION

The contour plots of the velocity (see Figure 10) show that the deflection angle decreases with an increasing height of the inflator jet hole. When the jet height reaches the height of the retainer ring, $y = 8 \text{ mm}$, the jet is no longer deflected, but passes the ring unobstructed and impinges on the right wall. The jet has the highest velocities around the inflow area (red area in the contour plots). It can be seen that when the high velocity part of the jet is completely in the cavity ($y = 3 \text{ mm}$), the jet is deflected vertically, but when the part is outside the cavity it forces the deflection angle down until there is no deflection anymore. The part of the flow that comes out of the cavity is rotated down by the part of the jet outside the cavity. The vertical position of the jet in the cavity therefore has a significant influence on the flow field (see Figure 10), as would be expected. The turning or deflection angle depends on the position of the jet in the cavity and is measured here from the contour plots of the velocity magnitude. It is defined as the turning of the centreline of the plume from the horizontal. The approximate decreasing angles with increasing height of the inflator hole are presented in Table 4.

<table>
<thead>
<tr>
<th>Height [mm]</th>
<th>Turn angle of the plume [°]</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>90</td>
</tr>
<tr>
<td>6</td>
<td>35</td>
</tr>
<tr>
<td>8</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>0</td>
</tr>
</tbody>
</table>

As can be seen in Figure 11, the pressure distribution varies with the inflator position. The deflection of the inflowing gas at the retainer ring causes a higher pressure at the fixed upper wall, whereas the unobstructed flow leads to a pressure peak at the right wall.

### 6.4 PERFORMANCE ASSESSMENT OF MPP

The MPP implementation of domain decomposition can be used to reduce the overall runtime of simulations. To assess the performance achievement for the inflator housing test case, a study is performed on a Linux cluster with four processors described in Table 3. The same input file was used on one, two, three and four processor runs. The test configuration employed is the single-species adaptive level 8 mesh. It has just over 700,000 active cells. The results of the tests are presented in Table 5.

<table>
<thead>
<tr>
<th>Number of processors</th>
<th>Runtime T [s]</th>
<th>Runtime T [h : m : s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>19907.9</td>
<td>5:31:47.9</td>
</tr>
<tr>
<td>2</td>
<td>11626.7</td>
<td>3:13:46.7</td>
</tr>
<tr>
<td>3</td>
<td>7671.2</td>
<td>2:07:51.2</td>
</tr>
<tr>
<td>4</td>
<td>5245.7</td>
<td>1:27:25.7</td>
</tr>
</tbody>
</table>

The speed-up and efficiency of the MPP process can be calculated using the data from Table 5 derived from the following general equations:

$$\text{Speed-up} = \frac{T_{1 \text{ proc}}}{T_{N \text{ proc}}},$$

$$\text{Efficiency} = \frac{\text{Speed-up}}{N_{\text{proc}}}.$$
where T is the runtime from Table 5. The calculated values are presented in Figure 16.

![Figure 16: Speed-up (top), dashed line is ideal speed-up, and efficiency (bottom) on a Linux cluster (Table 3)](image)

From Figure 16 it can be seen that the speed-up follows the ideal speed-up very closely, i.e. the dashed line. The results indicate that the speed-up can be further increased for more processors. The results also show that efficiency increases with the number of processors.

### 7 AIRBAG APPLICATION EXAMPLE

The previous sections provide information on the flow field and the pressure distribution in idealised inflator housing example scenarios with fixed walls. Since the deployment of a real airbag is also dependent on the pressure distribution, the fabric cushion is pressurised in reality after the inflator gas release and fluid-structure interaction causes the airbag deployment. Within this section, the advanced CFD airbag simulation method is illustrated by means of a numerical driver airbag example designed to cover the FMVSS 208 out-of-position (OoP) load case. First the numerical model is explained briefly followed by the simulation and test result discussion of the static free inflation of a non-folded driver airbag cushion.

#### 7.1 AIRBAG MODEL SETUP

This involves an advanced driver airbag model that features a cushion geometry which is initially separated into two chambers by internal tethers. Targeting less aggressive deployment, the internal chamber prevents inflation towards the occupant. The chambered cushion leads to a more lateral deployment in the plane parallel to the driver occupant’s head. The functionality of this advanced airbag technology is explained in more detail in [6]. Apart from its sophisticated cushion design, the cushion itself is fixed in the airbag module as introduced in section 2. Figure 17 shows the numerical model of the bag holder with mounted retainer ring and cylindrical gas generator with removed airbag cushion.
7.2 NON-FOLDED AIRBAG INFLATION RESULTS

To study the initial gas flow into the non-folded airbag, the cushion, inflator cartridge and retainer ring were mounted on a wooden mounting plate. The bag holder was not taken into account. In Figure 18, the test response of the non-folded airbag (with internal gas flow control chamber) deployment is compared with the simulation time history for the initial 12 ms in 2 ms time steps (left-hand and middle column).
The third column (right) shows the simulation time history of the non-folded cushion model with removed internal chamber structure to compare its deployment characteristics with the initially chambered cushion design.

### 7.3 SIMULATION RESULT DISCUSSION

The result in Figure 18 shows that for the first 6 ms, the released gas flows vertically into both cushions redirected at the retainer ring. At 8 ms the chambered cushion prevents further vertical deployment of the cushion and the gas starts to flow into the outer chamber through designed holes in a radial direction. The gas flow redirection at the internal chamber walls is reproduced by the simulation model and corresponds to the test response observation. To further explain the functionality of the advanced chambered cushion, Figure 19 plots the initial flow of the released gas at different inflator jet heights (left: 3 mm; right: 15 mm) for the first 6 ms visualising the gas velocity magnitude.
As observed for the idealised inflator housing volume in section 4.1, the extreme scenarios of the jet heights lead to significant differences of the initial chambered cushion inflation. At 3 mm jet height, the inflowing gas is redirected at the retainer ring into a vertical direction, which causes the internal chamber to inflate upwards (compare 2 ms state). At 15 mm jet height the inflowing gas is not affected by the retainer ring and therefore the main flow direction is horizontal as expected from the idealised inflator housing study. But there are also similarities in the inflation mode of both jet height results. At 6 ms the gas starts to flow through the designed holes into the outer chamber for both jet height scenarios. For the jet height of 3 mm the inner chamber can be understood as a mixing volume with the gas flow redirection through the designed holes in lateral direction similar to the 15 mm jet height deployment characteristics. Finally it is noted that the airbag simulation examples presented are based on a uniform CFD Euler mesh.

8 CONCLUSION

The computational analysis of the idealised inflator housing configuration shows that the vertical position of the holes in the gas generator cartridge has a decisive influence on the flow from the inflator housing and, accordingly, on the pressure distribution along the inner surface of the bag. This is mainly due to different angles of deflection of the gas jets by the retainer ring. As a consequence, the analysis suggests that airbag deployment can be adjusted by varying the vertical position of the inflator holes. This is confirmed by computational analysis of the early deployment of a driver airbag, which shows the influence of the jet deflection angle on the initial inflation of the inner chamber. It also highlights the purpose of this chamber as a mixing volume, which basically annihilates the influence of the jet deflection angle on deployment of the outer chamber. The flow analysis of the advanced initially chambered driver airbag provides a thorough insight into the physics of the airbag inflation process. Although not addressed within the framework of the paper, the CFD deployment analysis as part of a restraint system simulation allows prediction of the effectiveness of any kind of airbag (folding pattern) designs in order to mitigate occupant (dummy) injury values caused by the thermodynamic inflator energy transfer during early inflation through occupant (dummy) airbag contact interaction (OoP load case).

The study clearly reveals how Adaptive Mesh Refinement (AMR) can be employed for optimum usage of the available computational resources. For user-defined mesh size or run time limits, AMR redistributes the available gas flow cells according to built-in refinement rules to ensure increased
accuracy of the solution. The present study employs geometry-based mesh refinement to bypass the computational overhead of a uniform mesh approach. Solution-based AMR is not discussed within the framework of this paper in which the mesh is refined locally according to solution criteria (e.g. gradient of a flow variable etc.). Runtimes can be further reduced by Massive Parallel Processing (MPP) and the achieved speed-up is shown to be substantial. The multi-species gas simulation indicates small differences compared to single-species gas simulations. Although the basic flow features are the same, the pressure distribution along the idealised inner surface of the airbag is slightly different. However, the increased accuracy of a multi-species simulation has a price since each species requires the solution of an additional scalar transport equation.

ACKNOWLEDGEMENTS

The authors would like to thank Bart Wildenborg for the inflator housing simulations, Cindy Charlot and Peter Ritmeijer for the full deployment simulations.

LIST OF REFERENCES