Validation of the Simulation Methodology for a Mobile Explosive Containment Vessel

David Karlsson
DYNAmore Nordic AB, Sweden

Abstract

A Mobile Explosive Containment Vessel (MECV) is a chamber for protection against effects caused by explosions and is used to safely secure, contain, transport, store or test explosive materials. The MECV has been tested for a charge equivalent to 8 kg of TNT and strain levels at several positions were measured. These test data were used for comparison and validation of two simulation techniques and if necessary improve the simulation methodology.

The first technique uses a separate 2D-axisymmetric MMALE simulation for the explosive blast load calculation and it showed good agreement to the test. In this case, an axisymmetric blast simulation is first made and the pressure is recorded at the fixed boundary. Then an in-house developed program is used to map the blast load to the 3D structure simulation. The second, much more compute intensive technique, is to do a full 3D coupled MMALE simulation of the blast and structure. The second technique lead initially to lower strain levels compared to the test and a more detailed parameter study had to be performed to improve the simulation results.

As conclusion, we now have two validated simulation techniques and procedures to make realistic explosive simulations of containment vessels.

Introduction

The objective with this project is to validate two previously used simulation techniques against real tests of a Mobile Explosive Containment Vessel and if necessary improve the simulation methodology.

Mobile Explosive Containment Vessel (MECV)

The project is carried out for Dynasafe Protection Systems AB in Sweden which is a company that offers a wide variety of products in explosion protection and bomb disposal technologies. A Mobile Explosive Containment Vessel (MECV) is a chamber for protection against effects caused by explosions and is used to safely secure, contain, transport, store or test explosive materials. The MECV has been tested for an 8 kg TNT equivalent and strain levels at several positions were measured [1]. A typical MECV and the test setup are shown in Figure 1.

The MECV have an outer length of 1 500 mm, an outer diameter of 1 300 mm, a wall thickness of 30 mm and weight about 2 800 kg. It consists of a vessel, a cover and a locking ring. The chamber is opened or closed by rotating the locking ring 15° and sliding the cover end.

Figure 1: A typical MECV to the left and the test setup to the right.

Software

The analyses are performed with the nonlinear dynamic finite element software LS-DYNA® R6.0.0 [2]. The model was built for a previous project a few years ago with ANSA v13.1.2 [3].

FE-model

From the detailed drawing the most relevant load carrying parts are identified and used for the analysis, see Figure 2. This includes the vessel, the cover, the locking ring and some outer attachments. Details that are removed are pressure pipes, sealing rings and a fragment shield on the inside. The FE-model will be meshed with mainly hexa elements with a size of 6 mm in the locking mechanism and 10 mm in the rest of the chamber. This will result in about one million fully integrated volume elements which is a good compromise between simulation accuracy and simulation time.

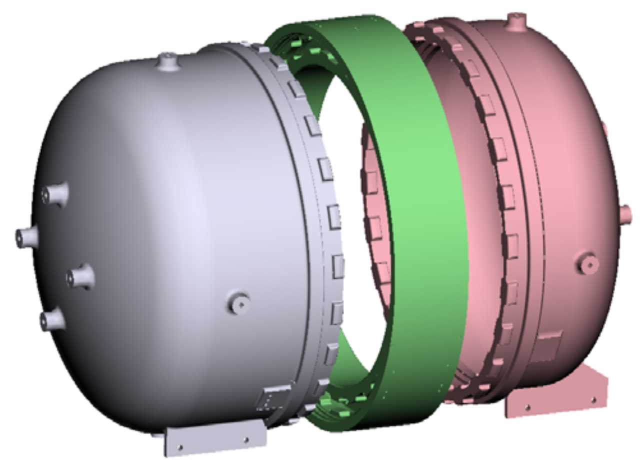


Figure 2: The most relevant load carrying parts.

The locking mechanism consists of two flanges and a locking ring with in- and outside teeth. During the creation of the FE-model a section of 15° is used due to symmetry. The section of the locking mechanism is finally mirrored and rotated to get a complete model. A cut through the meshed locking mechanism is shown in Figure 3.

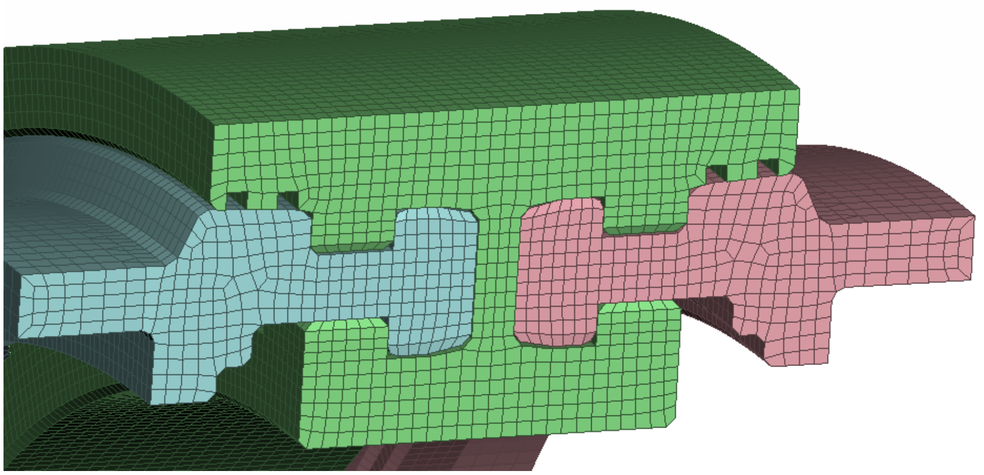


Figure 3: Cut through the meshed locking mechanism.

The complete model consists of 800 000 volume elements with 99.6 % hexas and 0.4 % pentas, illustrated in Figure 4 and 5. Two additional attachments have been added at the bottom, the yellow parts in the images below. These attachments were created directly in ANSA just from photos of the test setup.

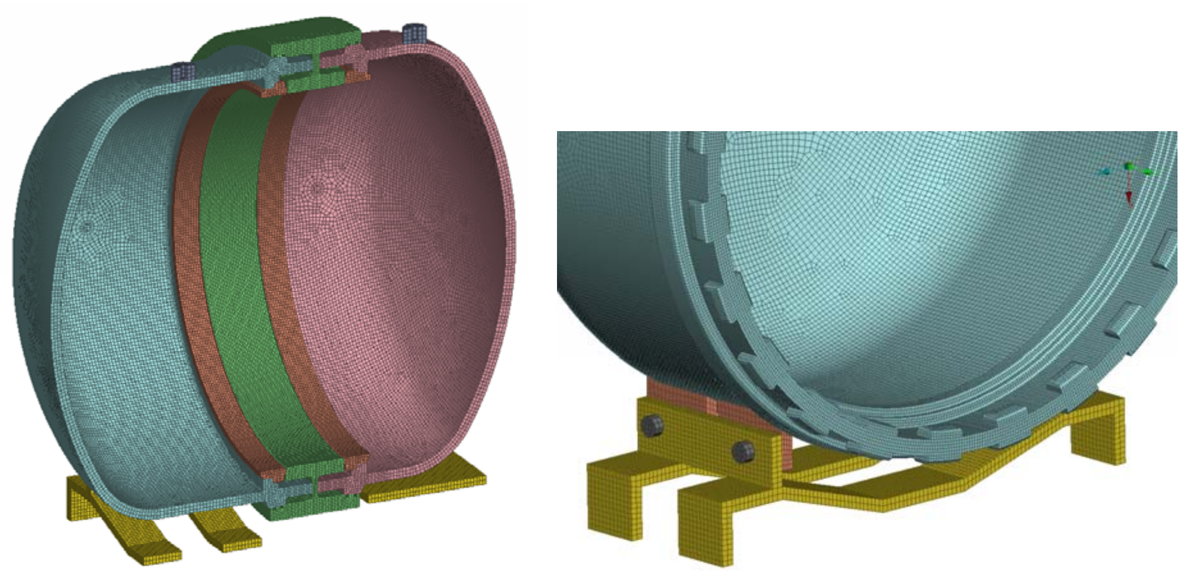


Figure 4: The mesh of the MECV model.

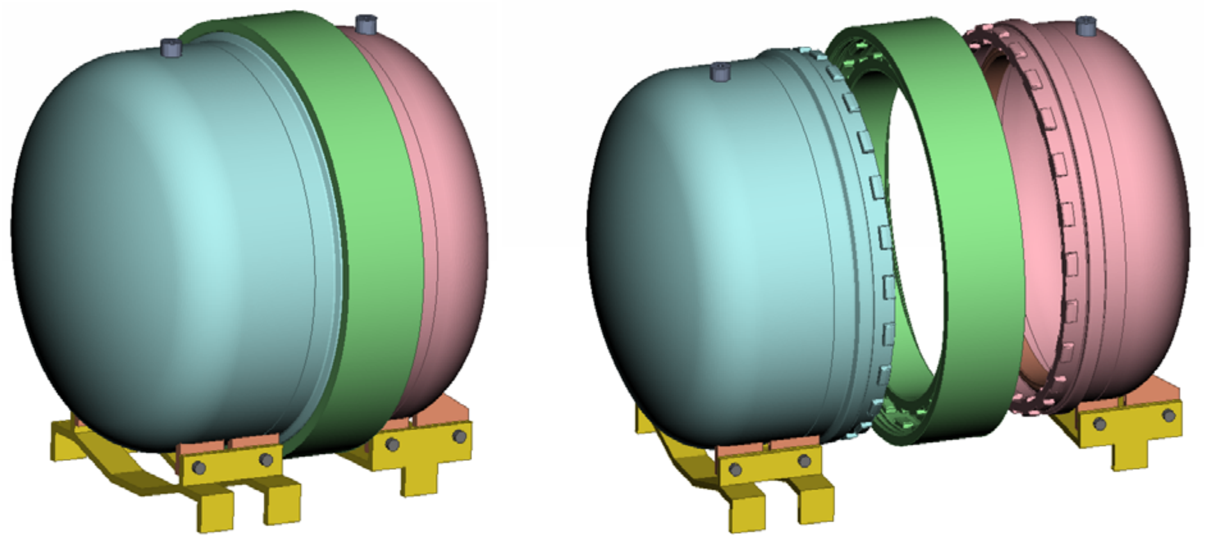


Figure 5: The complete MECV model.

Simulation setup

The load case is a spherical charge of 8.0 kg TNT equivalent placed in the center of the chamber, see Figure 6. The charge is initiated in the center. The outer attachments are constrained at the bottom feet with a fixed boundary condition.

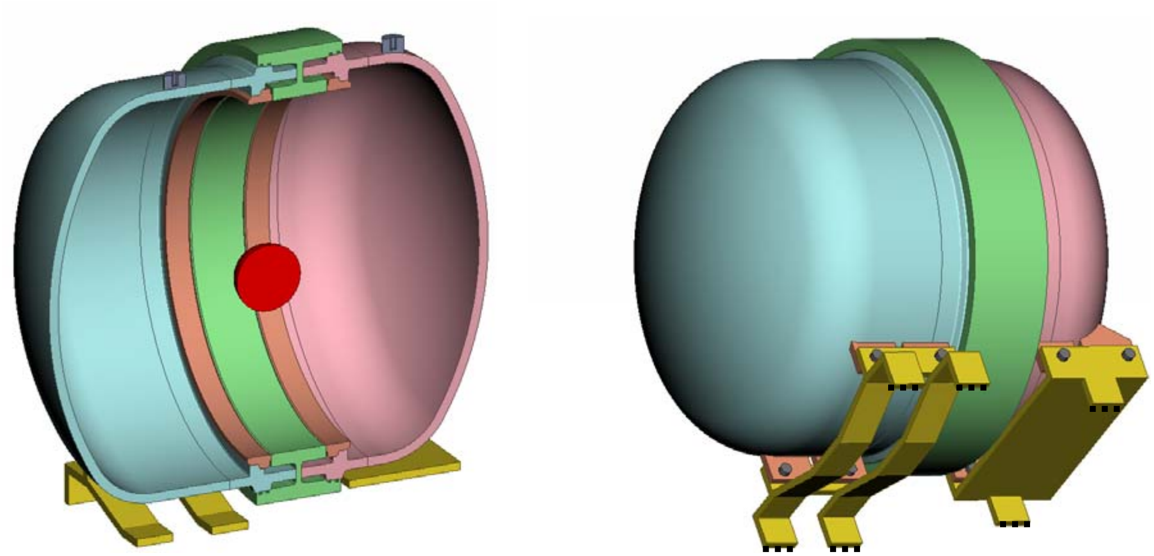
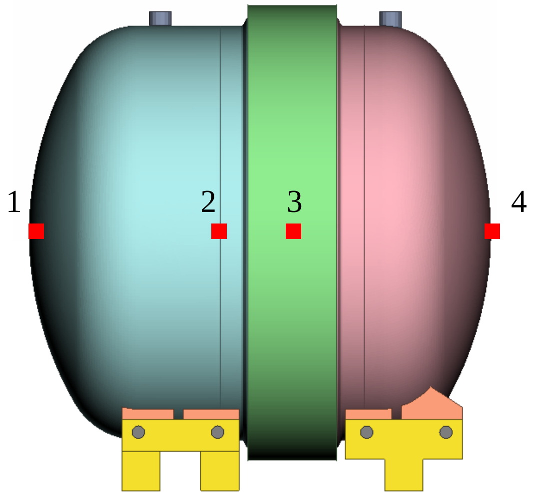


Figure 6: The charge position to the left and the fixed boundary conditions to the right.

The MECV is made of a high strength steel with a yield strength above 700 MPa. The material damping is set to 0.5-1.0 % of critical in the frequency range 100-3000 Hz. An elasto-plastic material model with linear hardening is used. The Jones-Wilkens-Lee-Baker (JWLB) equation of state is used for the fluid blast load analysis to describe the high pressure regime produced by the detonation. The air is modelled as an ideal gas.

Shell elements are used as strain gauges to capture the strain on the surface of the structure. The strain measurments are done in a similar way as the test [1] with respect to gauge location, sampling frequency and signal filter. The four strain gauges are placed as illustrated in Figure 7.



1. Vessel end
2. Vessel cylinder
3. Locking ring
4. Cover end

Figure 7: Location of the strain gauges.

Blast load technique

In the first technique "Axisymmetric mapped to 3D" the explosive blast load is done in a separate axisymmetric fluid simulation using Multi Material Arbitrary Lagrangian Eulerian (MMALE). This includes air and explosive interaction and blast wave reflections. The fluid element length is 5 mm and the fluid model consists of 31 000 elements. The inside of the chamber is modeled and the boundaries are fixed. The pressure as function of time is recorded at the fixed boundary with a specified interval, see the red marks in Figure 8. Then an in-house developed script is used to map the blast load to the 3D solid structure simulation. The output of the script is a file with load segments at the inside of the 3D structure model that refer to the load curves from the axisymmetric fluid simulation, see Figure 9. Each load segment will have an interpolated contribution from two load curves.

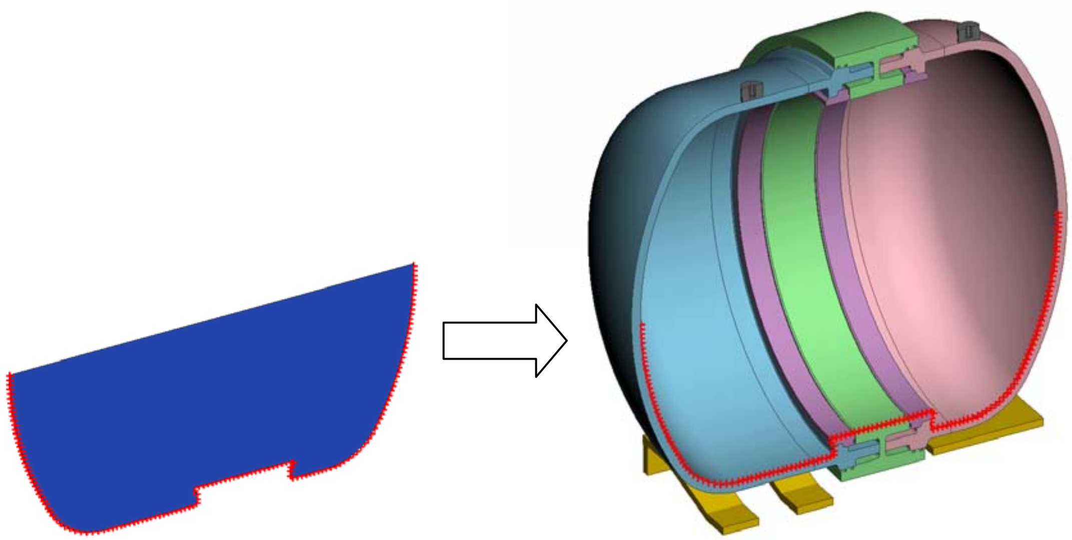


Figure 8: Axisymmetric mapped to 3D.

```
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4
        28 0.693E+00 0.000E+00    115556    115478    115640    115641
        29 0.307E+00 0.000E+00    115556    115478    115640    115641
```

Figure 9: Example of a load segment.

In the second technique "Fluid-structure coupling in 3D" the explosive blast load is done directly in the structure simulation using 3D MMALE with a fluid-structure coupling. This includes air and explosive interaction, blast wave reflections and fluid-structure interaction. The fluid element length is 15 mm and the fluid model consists of a block with 760 000 hexa elements, see Figure 10. Advantages and disadvantages with the two techniques are listed in Table 1.

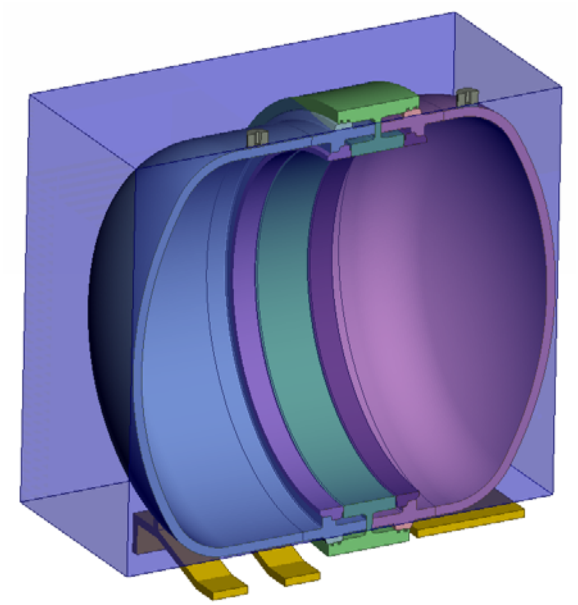


Figure 10: Fluid-structure coupling in 3D.

Table 1. Advantages and disadvantages with the two techniques

Axisymmetric mapped to 3D	Fluid-structure coupling in 3D
+ Fine resolution of the blast load + One blast load simulation can be used for many structure simulations + The safety factor can be changed in the mapping step + Fast simulation time - No fluid-structure interaction - Small deformations of the structure are assumed - Axisymmetric problems	+ Fluid-structure interaction + Can be used for non-axisymmetric geometry and/or load case + Large deformations of the structure - Coarse resolution of the blast load - Long simulation time - Several parameters for the fluid-structure coupling - Leakage in the fluid-structure coupling

Results

The first well-applied technique "Axisymmetric mapped to 3D" showed, after some small modifications in the material damping, relatively good agreement to the test. The termination time is 30 ms and four strain curves are compared in Figure 11. The Gauge 3 curve clearly shows that the oscillation frequency is in very good correlation between the test and the simulation. The total CPU time is about 3 hours with a 16 processor cluster.

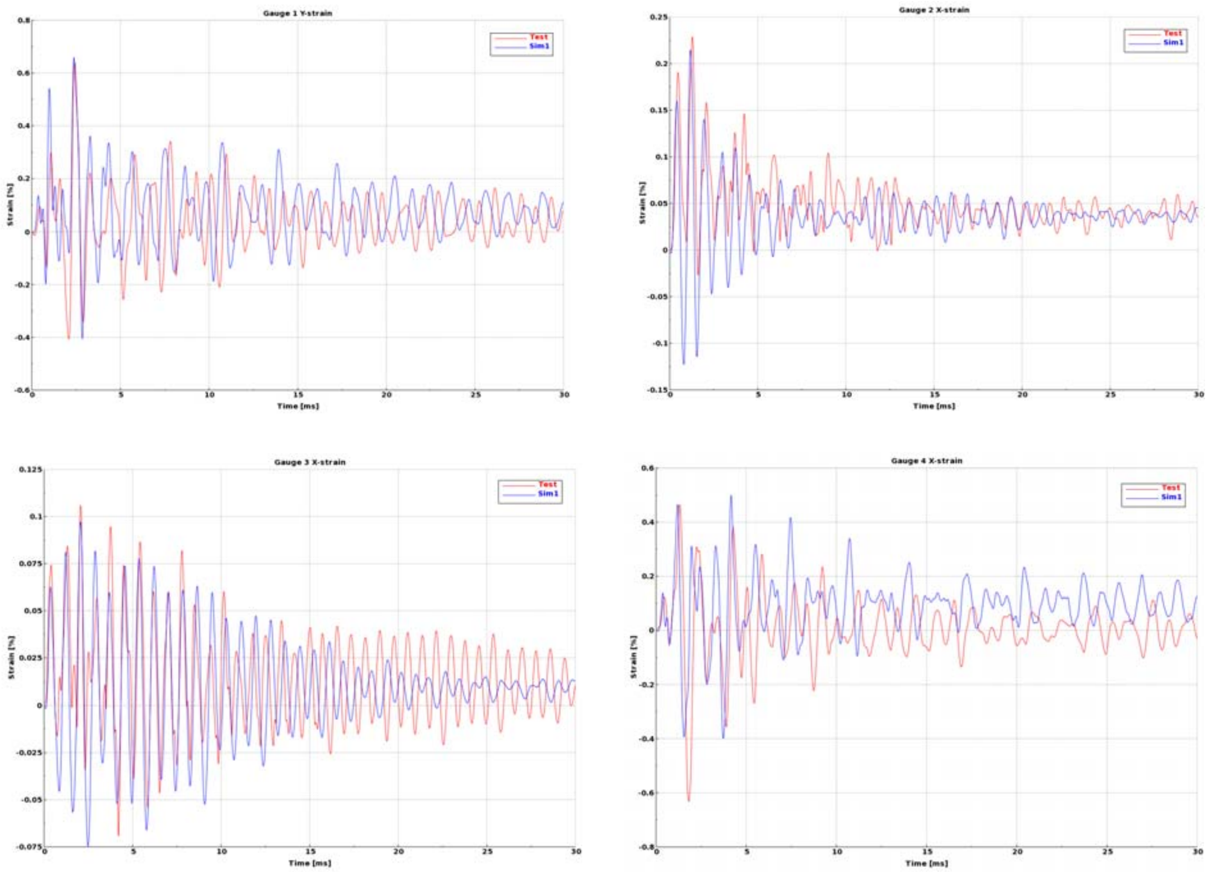


Figure 11: Strain comparison of the test vs. simulation "Axisymmetric mapped to 3D".

With the second technique "Fluid-structure coupling in 3D" the first analysis lead to significant lower strain levels compared to the test. A more detailed parameter study had to be performed to improve the simulation results. The fluid element length, the material damping and several coupling parameters were evaluated to improve the results. The final results are in relatively good agreement to the test. The termination time is 30 ms and four strain curves are compared in Figure 12. The influence of the fluid-structure coupling leakage gives a significant lower strain towards the end. The Gauge 3 curve shows that the oscillation frequency is in very good correlation.

The CPU time is about 24 hours with a 16 processor cluster. Two possibilities to speed up the CPU time is to either use a half model due to symmetry or remove the fluid after 5-10 ms. Both alternatives will give about the same results in a substantially shorter time.

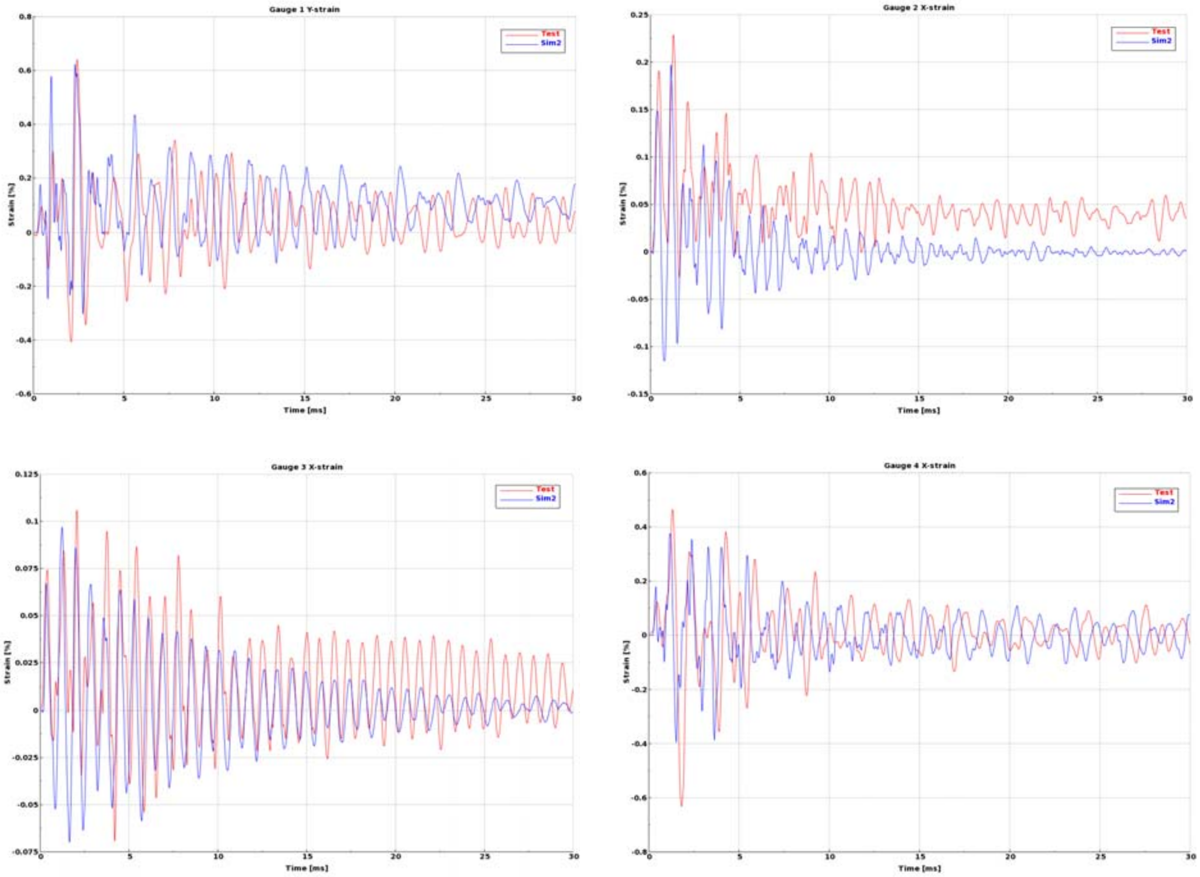


Figure 12: Strain comparison of the test vs. simulation “Fluid-structure coupling in 3D”.

Conclusions

As conclusion we now have two validated simulation techniques and procedures to make realistic explosive simulations of containment vessels.

References

[1] MECV8L Report from test campaign 2, Nr 064428, Dynasafe AB, 2012.
[2] LS-DYNA Keyword User’s Manual, Version 971, Livermore Software Technology Corporation, 2012.
[3] ANSA version 13.1.2 User’s Guide, BETA CAE Systems S.A., 2011.