New design of a pressure vessel subjected to blast loads

L.Cousin, P.Evrard

CEA, DAM, DIF, F91297 Arpajon, France

***Abstract**: A new design of containment vessel has been proposed to conduct confined detonation experiments. In order to guarantee the confinement of the detonation products, the risk expressed as a probability of failure has to be quantified. This quantification is done using probabilistic analyses which require data from experiments and simulations in order to be sustained. When a blast stresses a spherical vessel, many different mechanical phenomena appear and have to be studied using adapted models. A blast leads to reflected waves in the structure. A numerical chaining is used to access the vessel dynamic structural response. This chaining consists in linking CATIA V5 with Abaqus/CAE to realize the analysis models and in linking our hydrodynamic code to Abaqus /Explicit in a weak coupling: high speed hydrodynamic simulations provide transient pressures which are used to act on the inner shell of the vessel. Two mechanical phenomena and their associated finite-elements models are of interest. First, we focus on the nominal model used to study the dynamic response of the vessel itself, underlining the most important components (internal furniture, material laws, type of mesh...) to be taken into account. The spherical vessel has five ports. Each of them has a cover bolted to the vessel. Then, we review a model the aim of which is to understand the bolts behavior when the vessel is subjected to high dynamic stresses. This model consists in a first implicit analysis followed by an explicit analysis which continually takes into account the bolts pre-tension loads. Experiments were performed in order to validate this weak coupling. We end up showing the good agreement between experimental and numerical results.*

Keywords**: **Bolt Loading**, **Connectors**, **Constitutive Model**, **Coupled Analysis**, **Dynamics, Experimental Verification, Explosive**, **Impact, Safety**, **Vibration.

1. Introduction

The CEA has developed a new design of containment vessel for carrying out confined detonation experiments. This paper briefly presents the confinement vessel used for experiments and its associated instruments which allows understanding its behavior when submitted to blast loads. This behavior of the vessel has to be simulated because of the need existing in safety demonstration: to guarantee the confinement of the detonation products, the risk expressed as a probability of failure has to be quantified. This quantification is done using probabilistic analyses which require data from experiments and simulations in order to be sustained. The second point of the paper focuses on the numerical chaining used to master the blast effects on the vessel: this chaining goes from the Design Office, working with CATIA V5, to the Simulations Office, in which Abaqus models are prepared. Abaqus/Explicit is also linked in a weak coupling with our hydrodynamic code which provides information to load the inner shell of the vessel. The third point then describes the nominal Abaqus/Explicit model used to study the response of the vessel when submitted to a transient impulse loading. Results from the chaining are compared to experimental data. The fourth and last part consists in a description of another Abaqus model used to understand the behavior of the bolts which link five covers to the vessel body. The model consists in a first Abaqus/Standard analysis followed by an Abaqus/Explicit analysis using the `*IMPORT` functionality. A different bolts modeling is proposed because of the limitations of the `*Pre-tension Section` functionality only available in Abaqus/Standard, allowing for the bolts pre-tension loads to be taken into account in the dynamic simulation.

1. Containment vessel configuration

The containment vessel is presented in Figure 1. It consists of a steel spherical body with five ports each equipped with a cover:

- a steel cover at the top, used for general access,
- four aluminum side covers, used for diagnostics.

Each cover is connected to the vessel body by two concentric circles of bolts and three independent seals. The vessel is connected to a steel frame by its four side ports.

Figure 1 : Vessel for confined detonation experiments

The vessel is monitored during each experiment in the same way:

- several strain gauges are attached to the outer skin vessel, at strategic locations and at the center of the five covers,
- for each cover, a gauge is attached to a bolt belonging to the inner circle of bolts and another gauge is attached to a bolt belonging to the outer circle of bolts.

2. Numerical chaining

We use a numerical chaining (Figure 2) which links:

- the Design Office to the Simulations Office which uses Abaqus (Figure 2, path (1))
- and also the Simulations Office which uses our hydrodynamic code to our Simulations Office which uses Abaqus (Figure 2, path (2)).

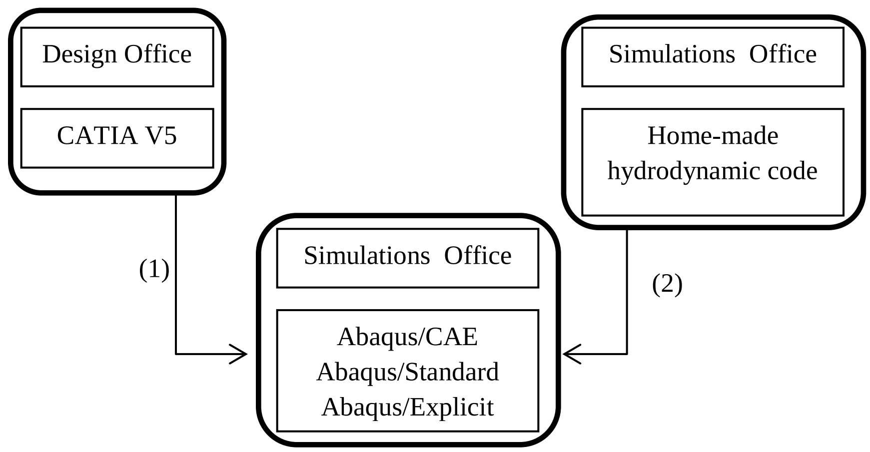

Figure 2 : Numerical chaining

2.1. From Design Office to Simulations Office

The first link enables to go from the geometric product assembly done with CATIA V5 in the Design Office to the finite elements model done in the Simulations Office using Abaqus/CAE (Figure 2, path (1)). A first VB script tool is used to put in the user's folder all native CATIA V5 files (CATParts and CATProducts) concerning the model as well as materials data associated with the parts. The materials data consist in a '.lib' file, an input file and a file giving the links between materials and their associated parts in the CATIA model. Materials are defined in a complete manner in the CATIA library: mechanical characteristics defining thermal behavior as well as dynamic behavior are known for most materials. Once the user's folder contains all these data, Abaqus/CAE is opened and the user accesses a plug-in allowed for the CATIA V5 and the materials data to be loaded in. Once the file to import is chosen:

- CATIA V5 geometries are imported in Abaqus/CAE,
- Imported parts are renamed following definitions names existing in the CATIA model,
- Instances are created automatically in Abaqus/CAE according to the imported parts,
- Materials data existing in the folder are imported and materials are assigned to the suitable parts,
- A table 'material/behavior' is available for the user: in this table one can choose the most appropriated behavior for the analysis he wants to lead (characteristics of a thermo-mechanic behavior, characteristics of a dynamic behavior type Johnson-Cook plasticity law...)

This gateway is useful to guarantee the quality of our Abaqus models: users are sure to work with the appropriated geometries and materials properties.

2.2 Weak coupling between a home-made hydrodynamic code and Abaqus/Explicit

The second link (Figure 2, path (2)) consists in a weak computational coupling between our hydrodynamic code and Abaqus/Explicit. In a mechanical way, the vessel is submitted to a blast load. This blast load is computed using a specific hydrodynamic code developed by the CEA. The code simulates explosive detonation, internal gas expansion and shock waves propagation in 2-D Eulerian analysis models. A characteristic load function of an explosive detonation inside the vessel is presented in Figure 3 (a): a transient impulse load can be observed, due to the shock wave, followed by a long term quasi-static load caused by the evolution and the heating of gaseous detonation products. Because of furniture inside the vessel, the pressure field versus time is not homogeneous on the inner shell. The pressure field must be computed at different points onto the inner shell in order to take into account the heterogeneous dynamic load and must be computed for a sufficient duration to cover the several reverberations inside the vessel. The way these pressure data are used in Abaqus is explained in paragraph 3.1.

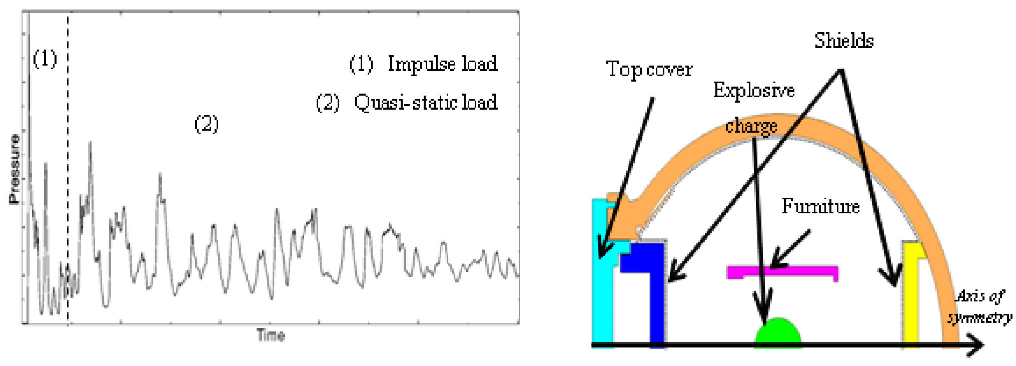

Figure 3 : (a) Typical pressure-time evolution; (b) Example of an 2-D Eulerian hydrodynamic model

3. Abaqus/Explicit analysis model in order to study the vessel response

3.1. Constitutive Abaqus/Explicit analysis model

Working assumptions

The modeling of the vessel presented in Figure 1 is accomplished using a half-structure shown in Figure 4. Furniture exists rounding the explosive charge which is not represented here but taken into account in the modeling.

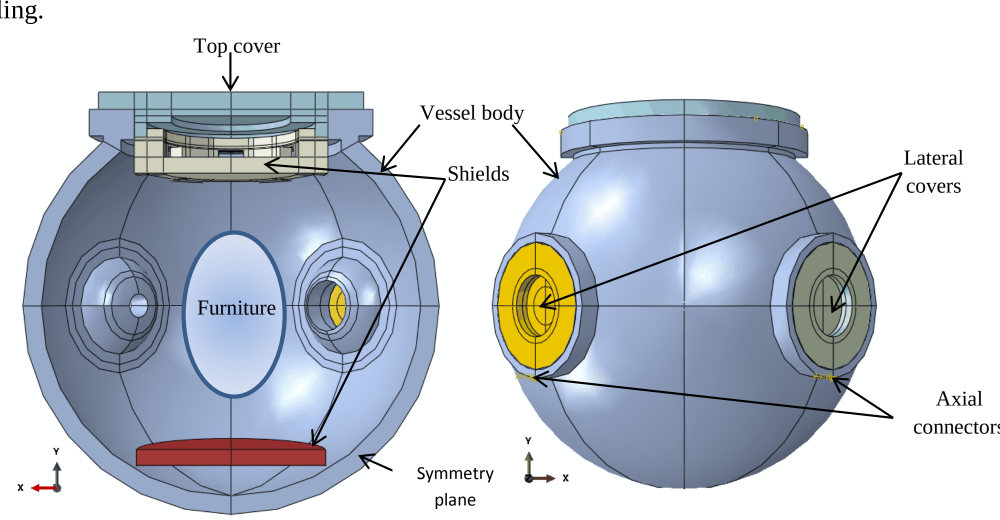

Figure 4 : Half-analysis model for the vessel's dynamic response

Boundary conditions

This half-vessel is fixed in the Y-direction by two AXIAL connectors type which have an elasticity modulus of one. Symmetry plane conditions are introduced for the nodes belonging to the cross-section plane (Figure 4).

Mesh

A 3D continuum elements mesh is applied. The analysis model contains 1,5 million 7 millimeter linear hexahedral elements C3D8R (Figure 5).

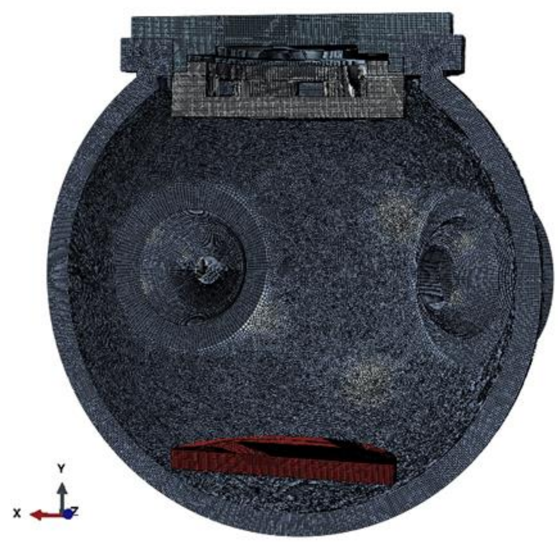

Figure 5 : Mesh of the half-model

Loads

Furniture is driven using speeds computed from the 2D-Eulerian hydrodynamic analysis. The speed data are used through a predefined field defined in the Abaqus/CAE Load Module. The resulting keyword is:

```
*Initial Conditions, type=VELOCITY
```

In the same time, all the pressure-time fields are used to load the inner shell of the vessel. We define one amplitude curve per pressure-time field. These fields are applied by slices which are defined creating:

- a surface: the inner shell,
- analytical fields to define the altitude of each slice in the Y-direction. This is done in the Abaqus/CAE Load Module through a table presented in Figure 6 (a), accessible from the Tools/Analytical Field menu,
- a load, type 'pressure' which allows to link the appropriated analytical field, surface and amplitude curve. The good definition of the slices can be monitored in the Visualization Module (Figure 6 (b)), loading the model of interest.

This functionality allows us to load the vessel without introducing any partitions of the structure which are a problem to keep a regular mesh.

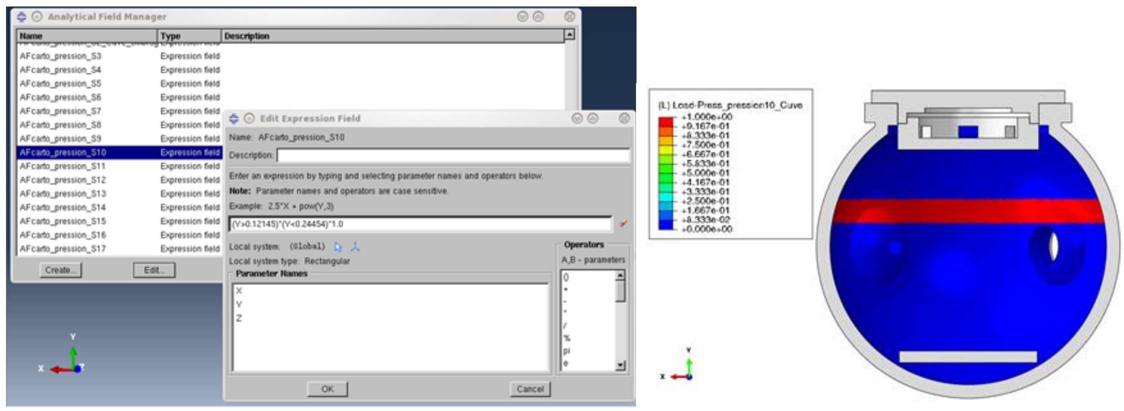

Figure 6 : (a) Definition of an analytical field describing a slice named ‘AF_carto_pression_S10’; (b) Verification of the definition of the slice in the Visualization Module

Materials

The high strength steel vessel behavior, the steel top cover behavior and the aluminum side covers behavior are supposed to be elastic perfectly plastic. The shields behavior is supposed to follow a Johnson-Cook law. The whole steel furniture behaviors are considered to be elastic perfectly plastic, coupled with a shear failure criterion to take into account their failure when impacting the inner shell of the vessel.

Interactions

A general frictionless contact is used in the model between the furniture and the vessel inner shell. In order to take into account impacts between the furniture, failed or not, and the vessel, we create different surfaces:

- an exterior surface on the inner shell of the vessel, named ‘exterior_vessel_inner_shell’
- an exterior surface on the whole furniture, named ‘exterior_furniture’
- interior surfaces in the whole furniture in order to take into account contacts existing between the future failed furniture and the inner shell vessel . These surfaces are named ‘interior_furniture’

Exterior surfaces are created on geometric entities, in a classical way in Abaqus/CAE, using the default selected tool. In order to create interior surfaces, the user has to define surfaces type ‘Mesh’ and change the default selected tool with ‘Select From interior Entities’ (Figure 7).

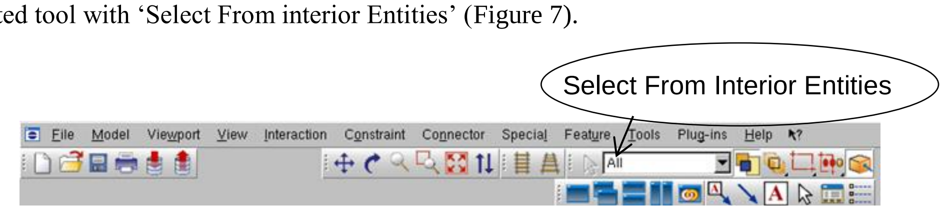

Figure 7 : Modification of the default selected tool to define interior surfaces

In the `*Contact` keyword, the following relations between surfaces must appear:

```
*Contact, op=NEW
*Contact Inclusions
Interior_furniture ,
Exterior_furniture,
Interior_furniture , Exterior_furniture
```

```
Interior_furniture, Exterior_vessel_inner_shell
Exterior_furniture ,Exterior_vessel_inner_shell
```

<u>Outputs</u>

Some of the elements belong to nodes sets according to the gauges attached to the outer shell of the vessel during experiments. The strain tensor is the output of interest. We assign a specific spherical material orientation (R, T, Z) on the vessel body so that the strain tensor is computed in the same orientation as the gauges orientation. This is done to be able to compare experimental and numerical data.

3.2. Analysis model validation

Several experiments were carried out using different explosive charges. The vessel is always monitored with gauges which locations are the same as the ones presented with the red dots in Figure 8. Experimental and numerical maximum Von Mises stresses values are compared for each gauge. One of the comparisons made is presented in Figure 8 . Good agreement can be observed excepted at the south pole and the north pole (top cover) of the vessel: the finite elements model underestimates the experimental data at the poles. These phenomena can be observed for each experiment comparison. Some more work is led at the moment in order to improve this fact.

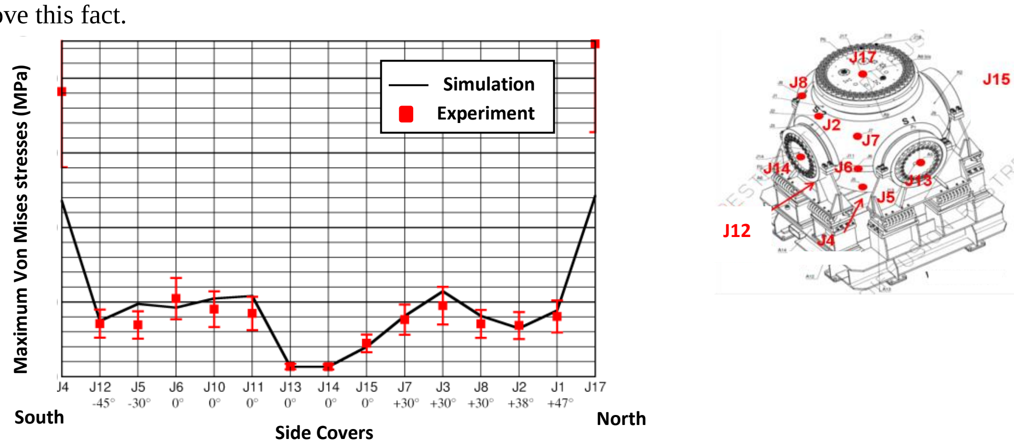

Figure 8 : Maximum Von Mises stresses- Experimental measures and numerical results comparison

4. An analysis model for the bolts behavior study

4.1. Constitutive Abaqus analysis model

Working assumptions

The Abaqus model used in order to understand the behavior of the bolts when the vessel is submitted to a blast load is presented in Figure 9. It consists in the same half-model as before, with the same symmetry plane and boundary conditions. No furniture is taken into account. Bolts used to assembly the covers to the vessel body are modeled. The top cover is assembled by two concentric circles of M36 bolts. The side covers are assembled by two concentric circles of M24 bolts.

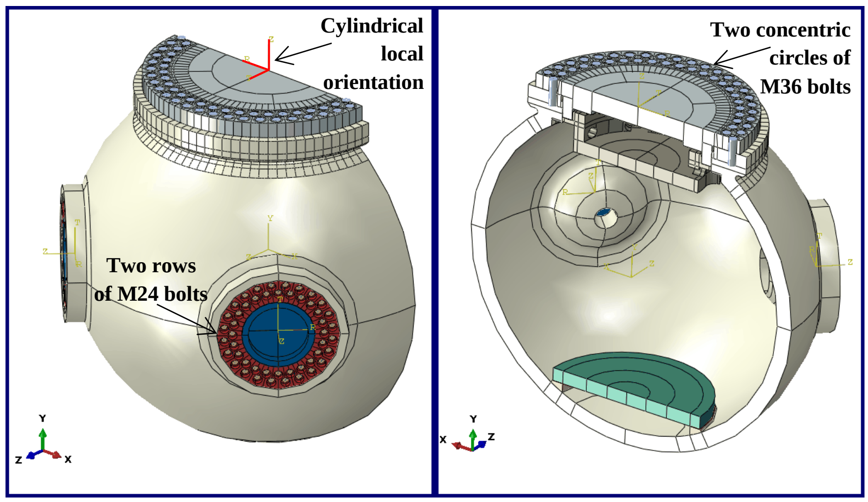

Figure 9 : Analysis model configuration

Interactions

The two shields are linked to the vessel using the *TIE functionality. Contacts defined between the covers and the vessel body take into account a friction of 0,1.

Mesh

All the parts are meshed with C3D8R linear hexahedral elements. The analysis model contains 1,5 million 7 millimeter linear hexahedral elements C3D8R.

Bolts modeling

General concept
The M36 bolts (top cover) and the M24 bolts (side covers) are modeled using beam elements (in red on Figure 10) associated with connectors in order to firstly impose the pre-tension load in the bolt and secondly monitor the existing axial force in the bolt when the vessel is submitted to a dynamic solicitation due to the blast.

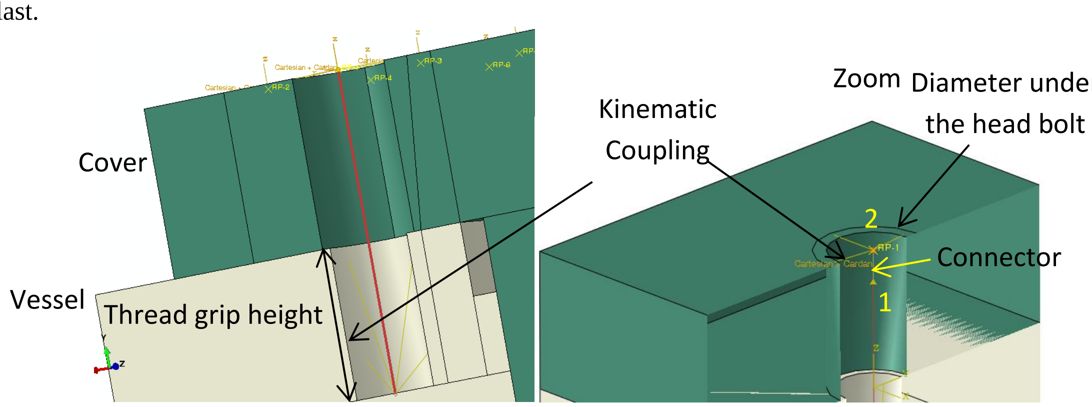

Figure 10 : Bolts modeling- Beam element associated with connector

The connector defines a kinetic relation between two points. One of these nodes is the end of the head bolt side beam. The second node is named differently from the first one but has the same location: the connector element length is null. Figure 10 shows a zoom of the modeling: the connector, built between node 1 and node 2, is connected in series with the beam (in red). The second node of the connector is the kinematic coupling node reference. This coupling is representative of the head bolt leaning on the covers: the slave nodes underlying surface is defined according to the diameter under the head bolt.   A kinematic coupling is also defined at the other end of the beam in order to take the bolt thread grip height into account: the slave nodes underlying surface is defined according to the real bolt thread grip height.

Beam elements characteristics
Linear beam elements, B31, are used to model the bolts. The behavior of these steel bolts is supposed to be elastic. The diameter, d, and the length, L, of the beam are computed in order to obtain the suppleness in tension and flexion of the M24 bolts and the M36 bolts respectively. These assembly parameters are computed using the COBRA software which follows the VDI2230 [1].

Connector elements characteristics
'Cartesian + Cartan' connectors type are used. They are associated with local cylindrical orientation (R, T, Z) such as defined in Figure 9. All translation and rotation degrees of freedom are activated (U1, U2, U3 + UR1, UR2, UR3). The designation '1' is equivalent to the local R-direction, the designation '2' is equivalent to the local T-direction, and the designation '3' is equivalent to the local Z-direction. The connectors are used to impose the pre-tension load in the bolt and secondly to know the axial force history when the vessel is submitted to a dynamic solicitation due to the blast so that their stiffness is infinite.

Sequence of analysis
The simulation is conducted in two times:

- A first implicit simulation is accomplished using Abaqus/Standard in which the pre-tension load is imposed. This time is a two steps analysis:
    - the first step is a step for contact initiation. The degrees of freedom of the connectors linked with beams on the top cover are all locked. The degrees of freedom of connectors linked with the beams of the side covers are all locked excepted the translation degree of freedom in the beams axis direction. A very small displacement is given to the side covers in this beam axis direction,
    - in the second step, the imposed displacements are off and the connectors linked with the beams of the top cover are relaxed in the beams axis direction. The pre-tension load is then passed through the whole bolts defining a load type 'Connector Force' in the third local direction (Z).
- The final state of this first Abaqus/Standard analysis becomes the initial state of a dynamic explicit simulation accomplished using Abaqus/Explicit using the `*Import` functionality. The vessel is submitted to the blast load following the same methodology as described in the paragraph 3.1, the bolts pre-tension load being taken into account. This pre-tension load state is maintained locking all the degrees of freedom of the connectors. Finally, we access the axial force history in the connectors which is equivalent to the axial force seen by the bolts.

4.2. Analysis model validation

Uniaxial gauges are attached to two M36 bolts and to two M24 bolts. In both cases, one of these bolts is located on the inner circle of bolts and the other on the outer circle of bolts. These gauges allow us to know

the initial force level obtained as a result of the tightening torque and to follow the time-force evolution during the experiment.
The measures accuracy is +/-13%. The finite elements model is evaluated on its capacity to estimate the greatest force passing through the bolts. Figure 11 shows the experimental and numerical maximum levels of force for a given experiment in:

- The M36 bolts of the top cover located in the inner circle (TC INT),
- The M36 bolts of the top cover located in the outer circle (TC EXT),
- The M24 bolts of the side cover located in the inner circle (SC INT),
- The M24 bolts of the side cover located in the outer circle (SC EXT).

Very good agreement is found. This notification is valid for others experiments.

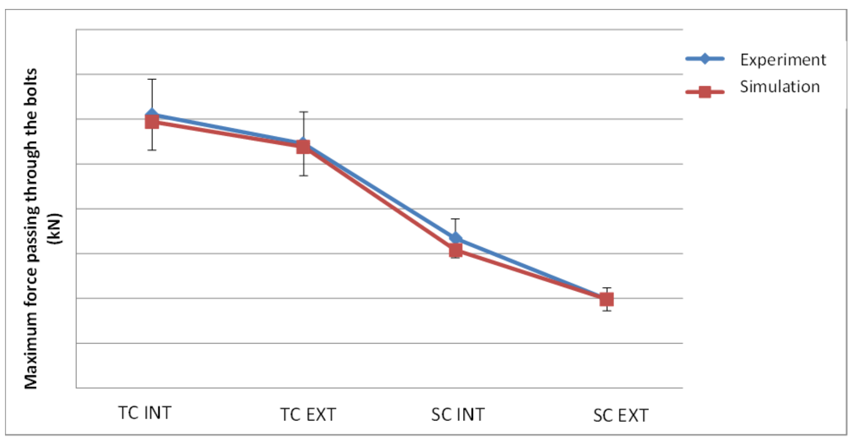

Figure 11 : Maximum forces passing through the bolts. Comparisons between experimental and numerical data.

5. Summary

This paper describes the tools of a numerical chaining used to design a confinement vessel submitted to blast load. This chaining links first CATIA V5 to Abaqus/CAE and secondly the CEA hydrodynamic code to Abaqus/Explicit in a weak coupling. The Abaqus /Explicit model dealing with the dynamic response of the vessel is presented and a comparison between experimental and numerical results is shown. The paper describes also an Abaqus/Standard-Abaqus/Explicit model the aim of which is to be able to estimate the higher axis force existing in the bolts of the covers when the vessel is solicited by a blast load. Comparisons with experimental results validate the two proposed models.

6. References

[1] VDI 2230 "Systematische Berechnung hochbeanspruchter Schraubenverbindungen Zylindrische Einschraubenverbindungen" Verein deutscher Ingenieure