# Tank Burst Modeling

**Amita Tripathi**

**fluidyn France**
**7, boulevard de la Libération**
**F-93200 Saint Denis**

## *Introduction*

One important constraint of the space structures is the weight of the structure which impacts directly on the payload. Therefore tanks on board of satellites and rockets are characterised by their thin walls. However, they are subjected to an hostile environment and are often subject to sudden thermal variations. For cryogenic fluids as well as for propergols, sudden and local vaporisation can bring excessive thermal loadings on the tank walls. Moreover, the state of weightlessness sometimes followed by sudden accelerations forces on the payload can also put a lot of stress on the walls. Finally, the operation of most of the industrial and propulsion energy conversion systems involve deflagration/detonation of chemically reacting mixtures. On one hand, the surrounding structures may be subjected to very high pressure, and, on the other hand, the deformation of structures might influence the propagation of pressure waves.
To be able to optimise the thickness of the walls, that is to decrease as much as possible their weight, without compromise the safety of the structure, it is necessary to study the behaviour of wieldings and walls to extreme and transient solicitations. Understanding of fluid-structure interaction is essential from the design and safety point of view.
Modelling of the transient evolution requires a strong coupling between fluids and structures. In the present paper, two demonstration cases are presented to assess the feasibility of such studies using a fluid-structure interaction software, *fluidyn*-MP. *fluidyn*-MP is a finite element/finite volume based solver for the steady state/transient analysis of fluid-structure interaction problems involving stress analysis, thermal analysis and fluid flow.
The first demonstration case concerns the exposure of the cryogenic tank to a solar radiation. The second one is an explosion of hydrogen inside the tank.

## *The numerical tool : fluidyn-MP*

*fluidyn*–MP is a commercially available software providing a single platform for multiphysics, fluid-structure interactions and conjugated heat transfer, by the strong coupling of the finite element methodology for the deformations and displacements of the structures and the finite volume methodology for the fluid flows. Fluidyn-MP is characterised by several features :

- Robust physical schemes adapted to each application
- Several numerical integration schemes from implicit to explicit
- Automatic exchange of boundary conditions between fluid and structure
- Adaptative mesh : non-conform, moving, structured and unstructured.

- Local timestep to optimise CPU

### Fluid solver

The fluid solver solves the 3D Navier-Stokes equations for compressible or incompressible fluids, discretized by the finite volume method. The fluids are modelled thanks to several numerical schemes (Roe, Van Leer, Roe Preconditioned, AUSM, Partial Donor Cell, etc.) which enables to adapt the scheme to the flow configuration. The flux splitting techniques are used for convective fluxes and pressure-related terms are evaluated implicitly by a SIMPLE or PISO scheme. The detonation modelling is done with an integrated model combining the characteristics of Chapman Jouguet at the detonation front. For the deflagrations, the software introduces the chemical kinetics from the database for the hydrocarbon fires.

### Structure solver

The structure solver uses the finite element method for non-linear transient analysis of structures by explicit or implicit time-integration schemes to model large and small displacements and deformations. The state equations of the structure elements can be elastic, elasto-plastic, or linear piecewise, with an isotropic behaviour, or orthotropic, etc. The plastification laws commonly used, like Steinberg Guinan et Johnson & Cook, depending on temperature or pressure, can be used.

### Coupling

The methodology of coupling is the following :

- The fluid solver marches to the first fluid time step.
- The pressure exerted by the fluid (mechanical load) are used by the stress solver to drive the structure to a new configuration with new velocities.
- The fluid takes up a new position in contact with the new structural position by remeshing the fluid domain to conform to the current configuration of the structure.
- The new distribution of fluid nodes leads to a new pressure field in the fluid. This is used in the next cycle of the fluid solver.

For explicit integration for stress solver, the stable time step for integration may be smaller than the fluid time step (use of a finite number of sub steps of the stress solver to complete one cycle of fluid solver).

## *Case 1 : Solar radiation of a tank*

### Physical model

The model consists of an elliptical gas enclosure containing two cubical boxes and exposed to ambient temperature outside (see Figure 1). The outer boundary of the enclosure is subjected to time-dependent heat flux (simulating solar heating) and radiation to the ambient, whereas, the inner boundary is subjected to convection to the gas inside and radiation to inner boxes.
The thickness of the enclosure and the boxes is of 10 mm. The conductivity of the enclosure and boxes are taken as 63 and 150 W/mK respectively. The outer boundary of the enclosure is subjected to radiation to the ambient at –50$^{o}$C with emissivity of 0.7. The inner boundary of the enclosure is also subjected to radiation to the inner boxes. In addition to radiation boundary condition, the outer boundary is also subjected to time dependent heat flux (solar radiation)

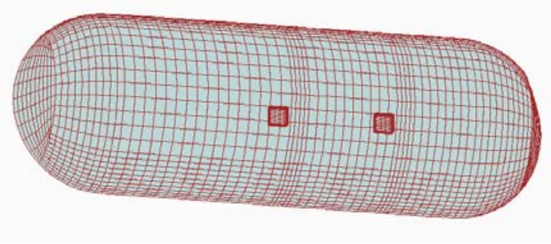


**Figure 1 :** Structural Mesh

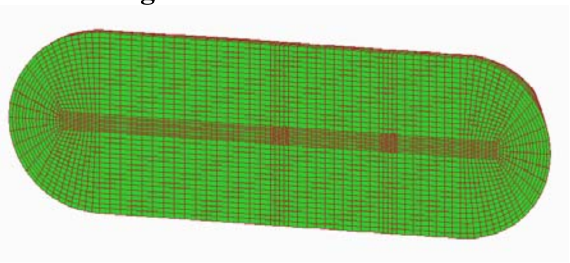


**Figure 2 :** Fluid Mesh

## Numerical model

Due to the symmetry of the problem, only one half of the model is modelled using 8-noded hexahedral elements.. The fluid domain consists of the gas in enclosure and gas inside the boxes and is shown in Figure 2. The structural domain (enclosure and two inner boxes) is shown in Figure 1. A total of 43060 hexahedral elements (2700 for the structure and 40360 for the fluid) are used for the discretisation of the domain.

The fluid is compressible and viscous. The natural convection within the enclosure is modelled by the Boussinesq model. The perfect gas equation of state is considered with a molecular weight of 4 and a gamma of 1.66, constant laminar viscosity = 1.97e-5. The first order, 6-stage Upwind Difference Scheme is employed and a time step of 14 sec is used for transient analysis.

The surface of the enclosure was divided in 12 sections (see Figure 3). The solar radiation was approximated by a triangular variation of the heat flux with peak value of 50 W/m$^2$ inside of each sections. Each section is submitted to the heat flux with a time lag in comparison with the previous one. Figure 4 shows two of the structural boundary face groups subjected to heat flux and the corresponding flux versus time curves. The fluid-structure interface is implicitly subjected to convection boundary condition with dynamically varying values of film coefficient and fluid temperature.

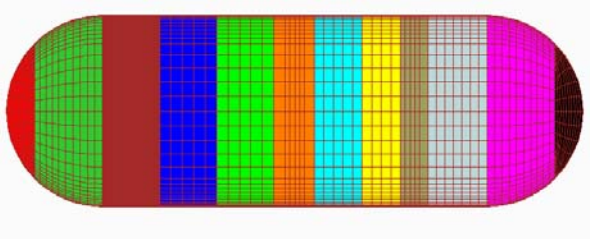


**Figure 3** : Time-dependent heat flux on outer surface

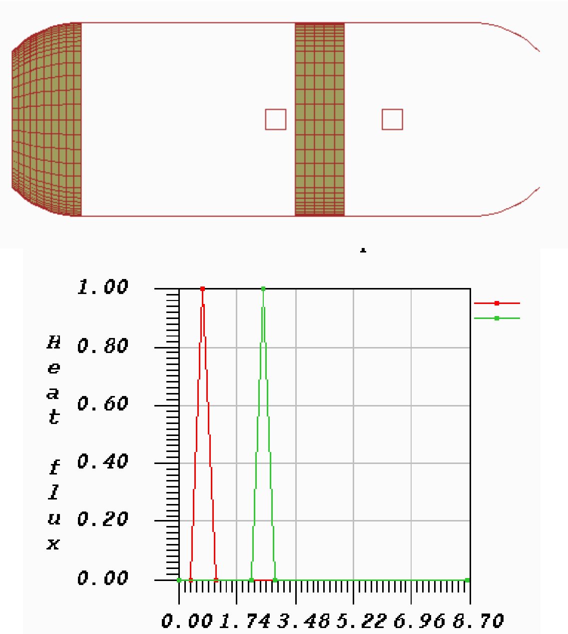



**Figure 4 :** a. Boundary face groups 2 and 7. b. Time dependent curves for groups 2 (red) and 7 (green)

## Results and Discussion

The transient simulation is done for a period of 24 hours. During that period, the outer boundary is subjected to heat flux for 12 hours. Figure 5 shows temperature in the enclosure after 6 hours. Temperature is localised around the narrow zone due to radiation condition to ambient at –50°C employed over the rest of outer surface. Figure 6 shows the temperature in the symmetric plane of fluid region at the same time. The stream line plot superposed on the temperature contour shows the convection effect within the enclosure.

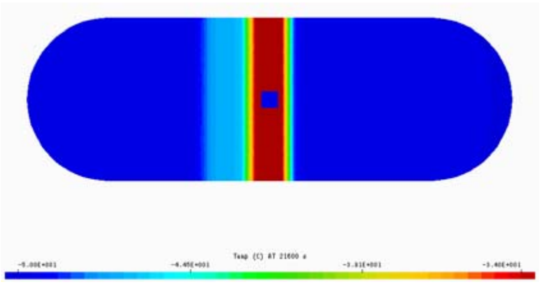


**Figure 5 :** Structural temperature after 6 hours

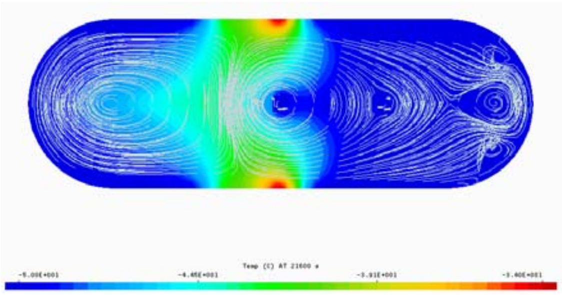


**Figure 6 :** Fluid temperature and streamline plot after 6 hours

## *Case 2 : Hydrogen explosion*

The purpose of this case was twofold:

- To compute the propagation of pressure waves generated due to explosion in a confined space and compare with experimental measurements.

- To study the effect of a deforming structure on the propagating pressure waves generated by explosion.

### Physical model

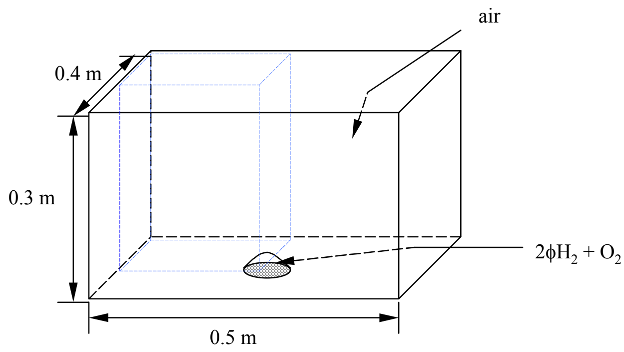



**Figure 7** : Physical Model

A hemispherical bubble of a mixture of hydrogen and oxygen is located at the bottom of a box (see Figure 7). The steel plates making up the box are 0.05 m thick. The bubble radius is 0.05 m. The ratio between hydrogen and oxygen is governed by the equivalence ratio Φ i.e. the ratio of the number of moles of hydrogen to the number of moles of hydrogen in the stoichiometric mixture.. Two different Φ have been used in the present case to make for two different ignition energies: one case will evolve into a deflagration and the other into a detonation.

### Numerical model

Due to symmetry only the region enclosed by the blue lines are considered (see Figure 8). Whole domain is discretized using uniform control volumes of 0.005 m x 0.005 m x 0.005 m. No-slip conditions for velocity with zero flux for scalars are used at the walls. Also, the walls are assumed to be adiabatic. The enclosure is fixed at the bottom surface to prevent the overall displacement. Initially the whole domain is at a pressure of 101325 Pa and a temperature of 300 K.

In the case of detonation, the initial composition of the reacting mixture is given

by $\Phi = 1$. For deflagration $\Phi$ is equal to 1.25.
Chemical reaction in the $H_2/O_2$ mixture is assumed to take place through a single-step global reaction of the form :

$$2H_2 + O_2 \rightarrow 2H_2O + \Delta Q,$$

where, $\Delta Q$ is the heat released due to the reaction. Rate of change of mass of different species are computed using the equations of the Arrhenius form.
Other parameters are listed below :
Kinematic viscosity = $1.89 \times 10^{-5}$ $m^2/s$
Prandtl number of the mixture = 0.72
Mass diffusivity of $H_2$ = $12.6 \times 10^{-5}$ $m^2/s$
Mass diffusivity of $H_2O$ = $2.63 \times 10^{-5}$ $m^2/s$

Specific heats for all the species are calculated from the enthalpy values given in JANNAF tables.
The structural properties of the steel used are:

- Yield strength = $2.1 \times 10^{11}$ Pa
- Poisson's ratio = 0.3
- Density = 7850 $kg/m^3$
- Tensile strength = $1.8 \times 10^9$ Pa

## Results and Discussion

*Detonation case*

Monitoring points have been located in the computational domain. The coordinates o these points are given in the table below (the origin located at the center of the spherical bubble):

Table 1 : Location of monitoring points

| Point | X (m) | Y (m) | Z (m) |
|---|---|---|---|
| P0 | -0.125 | 0 | 0 |
| P7 | -0.19 | 0.045 | 0.2 |
| P11 | 0.25 | 0.16 | 0 |
| P10 | 0 | 0.3 | 0 |
| P1 | 0.19 | 0.255 | 0.2 |

The figure 8 shows the time evolution of the pressure at the monitoring points. A comparison with experimental data shows a good agreement between the numerical and the experimental results. A high ignition energy caused the whole reactive mixture to burn in 10-4 s. This results in a wave front with very high pressure (about 10 bar).

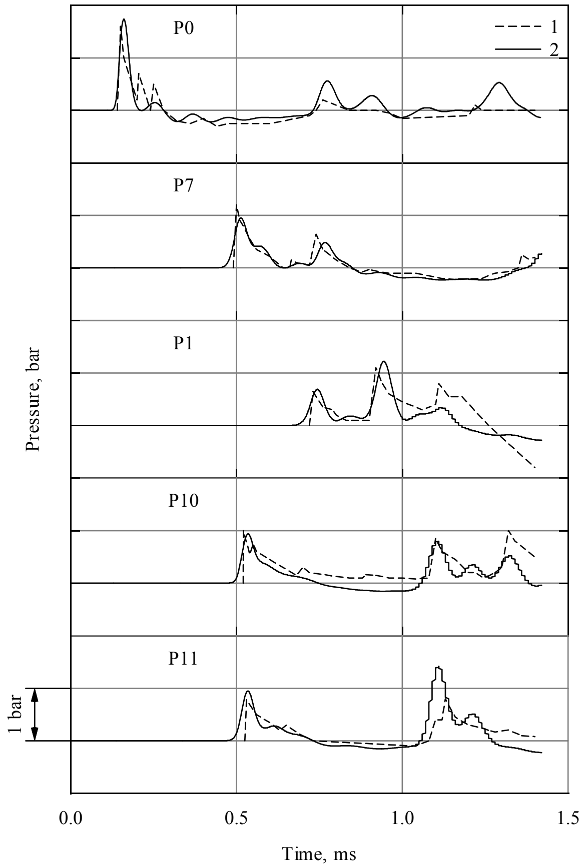



**Figure 8** : Comparison of the evolution of pressure between the experiment (plain line) and the computation (dotted line).

It is found that in this case the propagation of the pressure wave is significantly influenced even by a very small displacement (<1 mm) of the enclosing walls (Figures 9 to 12 and table 2).

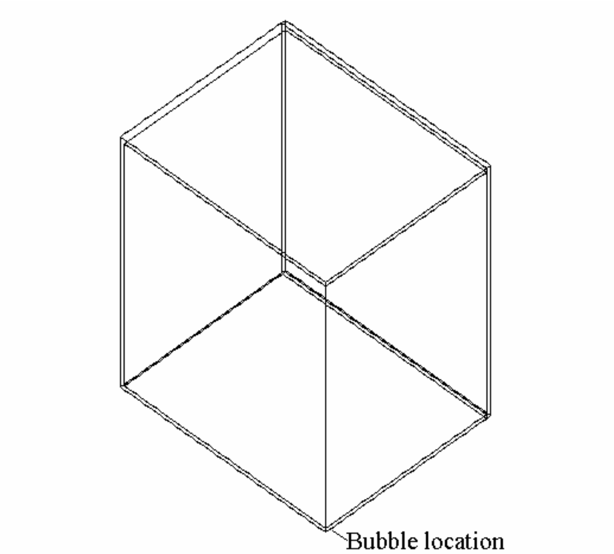



**Figure 9**: Initial shape of the box at t = 0 ms

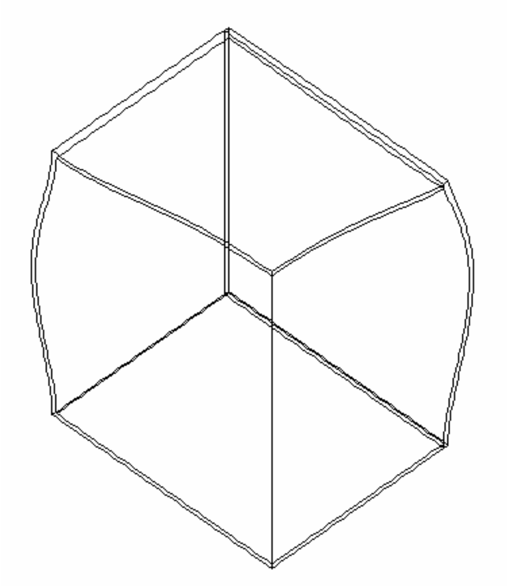


**Figure 11** : Box deformation at t= 1 ms

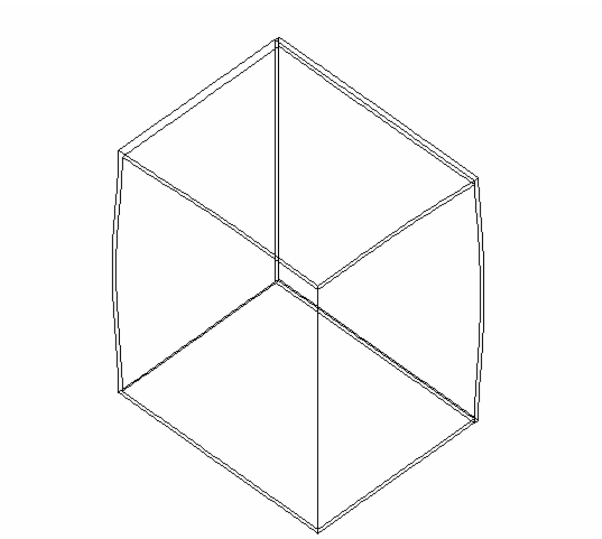


**Figure 10** : Box deformation at t= 0.5 ms

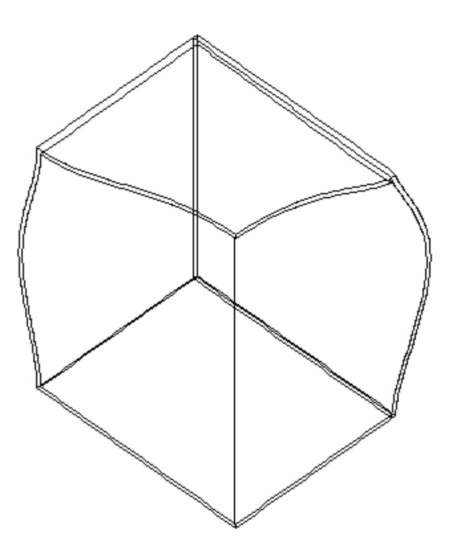


**Figure 12** : Box deformation at t= 1.5 ms

Table 2 : Maximum displacements in each direction

| **T (ms)** | **X (mm)** | **Y (mm)** | **Z (mm)** |
|---|---|---|---|
| 0.5 | 0.4 | 0.4 | 0.4 |
| 1 | 1.2 | 1.9 | 1.5 |
| 1.5 | 1.11 | 3.5 | 2.3 |

*Deflagration case*

A lower value of ignition energy resulted in deflagration of the mixture. This results in a wave front with moderately high pressure (about 2 bar). Figure 9 shows the time evolution of the pressure at the monitoring points. A comparison with experimental data shows that though the frequency of the pressure waves matches with the measurements, it is not the case for magnitude. This is probably due to the difference in ignition energy.

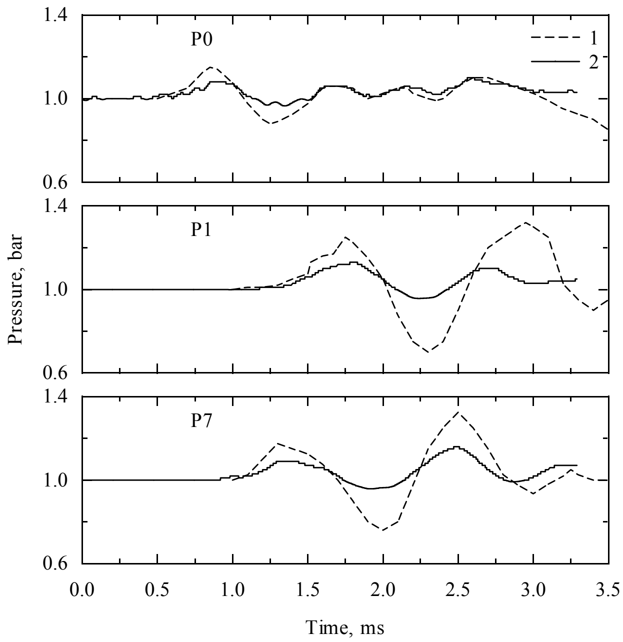



**Figure 9** : Comparison of the evolution of pressure between the experiment (plain line) and the computation (dotted line).

## Conclusions

A general-purpose code for the reactive fluid flow analysis, structural dynamics, and fluid-structure mechanical/thermal interaction, *fluidyn*-MP, has been developed and is presented here through two cases pertinent to space applications.

The first case was simulating the solar radiation on the thin shell of a space structure. The coupling between the conduction, radiation and convection was demonstrated.

The second case was simulating the deflagration and detonation of $H_2/O_2$ mixtures and the results compared with the experimental measurements. A typical fluid-structure interaction problem has been simulated by allowing the outer shell of the structure to deform under pressure differences. It is found that, in the case considered the structural deformation significantly affected the qualitative and quantitative behaviour of the pressure wave propagation inside the confined volume. Fluid structure interaction decreases the magnitude of the pressure wave. It also changes the arrival time and frequency of the wave.

It is therefore essential, for a proper optimisation of space structures (minimum weight for maximum fiability) to take into account multiphysical interactions between fluid-structure and heat transfer.

## References

# TANK BURST MODELLING

A. Kumar and A. Tripathi

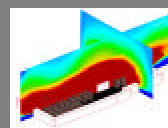
FLUIDYN France

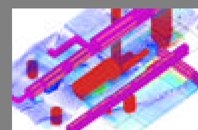
*On-Board Energetic Equipment*

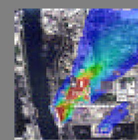
*Avignon, October 18-20, 2004*

## FLUIDYN-France / TRANSOFT International

- French company created in 1987
- Two poles :
  - Industrial processes : CFD, Fluid-structure interactions, Heat transfer
  - Environment / industrial risks : Toxic gas dispersion, explosions, fires ...







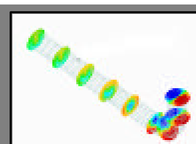


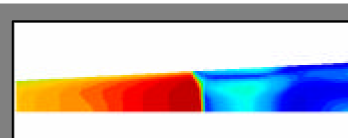
## FLUIDYN-France / TRANSOFT International

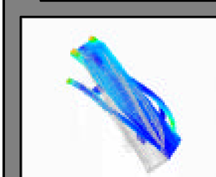
- Three main activities :
  - Provider of general software (*fluidyn* series)
  - Development of customized software dedicated to specific applications
  - Consultancy studies
- Several subsidiairies in the world : South-East Asia, US, UK
- Interaction with universities / laboratories
- Clients include : Framatome, CEA, TOTAL, SNCF, etc.

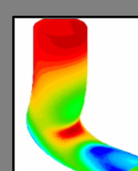


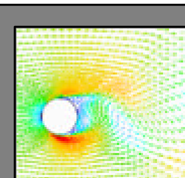
## *fluidyn*-MP : Software for Multiphysics





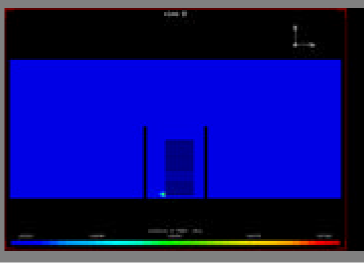






## Example of dedicated software : *fluidyn*-EXPLODE

- Explosions in open and confined spaces
- Based on general software : fluidyn-MP
- Interface, numerical schemes and physical models adapted





## OUTLINE

- Introduction
  - Numerical tool : *fluidyn*-MP
- Case 1 :
  - Demonstration case of explosion inside a tank
- Case 2 :
  - Demonstration case of heat transfer on a tank
- Conclusion



## *fluidyn*-MP : Functionalities

- Single platform for multiphysics : fluid-structure interactions
- Strong coupling of Finite Volume with Finite Element methods
- Robust physical schemes adapted to each application
- Several numerical integration schemes from implicit to explicit
- Automatic exchange of boundary conditions between fluid and structure
- Adaptative mesh
- Local timestep to optimize CPU



## Fluid solver

- 3D Navier-Stokes equations for compressible or incompressible fluids.
- Discretization by Finite Volume Method
- Structured (multiblock) and unstructured mesh
- Flux splitting techniques for convective fluxes
- Pressure-related terms evaluated implicitly by a SIMPLE or PISO scheme.
- A second order accurate central difference scheme for diffusion terms
- Several turbulence models
- Chemical reactions can be modeled (Arrhenius, EDC, JWL, ..)
- Multiphase flows

## Structural solver

- Finite Element Method for non-linear transient analysis of structures
- Geometric and material non-linearities
- Explicit or implicit time-integration scheme
- Large and small displacements and deformations

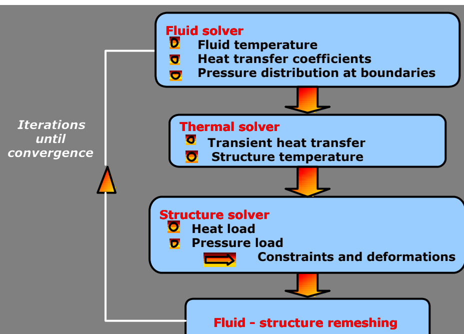


## Fluid-Structure Interactions

- Fluid solver to the first fluid time step.
- The pressure exerted by the fluid (mechanical load) are used by the stress solver to drive the structure to a new configuration with new velocities.
- The fluid takes up a new position in contact with the new structural position by remeshing the fluid domain to conform to the current configuration of the structure.
- The new distribution of fluid nodes leads to a new pressure field in the fluid. This is used in the next cycle of the fluid solver.
- For explicit integration for stress solver, the stable time step for integration may be smaller than the fluid time step (use of a finite number of sub steps of the stress solver to complete one cycle of fluid solver).



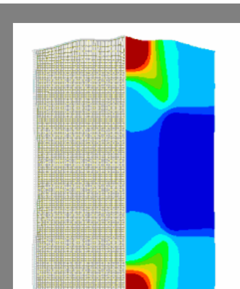
## Fluid-structure interactions with heat transfer






**Case 1 :**

**Explosion in a tank**

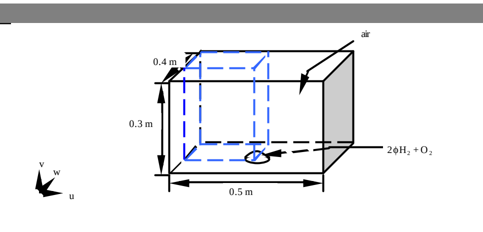


## Objectives

- Operation of most of the industrial and propulsion energy conversion systems involve deflagration/detonation of chemically reacting mixtures
- Surrounding structures may be subjected to very high pressures.
- Deformation of structures can influence the propagation of pressure waves



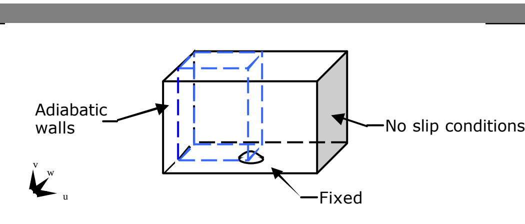
## Geometry




- Due to symmetry only the region enclosed by the blue lines are considered
- Bubble radius is 0.05 m
- Steel plates of 5 mm thickness for walls
- Deflagration or detonation is obtained by using different ignition energies



## Geometry




- No-slip conditions for velocity with zero flux for scalars at the walls.
- Walls assumed to be adiabatic.
- Fixed bottom to prevent displacement
- Initial conditions : ambient temperature and pressure



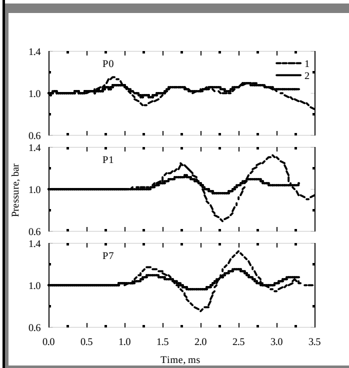
## Chemical reactions

- $f$ : mixture ratio
- In the case of detonation, $f = 1$.
- For deflagration $f = 1.25$.
- Chemical reaction is a single-step global reaction :

$$2H_2 + O_2 \longrightarrow 2H_2O + \Delta Q$$

- Rate of change of mass with Arrhenius equation

## Comparison for deflagration



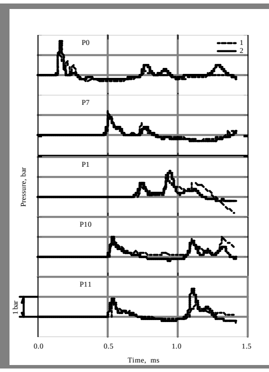

A low value of ignition energy results in deflagration of the mixture

Wave front with low pressure (about 2 bar)

P0(-0.125,0,0);
P1(-0.19,0.255,0.2),
P7(-0.19,0.045,0.2);
Origin located at the center of the spherical bubble.



## Comparison for detonation



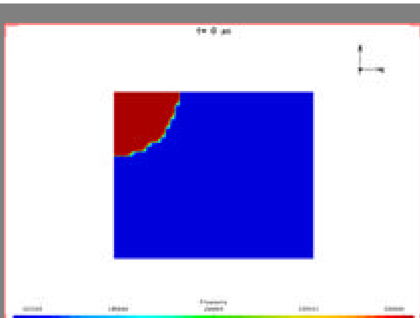

A high ignition energy caused the whole reactive mixture to burn in $10^{-4}$ s

Wave front with very high pressure (about 10 bar).

P0(-0.125,0,0);
P7(-0.19,0.045,0.2);
P11(-0.25,0.16,0);
P10(0,0.3,0);
P1(-0.19,0.255,0.2),
Origin located at the center of the spherical bubble.



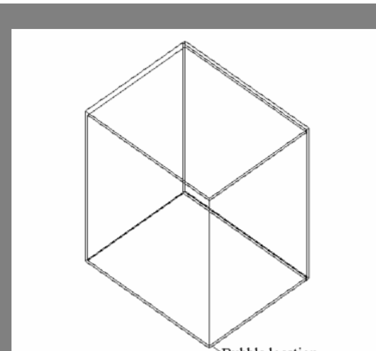
## Wave propagation





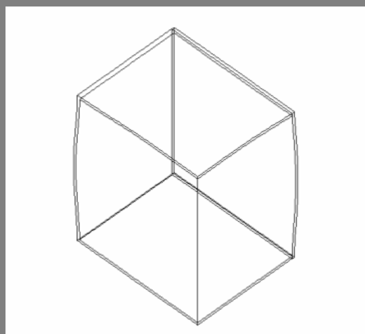
## Structure deformation (detonation)

t = 0



## Structure deformation (detonation)

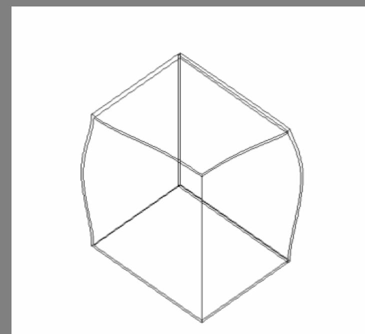
t = 0.5 ms
Max Displacement : 0,4 mm





## Structure deformation (detonation)

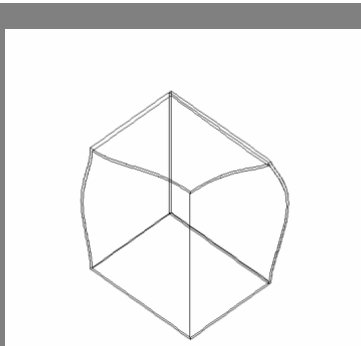
t = 1 ms
Max Displacement :
X – 1.2 mm
Y – 1.9 mm
Z – 1.5 mm





## Structure deformation (detonation)

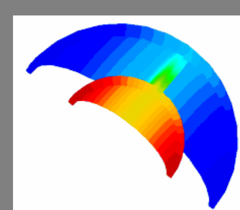
t = 1,5 ms
Max Displacement :
X – 1.11 mm
Y – 3.5 mm
Z – 2.3 mm





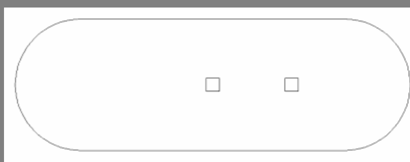
**Case 2 :**

Heat transfer between an airship structure and a gaseous medium : Conduction, convection and radiation



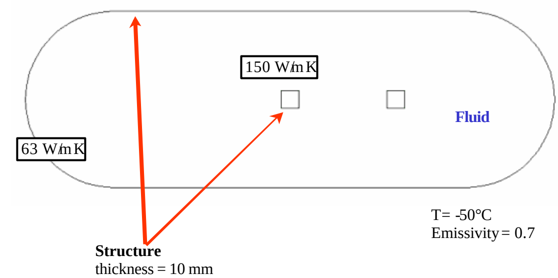
## Model

- Elliptical gas enclosure containing two cubical boxes
- Exposed to ambient temperature outside
- The outer boundary of the enclosure is subjected to time-dependent heat flux (simulating solar heating) and radiation to the ambient
- The inner boundary is subjected to convection to the gas inside and radiation to inner boxes





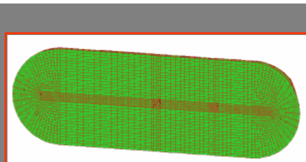
## Model

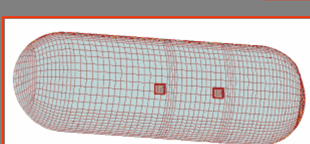





## Mesh

Fluid mesh : 43630 elements



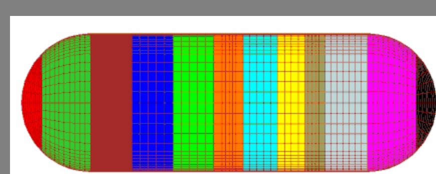


Structural mesh : 2700 elements



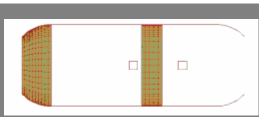
## Time-dependent flux (solar radiation)

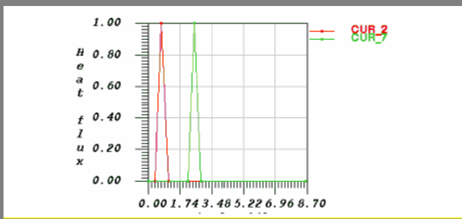
- Outer boundary divided in sections



- Solar radiation simulated by a triangular variation with peak of 50 $W/m^2$
- Dynamically varying film coefficient values (fluid temperature)

## Time-dependent flux (solar radiation)





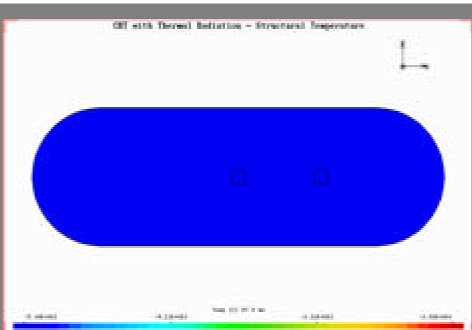



## Numerical parameters

- Compressible and viscous fluid
- Natural convection by Boussinesq model
- First-order 6-stage Upwind Difference scheme
- Transient timestep of 14 sec.
- Transient simulation for 24 hours
- Outer radiation of 12 hours

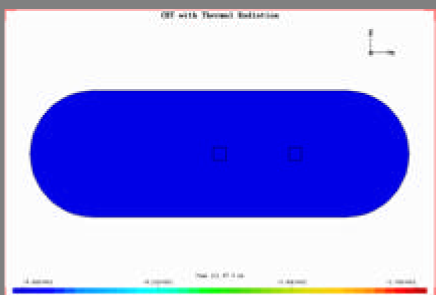
On-Board Energetic Equipment Avignon, October 18-20, 2004

## Structural temperature



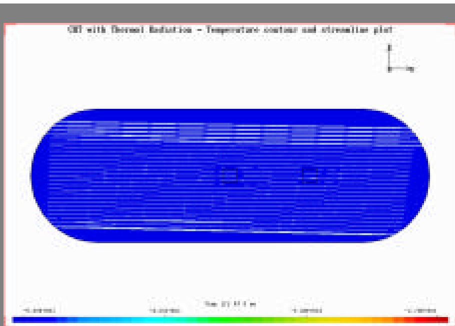

On-Board Energetic Equipment Avignon, October 18-20, 2004

## Fluid temperature




On-Board Energetic Equipment Avignon, October 18-20, 2004

## Streamlines





## Conclusions

To demonstrate the ability to simulate the deflagration and/or detonation of highly reactive mixtures and the response of the encompassing structures to the pressure waves thus generated.

To demonstrate the ability to simulate the response of a structure exposed to a transient radiation.

Understanding of fluid-structure interaction is essential from the design and safety point of view.

Lead to a better understanding of normal functioning and a better planning of potential problems

On-Board Energetic Equipment Avignon, October 18-20, 2004

**Thank you for your attention**

*On-Board Energetic Equipment*

*Avignon, October 18-20, 2004*