Vented explosion of H_2/air mixture : a comparison benchmark study

ICHS • Hamburg • 2017/09

E.Vyazmina/S.Jallais/L.Krumenacker/A.Tripathi/A.Mahon/J.Commanay/
S.Kudriakov/E.Studer/Th.Vuillez/F.Rosset

Vented explosions

- Vented explosions are widely studied, both experimentally and numerically
- Several analytical models exist for the overpressure inside the enclosure
- In complicated cases it is very difficult to find a proper analytical model:
  - presence of multiple vents
  - obstacles
  - flammable layer
  - gradient

Maximum internal and external overpressures, the length of the external flame etc. => definition of the safety distances requires an accurate and validated prediction based on CFD modeling

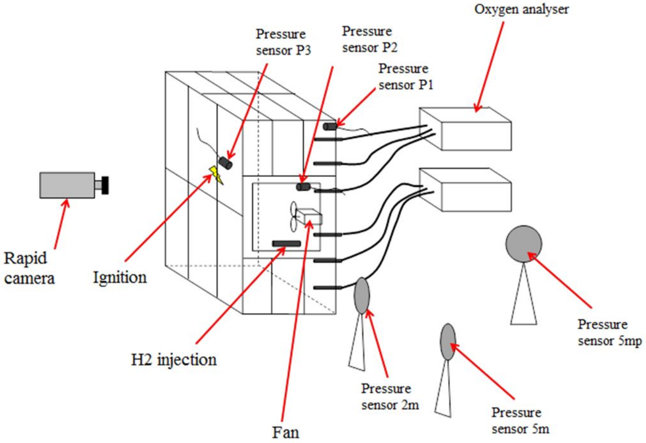
- comparison with experimental data
- several recommendations for CFD modeling of vented explosions

Contents

1

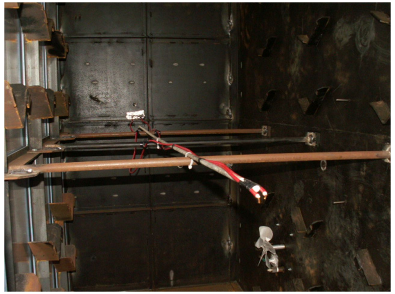
Bench description

Experimental chamber[1]



Experimental set-up:

- Test chamber of 2m x 1m x 2m and square vent of $0.49m^2$
- Homogeneous H_2/air mixture of 16.5% (±0.4%)
- BW ignition
- Fresh gas movement was visualized by adding particles of NH_4Cl



Measurements:

- 3 piezo-resistive sensors (0-10 bar) for overpressure inside
- 3 piezo-resistive sensors (0-2 bars) for overpressure outside at 2m, 5m (at the axis of the vent) and 5 m away from the vent (on the axis perpendicular to the chamber)
- 100Hz low-pass filter is used for the pressure signal

[1] Daubech, J., Proust, Ch., Gentilhomme, O., Jamois, D., Mathieu, L., Hydrogen-air vented explosions: new experimental data, Proc. of 5th ICHS, 2013.

2

Simulations description

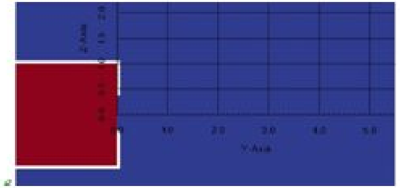
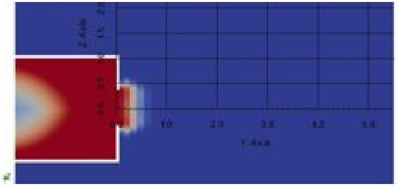
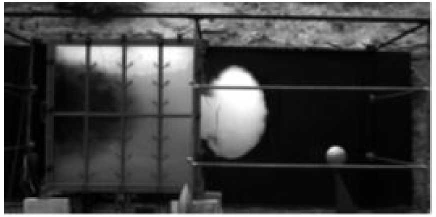
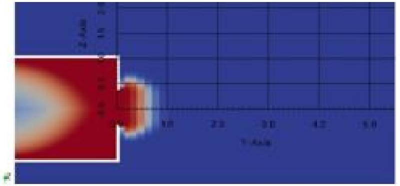
Bench participants and code description

Participant / Code	Domain (LxWxH)	Mesh	Number of grid cells	Turbulence modelling	Boundary conditions
Air Liquide / FLACS v10.4	10m by 5.5m by 5.5m	inside the box and in the evacuated cloud: 2.5cm	~6 M	RANS, k-eps	open outlet "plane wave" & wall boundaries for obstacles
APSYS / OpenFOAM 3.0.0	7.5m by 7m by 3.5m	Grid size 1.5cm close to walls, inside the box 3.125cm, outside 6.25cm	~1.2 M	LES - k-equation eddy viscosity model	open outlet boundaries & wall boundaries for obstacles
CEA / EUROPLEXUS	7.5m by 2.5m by 3m	Uniform 5cm	~1 M	Euler	Absorbing boundary conditions
Fluidyn / Fluidyn-VENTEX	7.5m by 8.5m by 4.5m	inside the box: 3cm; Refined in the axes of the explosion	~750k	RANS, k-omega SST	Open boundaries
ODZ-Consultants / FLACS v10.3	8m by 7.5m by 3m	Uniform 3cm	~6.2 M	RANS, k-eps	open outlet "plane wave"& wall boundaries for obstacles

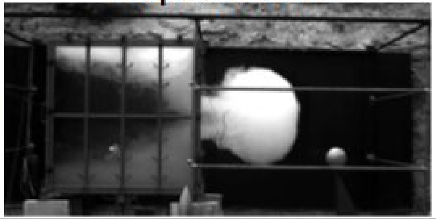
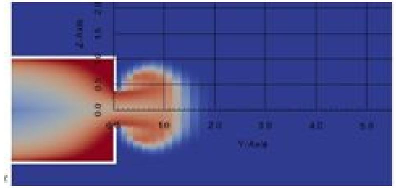
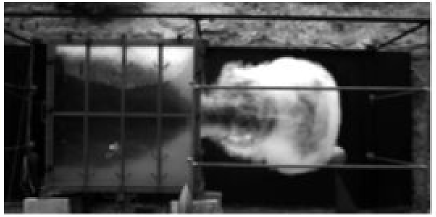
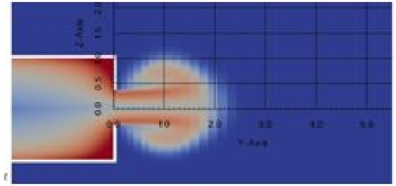
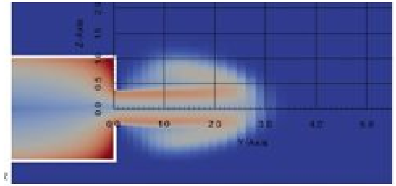
3

Simulation results

Results from numerical simulations: development of VEX

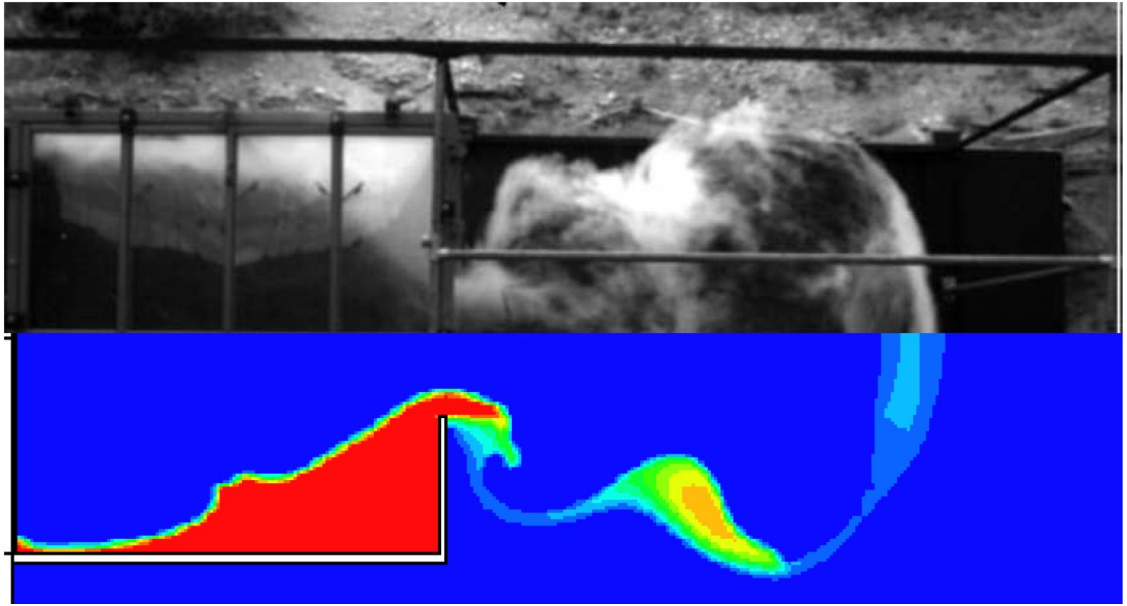
	Experiment	Fluidyn
Ignition		
100 ms		
110 ms		

Results from numerical simulations: development of VEX

	Experiment	Fluidyn
130 ms		
140 ms		
150 ms		

The moment of the external explosion

EXP

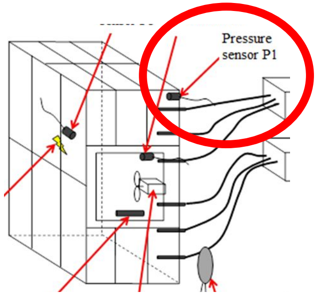


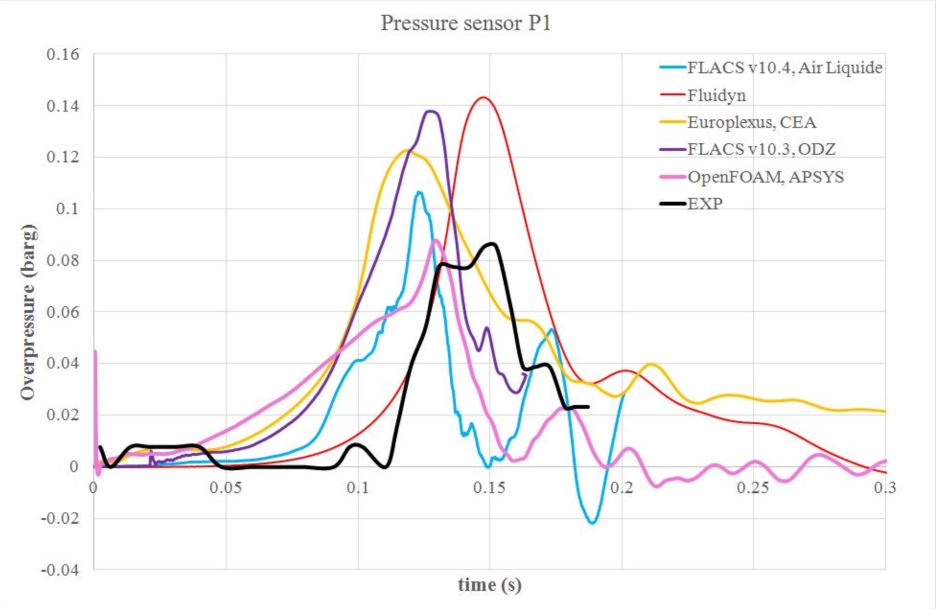
AL
FLACS
10.4

Experimental and numerical snapshots closely match

Air Liquide

Pressure evolution inside the chamber : P1



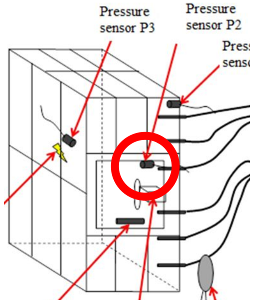


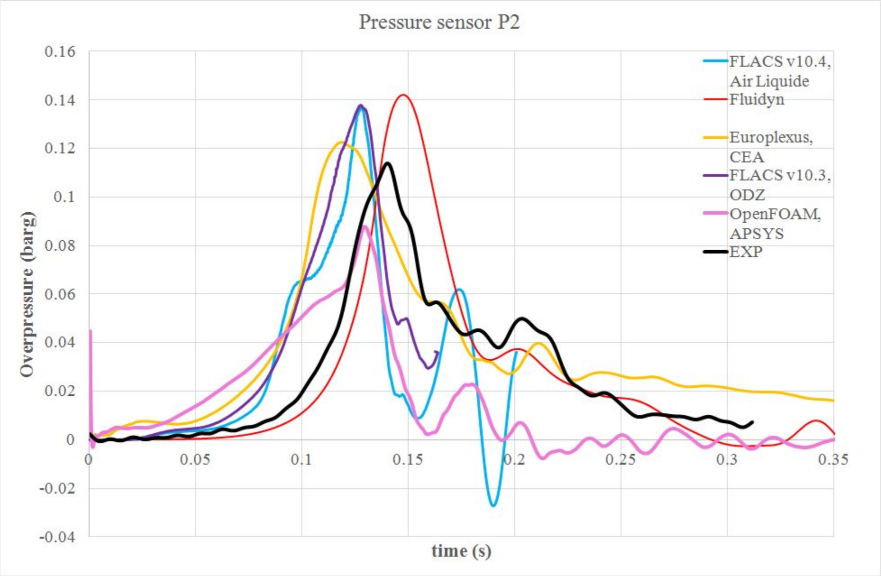
All CFD codes are in reasonable agreement with exp:

- the magnitude is overestimated by ~60% (**Fluidyn** and **ODZ**), 40% (**CEA**), 25% (**AL**), **~3%** (**APSYS**)
- all codes except Fluidyn predict the appearance of the spike in advance compare to experiment
- whereas Fluidyn overpressure maximum is slightly delayed in time

Pressure evolution inside the chamber : P2

P2 (114 mbarg) is larger than P1 (85 mbarg) ⇒ more important to obtain a better agreement on P2





All CFD codes are in reasonable agreement with exp:

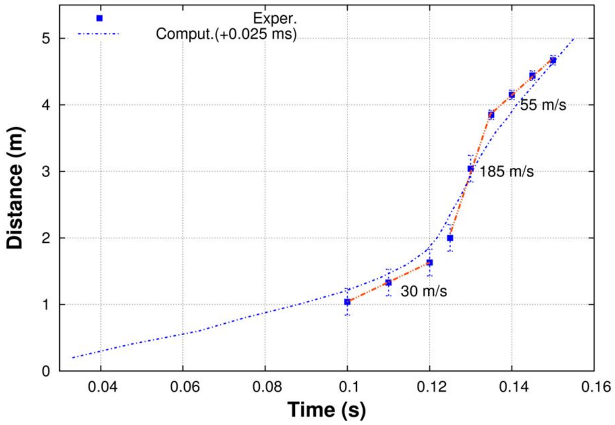
- the magnitude is overestimated by 25% (**Fluidyn**), 20% (**ODZ** and **AL**), and 7% (**CEA**)
- **APSYS** underestimates by 23%
- all codes except Fluidyn predict the appearance of the spike in advance compare to experiment
- whereas Fluidyn overpressure maximum is slightly delayed in time

Flame propagation distance vs. time

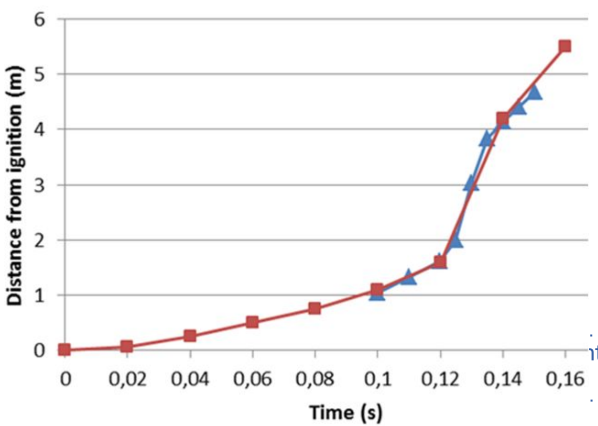
Simulated flame shows the same tendency as the experiment:

- slight acceleration approaching the vent (flame velocity is approximately 30 m/s),
- a violent acceleration up to 185 m/s due to the rapid burning of the evacuated cloud of fresh gas
- a deceleration of the flame at the end due to a slow burning of the rest of the mixture (less reactive and less turbulent)

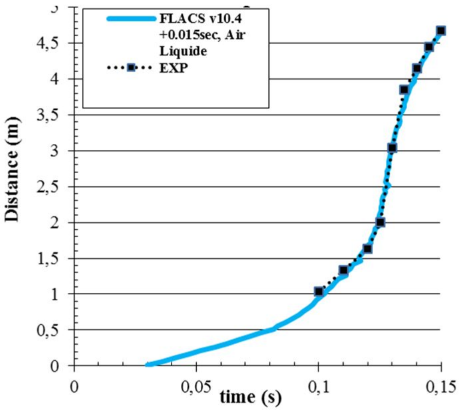
CEA



APSYS



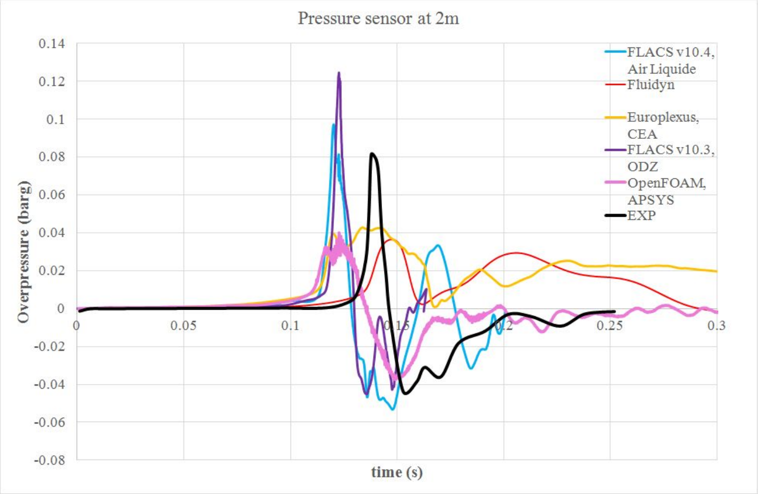
Air Liquide



Pressure evolution outside the chamber at 2m and 5m

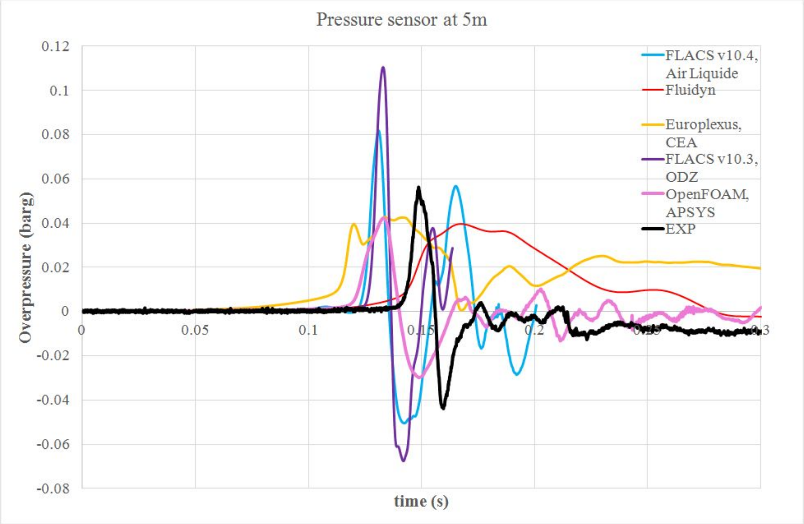
A 2m P is:

- overestimated by 50% (**ODZ**), 20% (**AL**) ⇐ presence of external walls
- underestimated by 50% (**Fluidyn** and **CEA**) ⇐ absence of external walls
- **APSYS** underestimates by 50% ⇐ mesh stretching in the this region ⇒ too diffusive



A 5m (close to the wall) P is:

- overestimated by 90% (**ODZ**), 50% (**AL**) ⇐ presence of external walls ⇒ increase of the P
- underestimated by 30% (**Fluidyn** and **CEA**) ⇐ absence of external walls
- **APSYS** underestimates by 25% ⇐ mesh stretching in the this region ⇒ too diffusive



4
Conclusion and Discussion

Air Liquide

Conclusion

- Inside the combustion chamber CFD matches closely experiment
- Outside the overpressure is underestimated:
  - CEA and Fluidyn considered that the combustion chamber is installed in a free field, without any interaction with outside structure. But the experimental facility is confined by two walls:
    - one in the streamwise direction (50 cm away from the detector at 5 m)
    - another all along the lateral direction (50 cm away of the chamber wall)

    ⇒ **extra confinement leads to the increase the overpressure outside the chamber**

    ⇒ **no effect on the pressure inside the chamber (in the absence of flame-structure interaction)**
  - Simulations performed with a stretched grid (APSYS) in the region of pressure detectors lead to an extra numerical diffusion and giving lower overpressure

Recommendations for CFD modelling of VEX

Based on the comparison sim/exp several **best practice recommendations** can be given :

- CFD can be used for large vent area and BWI
- The grid must be uniform inside the chamber and in the region of the evacuated cloud
- For the correct estimation of the overpressure outside the enclosure, all confinements and external rigid structures must be taken into account (represented in CFD simulations or a correction factor must be suggested)
- The grid must be uniform without any stretching in the region of interest (the region of monitoring points)

Results must be verified for other concentrations, gradient mixtures, CI and a presence of obstacles in the chamber

Thank you for your attention!!!

Air Liquide