Digital stakeholders' workshop
HyTunnel-CS project
4-5 May 2020

The pressure peaking phenomenon in garages: CFD model

D. Cirrone, D. Makarov, V. Molkov
Ulster University

Aim of the research

Pressure Peaking Phenomenon for jet fires

- This research aims at the development and validation of a CFD model for predictive simulation of pressure peaking phenomenon for hydrogen jet fire in large scale scenarios.
- CFD modelling complements a engineering tool:
  - Simulation of diversified scenarios that can not be represented by engineering tool assumptions.
  - Calculation of thermal load on the structure.
  - Calculation of hazard distances by thermal effects in the external surroundings of the enclosure.

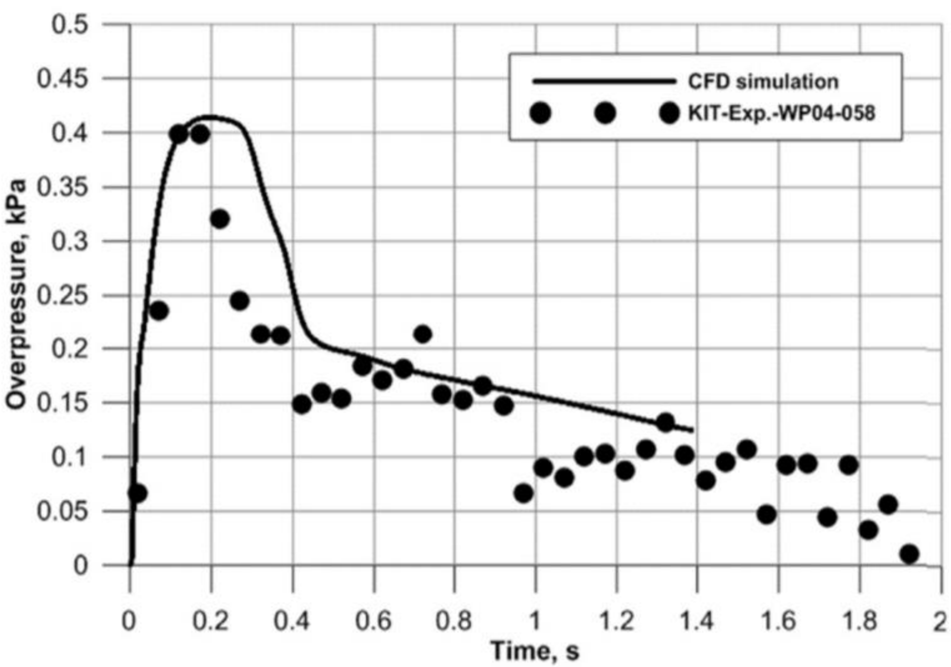
Previous work on PPP

CFD model validation against KIT tests

CFD model was validated against 4 experimental tests performed in KIT for a 1 m³ enclosure.

Results for test:

- $\dot{m} = 0.55\ g/s$
- Vent size: 3x30 cm



INTERNATIONAL JOURNAL OF HYDROGEN ENERGY 43 (2018) 17954–17968

Available online at www.sciencedirect.com

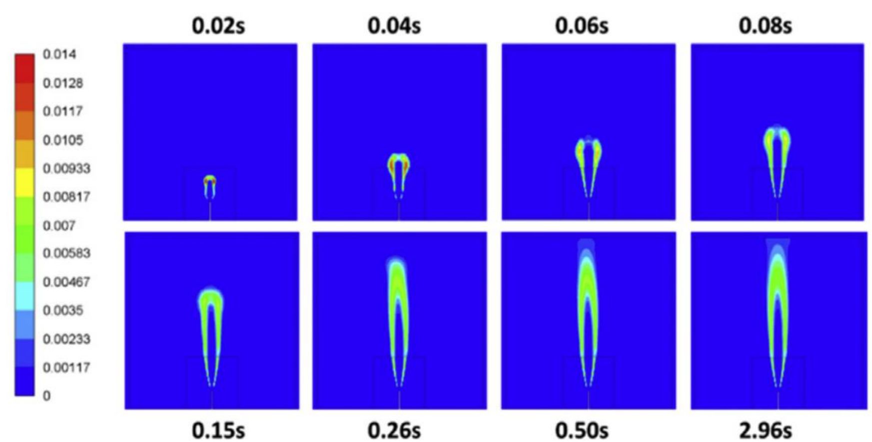
ScienceDirect

journal homepage: www.elsevier.com/locate/he

Numerical validation of pressure peaking from an ignited hydrogen release in a laboratory-scale enclosure and application to a garage scenario

H.G. Hussein, S. Brennan, V. Shentsov, D. Makarov, V. Molkov*

HySAFER, Belfast School or Architecture and the Built Environment, Ulster University, BT37 0QB, UK



OH mole fraction distribution

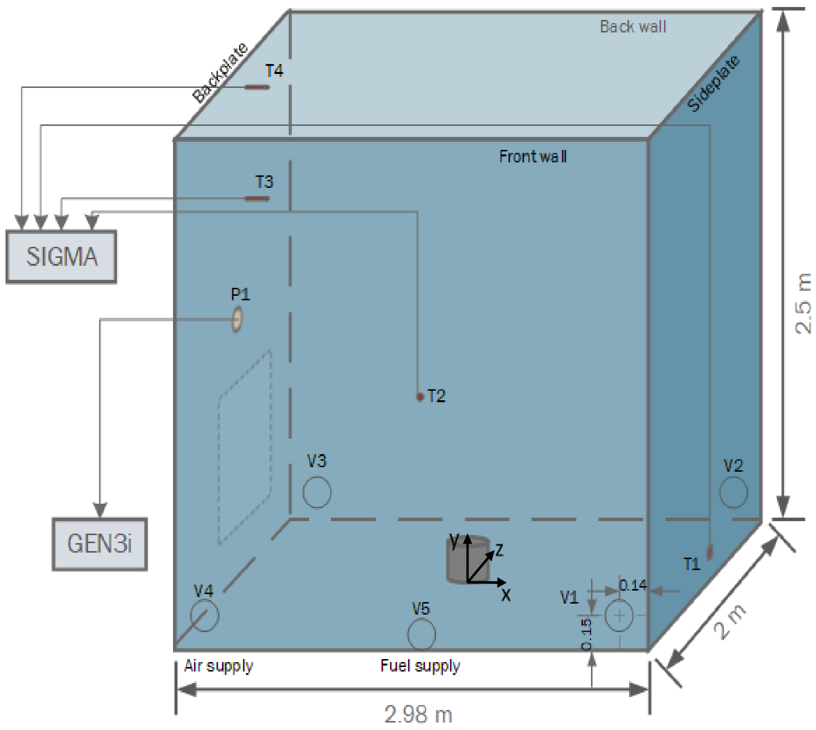
USN validation experiments

Details and advancements

- Large scale scenario: 15 m³ enclosure.
- Release mass flow rate up to 11.5 g/s.
- Hydrogen release duration is 6 s from 4 mm diameter.
- Selection of 3 tests for CFD modelling.

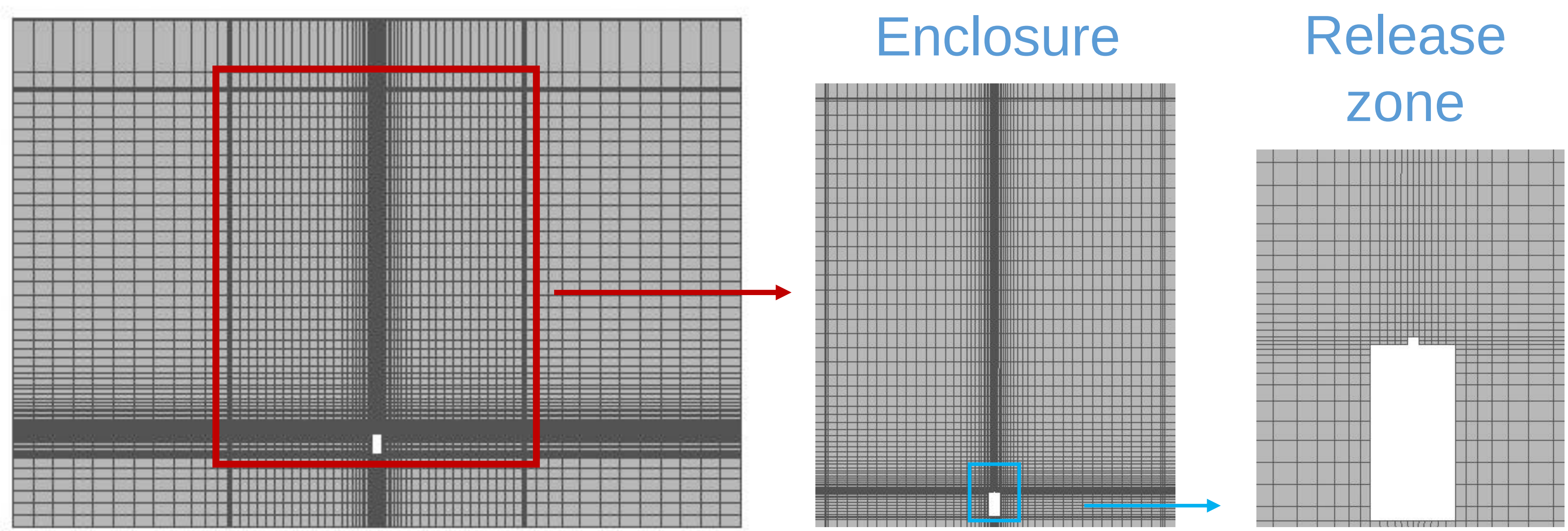
Test	Pressure, bar	Mass flow rate, g/s	N. of vents
14	117.8	11.37	3
18	124.6	11.47	2
19	89.3	8.62	1

- Pressure dynamics
- Temperature dynamics



CFD model
Numerical details

- ❖ ANSYS Fluent platform is used for simulations.
- ❖ Pressure-based solver with compressible ideal gas.
- ❖ RANS approach for turbulence modelling.
- ❖ Eddy Dissipation Concept for combustion modelling.
- ❖ Discrete Ordinates for radiation modelling.



Release source modelling

Notional nozzle approach

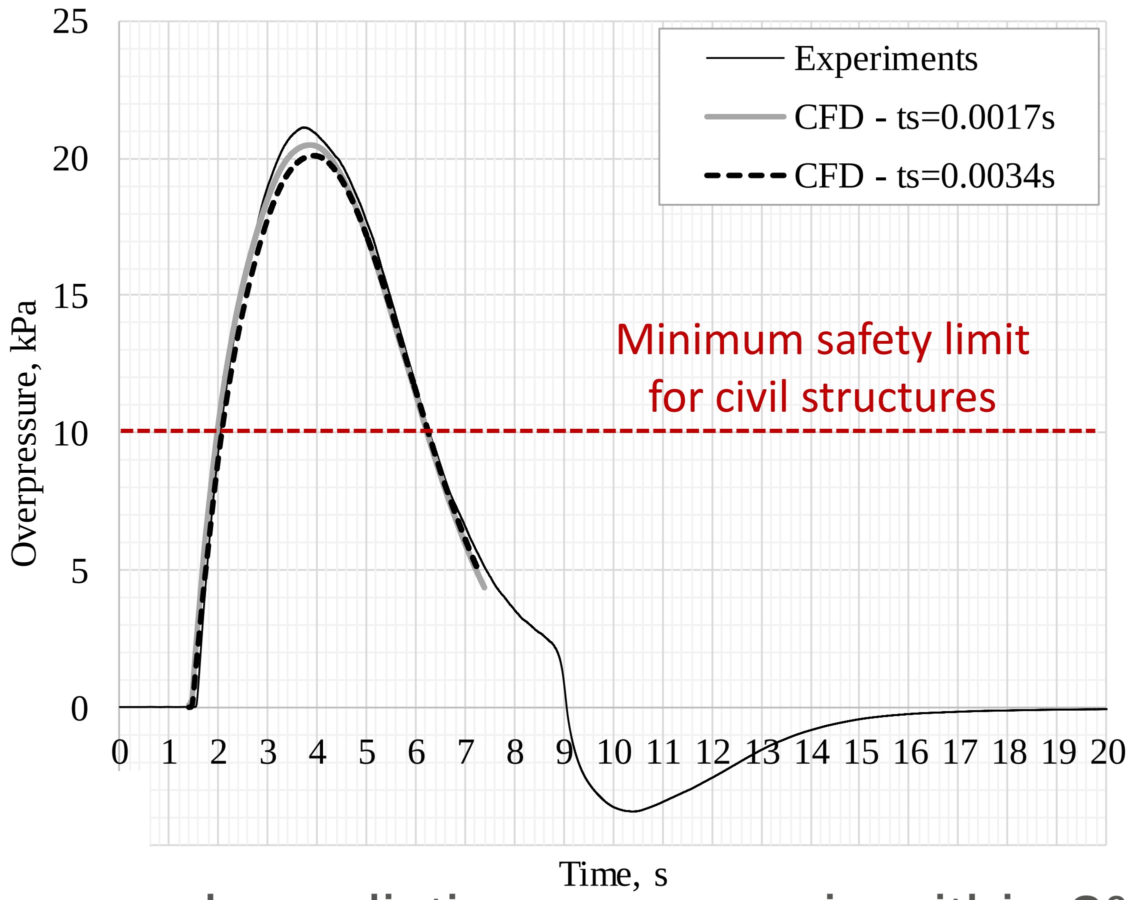
- Storage pressure >1.9 bar, thus an under-expanded jet is expected.
- Release source is modelled using the Ulster's notional nozzle theory.
- A discharge coefficient is introduced to reproduce the experimental mass flow rate and define notional nozzle parameters.

Test	Mass flow rate, g/s	C_d	D_{not}, mm	U_{not}, m/s
14	11.37	0.13	11	1154.5
18	11.47	0.12	11	1156.6
19	8.62	0.12	9.5	1156.6

Pressure dynamics

Simulation results

❖ **Test 14.** Pressure = 117.8 bar, mass flow rate = 11.37 g/s



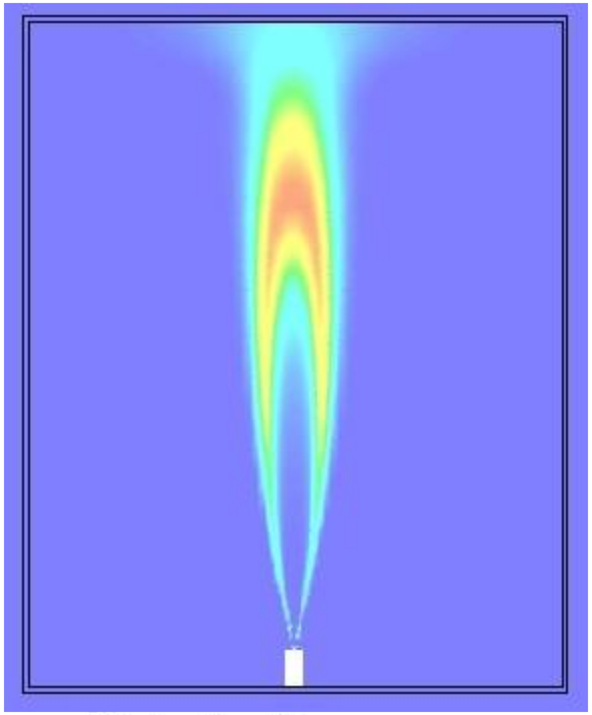
❖ Pressure peak prediction accuracy is within 3% (ts=0.0017s)

OH mole fraction dynamics

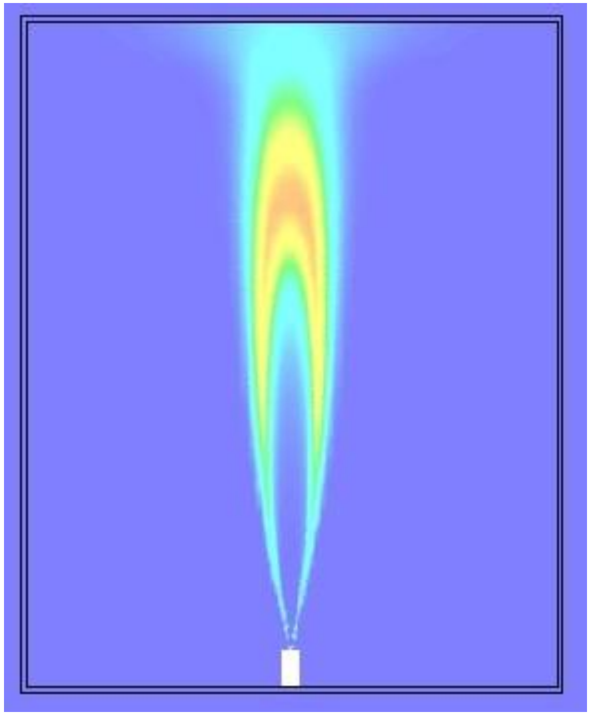
Simulation results

❖ **Test 14.** Pressure = 117.8 bar, mass flow rate = 11.37 g/s

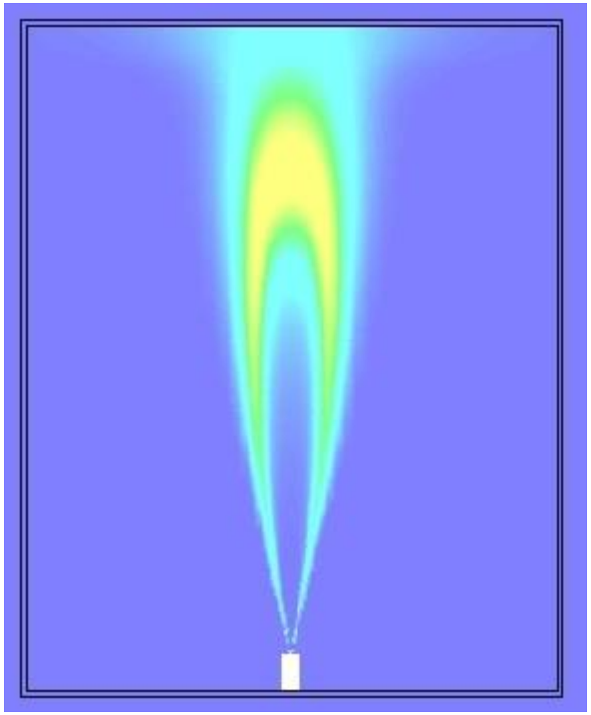
Time = 2.4 s



Time = 3.4 s

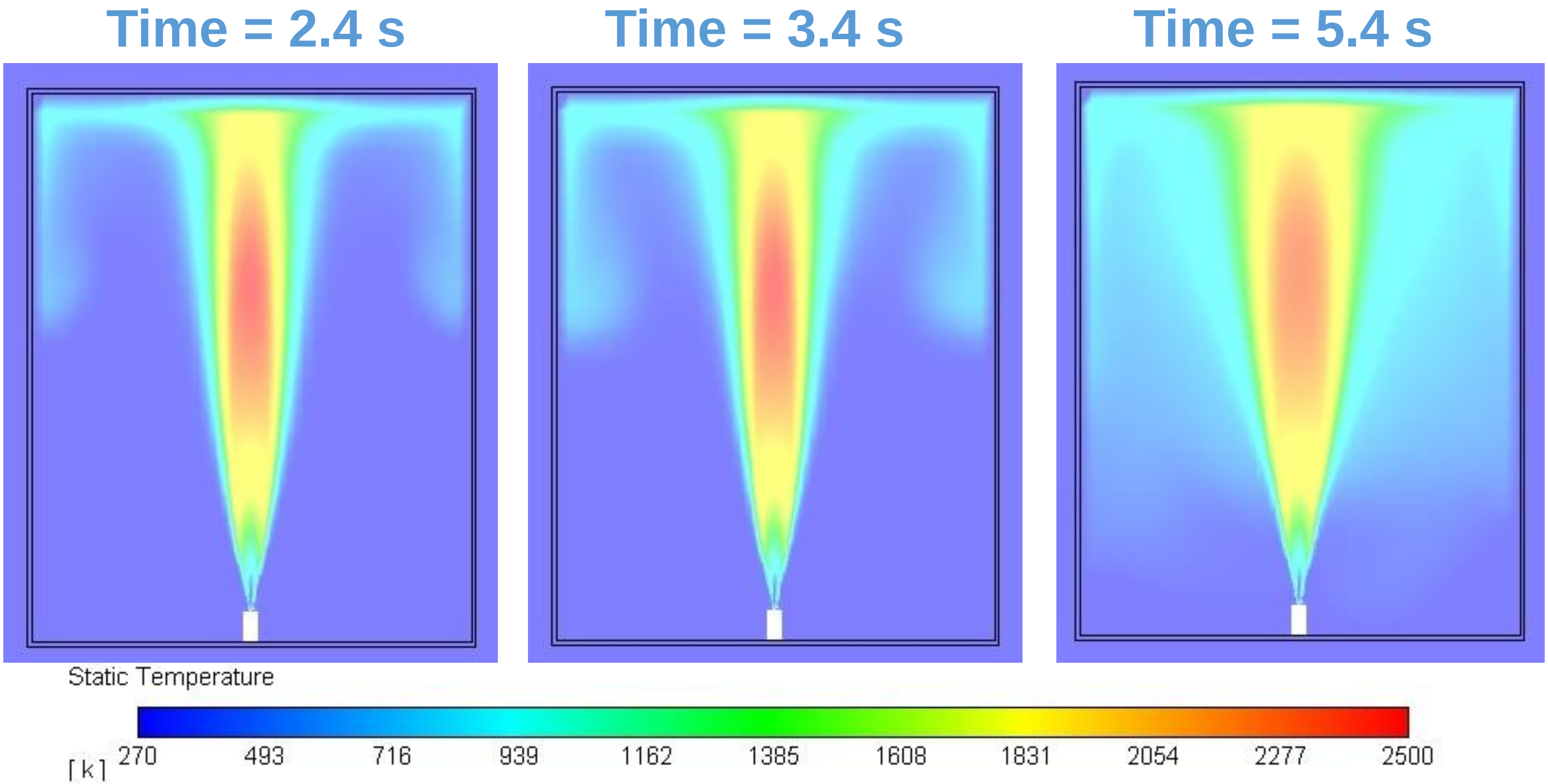


Time = 5.4 s



Temperature dynamics
Simulation results

❖ **Test 14.** Pressure = 117.8 bar, mass flow rate = 11.37 g/s



Concluding remarks
… and future works

- A CFD model is being validated against large scale experiments performed by USN within HyTunnel-CS.
- The CFD model agrees well with the experimentally measured pressure peak and dynamics in the enclosure.
- CFD modelling can be used to model the flame dynamics and thermal load on the enclosure structure.
- The validation range of a CFD model is envisaged to be further expanded to tests with pressure up to 700 bar.
- The CFD model can be employed in hydrogen safety engineering to determine the maximum TPRD size to prevent damage of a structure due to PPP.

Acknowledgements

This project has received funding from the Fuel Cells and Hydrogen 2 Joint Undertaking (JU) under grant agreement No 826193. The JU receives support from the European Union's Horizon 2020 research and innovation programme and United Kingdom, Germany, Greece, Denmark, Spain, Italy, Netherlands, Belgium, France, Norway, Switzerland.