

Hydrogen Vehicle Leak Modelling in Indoor Ventilated Environments

Presented by:

Dr Andrew Hallgarth (ahallgarth@HazRes.com)

Mr Moumen Zayer (azayer@HazRes.com)

Hazard Research and Risk Consultants Ltd (HazRes)

Presentation to:
Comsol Conference 2009, Milan, Italy

Date:
Thursday, 15th October 2009

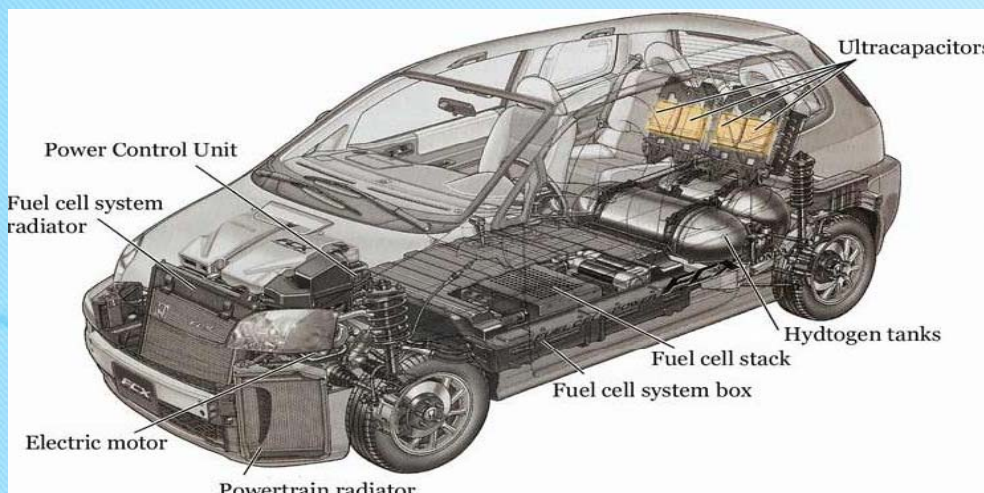
Contents

- Background & Introduction
- Description of Experimental Study Used for Validation
- Mathematical Model
- Validation Against Experimental Results
- Simulation of a Hydrogen Leak from a Vehicle in a Garage
- Conclusion

Background & Introduction

Background & Introduction (1/3)

- Hydrogen is an alternative fuel source instead of using conventional hydrocarbon-based fuels.
- To reduce carbon dioxide emissions, a number of international vehicle manufacturers are developing and building vehicles that use hydrogen.



Picture courtesy of techlover.com

Background & Introduction (2/3)

- Hydrogen has a high diffusion coefficient and a relatively low density compared to air, hence it is naturally very buoyant.
- Hydrogen also has a large flammability range from 4% to 75% in air (under normal gravitational conditions).

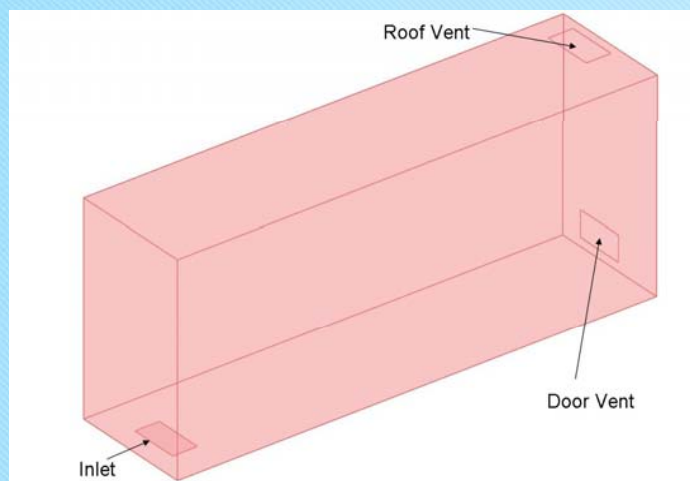
Background & Introduction (3/3)

- Using hydrogen as a mainline transportation fuel has inherent safety problems.
- In this presentation, HazRes looks at just one of many potential safety issues.
- The safety case investigated is a leak from a hydrogen vehicle in a garage.
- This work has potential implications for the storage of hydrogen vehicles with respect to close proximity to occupied buildings (such as our houses).

Description of Experimental Study Used for Validation

Experimental Description

- The experimental set-up (1) is in the form of an indoor, natural ventilated hallway measuring 2.90m by 0.74m, by 1.22m high (i.e. volume = 2.62m³).
- The hydrogen is released vertically, with a velocity 0.02m/s, from a 0.05m² area vent in the floor. There is a roof vent and a lower door vent for the gas ventilation at the right end of the hallway.



1. Kanayama Hiroshi et al, Finite Element Simulation of Hydrogen Dispersion by the Analogy of the Boussinesq Approximation, Journal of Computational Science and Technology, **2(4)**, 643-654 (2008)

Mathematical Model

- The governing conservation equations

$$\nabla \cdot \mathbf{U} = 0$$

$$\rho \frac{\partial \mathbf{U}}{\partial t} - \nabla \cdot \left[\left(\eta + \rho C_\mu \frac{k^2}{\varepsilon} \right) (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) \right] + \rho \mathbf{U} \cdot \nabla \mathbf{U} + \nabla P = \mathbf{F}$$

$$\delta_{is} \frac{\partial c_i}{\partial t} + \nabla \cdot (-D_i \nabla c_i) = R_i - \mathbf{U} \nabla c_i$$

- The density equation

$$\rho = \frac{P_m}{(CR_{H_2} + (1-c)R_{air})T_m}$$

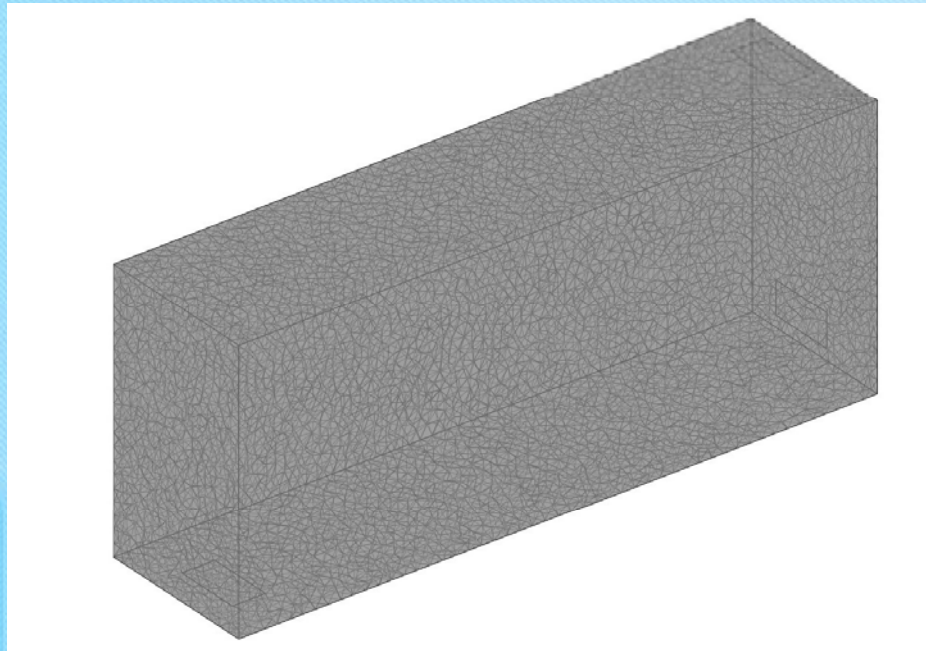
- The buoyancy force was modelled using a formulation similar to the Boussinesq approximation

$$F = -\beta c g$$

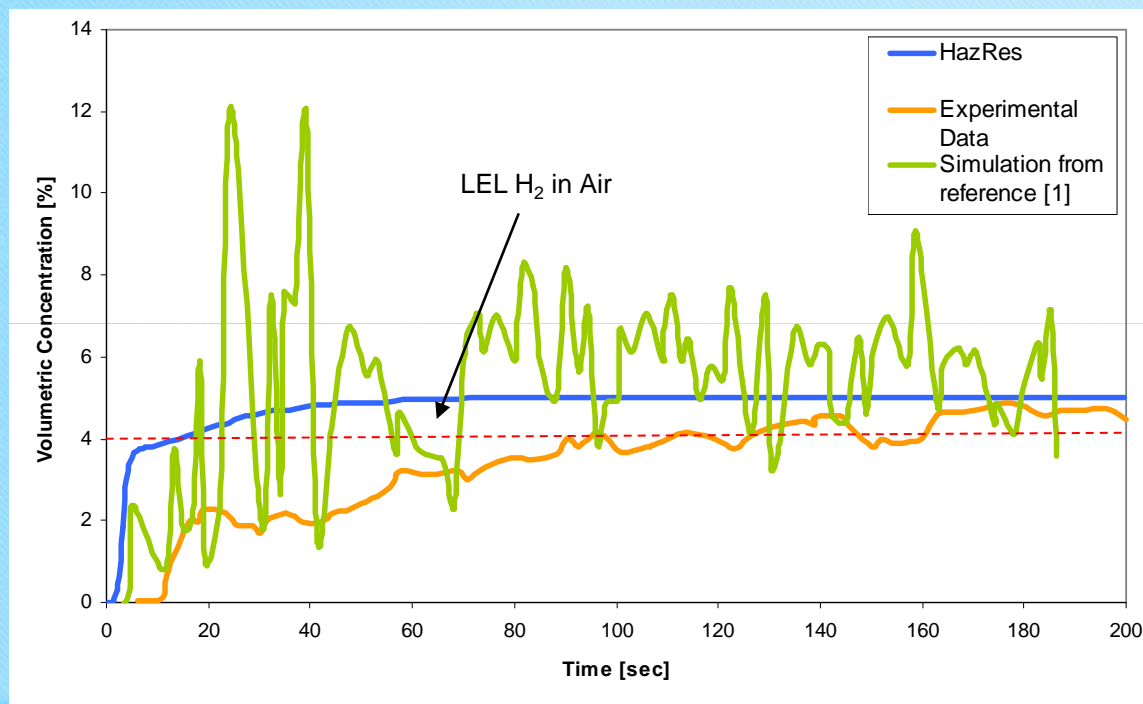
Validation Against Experimental Results

Computational Volume & CFD Dispersion

- A tetrahedral mesh was used. The number of elements (NoE) equates to 43,565 and the associated degrees of freedom (DoF) equate to 385,477.



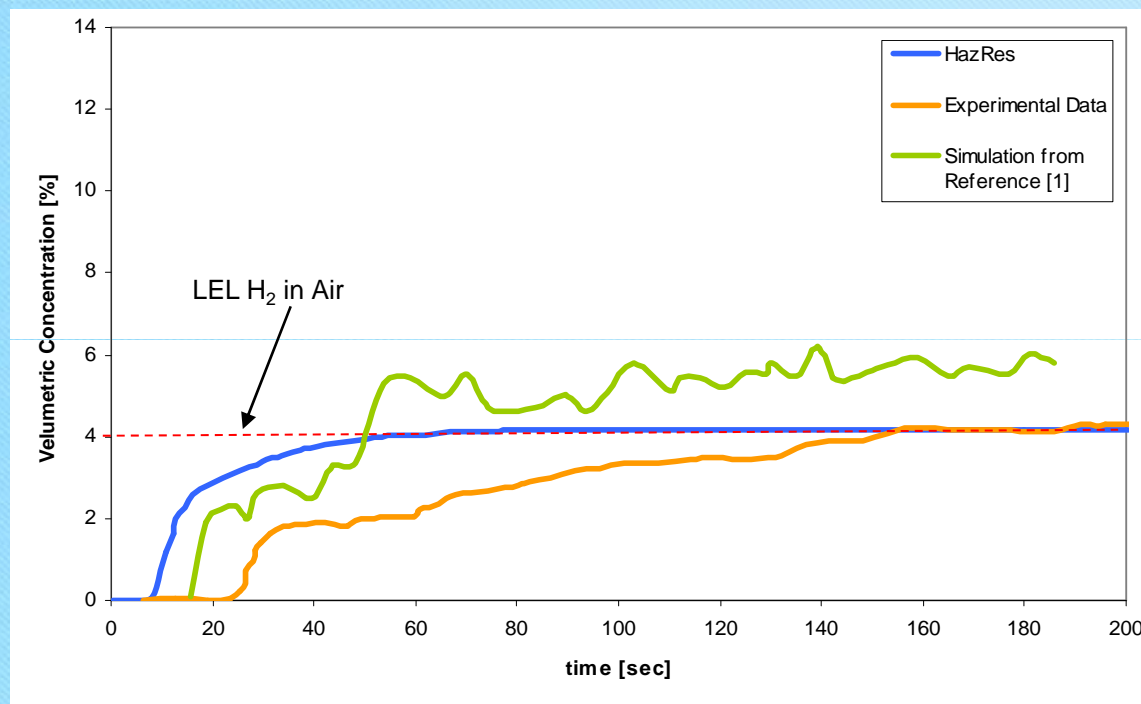
Comparative Study Results



Variation of concentration with time at the sensor located in the ceiling directly above the inlet.

1. Kanayama Hiroshi et al, Finite Element Simulation of Hydrogen Dispersion by the Analogy of the Boussinesq Approximation, Journal of Computational Science and Technology, **2(4)**, 643-654 (2008)

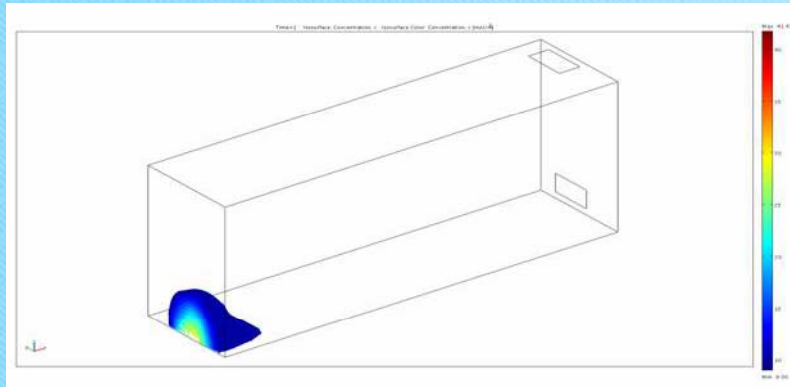
Comparative Study Results



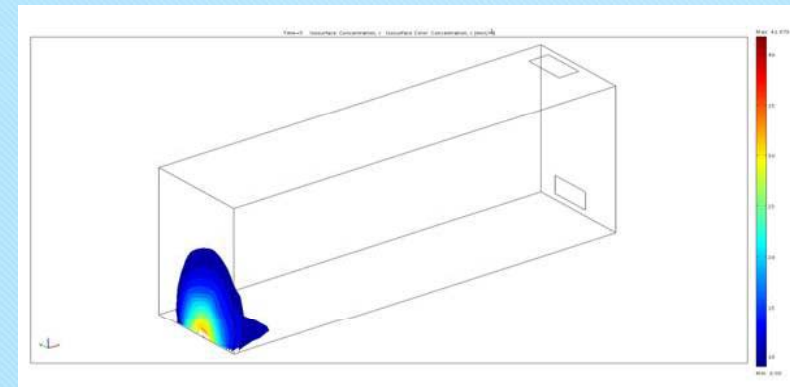
Variation of concentration with time at the sensor located in the ceiling adjacent to the roof vent.

1. Kanayama Hiroshi et al, Finite Element Simulation of Hydrogen Dispersion by the Analogy of the Boussinesq Approximation, Journal of Computational Science and Technology, **2(4)**, 643-654 (2008)

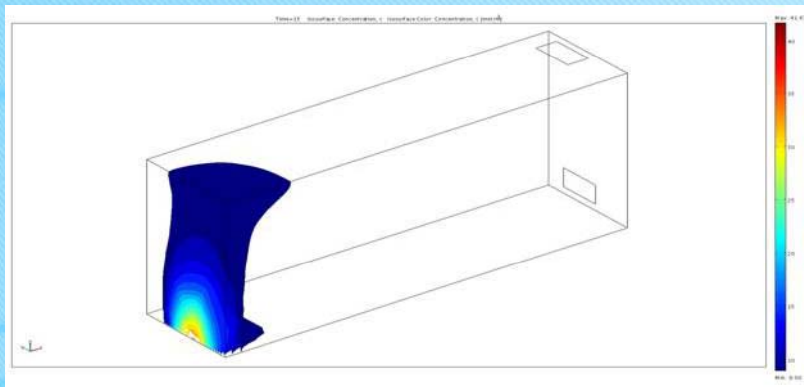
Transient Analysis (1/2)



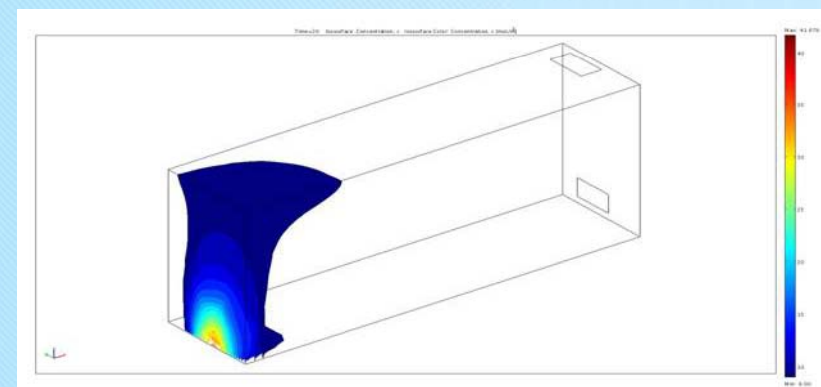
t = 1 sec



t = 5 sec

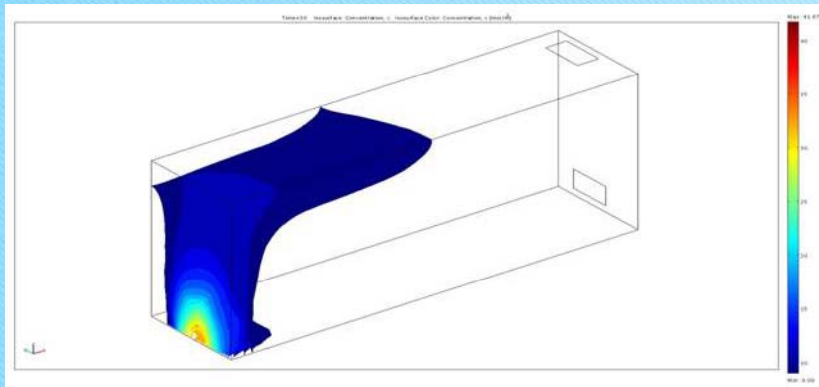


t = 15 sec

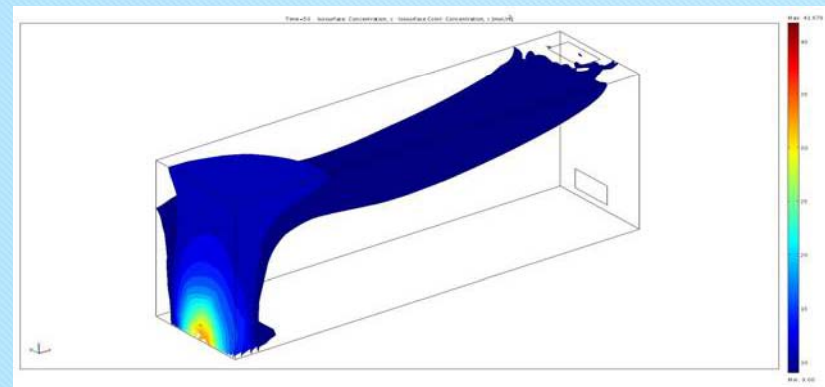


t = 20 sec

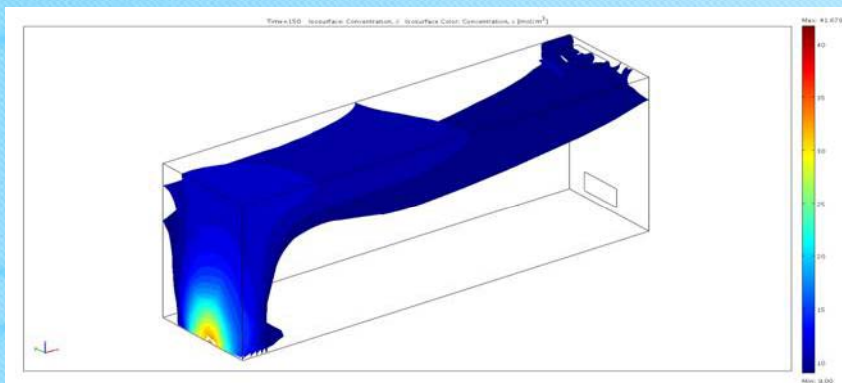
Transient Analysis (2/2)



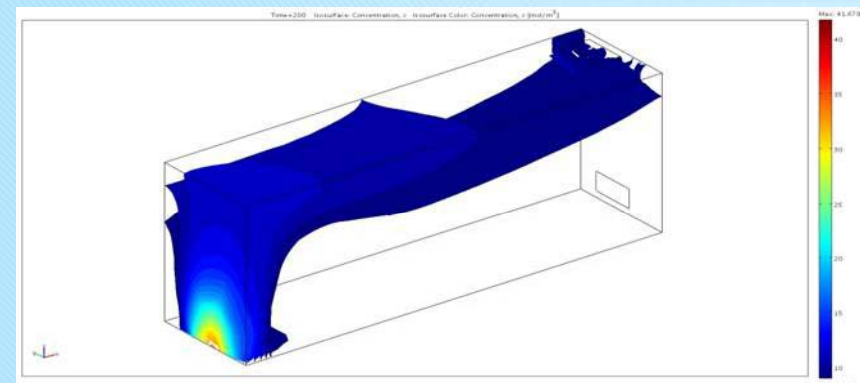
t = 30 sec



t = 50 sec



t = 150 sec



t = 200 sec

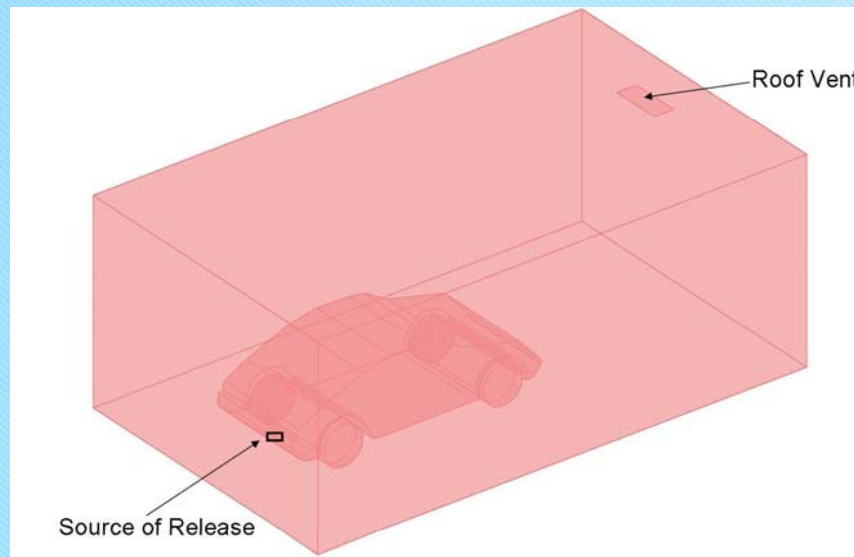
Analysis of Results

- The results agree well with the experimental data, especially when the cloud reaches the steady state where the volumetric concentration is between 4% and 5%.
- The slight discrepancy that can be observed at the start of the simulation can be explained by the fact that we used a stationary solution for the wind profile in order to make the solver converge initially and that we solved the concentration profile transiently, thereafter.

Simulation of a Hydrogen Leak from a Vehicle in a Garage

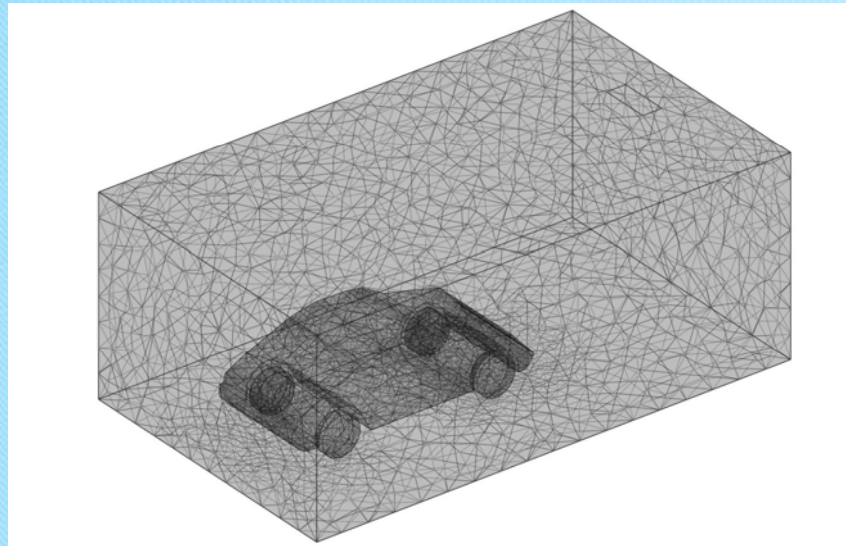
Simulation of a Hydrogen Leak from a Vehicle

- Simulation of hydrogen gas dispersion from a parked car in an enclosed area measuring 10m by 6m, by 4m high (i.e. volume = 240m³).
- The hydrogen was released, with an arbitrary velocity of 5m/s, from a circular hole diameter (0.01m) located underneath the rear of the car.

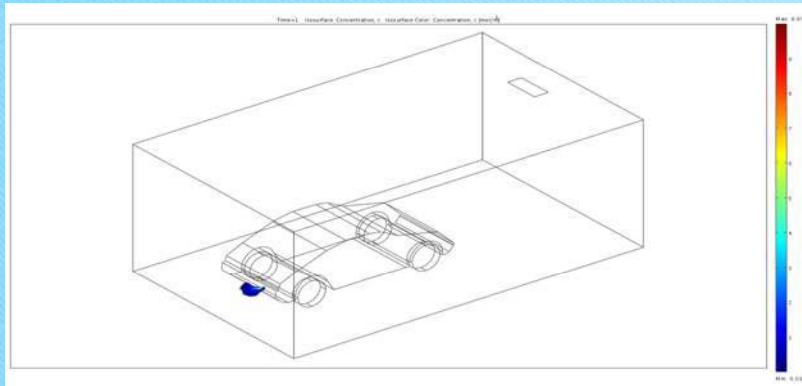


Computational Volume & CFD Dispersion

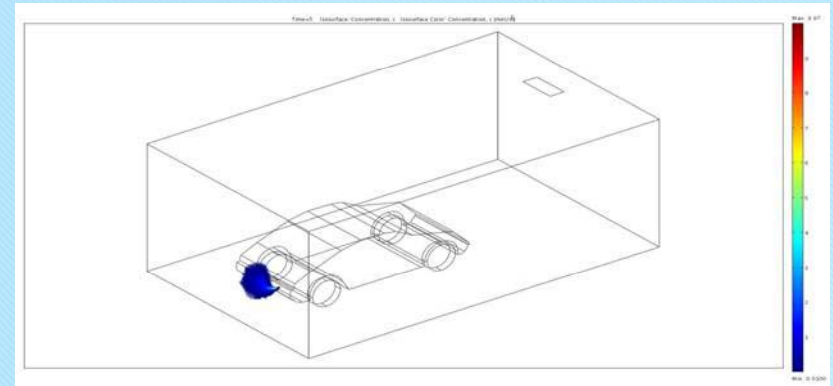
- A tetrahedral mesh was used. The NoE is equal to 272,687 and the DoF is equal to 1,664,351.
- Iso-surface concentration for Hydrogen in Air: 0~10 [volume%].



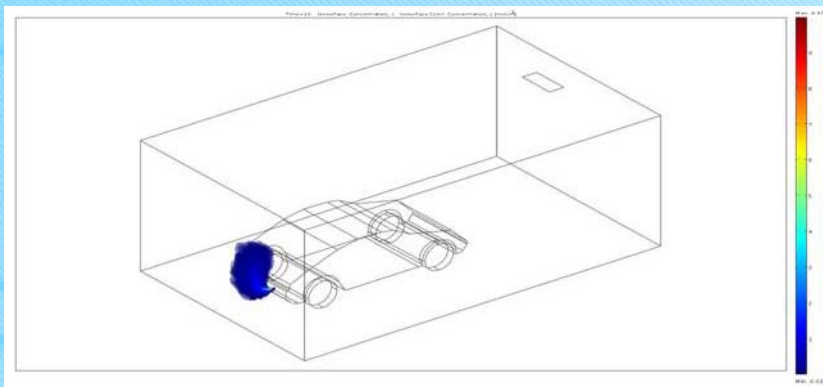
Transient Analysis of a Hydrogen Leak from a Vehicle (1/3)



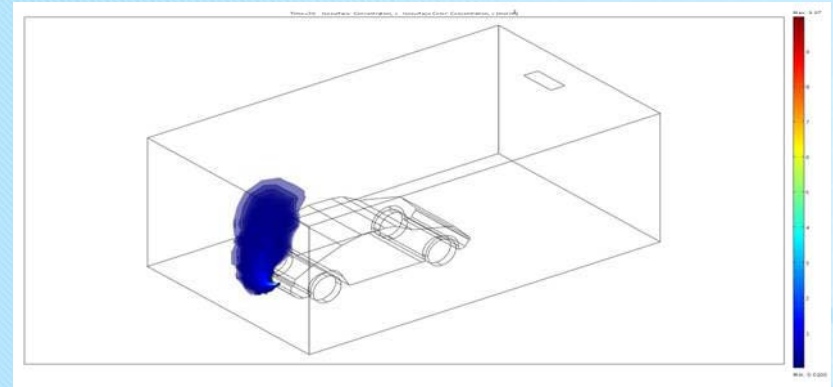
t = 1 sec



t = 5 sec

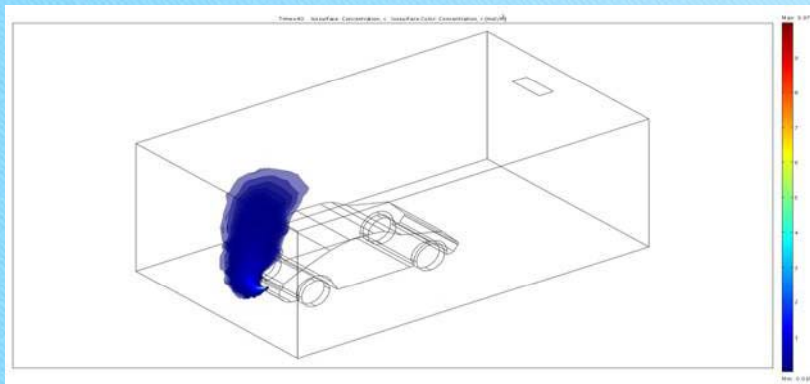


t = 10 sec

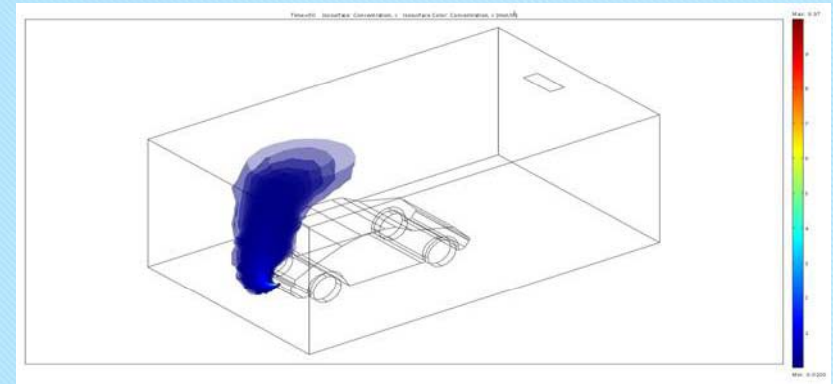


t = 30 sec

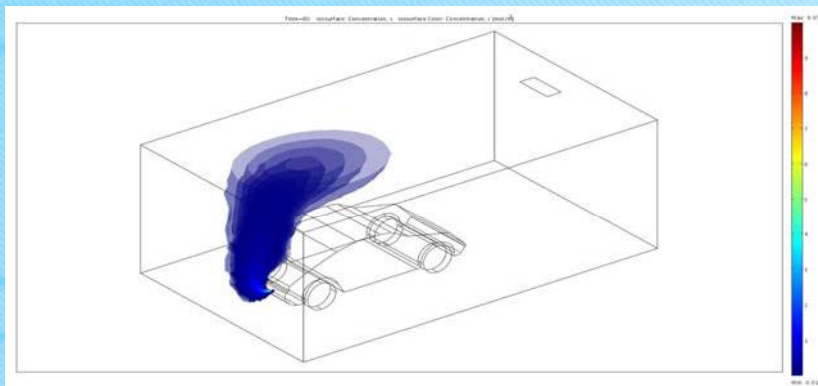
Transient Analysis of a Hydrogen Leak from a Vehicle (2/3)



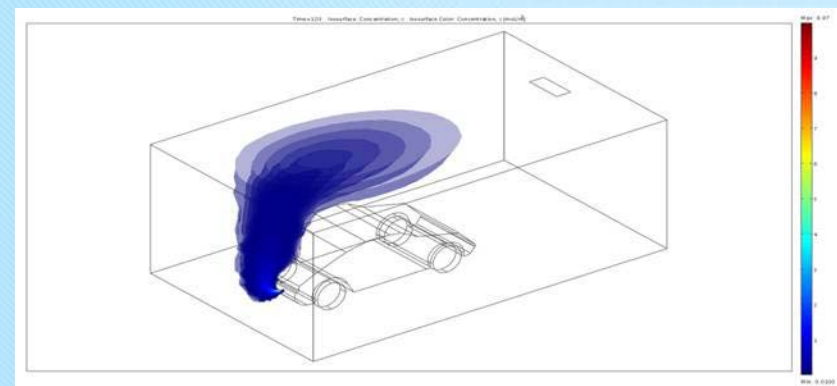
t = 40 sec



t = 60 sec

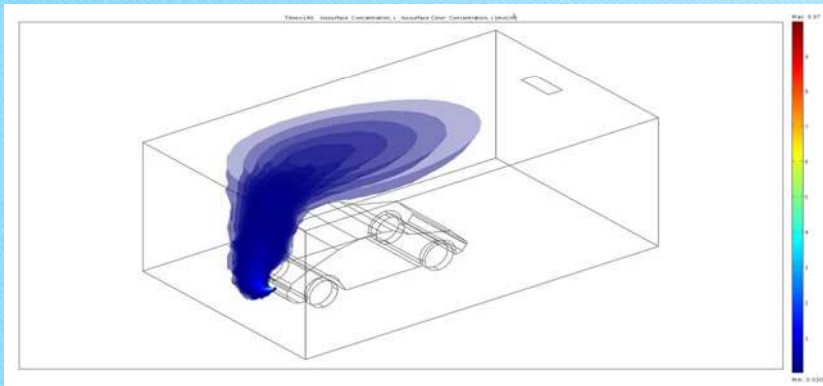


t = 80 sec

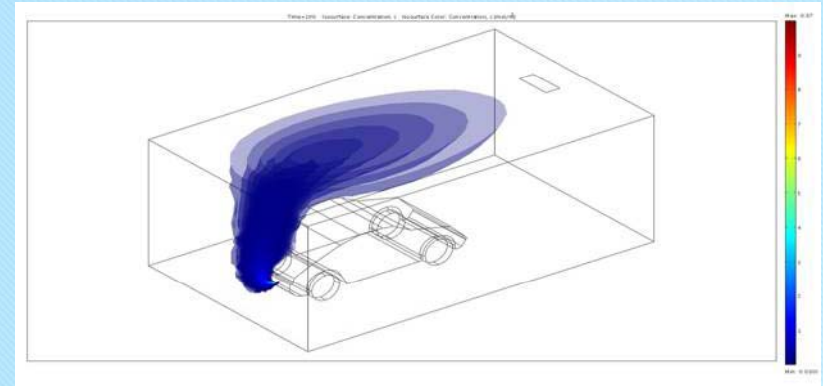


t = 120 sec

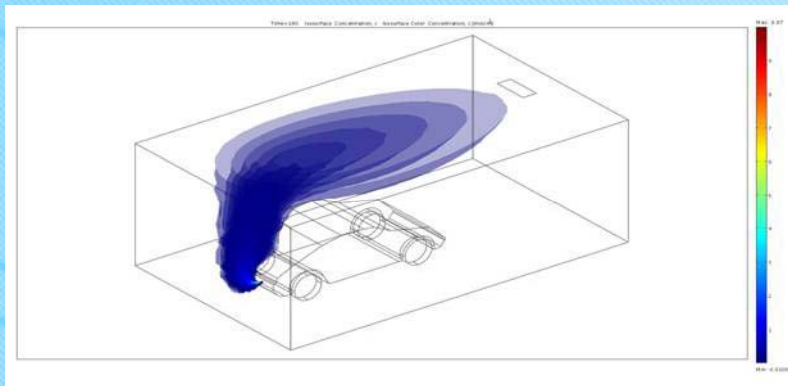
Transient Analysis of a Hydrogen Leak from a Vehicle (3/3)



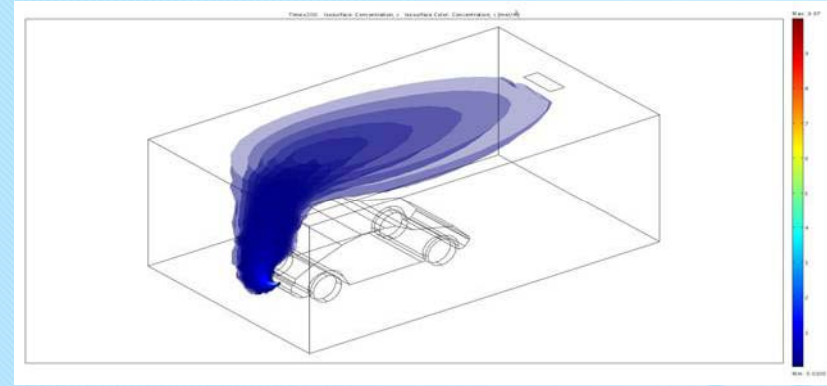
t = 140 sec



t = 160 sec



t = 180 sec



t = 200 sec

Conclusion

- HazRes has developed a CFD gas dispersion model, using COMSOL Multi-Physics, to simulate buoyant gas dispersion.
- The model is in good agreement with the experimental data, especially when the cloud reaches the steady-state, and has been applied to simulate hydrogen gas leaks and dispersion from a vehicle in a garage.
- This analysis shows that HazRes' CFD gas dispersion model, using COMSOL Multi-Physics, is a cost-effective tool for evaluating complex safety case problems, which involve hydrogen due to its unique characteristics as a renewable fuel energy carrier.
- HazRes' CFD model using COMSOL Multi-Physics can be used for other hydrogen safety case scenarios relevant to the energy sector.

Thank You for Your Attention !

Questions?

Please visit our website at: www.HazRes.com

E-mail: info@HazRes.com

Tel.: +44 (0)845 838 2026 / Fax: +44 (0)845 838 2027