HyTunnel-CS dissemination conference
14-15 July 2022, Brussels, Belgium

Effect of counter- and co-flow on hydrogen jets: simulations versus experiment

S.G. Giannissi[1], I.C. Tolias[1], H. Ebne Abbasi[2], D. Makarov[2], J. Grune[3], K. Sempert[3]

[1] Environmental Research Laboratory, National Center for Scientific Research Demokritos, Aghia Paraskevi, Athens, 15341, Greece, sgiannissi@ipta.Demokritos.Gr

[2] HySAFER centre, University of Ulster, Newtownabbey BT37 0QB, UK

[3] Pro-Science GMBH, Parkstrasse 9, Ettlingen, 76275, Germany

Clean Hydrogen Partnership

Co-funded by the European Union

Scope

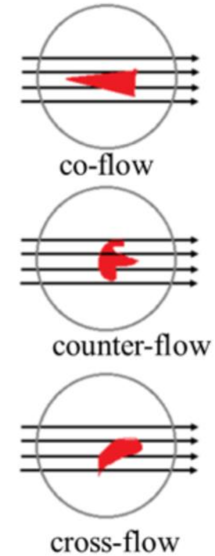
Efficiency of mechanical ventilation using CFD

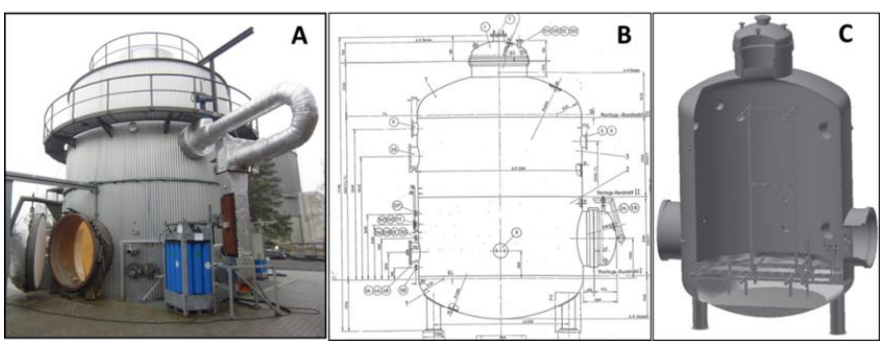
- Inter-comparison of CFD simulations to model ventilated hydrogen dispersion in presence of different ventilation configurations: co-flow and counter-flow.
- Provide recommendations on proper modeling of airflow generated from fans.
- Recommend tools and methodologies that can be used to perform safety assessments and to contribute to RCS development for hydrogen powered vehicles in confined spaces.

Description of experiments

PS experiments

- Pro-Science conducted experiments with compressed H2 release inside a safety vessel of with total volume of 220 m^3.
- Two large flanges with D=1890 mm were open.
- The propeller fan for mechanical ventilation was placed in one of the vent, while the other vent was kept open.





H_2 jet nozzle	1 mm				1 mm				4 mm				4 mm			
H_2 mass flow rate, g/s	1				5 (cross-flow: 1.5)				1				5 (cross-flow: 2.5)			
Ventilation flow velocity, m/s	0	1.5	3.5	5	0	1.5	3.5	5	0	1.5	3.5	5	0	1.5	3.5	5
Co-flow	1	2	5	8	11	12	15	18	21	22	25	28	31	32	35	38
Counter-flow		3	6	9		13	16	19		23	26	29		33	36	39
Cross-flow		4	7	10	41	14	17	20		24	27	30	42	34	37	40

Simulation set-up

Comparison between approaches

NCSRD

- ADREA-HF code. Solves the time dependent 3D conservation equations with k-ε turbulence model
- Birch 84 approach for notional nozzle
- MUSCL scheme for the convective terms, central differences for the diffusive terms and 1st order implicit scheme for time integration
- The entire safety vessel not modelled
- Cartesian grid. The total number of cells is ranged from 180,576 to 264,384.

UU

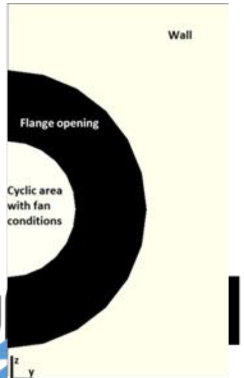
- ANSYS Fluent 2020R1. Solves the 3D conservation equations with k-ε turbulence model
- Molkov, 2012 approach for notional nozzle. However, due to long piping T=Tamb similar to Birch84
- Steady-state formulation of the pressure-based coupled solver. MUSCL scheme was used for discretization of all transported variables with the second-order approximation of pressure gradients
- Hexahedral mesh for the 1st airflow modelling approach with 244 944 cells and hybrid mesh (hexa and tetra) for the 2nd airflow modeling approach where the entire safety vessel is modeled with 781,107 cells

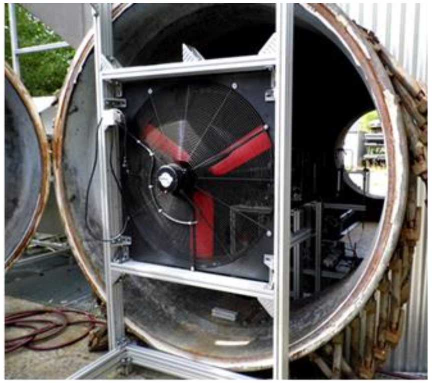
Simulation set-up

Comparison between approaches

NCSRD

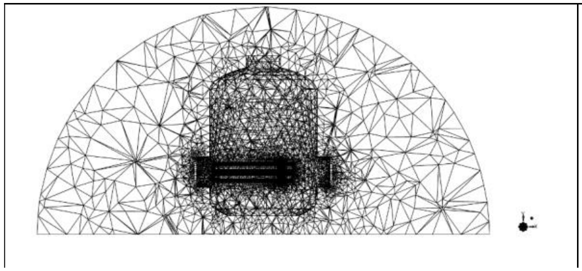
- Airflow modeling
  - **Uniform velocity profile** across airflow plane: uniform velocity in the entire airflow plane with BC given value for velocity, turbulent kinetic energy and dissipation rate. TI= 10% and 50% was examined. L=the propeller diameter.
  - **Gaussian velocity profile** across fan plane: A simulation without release was first carried out to calculate the 3D steady state flow field generated by the fan. Gaussian velocity profile was established. The 3D steady state simulation was set as initial and inflow boundary condition in the simulation with hydrogen release.

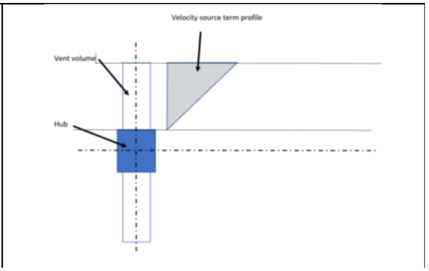




UU

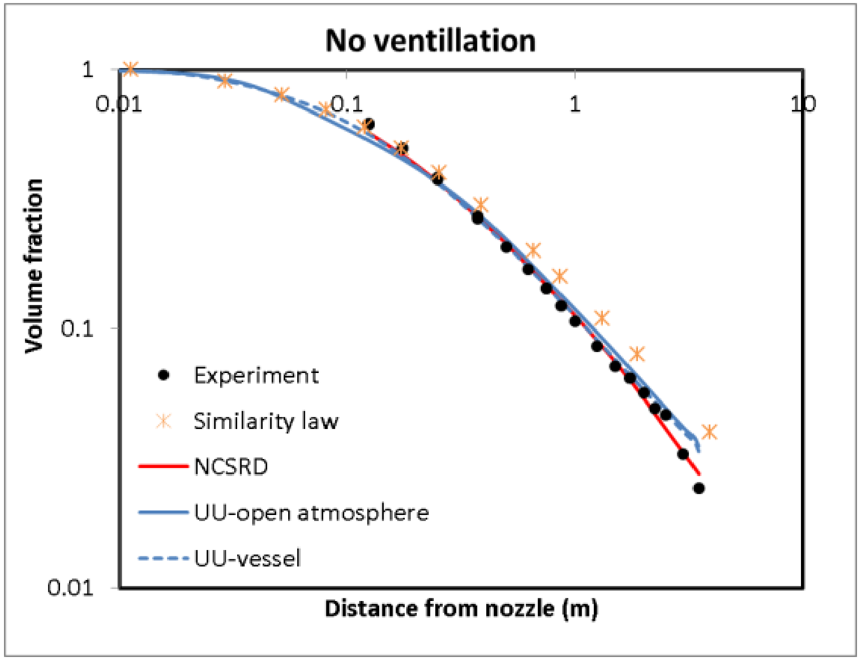
- Airflow modeling
  - **Uniform velocity profile** in open atmosphere: the effect of the fan was simulated imposing pressure gradient 15 Pa to reach flow velocity 5 m/s at the corresponding inflow boundary. TI=50%. L=0.2 m (distance between two strokes of propeller blades with three blades)
  - **Linear source term in vessel geometry**: the domain included the vessel itself. For co-flow and counter-flow volumetric source term for x-momentum conservation equation was set, whose value was growing linearly with radius Su=6201.6(R-Rhub) N/m^3, where R is the radius from the fan centre and Rhub=0.133 m is the radius of the fan hub.

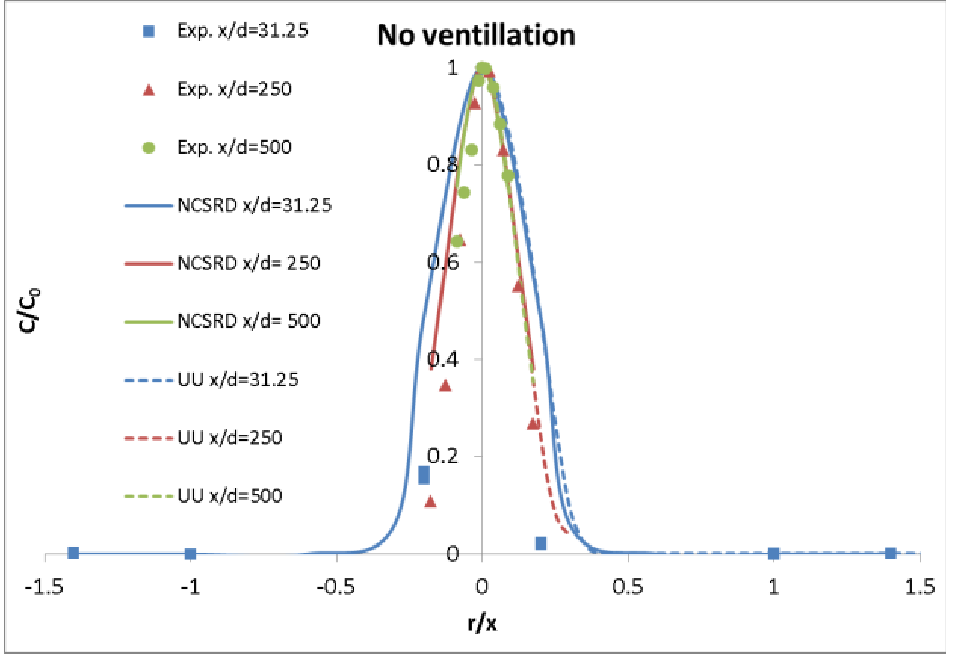




Results

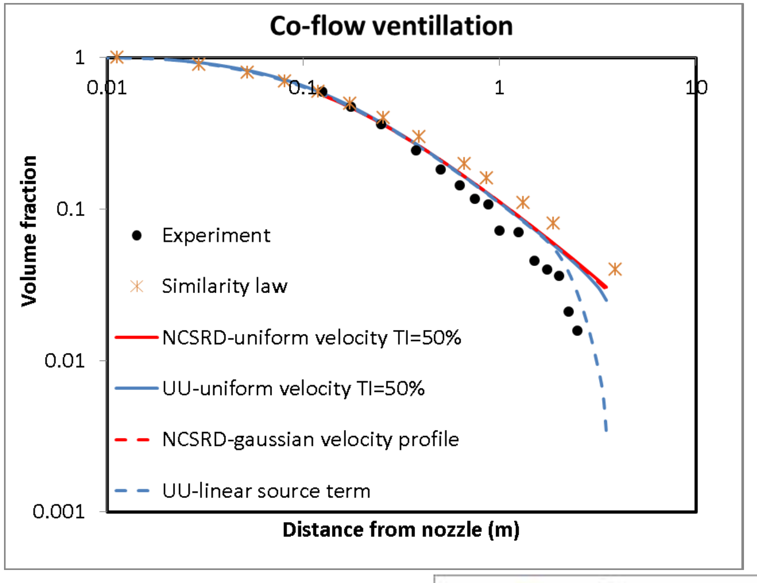
No ventillation

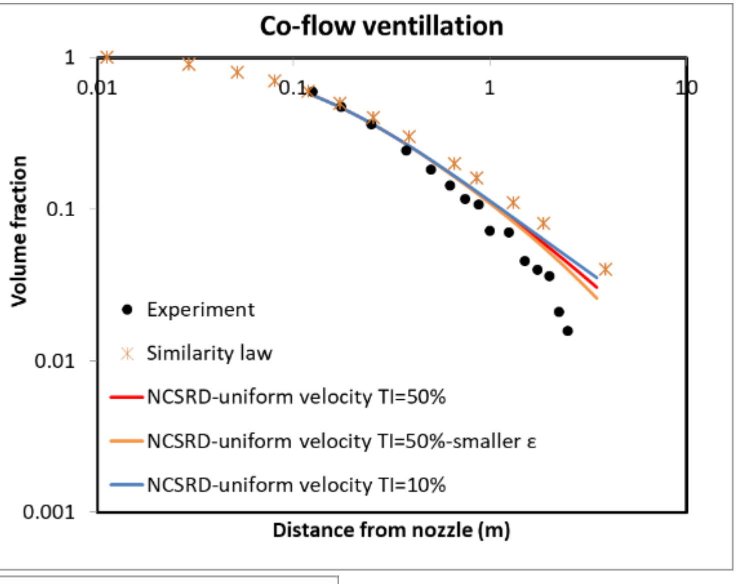


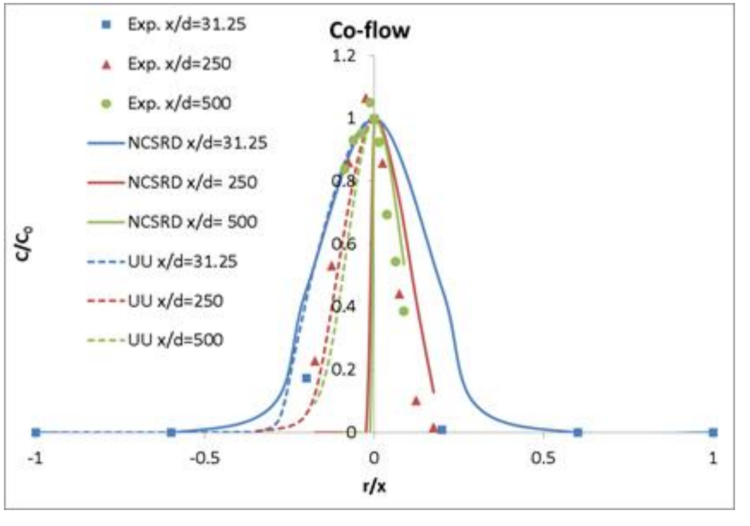


Results

Co-flow

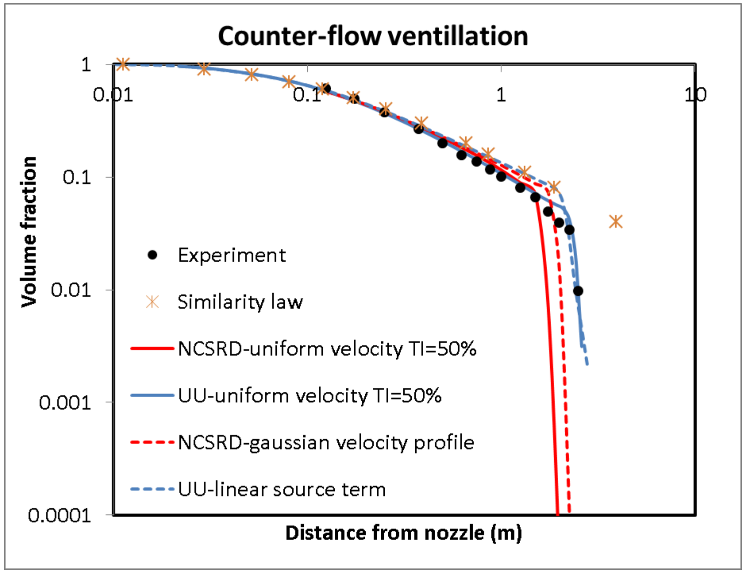


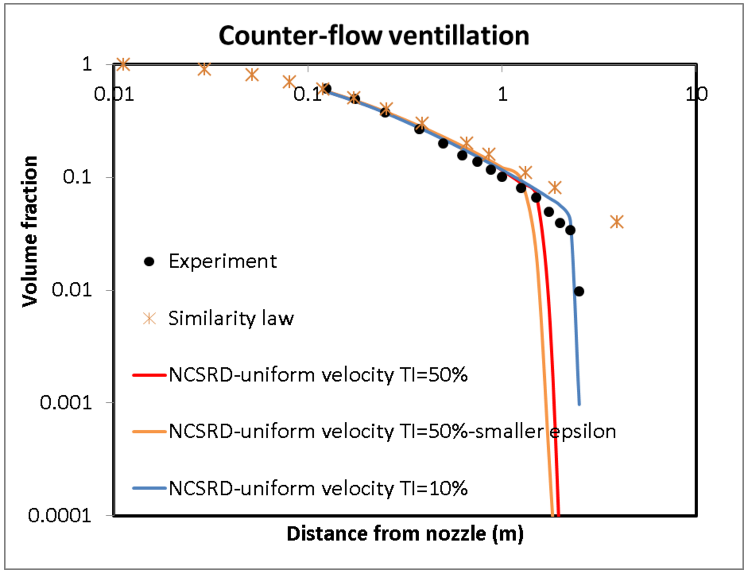


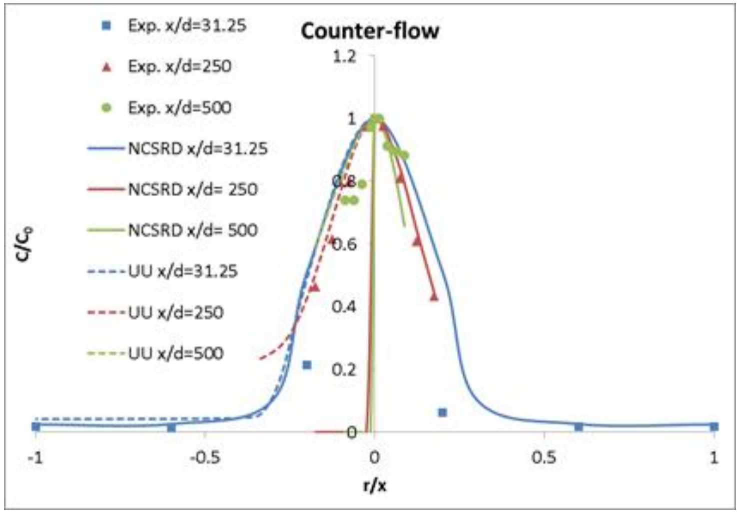


Results

Counter-flow





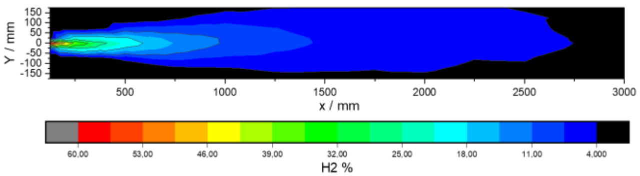


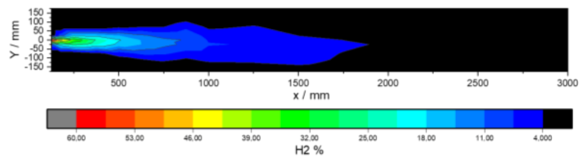
Discussion

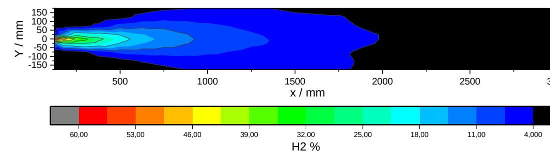
Concentration contours

No ventilation | **Co-flow** | **Counter-flow**

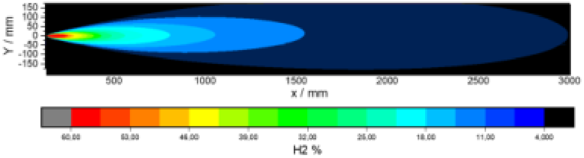
Experiment

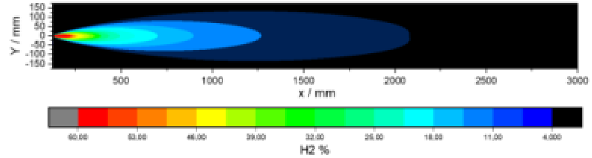


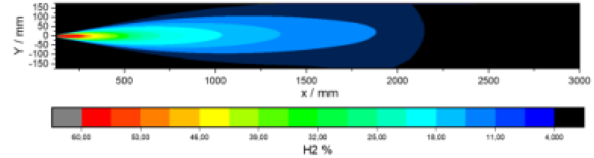




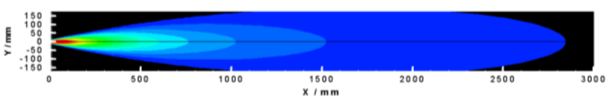
UU-linear source term

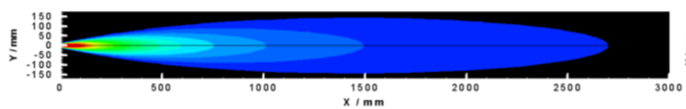


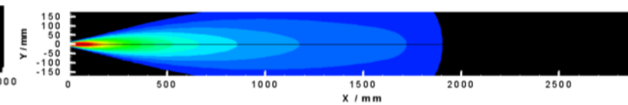




NCSRD-gaussian profile



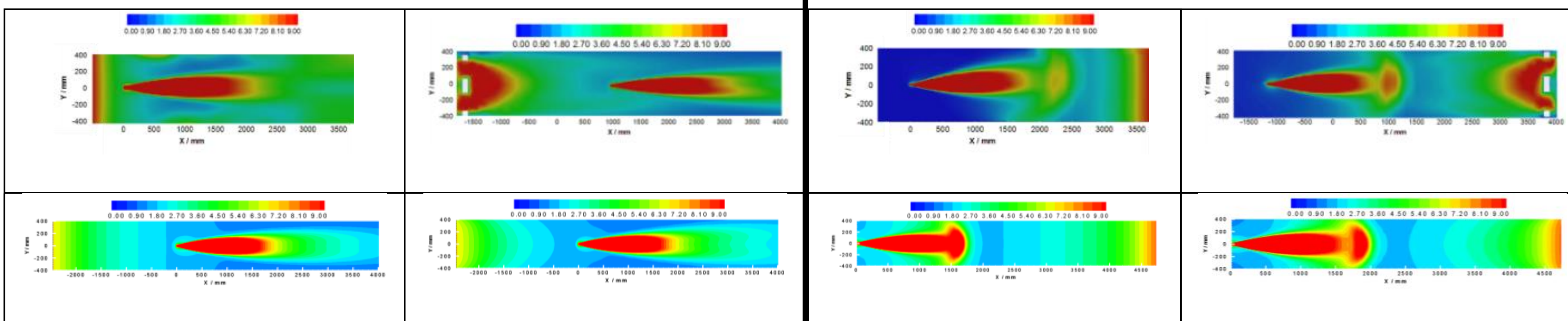




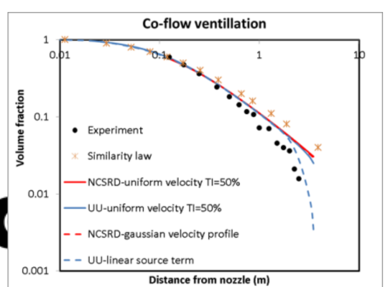
Discussion

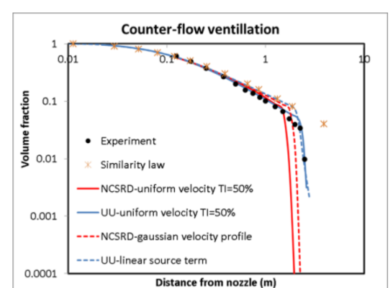
Turbulent kinetic energy contours

Co-flow		Counter-flow	
UU-uniform profile	UU-linear source term	UU-uniform profile	UU-linear source term



NCSRD-uniform profile TI=50%	NCSRD-gaussian profile	NCSRD-uniform profile TI=50%	NCSRD-gaussian profile





Conclusions

…and recommendations

- Inter-comparison between NCSRD and UU simulations based on Pro-Science experiments with the aim to:
  - **study the efficiency of mechanical ventilation** in high-pressure hydrogen release inside an enclosure, such as parking or tunnel, and
  - **provide recommendations** on modeling ventilated hydrogen dispersion.
- Scenarios simulated: hydrogen release (**d=4 mm, $\dot{m}$=5 g/s**) with no ventilation, **co-flow** and **counter-flow** (u = 5 m/s).
- In the **no ventilation** configuration,
  - good agreement with the experiment with UU simulations to over-predict the LFL distance. **Modelling the entire vessel is not essential**, because the large volume of the vessel does not affect dispersion.
- In the **co-flow** configuration,
  - **linear source term approach** mimics best the actual fan flow and gives the best agreement with experiment. The rest approaches over-predict the concentration at further distances downwind the nozzle. Uniform velocity profile requires large imposed turbulence intensity (50%) and small values of ε for better predictions.
- In **counter-flow** configuration,
  - the approaches **UU-uniform profile with TI=50%** and **NCSRD-uniform profile with TI=10%** capture better the interaction of airflow with the jet. However, the linear source term approach and the gaussian velocity profile approach predict more accurately the LFL distance (5% deviation).

Conclusions

…and recommendations

- The predicted **radial profiles** of volume fraction up to 2 m from the nozzle were in close agreement with the experiment with a tendency to predict a slightly wider cloud.
- **Similarity law** provides conservative results. Satisfactory agreement with the **no ventilation case**
- **To sum up:**
  - to mimic the fan flow in simulations, either high turbulence should be imposed or approaches with non-uniform velocity field across inflow boundary should be applied.
  - The position of the ventilation boundary in the co-flow and counter-flow configuration can have also an impact on results.
- **Some recommendations for the safer use of hydrogen**:
  - Both co-flow and counter-flow configuration lead to reduction of the LFL distance compared to no ventilation (up to 30% for hydrogen jet with a mass flow rate of 5 g/s through 4 mm nozzle). This is attributed to the better mixing and dilution of the cloud.
  - CFD codes can be used as predictive safety tools for high-pressure hydrogen release inside confined spaces with mechanical ventilation provided that the generated by the fan turbulence is properly modeled.
  - The similarity law can be used as a conservative tool for cases with ventilation (however, the decay was not well reproduced in the counter flow). Validated CFD models are recommended as hydrogen safety engineering tools.

Thank you for your attention

Acknowledgments

This project has received funding from the Fuel Cells and Hydrogen 2 Joint Undertaking (now Clean Hydrogen Partnership) under grant agreement No 826193.
The JU receives support from the European Union's Horizon 2020 Research and Innovation program, Hydrogen Europe and Hydrogen Europe Research.

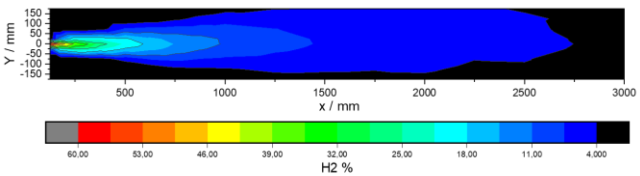
.

Discussion

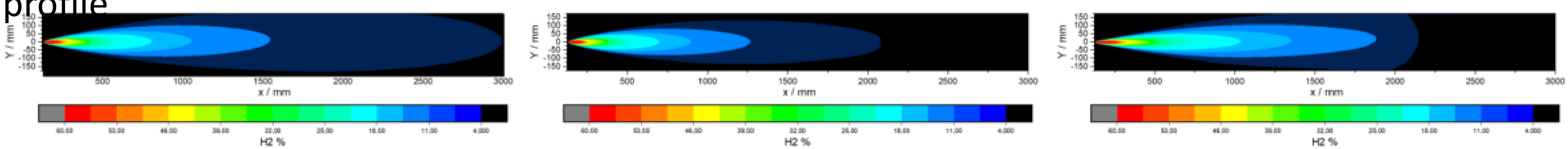
Concentration contours

No ventilation — Co-flow — Counter-flow

Exp

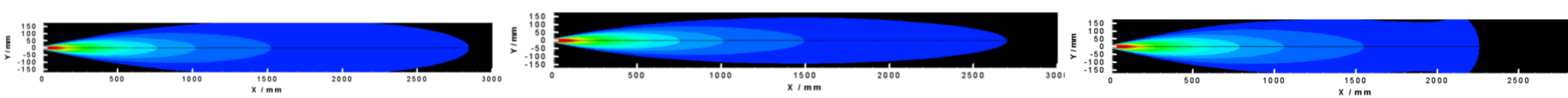


UU-linear source profile



NCSRD-gaussian profile

NCSRD-uniform profile with TI=10%



Outline

- Scope
- Description of experiments
- Simulation set-up
- Results
- Discussion
- Conclusions

Conclusions

…and recommendations

- Inter-comparison between NCSRD and UU simulations based on Pro-Science experiments with the aim to:
  - **study the efficiency of mechanical ventilation** in high-pressure hydrogen release inside an enclosure, such as parking or tunnel, and
  - **provide recommendations** on modeling ventilated hydrogen dispersion.
- Scenarios simulated: hydrogen release (**d=4 mm, $\dot{m}$=5 g/s**) with no ventilation, **co-flow** and **counter-flow** (u = 5 m/s).
- In the **no ventilation** configuration,
  - good agreement with the experiment with UU simulations to over-predict the LFL distance. **Modelling the entire vessel is not essential**, because the large volume of the vessel does not affect dispersion.
- Special focus on the modelling of the flow generated by the fan:
  - UU modelled the fan airflow by

    1) imposing uniform velocity profile and TI=50% in open atmosphere and

    2) imposing linear source term with zero velocities at the fan centre and include the entire vessel geometry.
  - NCSRD modelled the fan airflow by

    1) imposing uniform velocity profile and TI=50% and

    2) imposing gaussian velocity profile. The vessel was not modelled in both NCSRD approaches.

    In the approach with uniform profile, different length scales (consequently epsilon values) and smaller TI (=10%) were also tested.

Conclusions
…and recommendations

- In the **co-flow** configuration,
  - **linear source term approach** mimics best the actual fan flow and gives the best agreement with experiment. The rest approaches over-predict the concentration at further distances downwind the nozzle. Uniform velocity profile requires large imposed turbulence intensity (50%) and small values of ε for better predictions.
- In **counter-flow** configuration,
  - the approaches **UU-uniform profile with TI=50%** and **NCSRD-uniform profile with TI=10%** capture better the interaction of airflow with the jet. However, the linear source term approach and the gaussian velocity profile approach predict more accurately the LFL distance (5% deviation).
- The predicted **radial profiles** of volume fraction up to 2 m from the nozzle were in close agreement with the experiment with a tendency to predict a slightly wider cloud.
- **Similarity law** provides conservative results. Satisfactory agreement with the **no ventilation case**
- **To sum up:**
  - to mimic the fan flow in simulations, either high turbulence should be imposed or approaches with non-uniform velocity field across inflow boundary should be applied.
  - The position of the ventilation boundary in the co-flow and counter-flow configuration can have also an impact on results.

Conclusions

…and recommendations

- **Some recommendations for the safer use of hydrogen:**
  - Both co-flow and counter-flow configuration lead to reduction of the LFL distance compared to no ventilation (up to 30% for hydrogen jet with a mass flow rate of 5 g/s through 4 mm nozzle). This is attributed to the better mixing and dilution of the cloud.
  - CFD codes can be used as predictive safety tools for high-pressure hydrogen release inside confined spaces with mechanical ventilation provided that the generated by the fan turbulence is properly modeled.
  - The similarity law can be used as a conservative tool for cases with ventilation (however, the decay was not well reproduced in the counter flow). Validated CFD models are recommended as hydrogen safety engineering tools for cases of comparatively low release pressure when the jet is “less momentum-dominated”.