# IA-HYSAFE STANDARD BENCHMARK EXERCISE SBEP-V21: HYDROGEN RELEASE AND ACCUMULATION WITHIN A NON-VENTILATED AMBIENT PRESSURE GARAGE AT LOW RELEASE RATES

**A.G. Venetsanos[1], I. Tolias[2], D. Baraldi[7], S. Benz[5], B. Cariteau[2], J. Garcia[3], O.R. Hansen[4], C. Jäkel[6], S. Ledin[8], P. Middha[4], E.A. Papanikolaou[7]**

[1] Environmental Research Laboratory, National Centre for Scientific Research Demokritos (NCSRD), 15310 Aghia Paraskevi, Attikis, Greece, *venets@ipta.demokritos.gr*

[2] C.E.A. Saclay, D.E.N., D.M.2S., S.F.M.E., Laboratoire d'Etude Expérimentale des Fluides, 91191 Gif/Yvette cedex FRANCE

[3] Escuela Técnica Superior de Ingenieros Industriales, Universidad Politécnica de Madrid (UPM), José Gutiérrez Abascal, 2, E-28006 Madrid, Spain

[4] GEXCON AS, Fantoftvegen 38 Box 6015 Postterminalen N-5892 BERGEN Norway

[5] IKET, KIT, Postfach 3640, 76021 Karlsruhe, Germany

[6] Forschungszentrum Juelich (FZJ), 52425 Juelich, Germany

[7] Joint Research Centre of the European Commission (JRC), Institute for Energy, 1755 ZG Petten, The Netherlands

[8]Health and Safety Laboratory (HSL), Harpur Hill, Buxton, Derbyshire, SK17 9JN, UK

**Abstract**

The successful Computational Fluid Dynamics (CFD) benchmarking activity, originally started within the EC-funded Network of Excellence HySafe (2004-2009), continues within the research topics of the recently established "International Association of Hydrogen Safety" (IA-HySafe). The present contribution reports the results of the standard benchmark problem SBEP-V21. Focus is given to hydrogen dispersion and accumulation within a non-ventilated ambient pressure garage both during the release and post-release periods but for very low release rates as compared to earlier work (SBEP-V3). The current experiments were performed by CEA at the GARAGE facility under highly controlled conditions. Helium was vertically released from the centre of the 5.76 m (length) x 2.96 m (width) x 2.42 m (height) facility, 22 cm from the floor, from a 29.7 mm diameter opening at a volumetric rate of 18 L/min (0.027 g/s equivalent hydrogen release rate compared to 1 g/s for SBEP-V3) and for a period of 3740 seconds. Helium concentrations were measured with 57 catharometric sensors at various locations for a period up to 1.1 days. The simulations were performed using a variety of CFD codes and turbulence models. The paper compares the results predicted by the participating partners and attempts to identify the reasons for any observed disagreements.

## 1 INTRODUCTION

The recently established International Association of Hydrogen Safety (IA-HySafe: www.hysafe.info) includes within its Research Committee activities, the continuation of the successful Standard Benchmark Exercise Problems (SBEPs) first introduced within HySafe NoE (www.hysafe.org). The scope of the SBEPs is to validate, inter-compare and further develop existing Computational Fluid Dynamics (CFD) codes and models in predicting hydrogen related release, dispersion and combustion phenomena. In the present work, focus is given to the understanding and prediction of the dispersion / accumulation of hydrogen releases in confined spaces under low release conditions. Understanding the conditions under which small to medium hydrogen releases (up to 1g $s^{-1}$) in confined spaces become dangerous was a key objective of the InsHyde internal project (www.hysafe.org/inshyde) of HySafe has been initially addressed by SBEP related publications [1, 2, 3].

The experiments of the current SBEP-V21 were performed by CEA within HySafe and are described as test 5 in the HySafe deliverable D65 [4] and as test 4 in Gupta et al. (2007) [5]. Helium was used instead of hydrogen for safety reasons and a large number of sensors (57) were used to monitor the helium concentration distribution during both the release and diffusion phases within the facility.

## 2 EXPERIMENTAL DESCRIPTION

The GARAGE facility is representative of a realistic single vehicle private garage. The GARAGE facility is situated indoors to attenuate the variations of the meteorological conditions. It is of rectangular shape with interior dimensions of 5.76 m (length) x 2.96 m (width) x 2.42 m (height). The internal volume of the GARAGE is 40.92 $m^3$. The facility is equipped with a door in the back for technical access, and a tilting door on the front side (Figure 1). Concerning its design and construction, a stainless steel (Grade 304) skeleton (tubes of square section of 40 mm side width) was chosen, in order to have adjustable walls (Figure 1a). The material chosen for the panels is Styrodur® (2500 C), a green coloured extruded rigid polystyrene foam of 30mm thickness (40mm thickness at the floor level) as shown in Figure 1b. Styrodur® modules are fixed with silicon grease and the sealing is ensured with aluminum tape. Electric connections are optimized to assure minimum entry points inside the GARAGE and possible holes are sealed with silicon grease. Test 5 parameters are given in Table 1.


Figure 1. Description of GARAGE set-up at CEA Saclay (a) Left: structural steel skeleton and (b) Right: interior of the facility with the panels

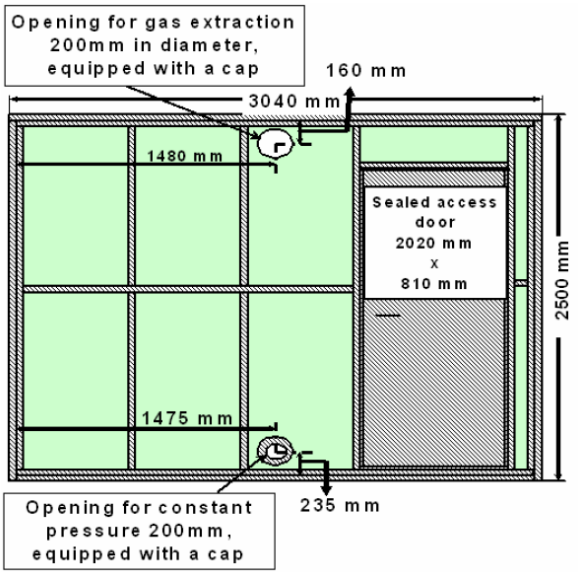


Figure 2. Openings (upper vent closed, lower vent open throughout the experiment)

| | |
|---|---|
| Garage x-dimension (mm) | 5760 |
| Garage y-dimension (mm) | 2960 |
| Garage z-dimension (mm) | 2420 |
| x release (mm) | -2880 |
| y release (mm) | 1480 |
| z release (mm) | 220 |
| Exit diameter (mm) | 29,7 |
| Volumetric flow rate - STP | 18 |
| He mass flow rate (g/s) | 0,054 |
| Garage Temperature T (°C) | 24,1 |
| Exit velocity (m/s) | 0,47 |
| Release Direction | Upwards |
| Release Type | Continuous |
| Release duration (s) | 3740 |
| Released volume - STP (NL) | 1122 |
| He released mass (gr) | 200,28 |
| Target concentration (%) | 2,94% |
| Total measurement time (s) | 90440 |

Table 1. Test-5 Parameters

The test was performed without ventilation. The upper vent was kept closed. The lower vent was kept open in order to maintain the facility at constant atmospheric pressure for the duration of the tests. Regarding the sealing efficiency, the leakage rate from the fully sealed garage was separately estimated to be 0.01 ACH. Concentrations were measured using thermal conductivity gauges (mini-catharometers) at 57 positions within the garage (see Figure 3). Table 2 presents the coordinates of the sensors of masts M1 and P1.

Table 2: Sensor coordinates

| Sensor name | x (mm) | y (mm) | z (mm) |
|---|---|---|---|
| M1N1 | -1000 | 1001 | 315 |
| M1N2 | -1000 | 1001 | 630 |
| M1N3 | -1000 | 1002 | 945 |
| M1N4 | -1000 | 1000 | 1260 |
| M1N5 | -1000 | 995 | 1575 |
| P1N1 | -190 | 155 | 1900 |
| P1N2 | -190 | 151 | 2135 |
| P1N3 | -190 | 150 | 2370 |

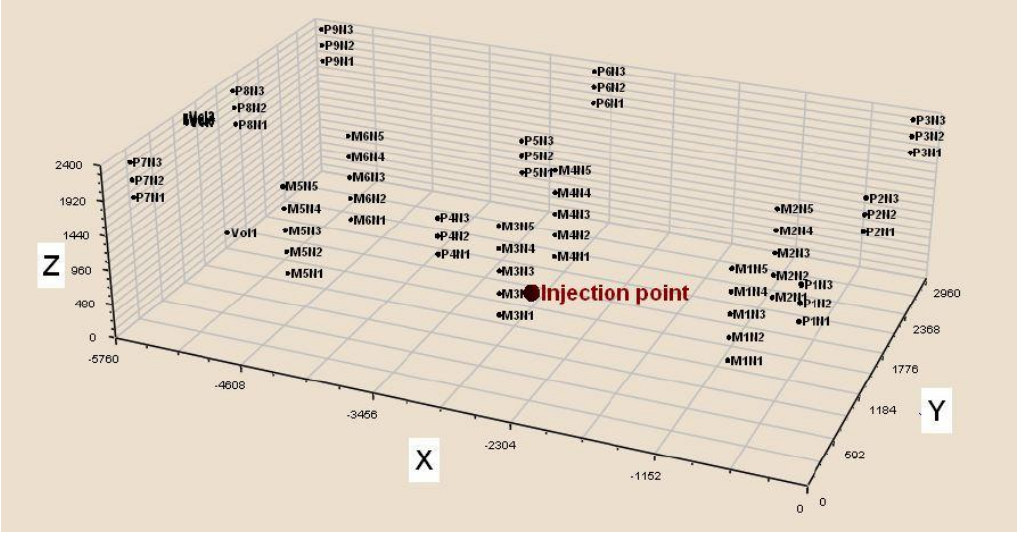


Figure 3. Sensor locations

## 3 MODELLING STRATEGY

### 3.1 HSL

HSL used the Ansys CFX 12.1 code. The computational domain did not extend beyond the garage. Symmetry was not assumed. Simulations were performed for 2 hybrid meshes with prismatic cells in the near–ceiling region and tetrahedral cells elsewhere, with 88,840 and 46,577 nodes respectively. Discretization of the convection terms has been performed using the High resolution scheme for all solved variables except those related to turbulence, for which the first order upwind scheme was used. This is an option in CFX where the discretization is a blend between first-order upwind and a second-order accurate discretization depending on gradients in the solution. The code decides for itself what should be the blend for any particular computational cell. Discretization of the transient terms was performed using the 2nd order backward Euler scheme. Turbulence was modeled using the Shear Stress Transport model (SST) proposed by Menter. The turbulent Schmidt number was $Sc_t$ = 1.0. The vent was modeled as opening with constant pressure, allowing entrainment. Simulations were performed up to 12000 s for the fine mesh with CPU time 19 days and 15 hours (4 days and 11 hours for the release

and 15 days and 4 hours for the diffusion phase). For the coarse mesh simulations were performed up to 15000 s, with CPU time 18 days and 5 hours (6 days and 7 hours for the release and 11 days and 22 hours for the diffusion phase). The longer run times in the release phase on the coarse mesh was due to poor convergence. The simulations were carried out on two cores on a Dell Precision 5400 machine with two Intel Xeon processors and running Windows XP SP3 64-bit operating system.

### 3.2 FZJ

FZJ used the Ansys CFX 12.1 code. The computational domain did not extend beyond the garage. Symmetry was assumed in the Y-direction. The mesh consisted of 212272 nodes with hexagonal elements (unstructured). A second order scheme was used for convection terms discretization. The transient terms were discretized using the second order backwards Euler scheme. In the release phase the maximum time step was 0.1 s, the maximum Courant number 21 and the RMS Courant 0.3. In the diffusion phase the maximum time step was 0.5 s, maximum Courant number 6.4 and RMS Courant 0.12. Turbulence was modeled using the SAS (Scale Adaptive Simulation) SST model incl. buoyancy and turbulence production and dissipation terms [6].The molecular diffusivity of He to air was taken equal to 8e-05 m2/s. The vent was modeled as an "opening" boundary condition with flow direction normal to boundary, relative pressure equal to zero and temperature equal to initial temperature. Simulations were performed up to 20000 s, using 3 threads. CPU time was 15 days and 21 hours (9 days and 21 hours for the release phase) on an Intel i7 860 CPU.

### 3.3 GEXCON

GEXCON used the FLACS code R&D version flacs2.2.8extended. The main reason for using this R&D version and not the official commercial release is that the incompressible solver used is only available in the R&D version of FLACS. For this type of scenarios with low velocity releases, a factor of 10 speed-up of simulation time can be achieved using an incompressible solver. As He does not exist among the predefined gases in FLACS, it was modeled indirectly as a constant composition mixture of hydrogen and methane, having the same molecular weight as that of He. This approach was considered acceptable on the grounds that for low momentum releases the main parameter to characterize a gas is the molecular weight and that the effect of the "slightly wrong" other gas physical properties can be considered as negligible. The computational domain did not extend beyond the garage. Symmetry was not assumed. The geometry was modeled in quite some detail to look realistic. Wall plates and structure were modeled as accurately as possible and also the instrumentation holders were included in the geometry model. The typical grid cell size was approximately 20x20x10 cm. The grid was refined around the release according to FLACS guidelines to 2cm x 2cm in horizontal directions. The total number of cells was 30888 during the release phase and 4560 during the diffusion phase. The time step was fixed to 0.05 s. Discretization of the convective terms was performed using Kappa schemes with weighting between 2nd order upwind and 2nd order central difference and delimiters used for some equations. Turbulence was modeled using the standard k-ε model, modified for gravity effects. The opening was modeled with the "Nozzle" boundary condition (FLACS default passive outflow condition for no wind scenarios). Simulations were performed up to 100000 s. CPU time was around 12 hours during the release phase (up to 4000 s). If the same grid and time step would be used, the simulation time would have been 30h until 10,000s and 60h till 20,000s. To accelerate the diffusion phase the grid resolution was changed by removing the refined region around the jet. In this way the 2nd part of the simulation from 4000−20000s took 4-5 h and the simulation until 100,000s roughly 24h. Ordinary 1 CPU Linux-PCs (Intel Xeon W3550 3.07 GHz) were used, i.e. no parallel computing was utilized. GEXCON also performed revised simulations with corrected Helium gas constant (corrected proportion between H2 and CH4) and extension of the computational domain beyond the garage. The incompressible parallel code was used and the simulation time on a quad-core desktop took 3h and 15min for 0-4000s and 2h and 11min for 4000s-20.000s.

### 3.4 JRC

JRC used the Ansys CFX 12.1 code [7]. The computational domain extended beyond the garage by a box of 2 m × 3 m × 3 m dimensions in the X, Y and Z. Symmetry was not assumed. Two meshes were

used one for the release phase and one for the diffusion phase. The release phase mesh (unstructured) consisted of 96228 nodes inside the garage. The diffusion phase mesh (mainly structured) consisted of 65256 nodes inside the garage. In both cases the mesh (unstructured) outside the garage consisted of 5173 nodes. A second order scheme was used for convection terms discretization. The transient terms were discretized using the second order backwards Euler scheme. Maximum time step was set to 0.5 s. Turbulence was not modeled (laminar flow approach). Benintendi (2011) [8] gave a short description of relevant works on the transition from laminar to turbulent regime for jets and referenced that laminar flow jets can be found with Reynolds number as high as 300, 600 and even 1000. Being the Reynolds number of the studied experiment equal to 115 at the release nozzle, JRC assumed a laminar regime. It should be noticed here that the complete understanding of the transition from laminar to turbulent jet is still an open issue [8]. During the diffusion phase, long enough after the jet release is stopped, the flow is laminar. Outside the garage the top plane was modeled with an "opening" boundary condition with flow direction normal to boundary, relative pressure equal to zero and temperature equal to the initial temperature. Symmetry boundary conditions were applied at the lateral planes and the rest of the planes were treated as walls (no slip condition). Simulations were performed up to 15000 s, using 8 threads. CPU time was 9 days and 20 hours. CPUs clock was at 3.25 GHz.

### 3.5 KIT

KIT used the GASFLOW 3.01 code. The computational domain did not extend beyond the garage. Symmetry was not assumed. The computational mesh consisted of regular orthogonal cells in Cartesian geometry. The mesh had 51 cells in x-direction, 33 cells in y-direction, and 35 cells in z-direction. The cell size varied in x–direction from 2.632 cm (source location) to 21.04 cm (border of grid), in y–direction from 2.632 cm (source location) to 16.57 cm (border of grid), and in z–direction from 11 cm (lower border of grid) to 3.0 cm (source location) to 8.367 cm (upper border of mesh). During the He-release phase the time step was approximately 0.01 s, whereas afterwards the time steps amounts to roughly 0.08 s. Convection terms were discretized using the first order upwind scheme. For the time discretization the linearized Arbitrary Lagrange Euler (ALE) integration scheme was used. This is a pressure based semi-implicit methodology, which is subdivided into three steps: an explicit Lagrangian phase for the multidimensional finite control volumes (phase A), an implicit pressure iteration phase (phase B) and a rezone phase (phase C). The purpose of phase B is to compute time-advanced pressures to allow calculations of low-speed (low Mach-number) flows without any time-step restrictions from the fluid sound speeds. Turbulence was modeled using the k-ε model with $\mathrm{Pr}_{\varepsilon} = 1.4245$, $C_{\varepsilon 2} = 1.8333$ (standard values are 1.3, 1.92) and modified for stability with $C_{\varepsilon 3} = 1.96$. A constant pressure boundary condition was used for the opening. Simulations were performed up to 20000 s. CPU time was 2d, 15h and 40 min, using 1 thread. Computer characteristics were PC, OS: Suse Linux 11.2, Intel i7-950 processor, 3.06GHz.

### 3.6 NCSRD

NCSRD used the ADREA-HF code [9]. The computational domain extended 1.02 m beyond the garage back wall (where the opening is located). The grid was Cartesian consisting of 48x35x32 (53760) cells in the X, Y and Z directions. Symmetry was not assumed. The opening was treated using the porosity approach. The smaller cell was at the source with a dimension of 3 cm in all directions. Far from the source the grid expanded with ratio 1.12. In the vertical direction the cells were not permitted to exceed 10 cm. Outside of the garage 3 equidistant cells were used in the Y-direction. Following earlier validation work [2] the SMART scheme was used for convection discretization and the first order fully implicit Euler for transient terms with a maximum permitted CFL of 2. Turbulence was modeled using the standard k-ε model, modified for stability. The molecular diffusivity of He to air was 5.649e-05 m2/s. In the free domain planes outside the garage a constant pressure boundary condition was used for the normal velocities. For the remaining variables either zero gradient if outflow or initial value if inflow. Simulations were performed up to 7080 s, using 2 threads and the OpenMP ADREA-HF parallel version. CPU time was 10 days on a Windows 7 portable workstation equipped with an Intel Core i7 M620 CPU at 2.67 GHz.

### 3.7 UPM

UPM used the ANSYS-Fluent 13.0 code. The computational domain did not extend beyond the garage. Symmetry was not assumed. A hexahedral mesh was used consisting of 125516 cells. The grid was refined near the He release. The minimum size corresponds to the release exit (2.63 cm) and the maximum size to the points close to the upper corners (10 cm). The convective terms were discretized using a second order scheme. The first order fully implicit Euler scheme was used for the transient terms. Time step was 0.05 s in the period 0-5000 s and 0.5 s in the period 5000-20000 s. An additional run was performed with time steps 0.1 s and 1 s respectively and was shown to have negligible differences compared to the first one. Simulations were performed in laminar flow mode. The molecular diffusivity of He to air was 5.649e-05 m2/s. At the opening an "outflow" boundary condition was used, i.e. the diffusion fluxes in the direction normal to the opening plane were assumed to be zero and an overall mass balance correction was imposed to obtain the mass flow rate through the opening. Simulations were performed up to 20000 s, using 4 threads. CPU time was 13 days on a Windows 7 machine equipped with 2 Intel Xeon Quad Core CPUs at 2.4 GHz. UPM also performed revised simulations with extension of the computational domain beyond the garage.

## 4 RESULTS AND DISCUSSION

The SBEP was performed in two phases. In the first phase the partners performed the simulations without any knowledge of the results or the modeling assumptions of other partners. In the second phase after distribution/discussion of the results some partners performed additional/revised simulations and obtained clearly better results. The first phase of the exercise was partly blind, because although in the SBEP specifications the lower vent was defined as open during the entire experiment, the experimental results initially distributed corresponded to another CEA experiment (Test-5-vent open/closed) with identical conditions as the above described Test-5, but with the bottom vent "tap-closed" after the end of the release, (obviously with the intention to maintain the released helium within the garage as much as possible). It should be clarified here that both the experimental results and the associated calculations discussed below correspond to the case of the bottom vent open during the entire experiment (release and diffusion phases). The main modeling strategy parameters are summarized in Table 3.

Table 3: Modelling strategy parameters

| Participant/ Code | Turbulence model | Number of computational cells | Vent model | Molecular diffusivity ($m^2/s \times 10^{-5}$) |
|---|---|---|---|---|
| FZJ/CFX | SAS-SST | 212272 (half garage) | Opening | 8.0 |
| GEXCON/ FLACS | k-ε | Release: 30888<br>Diffusion: 4560 | Opening | 2.0 |
| GEXCON_revised/ FLACS | k-ε | Release: 44352<br>Diffusion: 7200 | Extension | 2.0 |
| HSL/CFX | SST | 88840 | Opening | 1.86 |
| JRC/CFX | Laminar | Release: 101401<br>Diffusion: 70429<br>Outside: 5173 | 2 m × 3 m × 3 m extension | 11.7 |
| KIT/ GASFLOW | k-ε | 58905 | Opening | 7.5 |
| NCSRD/ ADREA-HF | k-ε | 53760<br>Outside: 3360 | 1 m × 2.96 m × 2.42 m extension | 5.65 |
| UPM/ FLUENT | Laminar | 125516 | Opening | 5.65 |
| UPM_revised/ FLUENT | Laminar | - | 1 m × 2.96 m × 2.42 m extension | 5.65 |

Figure 4 shows the predicted mass of He inside the garage, as function of time compared to the experiment. The line "experiment" in Figure 4 was produced by horizontally averaging the measured concentrations (over the masts) at each of the 8 available elevations (see Table 2) and then by performing a vertical integration assuming 8 homogeneous concentration layers from the floor to the ceiling. The observed experimental He loss can be attributed to both the bottom vent and to a small but existing leakage from the facility. The facility leakage was estimated, using the data from the Test-5-vent open/closed experiment, to be approximately 5% for a time period of 20000 s. The value of 5% is consistent with the loss obtained from an initial He concentration of 2%, within a period of 20000 s with the given (in the above experimental description) value of 0.01 ACH (air changes per hour). The line "experiment-corrected" shown in the figure has been therefore produced by adding to the line "experiment" the abovementioned facility leakage. Therefore, model predictions in Figure 4 should be compared to the line "experiment-corrected" as they do not account for facility leakages. The model predictions show roughly 2 kinds of behavior. Either a negligible predicted He loss or a finite loss. The second behavior is consistent with the experimental trend but the predicted He loss seems to be rather overestimated in some cases (for FZJ it goes up to 36% at 20000 s). The abovementioned behavior of the He loss can be associated mainly to the way that the vent was modeled.

In four of the cases (JRC, NCSRD, GEXCON_revised and UPM_revised) the computational domain was extended beyond the GARAGE to avoid specifying boundary conditions at the opening. In both cases a finite He loss was predicted (although not entirely clear how much it was for NCSRD as the simulation time was up to 7000 s). In 3 of the cases (GEXCON, KIT and UPM) the He loss was negligible (the GEXCON prediction for the He loss goes up to 12000 s). In these cases the vent was modeled with an "opening" boundary condition, which assumes given pressure at the boundary (or immediately after the boundary) and is able to produce either solely outflow, solely inflow or both inflow and outflow through the vent. In the remaining 2 cases (FZJ and HSL) a finite He loss was observed despite the fact that the same, as in the previous group of simulations, type of boundary condition was used, i.e. opening. This could be due to the turbulence model used, which is roughly similar in these 2 cases (SAS-SST and SST respectively). However, the effect of the turbulence models should be investigated further.

In an attempt to test the sensitivity of the results to the vent boundary condition, FZJ performed an additional simulation using the "outlet" boundary condition, which permits only outflow through the vent and found that the predicted He concentration at the lower elevation ($z = 0.315$ m) increases and that the predicted He loss decreases, which leads to better agreement to the experimental data. The reason for this improvement is that the "opening" boundary condition permits inflow of fresh air (zero He concentration) and thus for the pressure in the facility to be kept constant the He-air mixture outflow through the vent is increased as compared to the pure "outlet" boundary condition case. Still between the "opening" and the "outflow" boundary conditions the first seems to be more consistent to the present experiment. As far as the sensitivity of the "opening" boundary condition is concerned, Matsuura et al. [10] showed that even a small decrease by 0.5 Pa of the pressure at the lower vent of a hallway results in a substantial increase of the concentrations, especially at the lower sensors, suggesting that the "opening" boundary condition is very sensitive to the pressure level assumed. The sensitivity of the results to the assumed boundary conditions was also verified in SBEP-V20 [3].

Figure 5 to Figure 12 presented below show the predicted He concentrations at masts P1 (3 elevations) and M1 (5 elevations). It should be mentioned that the readings of the 57 sensors deployed during the experiments (masts M1-M6 and P1-P9) showed large horizontal homogeneity and therefore the authors selected the 2 masts as being representative of the overall experimental data.

In general it can be observed that the predicted concentration levels agree with the experiments within a range of deviation of about 0.5 to 1.0% v/v and that the disagreement with the experiment data tends to be higher at the lowest sensors as expected from previous experience (e.g. SBEP-V3).

The effect of the vent modeling approach on the predicted concentrations can be observed by comparing the first-phase predictions of partners UPM and GEXCON (performed with the "opening" boundary condition at the vent) with their second phase results (performed by extending the computational domain beyond the GARAGE). It should be noted that in the case of GEXCON the revised calculation included also a correction regarding the modeled molecular weight of He, which was overestimated during the first phase of the SBEP. The above mentioned comparison shows that the extension of the computational domain beyond the garage leads to a higher predicted decrease rate of the He concentration at large times, which is more consistent with the experiments. Such an approach is expected to improve the predictions of KIT, HSL and FZJ.

In the release phase the predictions show a delayed concentration increase and associated underestimation of concentrations compared to the experiment especially at the lower sensor elevations. This could be attributed to excessive diffusion introduced by the turbulence model for those partners using a turbulence model. Two partners used the "laminar" approach (JRC and UPM). UPM over-predicted the concentrations, while the prediction is rather in good agreement with the experiments for JRC. It should be noted that JRC used the highest value for molecular diffusivity of helium in air (see Table 3). The correct He to air molecular diffusivity at 24.1 °C is 7.2 $10^{-5}$ cm/s. The data submitted by JRC results were obtained by a 62% higher value. After the end of the deadline of the benchmarking activities for all partners, JRC performed some additional simulations with the correct diffusion coefficient. The new results cannot be presented in the paper because they were obtained after the deadline. Nevertheless it can be stated that the new simulations (with the correct molecular diffusivity) shows that the effect of the diffusion coefficient on the simulation results is mixed, slightly improving the results for the sensors that are located in the upper and middle part of the garage and slightly decreasing the accuracy of the results in the sensors in the lower part of the garage.

In the diffusion phase simulations performed by HSL using the SST turbulence model show a very slow concentration decrease rate with time and associated overestimation of concentrations at all levels except the lowest one. This could be partly due to the "opening" boundary condition used for the vent and partly due to underestimation of the diffusion. Table 3 shows that HSL uses the lowest molecular diffusivity value for helium diffusion in air. An overestimation of the upper sensors concentration and underestimation at the lower ones, with good agreement with experiment in between, is also observed in the GEXCON_revised predictions using the k-ε model. Although the concentration decrease rate is not as slow as in HSL simulation this behavior suggests underestimation of mixing, which again could be due to the low value of the molecular diffusivity used by GEXCON.

One of the parameters requested to be reported by the partners in the SBEP specifications was the CPU time. The performed simulations required large amounts of CPU time of the order of 10-20 days when performed using the fully compressible approach. The incompressible approach on the other hand used by partners GEXCON and KIT reduced drastically the CPU time by at least a factor of 10.

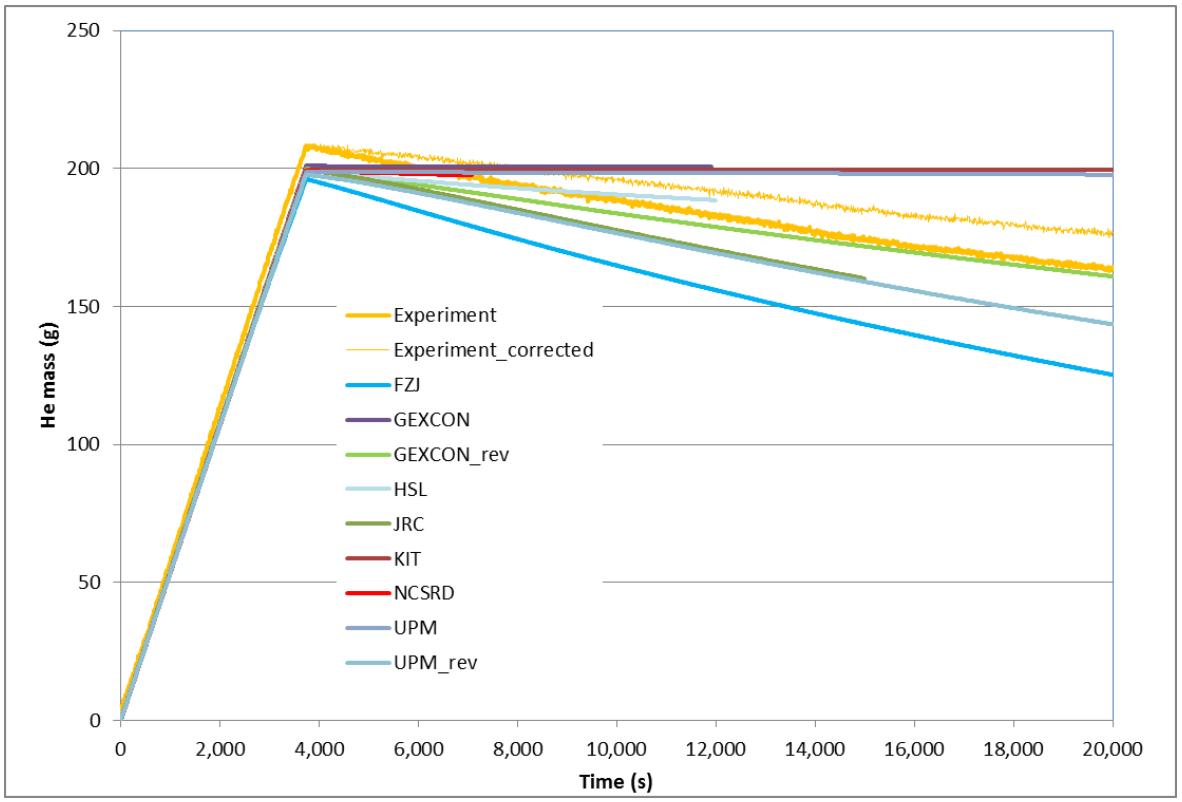


Figure 4 He mass history inside the garage

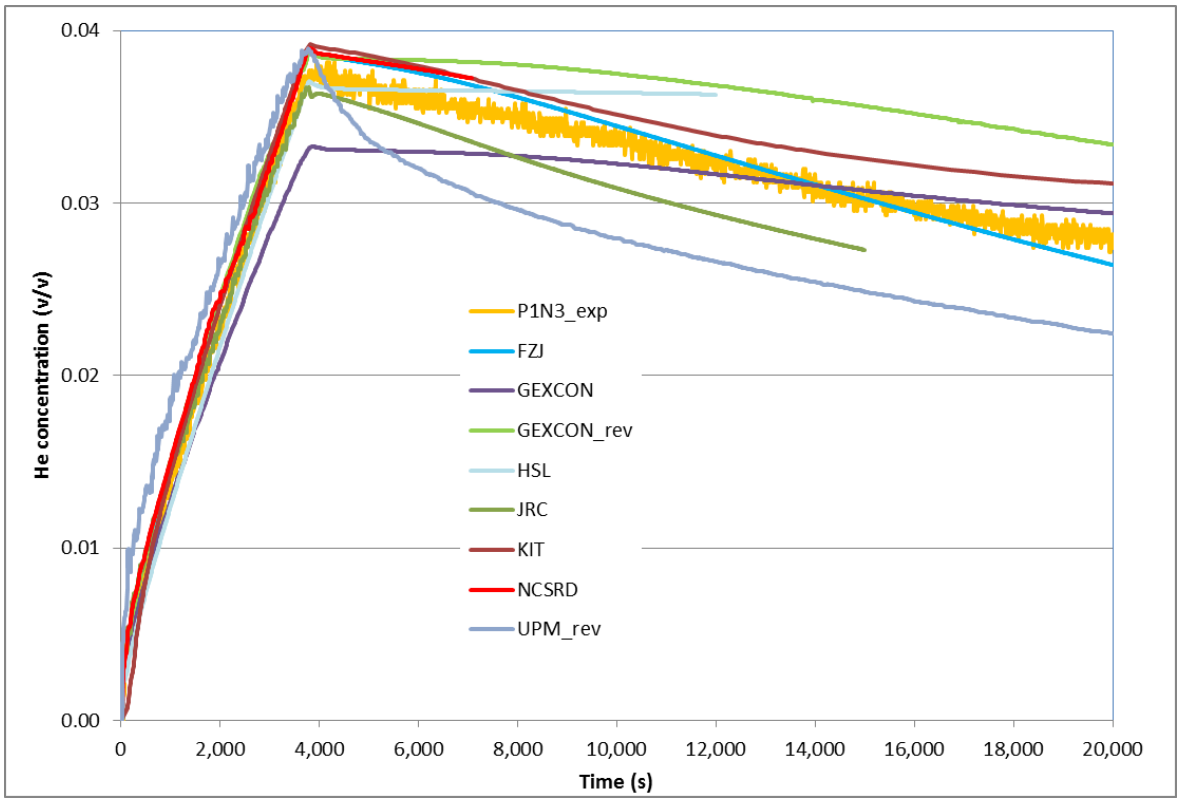


Figure 5 Predicted/observed concentrations at sensor P1N3 (z = 2.37 m)

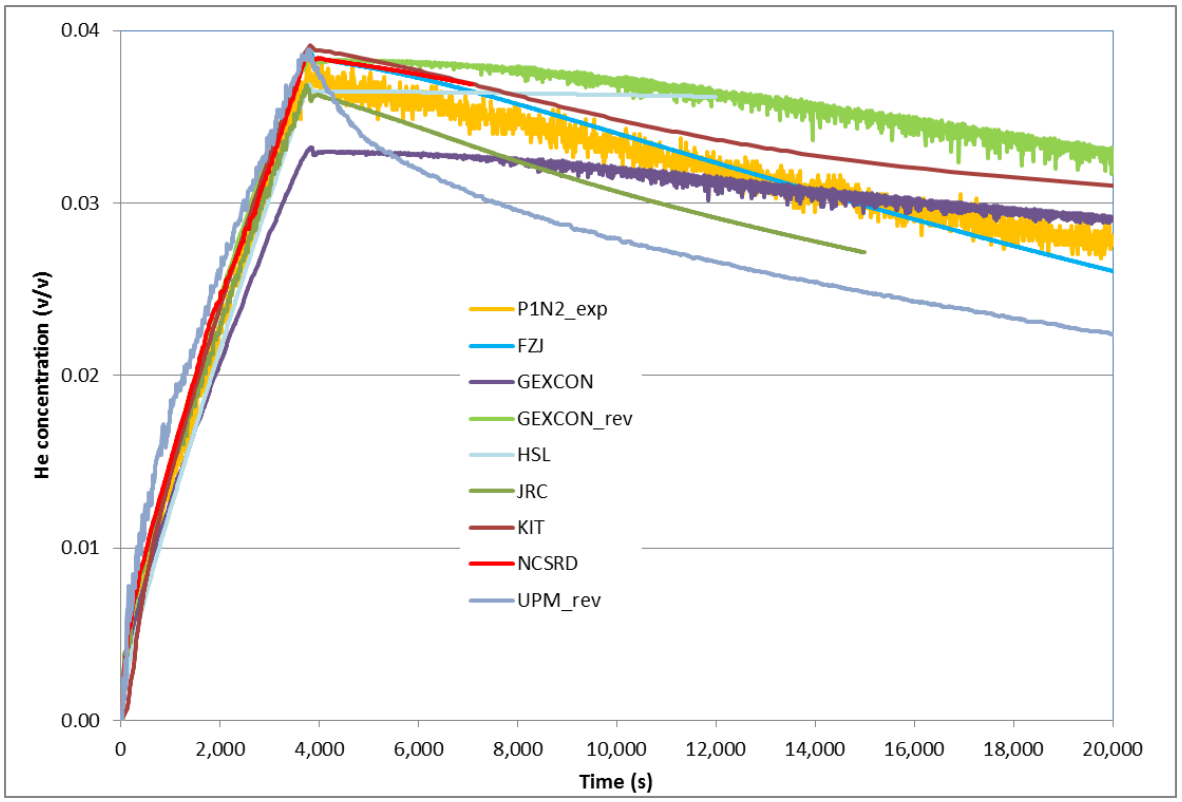


Figure 6 Predicted/observed concentrations at sensor P1N2 (z = 2.135 m)

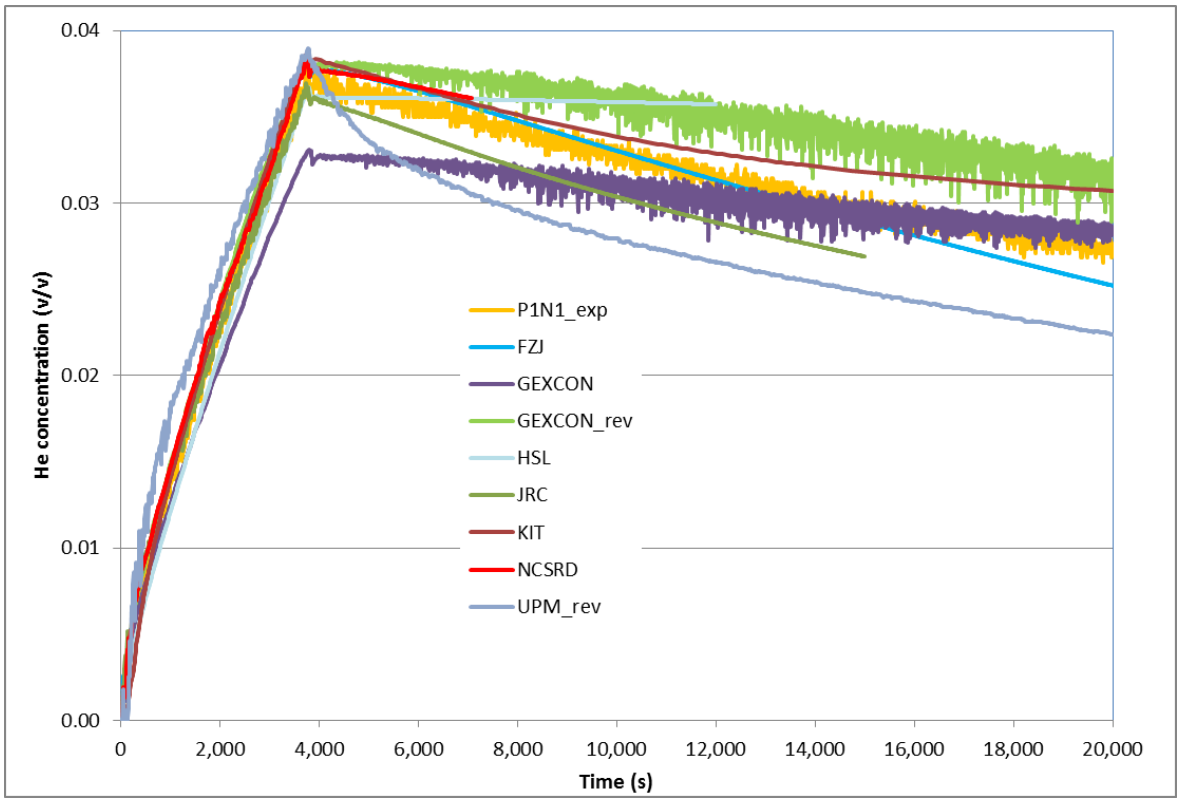


Figure 7 Predicted/observed concentrations at sensor P1N1 (z = 1.9 m)

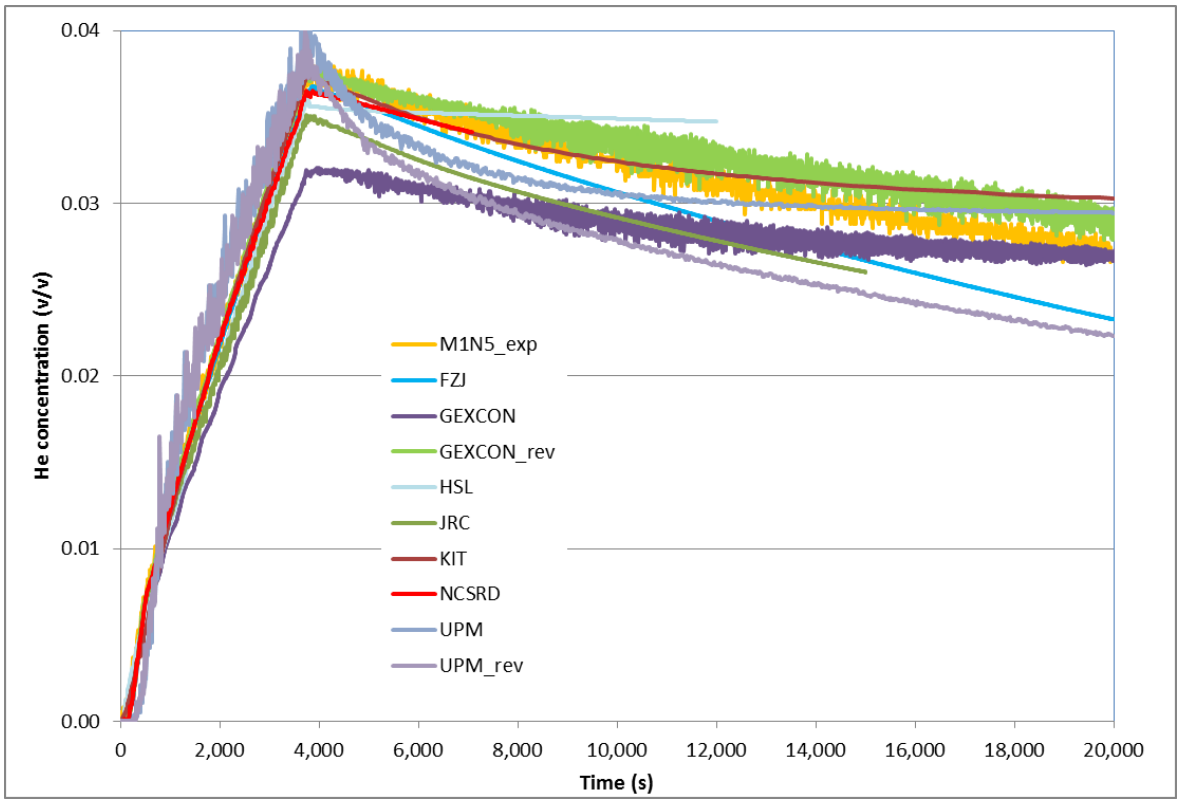


Figure 8 Predicted/observed concentrations at sensor M1N5 (z = 1.575 m)

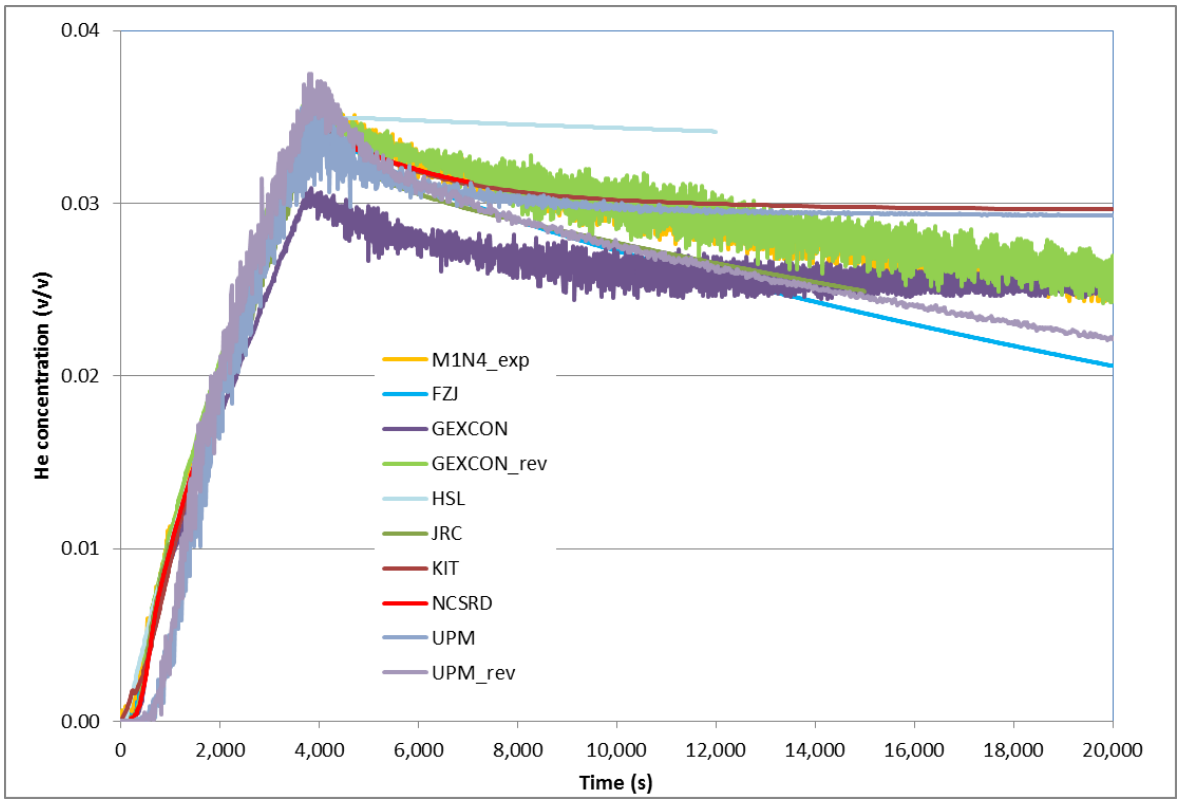


Figure 9 Predicted/observed concentrations at sensor M1N4 (z = 1.26 m)

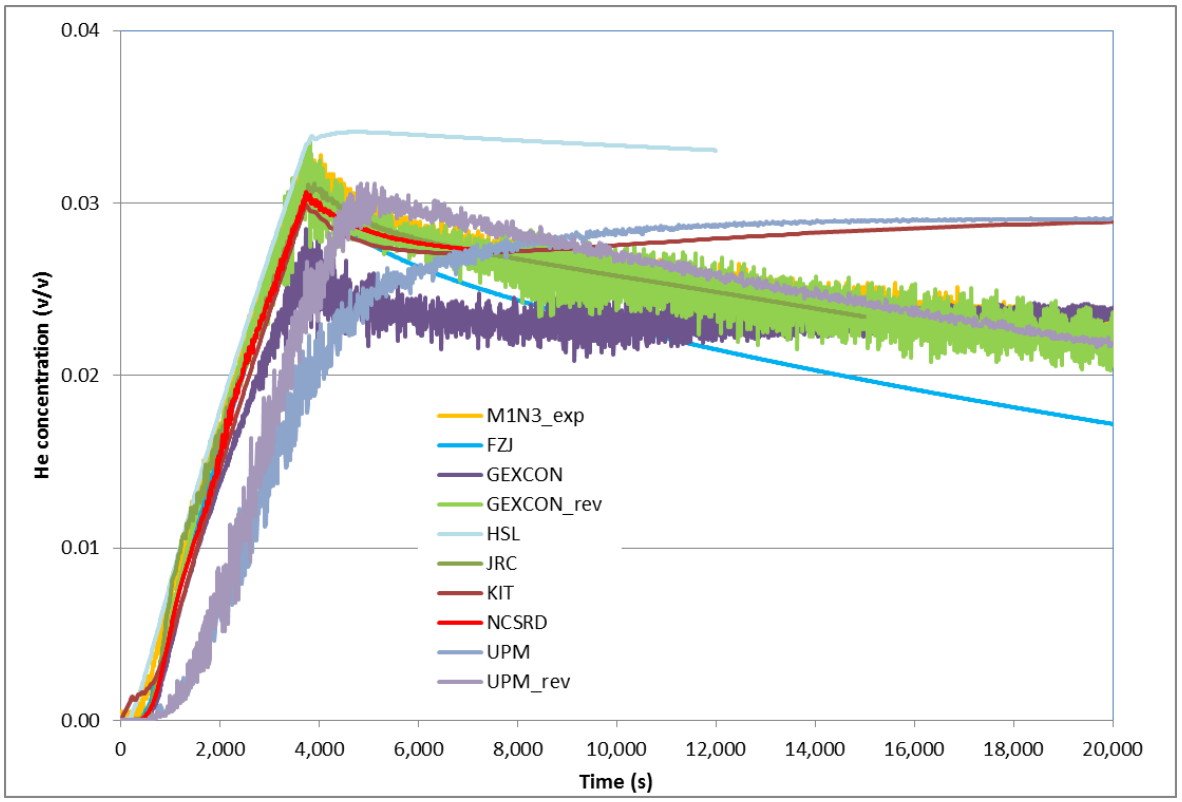


Figure 10 Predicted/observed concentrations at sensor M1N3 (z = 0.945 m)

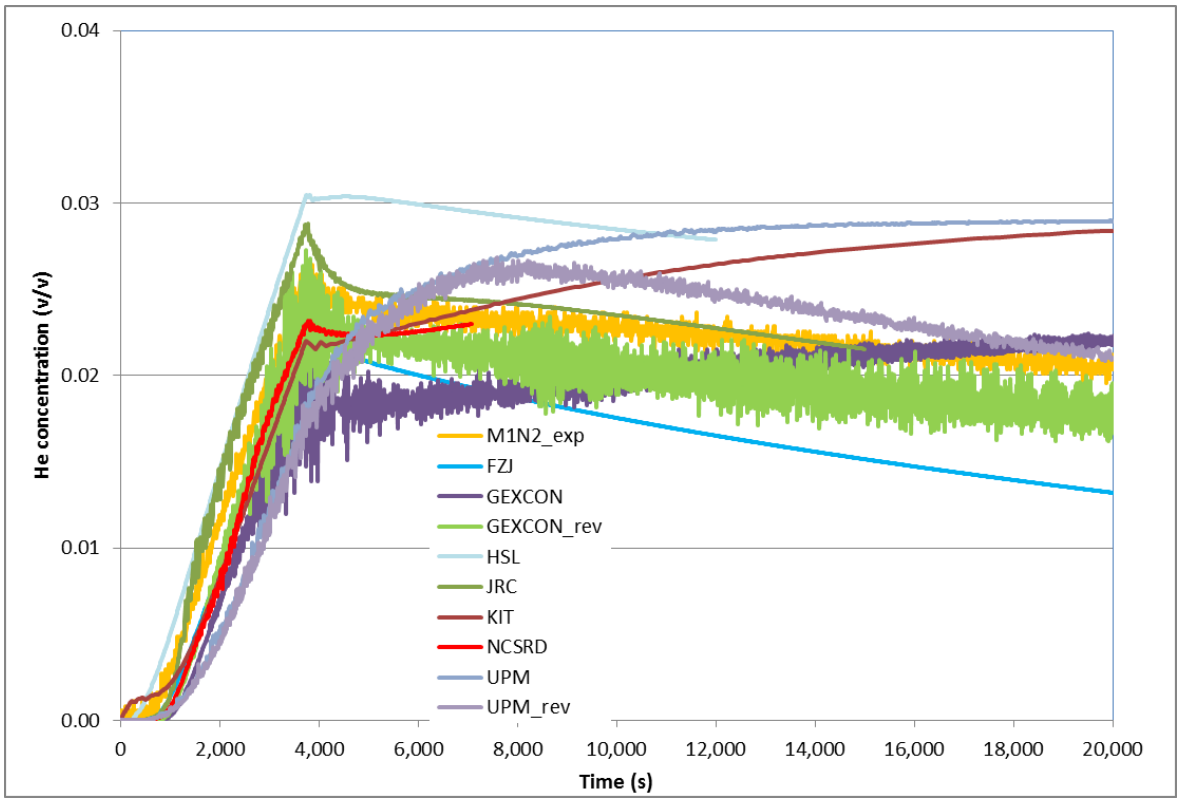


Figure 11 Predicted/observed concentrations at sensor M1N2 (z = 0.63 m)

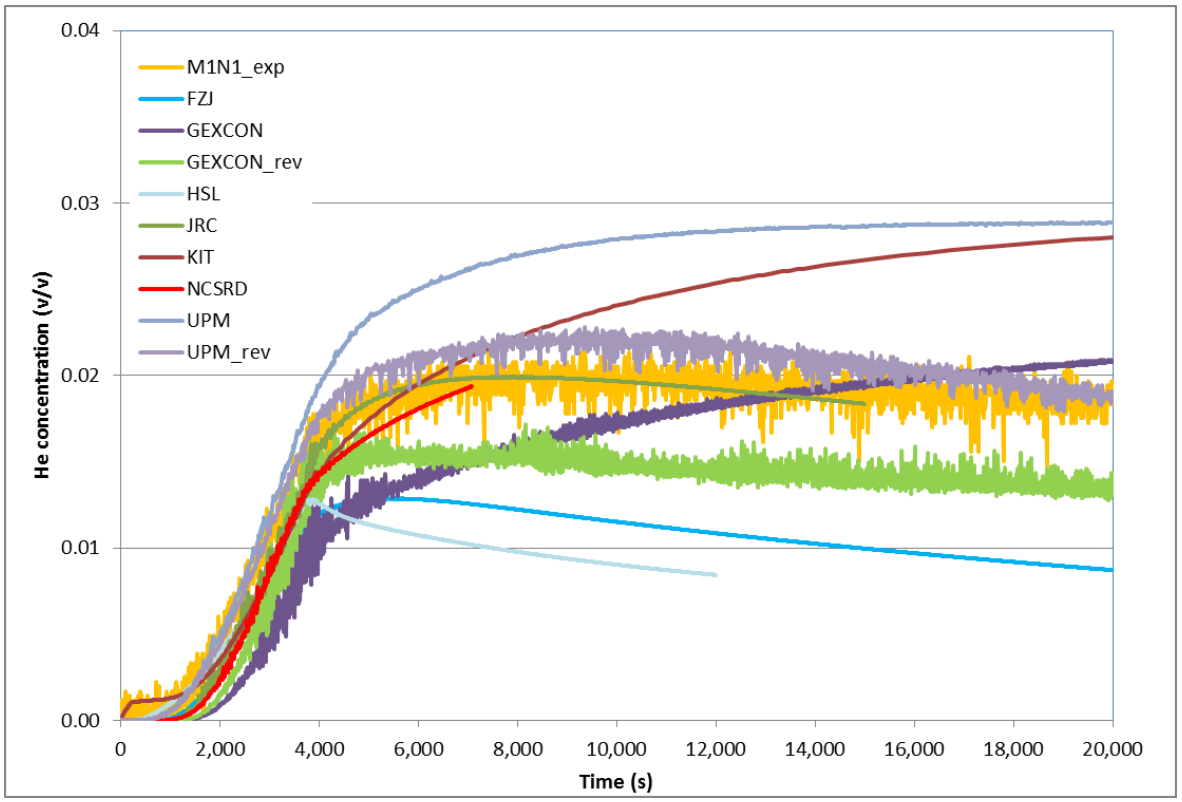


Figure 12 Predicted/observed concentrations at sensor M1N1 (z = 0.315 m)

## 5 CONCLUSIONS

A CFD benchmark exercise was organized within IA-HySafe in order to evaluate various modeling approaches in predicting the physical phenomena associated to the short and long term mixing and distribution of hydrogen releases in confined spaces. The experiment simulated was the CEA-TEST-5. The performed analysis led to the following conclusions:

The predicted concentrations at large times were found sensitive to the method used to model the vent. Extension of the computational domain beyond the garage's physical boundaries where the vent is located clearly gave better agreement between predicted and measured concentrations.

In general, the predicted concentration levels were found to agree with the experiments within a range of deviation of about 0.5 to 1.0% v/v, the disagreement with the experimental data being higher nearer to the ground.

Simulations were performed using both the fully compressible approach and the incompressible approach. The CPU times reported using the fully compressible approach, were of the order of 10-20 days. These were drastically reduced by at least a factor of 10, when using an incompressible approach.

## 6 ACKNOWLEDGEMENTS

NCSRD would like to acknowledge partial funding by the EC ($7^{th}$ FP) Grant Agreement No 229773 (PERL).

## 7 REFERENCES

1 Gallego E., Migoya E., Martin-Valdepenas J.M., Crespo A., Garcia J., Venetsanos A.G., Papanikolaou E., Kumar S., Studer E., Dagba Y., Jordan T., Jahn W., Oíset S., Makarov D., An

Intercomparison Exercise on the Capabilities of CFD Models to Predict Distribution and Mixing of H2 in a Closed Vessel, Int. J. Hydrogen Energy, 32, No 13, 2007, pp. 2235-2245.
2 Venetsanos A.G., Papanikolaou E., Delichatsios M., Garcia J., Hansen O.R., Heitsch M., Huser A., Jahn W., Jordan T., Lacome J-M., Ledin H.S., Makarov D., Middha P., Studer E., Tchouvelev A.V., Teodorczyk A., Verbecke F., Van der Voort M.M., An Inter-Comparison Exercise On the Capabilities of CFD Models to Predict the Short and Long Term Distribution and Mixing of Hydrogen in a Garage, International Journal of Hydrogen Energy, 34(14), 2009, 5912-5923.
3 Papanikolaou, E.A., Venetsanos, A.G., Heitsch, M., Baraldi, D., Huser, A., Pujol, J., Garcia, J., Markatos, N. HySafe SBEP-V20: Numerical studies of release experiments inside a naturally ventilated residential garage (2010) International Journal of Hydrogen Energy, 35 (10), pp. 4747-4757
4 HYSAFE deliverable D65, Experimental results on helium release inside a full scale GARAGE set-up, prepared by Benjamin Cariteau, CEA
5 Gupta S., Brinster, J., Studer, E., Tkatschenko, I., Hydrogen Related Risks Within a Private Garage: Concentration Measurements In a Realistic Full Scale Experimental Facility, ICHS-2, San Sebastian, 2007
6 Ansys CFX-Solver Theory Guide, Release 13.0, November 2010
7 ANSYS, CFX-12.1, 2009, Canonsburg Pennsylvania, http://www.ansys.com
8 Renato Benintendi, Laminar jet modelling for hazardous area classification, Journal of Loss Prevention in the Process Industries 24 (2011) 123-130
9 Venetsanos A.G., Papanikolaou E., Bartzis J.G., The ADREA-HF CFD code for consequence assessment of hydrogen applications, Int. J. Hydrogen Energy 35 (2010) 3908–3918
10 Matsuura K, Kanayama H, Tsukikawa H, Inoue M. Numerical simulation of leaking hydrogen dispersion behaviour in a partially open space. International Journal of Hydrogen Energy 2008;33:240–7