STUDY OF HYDRODYNAMICS PERFORMANCE OF A MULTI-VENTURI FILTERING SYSTEM FOR BWR SEVERE ACCIDENT VENTING STRATEGIES

by

Alejandro Antonio REYES-GARCIA, Eduardo SAINZ-MEJIA, Javier ORTIZ-VILLAFUERTE *, Javier C. PALACIOS-HERNANDEZ, and Roberto Carlos LOPEZ-SOLIS

National Nuclear Research Institute, Carretera Mexico-Toluca,
La Marquesa s/n Ocoyoacac, Estado de Mexico, Mexico

Scientific paper
https://doi.org/10.2298/NTRP2001016R

The aim of this project was to determine the capacity of a multi-venturi scrubber filtering system to cope with vented gas mass-flow rate coming from a BWR Mark II primary containment during a long-term station blackout. The multi-venturi filtering system CFD models were developed in the environment of the open source platforms SALOME and OpenFoam. The first geometrical model was created based on the dimensions of a well-known experimental setup, and the results of the pressure drop along the streamwise co-ordinate showed a maximum difference of 10 % in relation to the experimental values for different cases of liquid to gas mass ratios. Then a full scale multi-venturi model was developed. To study the performance of this system during conditions expected in a severe accident, a gas mixture similar to that occurring in a BWR Mark II containment at venting pressure was used as inlet gas. The gas mass-flow that can be cleansed by individual venturis and the pressure required to activate those venturis were computed. The pressure drop profiles in each sector were also determined as the function of different liquid loadings. The results showed good agreement with the capacity of the design taken as the reference model.

Key words: BWR, Mark II, severe accident, OpenFoam, SALOME, venturi scrubber

INTRODUCTION

During a severe accident scenario, a combination of strategies to keep the corium properly cooled and to vent the primary containment is considered the best approach to avoid uncontrolled release of radioactive material into the environment. Primary containment venting strategies are particularly significant for BWR with Mark I and Mark II primary containment designs, because of their volume [1]. Taking into account the requirement for the Mark I and Mark II designs to implement hardened containment venting systems (HCVS) from the wetwell to cope with over-pressurization conditions [2], a variety of venting strategies should be studied, for example, the frequency and duration of releases to provide better support for the actions considered in the severe accident management guidelines (SAMG). Additionally, those HCVS are required to properly operate under severe accident conditions [3]. Taking into consideration the possibility of losing the capability of venting from the wetwell, the option of implementing a HVCS directly from the drywell is also included in the SAMG. Although the implementation of engineered filtering systems is not required, their support in reducing offsite consequences has been recognized [4], and thus the capacity of these filtering systems must be evaluated under severe accident conditions.

Differently engineered technologies of filtering systems are available for application in nuclear power stations, such as the ones that use venturi scrubbers, or those based on sand-bed or metal fiber filtration, or the combination of the latter two [5]. Venturi scrubbers have been used for quite some time in different industries for the collection and filtration of small particles, so there is a large number of studies related to their performance under different operating conditions [6-9]. In particular, the efficiency of venturi scrubbers for aerosol capturing in filtered containment venting systems (FCVS) is an active research area [10, 11]. Some venturi scrubber FCVS are based on the use of multiple venturis distributed radially in a large water tank, so the analysis of the overall hydrodynamic system performance requires determining the capacity of different venturi designs in the system.

* Corresponding author; e-mail: javier.ortiz@inin.gob.mx

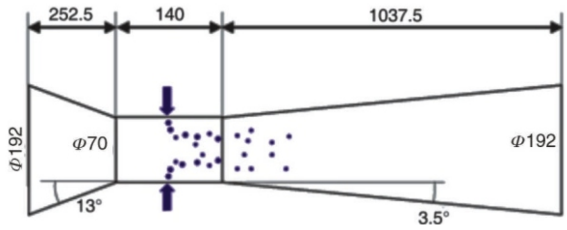

Figure 1. Dimensions of the venturi geometric model in mm

A venturi scrubber is a simple device with changing convergent/divergent flow sections, in which a gas-particle stream is accelerated into a zone where it mixes with a liquid film and liquid drops and this mixture is then decelerated in a diffuser. Figure 1 shows a schematic of a typical venturi scrubber. This device is quite robust for capturing very small particles with atomized liquid drops. The action starts with an incoming polluted gas-particle stream which is at first accelerated in the convergent entrance zone. In the throat of the venturi the liquid surrounding the device is sucked in through a number of small orifices and due to high kinetic energy of the gas stream, the liquid is atomized into very small drops which collect the particles flowing in the streaming gas. Finally, in the diffuser, the pressure is partly recovered as the flow decelerates.

In the multi-venturi concept, the efficiency of a single venturi is multiplied in order to withstand high flow rates at high pressure. In this engineering system all venturi devices have identical active sections (with the exception of the intake and discharge pipes), so it can be designed to operate at optimum gas mass-flow and pressure drop conditions, depending on the boundary conditions and the target decontamination factor. The multi-venturi scrubber being analyzed in this paper consists of an arrangement of radial rings that have different submergence depths at their entrance sections. One advantage of this design is that as primary containment pressure increases, more rings of venturi tubes can be put into action in a passive way and start distributing the total mass gas-flow among them, while keeping similar pressure drops in the active section. In this way, if the containment pressure increases, the multi-venturi system is still capable of handling the rise of discharge pressures.

We also studied the capacity of a FCVS design based on multi-venturi scrubbers to endure the mass-flow rate during a venting action from a Mark II containment. First, a CFD model of a venturi scrubber was developed in the environment of the open source platforms SALOME [12] and OpenFoam [13]. The results were then compared with experimental data. Then the dimensions of this single venturi model were modified to simulate venturi scrubbers immersed in water, as described in a design of a multi-venturi filtering system. Results of mass-flow rates of individual venturis and groups of venturis forming circular sectors are presented and discussed.

VENTURI DEVICE BASE CFD MODEL

A 3-D geometric Venturi model was constructed for CFD analysis based on the experimental setup by Haller *et al.* [14], as shown in fig. 1. Liquid is injected through 12 injectors having 2.5 10^{-3} m diameter. Figure 2 shows a diagram of sequential steps in the construction of the geometric model using SALOME GEOM and MESH modules. The venturi full geometry model consists of one straight pipe section to model the connecting duct to the venturi entrance, the convergent zone, the throat, the divergent (diffusor) section, and an additional straight pipe for the discharge zone. The first and the last straight pipe sections are not shown in fig. 1. All these components were modeled with a mesh of triangular prism cells using the extrusion technique. The PYTHON scripts in the SALOME platform environment have been generated to speed up the mesh generation process. This allows efficient accommodation of dimensional changes in the basic geometrical shape and changes in the mesh fineness at different longitudinal sections and in the input and output faces of the venturi. An iterative process was carried out to determine appropriate type and size of the final mesh, having the pressure drop along

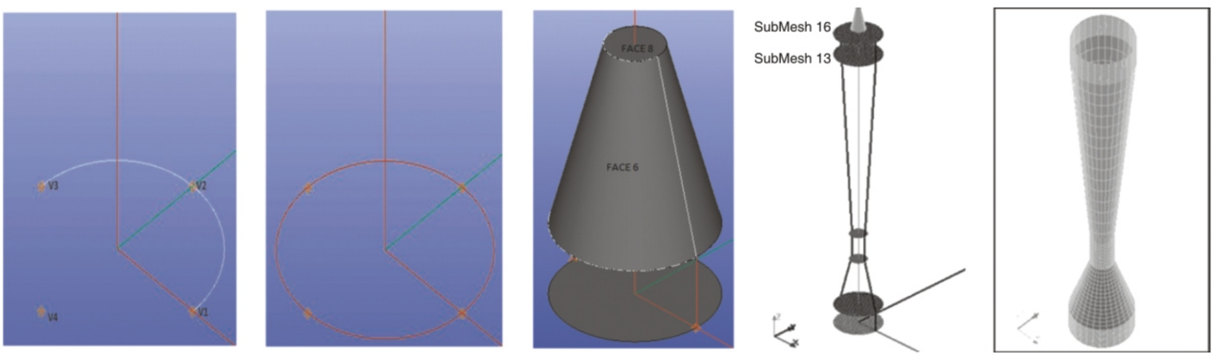

Figure 2. Stages in the development of the geometric model in SALOME

Table 1. Boundary conditions for simulation of Haller *et al*. [14] experiment

Gas mass-flow rate	0.2278 kgs^{-1} for a throat velocity of 50.0 ms^{-1} 0.3189 kgs^{-1} for a throat velocity of 70.0 ms^{-1} 0.4556 kgs^{-1} for a throat velocity of 100.0 ms^{-1}
Working gas	Air
Temperature	298.15 K
Pressure	101323.0 Pa
Liquid to mass ratios	0.5, 1.0, 1.5, 2.0, 2.5, and 3.0
Turbulence model	k-ε

the venturi device as the main reference parameter as shown later.

The boundary conditions were applied on sub-meshes of the appropriate sections of the main geometric model. Three boundary surfaces were defined: the pipe wall, the gas input cross-section, and the output cross-section. The corresponding boundary conditions were zero velocity at wall, constant input mass-flow and constant output pressure. Table 1 shows the boundary conditions used in the calculations. Since gas-flow rates are variables commonly used in containment venting calculations, it is preferred to keep this variable as a boundary condition, instead of venting gas velocities. Additionally, throat velocity was the main parameter in Haller *et al*. [14] experiment, so an iterative process was performed for the incoming gas-flow rate to obtain the target velocity value. The three gas-flow mass values used to obtain the throat velocities shown in Haller *et al*. [14] experiment are given in tab. 1.

The OpenFoam calculations were carried out with the Eulerian-Lagrangian transient solver reactingParcelFoam. This module can be applied to laminar or turbulent flow, and Lagrangian parcels. The k-ε was chosen for turbulence modeling. For the Lagrangian computations, constant diameter water droplets are injected at 12 orifices around the wall surface of the throat section. This diameter however is a function of gas velocity and the liquid to gas mass ratio. Further, no droplet growth or break up models have been implemented so far, so droplet size was kept constant during calculations. Finally, the injection of aerosol particles in the gas-flow was not considered at this time. Transient calculations were then executed until the mixture gas-liquid flow reached steady conditions for the specified boundary conditions. The turbulence parameters at the input face section (input boundary conditions) were calculated as the function of the Reynolds number Re with the following equations, where I is the turbulence intensity, $\overline{U}$ is the mean velocity and D_h is the hydraulic diameter

$$I = 0.16 Re^{\frac{1}{8}} \tag{1}$$

$$\kappa = \frac{3}{2}(\overline{U}I)^2 \tag{2}$$

$$\varepsilon = \frac{0.1643\kappa^{1.5}}{D_h} \tag{3}$$

Figure 3 shows the results of the pressure drop along the axis of the venturi from the OpenFoam simulations, for gas velocity at the throat of 70 ms^{-1}, *vs*. the data from the Haller *et al*. [14] experiments, for two liquid mass-flow rate to gas mass-flow rate ratios (lg^{-1}), also referred to as liquid load. The maximum relative error for l/g = 2.5 is 9.7 %, and 4.0 % when l/g = 1.0. Figure 4 shows a comparison of the overall pressure drop in the venturi as the function of liquid load for different gas throat velocities. The maximum deviations occur for the case of l/g = 0.5, and the rest of data points have relative errors bounded in a 10 % range. There is no information about experimental uncertainty.

There are many theoretical and numerical studies of the performance of this type of venturi devices, but the numerical simulations by Goniva *et al*. [15] are of particular interest to this work, because the reference venturi design is that described by Haller *et al*. [14] and they also used OpenFoam for computation. Their results are in better agreement with Haller *et al*.´s data, in comparison with the results shown in figs. 3 and 4. In the present work, to make the initial approx-

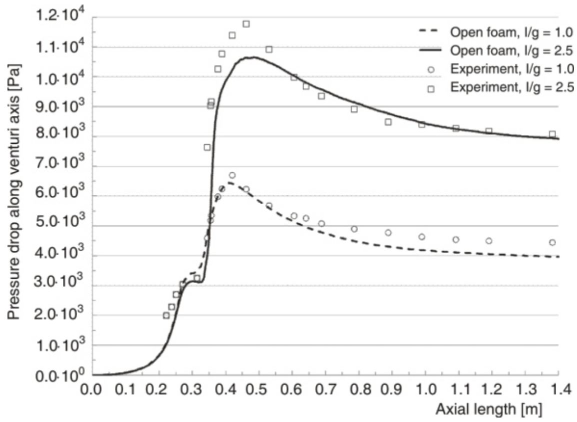

Figure 3. Comparison of calculated and experimental pressure profiles along the venturi

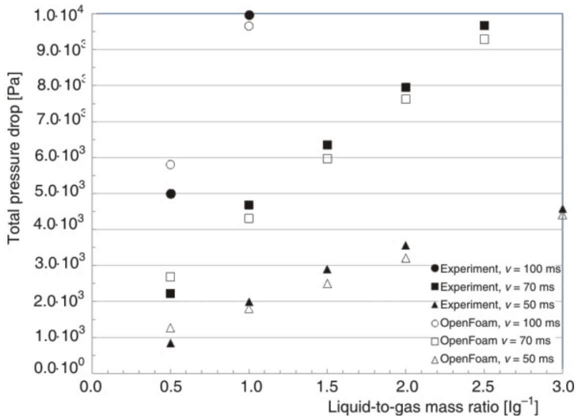

Figure 4. Comparison of calculated and experimental overall pressure drops in the venturi

imation and to save computation time, liquid film flow and droplet breakup models were not considered, but the differences between computed and experimental values were considered acceptable as a starting point to create a model of the venturi devices used in filtering systems.

MULTI-VENTURI SYSTEM

The multi-venturi filtering system taken as the base design for this study is the one described by Elisson [16]. This design is currently in use in Swedish nuclear power plants, and in one in Switzerland [5]. A schematic of the multi-venturi scrubber system developed in this work is shown in fig. 5. Table 2 shows some relevant design parameters.

By simply changing dimensions in the PHYTON scripts of the basic geometric model, a different geometric and mesh model of a single venturi device was constructed in SALOME, taking the model previously described as the starting point. Dimensions and SALOME mesh of the modified venturi model are shown in fig. 6. A total length of 1 m was chosen because the original designers found out that an equivalent pressure drop was necessary to achieve satisfactory cleansing of the entering gas/dust mixture. Thus, the base model only needs an increase in the entrance section length, as it can be seen on fig. 7, which shows the increased length of seven raiser tubes along transverse distribution ducts. All riser pipes in a corresponding concentric ring, or sector, have the same length, so it is only necessary to determine the hydrodynamic performance of one of them to know the overall sector performance, and then that of the whole system.

BWR VENTING CONDITIONS

The entire multi-venturi system just described was used to estimate its capacity of facing a typical venting action from a BWR Mark II primary containment, during a progressing severe accident. The boundary and initial conditions data were obtained from the simulation of a long-term station blackout by Gomez-Torres *et al*. [17]. In that research, the venting action started when containment pressure was 4.42 10^5 Pa (gauge) and the initial mass-flow was 28.75 kgs^{-1}, but flow quickly became critical, with the value of 13.2 kgs^{-1}. These choked conditions lasted for about 250 seconds. The containment depressurized rapidly too, falling to 3.92 10^5 Pa in about 20 seconds, and continued decreasing at about the same rate as transient evolved. Note that actual pressure at the scrubber system inlet would be less, because of the pressure drop along the venting line. Thus, comparing these values of pressure and mass-flow with those in tab. 2, it can be noted that the multi-venturi scrubber

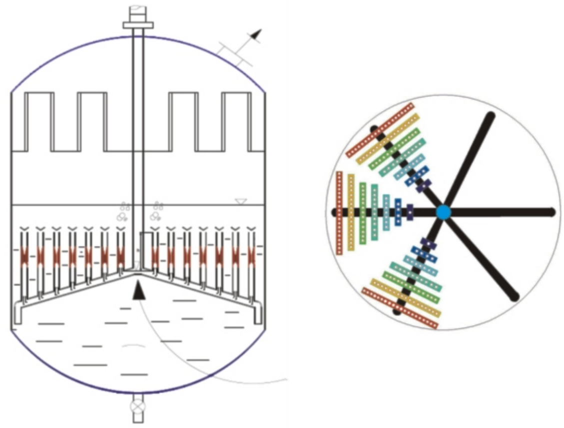

Figure 5. The multi venturi scrubber system design

Table 2. Multi-venturi scrubber design parameters [16]

Gas mass-flow rate	0.1-13 kgs^{-1}
Gas composition	Steam, N_2, H_2
Gas temperature	340.15-423.15 K
Design pressure	3.0 10^5 Pa + water column, for BWR
Rupture disc opening pressure	5.0 10^5-6.0 10^6 Pa
Aerosol mass mean diameter	1.5 10^{-6} m
Total amount of aerosols	90 kg, for BWR
Amount for radioactive aerosols	20 kg, for BWR
Minimum required decontamination factor for aerosols and elementary iodine	100, for BWR

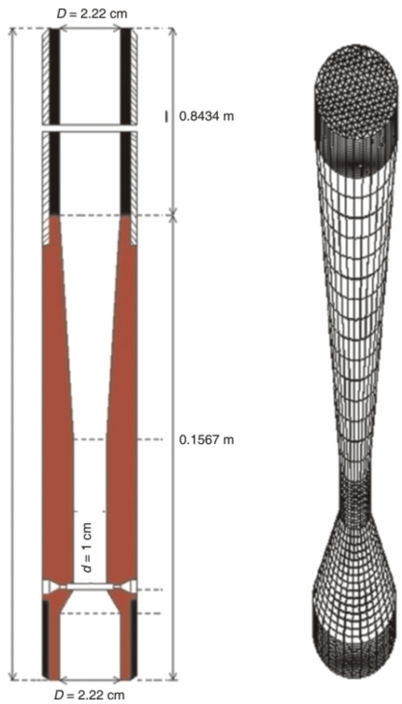

Figure 6. Dimensions of the base venturi model for the multi-venturi system

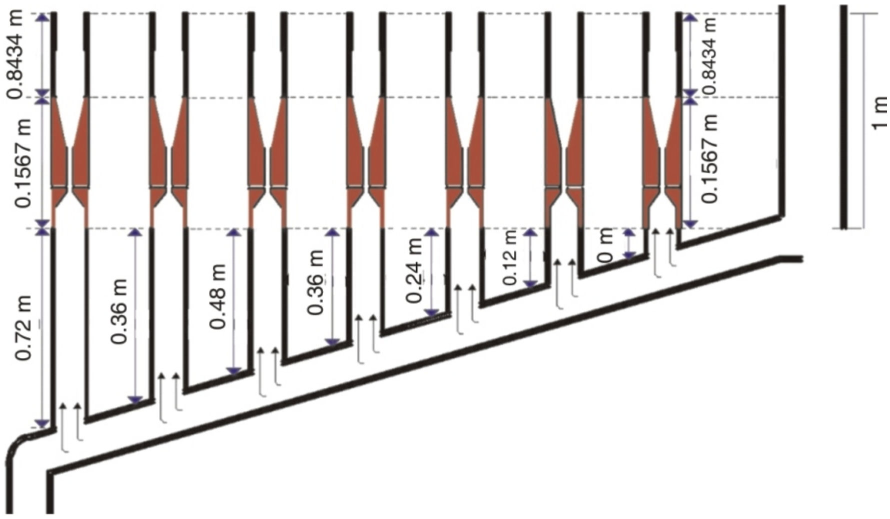

Figure 7. Length increase of the riser entrance section along a distribution duct

Table 3. Resulting mass-flow rates per venturi and per sector

Sector	Per individual venturi [kgs^{-1}]	Per sector [kgs^{-1}]
1	0.0182	0.1091
2	0.0236	0.5654
3	0.0259	1.0897
4	0.0274	1.6453
5	0.0285	2.2236
6	0.0294	2.9988
7	0.0302	3.6192

system could handle the incoming gas flow. However, it is still necessary to determine if it has the minimum pressure to move through the venturi devices and reach the surface of the tank water pool to become effectively cleansed.

To determine the gas mass-flow capacity of a single venturi in each ring, the necessary pressure boundary condition is 520945.0 Pa (absolute) at the surface of the water pool (output boundary). Additionally, the gas temperature was set at 423.15 K, and the gas mixture was composed of 38.7 % nitrogen, 3.9 % hydrogen, and 57.4 % steam. The computational procedure in OpenFoam started by setting the pressure at the surface of the water pool, and then iterating over different inlet gas-flow rates to obtain the inlet pressure necessary to sequentially activate each concentric sector. The calculation yielded the individual mass-flow for a single venturi in a ring and from that the total mass-flow was determined by the number of venturis in that sector (ring).

RESULTS

The first simulated test case corresponded to a gas stream through the venturi devices, but without the addition of water droplets, that is, as if the venturis did not have injection ports. Table 3 shows the mass-flows per individual venturi device in one of the transversal distribution pipes, and the total in that sector and fig. 8 shows the pressure required to activate each sequential sector, and the gas mass-flow that can be cleansed. Figure 9 shows pressure drop along the whole multi-venturi device, including the different entrance section lengths, as shown in fig. 7.

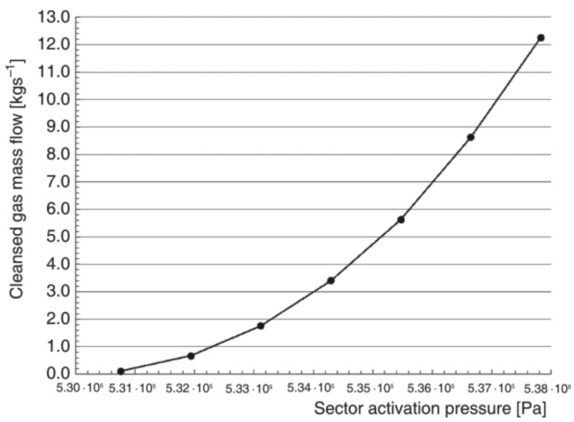

Figure 8. Cleansed gas mass-flow as the function of the activation pressure of each sector

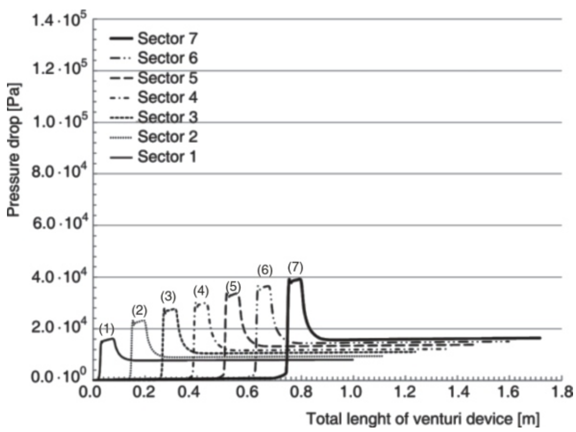

Figure 9. Pressure drop along flow direction. Case: no liquid injection

After that a parametric study was performed to determine the impact of liquid injection on the overall performance of the multi-venturi device, as the function of liquid to gas ratio at the throat. Figures 10-12, are similar to fig. 9, so they can be directly compared. The pressure drop profiles in figs. 10-12 are consistent with other similar studies, as for example the Eulerian-Lagrangian approach used by Pak and Chang [18]. All these figures show clearly how an increasing liquid load leads to higher pressure drops. This is due mainly to momentum exchange between the gas stream and the micro-droplets. This momentum exchange was explicitly modeled in the OpenFoam Eulerian and Lagrangian equations. The pressure drop at each of the venturi sections has been extensively researched and described. For example, Farbar [19] and Carlson [20] showed that pressure loss at high rates of particles (water micro-size droplets in this case) is due to interactions among those flowing particles. In pressurized systems, it is important to additionally include a model for the boundary-layer to avoid pressure drop under the predicted level, as shown by Sun and Azzopardi [21]. In the diffuser section of the venturi device there is a significant increase of high velocity droplets, increasing interaction probability among droplets and with the device wall, and thus leading to a noticeable pressure drop. Allen and Santen [22] showed that pressure drop and gas velocity depended strongly on the particle/gas mass loading, as it can be noted in figs. 10-12, where the total pressure drop (including the entrance and discharge pipe sections) for a single venturi in each sector is shown.

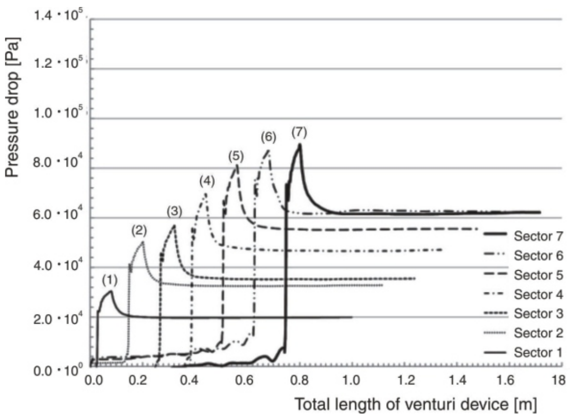

Figure 10. Pressure drop along flow direction. Case: l/g = 0.45

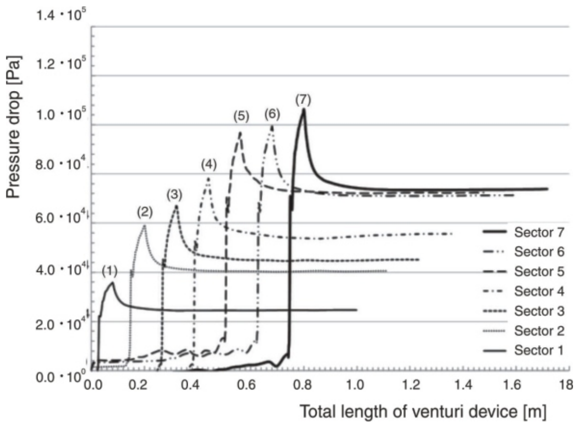

Figure 11. Pressure drop along flow direction. Case: l/g = 0.90

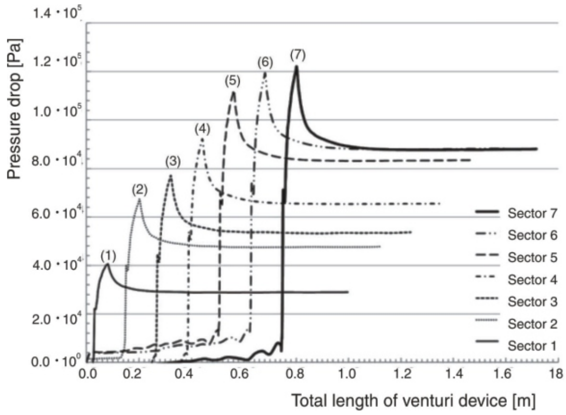

Figure 12. Pressure drop along flow direction. Case: l/g = 1.35

The Lagrangian approach was applied to the dispersed micro-droplets in order to study the efficiency of individual droplets to capture aerosol particles among other parameters, so Eulerian liquid velocity and volume fraction fields are not directly available, for the purpose of comparison with other studies. However, with OpenFOAM it is possible to recover the instantaneous volume fraction occupied by the droplets in the cells along their trajectories. In this way a time average value can be calculated from all time steps, assuming that the water mass balance has been reached. Figure 13 shows the time average droplet volume fraction along radial lines at different vertical positions of a venturi device in Sector 1. This venturi is basically the same as the one shown in fig. 6. As shown by Majid *et al*. [23], in their 3-D CFD simulation, both droplet size and distribution of liquid in the venturi depends greatly on the ratio of liquid to gas mass-flow rates. In that study, in the diffuser zone, for low

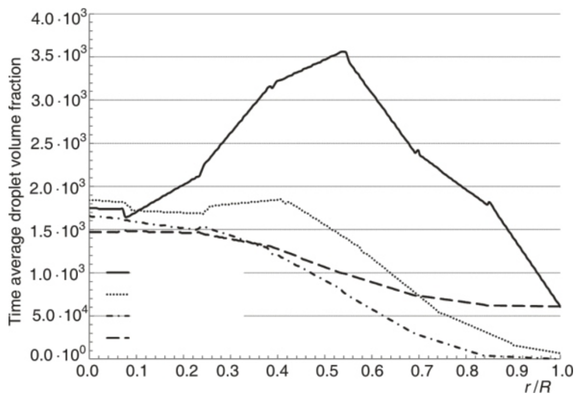

Figure 13. Radial profiles of droplet concentration at different heights of diffuser and discharge pipe. Case: l/g = 1.35

gas-flow rates, droplets tend to drift towards walls, and the higher the liquid flow rate, the stronger this effect becomes. On the other hand, higher gas-flow rates lead to higher concentration of droplets at venturi´s central region. In fig. 13, it can be noted that at the diffuser entrance, the higher droplet concentration is not at the central region, which corresponds to a mixture of high liquid flow rate and low gas-flow rate, but as the gas-liquid mixture flows upward, droplet concentration is higher at venturi´s center. In the discharge pipe entrance zone, droplet concentration is still higher at the center, but the profiles tend to become flatter.

The parametric studies can be extended to consider changes in geometry of the venturi device, in order to determine the capacity of the multi-venturi scrubber under different operational conditions. Additionally, the geometric model can be easily modified, if for example more distribution pipes are needed to cope with the incoming gas stream. In this work, only a typical venting pressure has been used in the analysis, but during a severe accident, the reactor operator and supporting technical staff decide about the time and duration of venting actions. This may strongly affect the pressure and gas flow being liberated. Once an initial venturi design satisfies primary needs of pressure drop, micro-droplet loading, throat velocity, *etc*., the physical models still missing in the present OpenFoam model can be implemented to optimize the initial design.

CONCLUSIONS

The hydrodynamic performance of a multi-venturi scrubber system has been studied in this project, under conditions of a BWR primary containment venting action. The multi-venturi system consisted of seven concentrically arranged sectors, and each sector had a different number of individual venturis. The base CFD model of a single venturi device was practically the same for all sectors, differing only in the length of the entrance section, to take into account the different depths at which each sector starts. Thus, all venturis of a given sector had the same hydrodynamic response to an incoming inlet gas.

The base CDF model of the single venturi device was developed in the environment of the SALOME and OpenFoam platforms. The hydrodynamic calculations were carried out with the physical and numerical models of the solver *reactingParcelFoam*. The results obtained with the base CFD model were tested against data from some experimental results of pressure drop profiles along the axis of the venturi and overall pressure drop as the function of liquid loading. The maximum relative errors were limited in a range of 10 %. This initial base CFD model was modified to create a representative model of a single venturi device as those used in the sequential sectors of a multi-venturi scrubber system. For given boundary and initial conditions corresponding to a venting action from a BWR Mark II primary containment and design conditions of a scrubber system, the gas mass-flow that can be cleansed by individual venturis and the pressure required to activate those venturis were computed. Also, the pressure drop profiles in each sector were determined as the function of different liquid loadings. The results obtained are in good agreement with those of the reference system design.

ACKNOWLEDGMENT

The authors wish to strongly acknowledge the support of SENER-CONACYT to the project FSE-2013-04 -213519 through the Energy Sustainability Fund. The support of ININ to the project CA-509 is also acknowledged.

AUTHORS' CONTRIBUTIONS

The main CFD modeling and calculations of the multi-venturi system performance was carried out by A. A. Reyes-Garcia for his MSc Thesis; the principal developer of CFD models in OpenFoam is E. Sainz-Mejia; J. Ortiz-Villafuerte was in charge of experimental data collection, comparison with numerical results and analysis, and is also the main developer of the manuscript; the challenges of applying CFD modeling to practical nuclear reactor issues were developed by J. C. Palacios-Hernandez; and R. C. Lopez-Solis gathered and organized the data, and wrote the first version of the manuscript. All authors participated in the final analysis and discussion.

REFERENCES

[1] Gabor, J., *et al*., Investigation of Strategies for Mitigating Radiological Releases in Severe Accidents - Mark I and Mark II Studies, 2012 Technical Report. 1026539 Final Report, Electric Power Research Institute, Palo Alto, California, U.S.A., September 2012

[2] ***, United States Nuclear Regulatory Commission, Installation of a Hardened Wetwell Vent. Generic Letter 89-16, Washington D.C., USA, 1989

[3] ***, United States Nuclear Regulatory Commission, Order to Modify Licenses with Regard to Reliable Hardened Containment Vents Capable of Operation Under Severe Accident Conditions, Order EA-13-109, Washington D.C., USA, 2013

[4] ***, United States Nuclear Regulatory Commission, Evaluation of The Containment Protection and Release Reduction for Mark I and Mark II Boiling Water Reactors Rulemaking Activities (10 CFR Part 50) (RIN-3150-AJ26). SECY-15-008, Washington D.C., USA, 2015

[5] ***, Nuclear Energy Agency – Committee on the Safety of Nuclear Installations, OECD/NEA/CSNI Status Report on Filtered Containment Venting. NEA/CSNI/R(2014)7, 2014

[6] Calvert, S., Venturi and Other Atomizing Efficiency and Pressure Drop Scrubbers, *AIChE Journal, 16* (1970), 3, pp. 392-396
[7] Yung, S. C., *et al*., Pressure Loss in Venturi Scrubbers, *Journal of the Air Pollution Control Association, 27* (1977), 4, pp. 348-351
[8] Goncalves, J. A. S., *et al*., Evaluation of the Models Available for the Prediction of Pressure Drop in Venturi Scrubbers, *Journal of Hazardous Materials, B81* (2001), pp. 123-140
[9] Ali, M., *et al*., A Review of Performance of a Venturi Scrubber. Research Journal of Applied Sciences, *Engineering and Technology, 4* (2012), 19, pp. 3811-3818
[10] Gulhane, N. P., *et al*., Experimental Study of Iodine Removal Efficiency in Self-Priming Venturi Scrubber, *Annals of Nuclear Energy, 78* (2015), pp. 152-159
[11] Zhou, Y., *et al*., Experimental Research on Aerosols Collection Performance of Selfpriming Venturi Scrubber in FCVS, *Progress in Nuclear Energy, 85* (2015), pp. 771-777
[12] ***, OPEN CASCADE, SALOME The Open Source Integration Platform for Numerical Simulation. https://www.salome-platform.org, (CEA, EDF & OPEN CASCADE) Copyright 2005-2019
[13] ***, OpenCFD Ltd (ESI Group), OpenFOAM: The Open Source CFD Toolbox, https://www.openfoam.com, Copyright 2004-2018,
[14] Haller, H., *et al*., Venturi Scrubber Calculation and Optimization, *Chemical Engineering Technology, 12* (1989), pp. 188-195
[15] Goniva, C., *et al*., Simulation of Off Gas Scrubbing by a Combined Eulerian-Lagrangian Model, *Proceedings*, 7th International Conference on CFD in the Minerals and Process Industries (CSIRO), Melbourne, Australia, 2009
[16] Elisson, K., A Multi Venturi Scrubber System for Containment Venting, *Proceedings*, Specialists' Meeting on Filtered Containment Venting Systems. OECD/NEA/CSNI Report No 148, Paris, France, 1988
[17] Gomez-Torres, A. M., *et al*., CDF Analysis of hydrogen volumetric concentrations in a Hard Venting Containment System of a Mark II BWR, *Annals of Nuclear Energy, 85* (2015), pp. 552-565
[18] Pak, S. I., Chang, K. S., Performance Estimation of a Venturi Scrubber Using a Computational Model for Capturing Dust Particles with Liquid Spray, *Journal of Hazardous Materials, B138* (2006), pp. 560-573
[19] Farbar, L., The Venturi As a Meter for Gas-Solids Mixtures. *Proceedings*, ASME Annual Meeting, New York, USA, 1952
[20] Carlson, H. M., *et al*., Meter for Flowing Mixtures of Air and Pulverized Coal, *Trans ASME, 70* (1948), pp. 65-79
[21] Sun, H., Azzopardi, B. J., Modeling Gas-Liquid Flow in Venturi Scrubbers at High Pressure, *Process Safety and Environmental Protection, 81* (2003), 4, pp. 250-256
[22] Allen, R., Santen, A., Designing for Pressure Drop in Venturi Scrubbers: The Importance of Dry Pressure Drop, *The Chemical Engineering Journal and the Biochemical Engineering Journal, 61* (1996), 3, pp. 203-211
[23] Majid, A., *et al*., CFD Simulation of Dust Particle Removal Efficiency of a Venturi Scrubber in CFX, *Nuclear Engineering and Design, 256* (2013), pp. 169-177

Received on June 10, 2019
Accepted on February 14, 2020

Алехандро Антонио РЕЈЕС-ГАРСИА, Едуардо САИНЗ-МЕХИА,
Хавиер ОРТИЗ-ВИЉАФУЕРТЕ, Хавиер Ц. ПАЛАСИОС-ХЕРНАНДЕЗ,
Роберто Карлос ЛОПЕЗ-СОЛИС

СТУДИЈА ХИДРОДИНАМИЧКИХ ОСОБИНА ВИШЕСТРУКОГ ВЕНТУРИ ФИЛТРАЦИОНОГ СИСТЕМА ЗА ПОТРЕБЕ СТРАТЕГИЈА ИСПУШТАЊА ГАСА У СЛУЧАЈУ ТЕШКИХ АКЦИДЕНАТА BWR РЕАКТОРА

Циљ овог пројекта је испитивање издржљивости система за филтрацију са вишеструким вентуријевим пречишћивачем при наиласку испуштеног гаса великих брзина протока из примарног суда BWR Mark II реактора, током дугорочног губитка електричне енергије. Развијени су модели система за филтрацију са вишеструким вентури пречишћивачима помоћу програма SALOME и OpenFoam слободног приступа. Први геометријски модел направљен је према димензијама познате експерименталне поставке и резултати губитка притиска дуж правца простирања протока показују максималну разлику од 10 % у поређењу са експерименталним вредностима, за различите односе течности и гаса. Након тога развијен је вишеструки вентуријев модел стварних димензија. Како би се испитале перформансе овог система при условима који се могу очекивати током тешког акцидента, као улазни гас коришћена је смеша гаса слична оној која се јавља у BWR Mark II реактору при притиску испуштања. Проток масе гаса тада се може прочистити појединачним вентури пречишћивачем при чему су израчунате вредности притиска за активирање пречишћивача. Одређени су и профили пада притиска за сваки сектор као функција различитих оптерећења улазних течности. Резултати показују добро слагање са могућностима дизајна одабраног као референтни модел.

Кључне речи: BWR Mark II, тешки акцидент, OpenFoam, SALOME, вентури пречишћивач