

## CAVITATING FLOW PATTERN CHARACTERIZATION IN SQUARE SECTION INJECTORS BY MEANS OF CFD

Miguel G. Coussirat<sup>a</sup>, Flavio H. Moll<sup>a</sup> and Alfred Fontanals<sup>b</sup>.

<sup>a</sup>*Grupo LAMA- Universidad Tecnológica Nacional-Facultad Regional Mendoza, Argentina;*

<sup>b</sup>*Departamento de Mecánica de Fluidos, Escola Enginyeria Barcelona Est, EEBE. Universitat Politècnica de Catalunya, España*

**Keywords:** Cavitating Flow, Turbulence, Square Section Injectors, Atomization, CFD.

**Abstract.** Cavitation in pressure atomizers strongly affects the liquid/spray jet behavior at their outlets. The liquid atomization at the outlet jet is promoted when the cavitation stage is fully-developed into the atomizer nozzle. The type of atomization induced by cavitation allows developing more efficient atomizers when this cavitation stage is controlled. Cavitating flow is related to turbulent and multi-phase flows with mass transfer between the liquid and gaseous phases, and it is affected by several factors related to the physical properties of the fluid and the characteristics of the fluid flow. Due to the high-speed flow and small spatial-time scales involved, cavitation studies in injectors by experiments is very expensive. On the other hand, despite that several codes for numerical modeling of cavitating flows have been developed, this kind of flow modeling is still a big challenge, because cavitating flows should not be modeled as typical turbulent flows. There is a close relation between the cavitation inception/developing condition and the turbulence level in the flow leading to a non-standard turbulence state. When the Reynolds Averaged Simulation (RAS) of turbulence is used, previous works showed that in general its influence over the obtained results is bigger than the influence of the selected mass transfer model for a suitable cavitating flow modeling. It is also demonstrated that this selection could be competitive compared to more sophisticated options, e.g., Large Eddy Simulations (LES), when only a general flow structure and the mean/global values are required. In this work more evidence related with the capability of the RAS turbulence models are presented, and the obtained conclusions could be useful to improve injectors design by numerical modeling because the detection of the incipient/quite developed cavitation flow conditions is captured accurately using low computational resources.

## 1 INTRODUCTION

Cavitation is a complex phenomenon that appears in liquid flows when the hydrodynamic pressure,  $P_c$  in some place falls down reaching the vapor pressure of the liquid,  $P_v$ . This low pressure provokes that the initial liquid flow becomes a two-phase flow, i.e., liquid-bubbles of vapour, [Knapp et al., 1970](#), [Brennen, 1995](#), [Brennen, 2005](#). The initiation of cavitation by vaporization of the liquid may require the existence of stresses lower than vapor pressure due to the surface stress tension in the bubble. However, the presence of undissolved gas particles, boundary layers, and turbulence will modify and often mask a departure of this critical pressure  $P_c$  from  $P_v$ , [Ardnt, 2002](#).

Several Computational Fluid Dynamics (CFD) codes based on a certain kind of ‘multi-phase flow modeling’ technique, involving both mass transfer (by cavitation) and turbulence submodels, have been developed for studying cavitating flows. For turbulence modeling a Reynolds Averaged Simulation plus an Eddy Viscosity (or Reynolds Stress Model), i.e., RAS+EVM(RSM), formulation for the mixture (liquid+vapor) have been developed in the two last decades and is commonly used now. It is frequently so-called RAS turbulent multi-phase (RAS-TMF) flow modeling, [Ruiz et al, 1999](#), [Yuan et al., 2001](#), [Schnerr et al, 2001](#), [Singhal et al., 2002](#), [Coutier-Delgosha et al, 2003](#), [Habchi et al, 2003](#), [Palau et al., 2004](#), [Martynov et. al, 2006](#), [Giannadakis et al, 2008](#), [Li et al, 2008](#), [Darbandi, et al, 2010](#), [Gonçalves et al, 2012](#), [Payri et al, 2012](#), [Zhang et al, 2013](#), [Duke et al., 2014](#), [Sou et al., 2014](#), [Congedo et al, 2015](#), [Naseri et al, 2015](#), [Coussirat et al., 2016a](#), [Koukouvinis et al, 2016](#), [Coussirat et al, 2017](#), [Ghorbani et al, 2017](#).

More recent options for turbulence modeling implying a different kind of filtering strategy, such as Large Eddy Simulation (LES) or Hybrid Formulations, (DES/VLES) for the mass, momentum and energy transport equations are available too, [Coussirat et al., 2016b](#), [Sou et al., 2014](#), [Koukouvinis et al, 2016](#).

In the RAS-TMF flow modeling the fluid mixture density  $\rho_m$ , and the turbulent dynamic viscosity of the flow  $\mu_{t,m}$ , are closed by a cavitation and a turbulence model respectively. This  $\mu_{t,m}$  is added to the molecular viscosity of the mixture  $\mu_m$ , to obtain an effective or total viscosity. Nowadays, there are several options and possible combinations of these submodels inside a commercial or an in-house CFD code, [Li et al, 2008](#), [Zhang et al, 2013](#), [Coussirat et al., 2016b](#), [Coussirat et al, 2017](#). In spite of this, cavitating flows are still a big challenge, because cavitating flow involves non-standard turbulent flow modeling, due to this modeling offers several difficulties for a suitable modeling by means of the available CFD codes, [Coussirat et al., 2016b](#), [Giannadakis et al, 2008](#), [Duke et al., 2014](#), [Sou et al., 2014](#), [Congedo et al, 2015](#), [Naseri et al, 2015](#). There is a close relation between the cavitation inception/developing condition and the turbulence level in the flow leading to a non-standard turbulence state, so cavitating flows should not be modeled as typical turbulence, [Coussirat et al., 2016a,b](#), [Coussirat et al, 2017](#).

A possible reason for this non-standard state is that turbulence affects cavitation inception since a nucleus may be found in the core of a vortex where the local pressure level is lower than the mean value of the pressure in the flow. Hence, the nucleus could cavitate when it should not do under the influence of the mean pressure level. This fact points out that cavitation may alter the global pressure field modifying the location of flow separation and the induced variations of the local turbulence level. Thus, turbulence may promote cavitation and vice versa. Although some details of these complex viscous effects on cavitation inception were extensively examined by several authors in the past, [Knapp et al, 1970](#), [Brennen 2005](#), the effects such as interaction of turbulence and cavitation inception have been recently identified more clearly, [Tseng et al., 2014](#), [Koukouvinis et al, 2016](#), [Ghorbani et al, 2017](#). Then, inappropriate predicted values for the pressure fluctuation due to the turbulence model selected could affect the cavity shape prediction and the general cavitation stage. A broad

discussion, details and an extensive list of references related to the aforementioned topics can be found in previous works, [Coussirat et al., 2016a,b](#) and [Coussirat et al, 2017](#).

Under the viewpoint of engineering applications, due to the high pressure drop in the injector orifice, cavitation occurs almost always in high-pressure Diesel injectors and it is of paramount importance to take advantage of the useful effects that cavitation could provide, i.e., sprays of good quality. In turn, the potential capability of the damage that the cavitating flow has on the injector has to diminish when the device works under design conditions. Due to difficulties (technical and costs) to carry out experiments in real Diesel injectors, the CFD tool could be an interesting option for nozzle designs. The interaction of turbulence and cavitation has been extensively studied either experimentally or by means of the CFD tool, see full details in [Coussirat et al., 2016a,b](#), [Coussirat et al, 2017](#). For Diesel injectors, the early and recent experimental results from [Nurick, 1976](#), [Nurick et al, 2008](#), and [Nurick, 2011](#), [Sou et al, 2006](#), [Sou et al, 2008a,b](#) and [Sou et al, 2014](#) (used also in this work), were extensively used in several of the aforementioned CFD works available in the literature.

Current CFD applications could be useful to understand the impact of some of the design/operating variables under cavitation conditions. It is important the role of these variables to identify the correct state of the internal orifice flow, because this state has a tremendous effect on the external spray.

## 2 CFD OF TURBULENT CAVITATING FLOW IN INJECTORS

This work is related to CFD modeling of cavitating flows in confined geometries, more specifically, dealing with high-pressure Diesel injectors of square sections. One of the main subjects here is to gain insight in the behavior and performance of the numerical two-phase flow models developed for general usage, applied to the design of devices where cavitating flows appear. The CFD code calibration is a necessary task here, and it must rely on a physical basis. To discuss with more details the physical basis of the interaction between turbulence and cavitation, some of the experiments used in the already mentioned works are revisited. The present work tries to obtain a deeper insight of the RAS+EVM(RSM) turbulence modeling for square section nozzles, analyzing both the mean velocity field and their fluctuations for subsequent calibration tasks taking into account the problems that EVM models have when they are applied to compute turbulent flow in square section ducts. Then, to analyze with more details the physical basis of the interaction between turbulence and cavitation, some of the experiments used in [Coussirat et al, 2017](#) were revisited bearing in mind the following considerations for turbulence modeling:

1)The primary role of the suitable prediction of  $\mu_{t,m}$ , because it provokes the mixing rising, related to the energy dissipation and the flow losses downstream of the orifice. Thus, the vapor is transported more slowly, increasing its dissipation and bringing the liquid phase towards a two-phase region, affecting the vapor convective transport and forcing the two-phase flow region to shorten. In this way, the flow pattern looks steadier and the two-phase regions of the turbulent simulations are smaller. This difference is generated due to the effect of the  $\mu_{t,m}$  level, computed by the turbulence model, being  $\mu_{t,m} \gg \mu_m$  which damps the unsteadiness occurred in the laminar simulations.

2)The turbulence field is qualitatively and quantitatively different under non-cavitation or cavitation conditions, [Ruiz et al., 1999](#). The general level of turbulence in cavitating flows presents higher values and goes down more slowly along the orifice length  $L$ . The turbulence is concentrated either nearer the orifice wall, or closer to the jet surface once outside the orifice. The spatial distribution and the slow decay of the turbulence produced in cavitating

flow in orifices could be related to some preferred turbulence scales in the process, and the main conclusion here is that cavitating flows should not be modeled as typical turbulence.

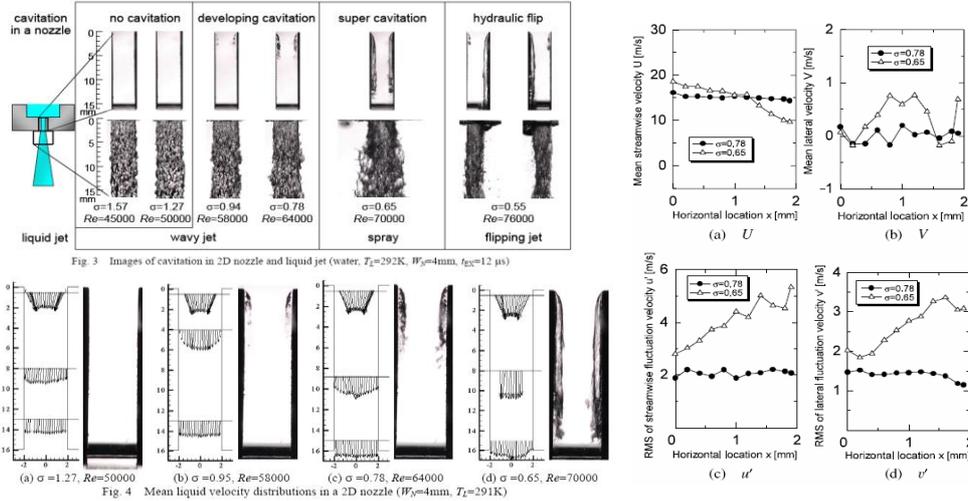
**3)**The flow separation into the injector nozzles may be caused either by the sharpness of the nozzle inlet lip or by the cavitation process, [Habchi et al, 2003](#). The separation formed by the primary phase flow influences the extension of the two-phase region. In zones nearer the sharp edged inlet orifice corners, the pressure goes down and vapor appears provoking a diminution of  $\mu_{t,m}$ . Hence, the small scales of turbulence do not have a relevant influence here. On the other hand, the interface is wrinkled downstream of the sharp edged orifice; and in the reattachment region the sub-grid scale turbulence certainly plays a major role in the break-up and coalescence of the collapsing bubbles. **4)**No clear conclusion about the superiority of one turbulence EVM model over other could be pointed out, despite many works related to the performance of several EVMs and cavitation models combinations exist, [Coussirat et al, 2017](#). In all cases, the CFD results obtained led to the stable cavity predictions, because higher levels of the  $\mu_{t,m}$  computed inside the cavity avoids the re-entrant jet formation, suppressing the instability of partial sheet cavity. The influence of the experimental data in the setting of turbulence and cavitation model coefficients was also studied as a possible solution to reduce the  $\mu_{t,m}$  levels. **5)**[Gonçalves et al, 2012](#), [Naseri et al, 2015](#) and [Coussirat et al, 2016a,b](#) showed some CFD results changing the production and dissipation coefficients in the turbulence transport equations of some EVM for steady and unsteady simulations. These corrections led to reduce the  $\mu_{t,m}$  in mixture regions, allowing reproducing some of the observed experimental cavitation conditions (inception and shedding pattern in unsteady cases) remarking the importance of an accurate capture of the pressure fluctuations induced by the turbulence is remarked.

### 3 CFD SETUP DEFINED FOR SQUARE SECTION INJECTORS

A commercial code [Ansys/Fluent v12 Software, 2018](#) was used for modeling a turbulent cavitating flow in a square section orifice injector. The geometry and flow data were extracted from the [Sou et al., 2006](#) and [Sou et al, 2008a,b](#), see [Fig.1](#). The geometry selected was a square nozzle (2D model) where:  $l=1.0$  mm,  $D/w=8$ ,  $L/w=4$  and  $l/w=0.25$ , being  $l \times w$  the outlet nozzle area (height  $\times$  width);  $L$ , the wall nozzle length and  $D$  the inlet nozzle diameter. The flow data to define the boundary conditions and to compare against the CFD results were extracted from the aforementioned references, being: **1)**The mean flow velocity at the outlet computed from the measured mass flow, measured by means flowmeters, being the uncertainty in the measured flow rates less than 3.7%; **2)**The static pressure upstream of the nozzle (for some cases), measured by means of a Bourdon pressure gages; **3)**The velocity profiles at some positions in the nozzle, measured by means of the Laser Doppler Velocimeter, LDV technique; **4)**The concentration of oxygen dissolved in the water was measured using a dissolved oxygen probe. **5)**Images for the vaporisation cavity structure when cavitation is present were taken using a digital camera and a flash lamp, but without any information about the vapour fraction values into the cavity. For the items **2-4** none information related to the probes/measurements uncertainties was given.

For the selected geometries, the CFD estimations for flow separation and reattachment are strongly dependent on a correct prediction for the development of the near-wall turbulence and its instability, leading to the possibility to have an unsteady flow, giving place to Unsteady Reynolds Averaged Simulations (URAS), more expensive in terms of CPU requirements. Fortunately, studies performed in a previous work from [Coussirat et al, 2017](#), based in the Strouhal number computation allow justifying the use of steady state CFD simulation in the cases selected to detect the incipient cavitation state.

Over the selected geometries, the boundary conditions were defined in the same way as experiments, i.e., a value of  $P_B=95,000$  Pa was set at the outlet in all cases, and for each case an inlet velocity value,  $c_I$ , was defined from its associated  $\sigma$  and Re number values, and applying the continuity principle between the inlet and outlet of the injector, see Table.1.



**Fig.1:** Experimental data for square nozzles, Sou et al., 2006, Sou et al., 2008a,b. **Left:** Cavity structure and mean velocity field in the nozzle measured (LDV technique). **Right:** Mean velocity streamwise and transversal components ( $U, V$ ) and their turbulent *rms* fluctuations ( $u', v'$ ) at some nozzle positions (only  $y=15mm$  is shown here). **Notation:**  $Re=c_B W/v_b$ , Reynolds number;  $\sigma$ , cavitation number,  $\sigma=(P_B-P_v)/(0.5\rho_l c_B^2)$ ;  $T_L$ , liquid temperature;  $W_N=w$ , nozzle width;  $t_{EX}$ , exposition time (photos);  $P_B, P_v$ , outlet and vapour pressures;  $\rho_l$ , liquid density;  $c_b$ , flow velocity at the nozzle  $v_b$ , liquid phase kinematic viscosity.

Table.1 also shows the different turbulence models used for each case. In all the modelled cases the Singhal cavitation model was used because in previous works, Coussirat et al., 2016a,b, Coussirat et al., 2017, it was demonstrated that the predictions of incipient cavitation stage in cavitating flows is more sensitive to the turbulence modeling than the cavitating flow modeling. For defining the mesh size, the sensitivity mesh analysis (i.e., comparison of CFD results between several meshes, and 2D and 3D cases) already performed in Coussirat et al., 2016a was used as reference.

The mesh defined was a hybrid mesh with quadrilateral/triangular cells 26,600 cells. In all the cases modeled the cell size near the wall (at the nozzle inlet zone) was computed in order to obtain values for  $y^+ < 15$ .

$l/w$	$c_B[m/s]$	12.50	14.50	16.00	17.50
4.0	$\sigma$	1.27	0.94	0.78	0.65
	$Re [\times 10^4]$	5.00	5.80	6.40	7.00
	models used	ABCD	ABCD	ABCD	ABCD

**Table.1:** Square nozzles,  $l=1mm$ ,  $D/w=8.0$ ,  $L/w=4.0$ , and  $l/w=4.0$ , (2D) from Sou et al., 2006, Sou et al., 2008a,b databases. CFD cases modeled using the EVMs consigned in the table combined with the Singhal model.

**Notation:** A, Spalart Allmaras; B, Standard  $k-\epsilon$  (i.e., Std  $k-\epsilon$ ); C, SST  $k-\omega$ ; D, RSM.

It is necessary to take into account that the grid convergence studies with the wall functions approach fail in some cases because the wall boundary condition is ill-posed. Therefore, for a well-posed boundary condition the inner limit was defined for a suitable application of the standard wall functions here, (i.e., at a value of  $y^+ \sim 11$ , laminar sublayer); although the SA and SST  $k-\omega$  do not use this wall treatment. On the other hand, the Std  $k-\epsilon$  and the RSM models

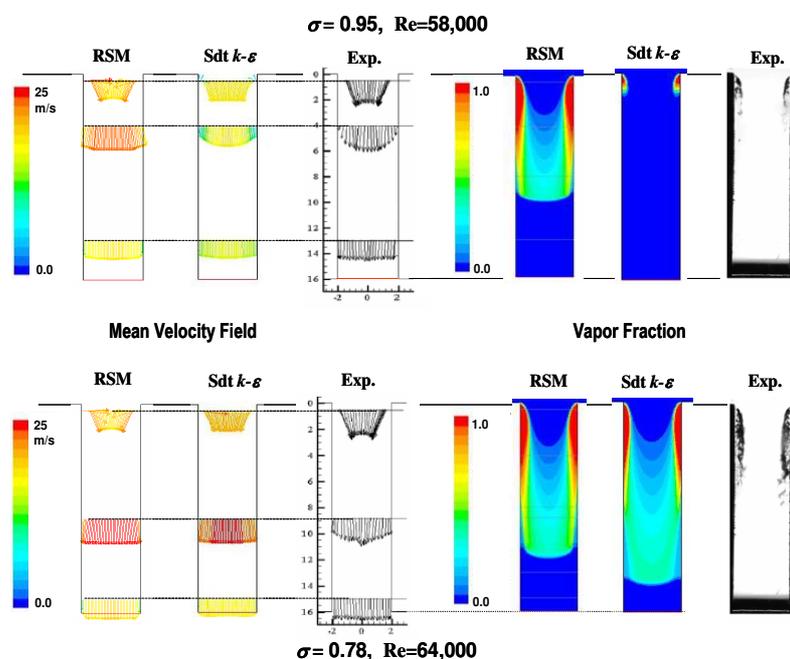
use standard wall functions. For the former two models, a specific ad-hoc near-wall treatment was selected (wall functions or simplified equations for boundary layer turbulence, see details in [Ansys/Fluent v12 Software, 2018](#)).

Once the geometry and the boundary conditions were define, the following CFD setup was also defined in each case: **1)**The selection of Second-order Upwind schemes for all the equations (flow and turbulence), except for the vapor transport equation, where the ‘QUICK’ scheme was selected. **2)**The selection of the ‘SIMPLE’ scheme for the pressure–velocity coupling; **3)**The definition of values for the dissolved gases contained in the liquid phase into a range of  $(1-9) \times 10^{-9}$  ppm depending on the available information from experimental data; **4)**An  $O(10^{-5})$  imposition for the normalized residuals; **5)**The prescription of double precision in the computed values.

#### 4 DISCUSSIONS OF THE OBTAINED RESULTS

The CFD obtained results for the general flow pattern and the cavity shape were compared against experiments, [Fig.2](#). The results for the averaged mean velocities at the outlet,  $c_B$  were obtained for the values of the  $\sigma$  coefficient extracted from [Table.1](#), and they were compared against the experimental results, [Table.2](#). Finally the mean and fluctuating velocity profiles for both velocity components (i.e.,  $U, V$  and  $u', v'$ ) were compared against experiments at some positions along the nozzle, [Fig.3](#) and [Fig.4](#).

[Fig.2](#) shows the general velocity pattern and the cavity shape obtained for incipient and quite developed cavitation stages. The general flow pattern looks similar to experiments and for the quite developed cavitation stage the ‘mean’ cavity shape is predicted with some accuracy. The experiments show that the cavity grows for the full developed cavitation stage. In both cases the general flow pattern looks qualitatively similar. The Std  $k-\epsilon$  shows better predictions of the cavity evolution. Despite that RSM recover more physics due to tensorial character the cavity evolution is not well captured.



**Fig.2:** Comparison of CFD results (vapor fraction and velocity fields), under two flow conditions, (i.e., **Top:**  $\sigma=0.95$  and **Bottom:**  $\sigma=0.78$ ), see [Table.1](#). CFD results obtained by using the Std  $k-\epsilon$  and the RSM models. Experimental data are from [Sou et al., 2006](#). **Notation:**  $\sigma=(P_B-P_v)/(0.5\rho_l c_B^2)$ ;  $Re$ : Reynolds number is based on  $c_B$  and  $w$ .

Table.2 shows comparisons between the experimental and the CFD obtained results for the averaged mean velocities at the outlet,  $c_B$  for several values of the  $\sigma$  coefficient, see Table.1. It was observed that for incipient and quite developed cavitation, the averaged mean velocity at the nozzle outlet is well predicted. The best results were obtained by using the Std  $k-\epsilon$  model. Instead, for fully developed cavitation stages these values are overpredicted. An analysis of the mean and fluctuating velocity profiles, see Fig.3, allows to say that in all cases the velocity profile is quite well predicted near the nozzle inlet (i.e., at  $y=0.5\text{mm}$ ). For the non-cavitating flow ( $\sigma=1.27$ ) the recirculation zone is overpredicted by the the SA, SST  $k-\omega$  and RSM models. Instead, the Std  $k-\epsilon$  predicts more accurately the recirculation zone, but the velocity maximum at this zone is underpredicted.

$\sigma$	Exp. Sou et al, 2006	SA (A)	Std $k-\epsilon$ (B)	SST $k-\omega$ (C)	RSM (D)
1.27	12.50	12.53	12.50	12.53	12.36
0.95	14.50	14.22	14.49	14.19	14.69
0.78	16.00	19.64	16.67	19.53	19.38
0.65	17.50	26.26	23.53	25.27	25.62

Table.2: Comparisons of the averaged mean velocity at the outlet, i.e.,  $c_B$  [m/s].

Notation: SA, Spalart Allmaras; Std  $k-\epsilon$ , Standard  $k-\epsilon$ ; SST, SST  $k-\omega$ ; RSM, Reynolds Stress Model.

For the quite developed cavitation ( $\sigma=0.95$ ) the general flow pattern is well predicted by all turbulence models in this zone, except by the SST  $k-\omega$ . When the cavitation stage becomes fully developed the predictions get worse and the zone with velocity recirculation around the cavity is overpredicted.

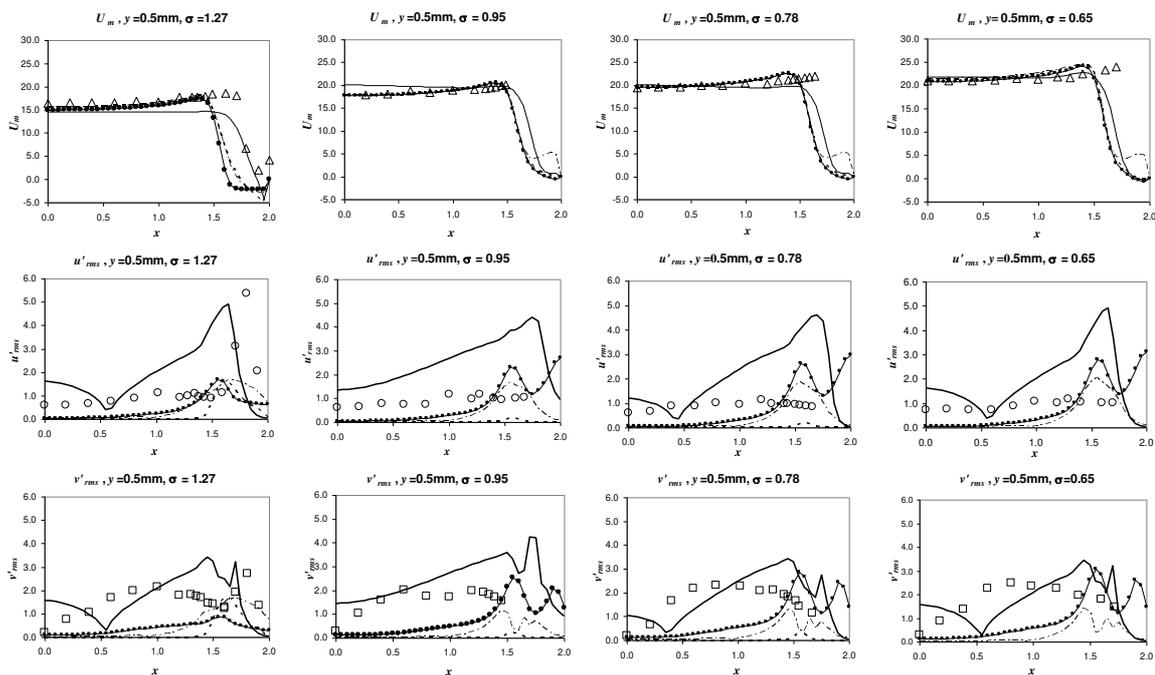
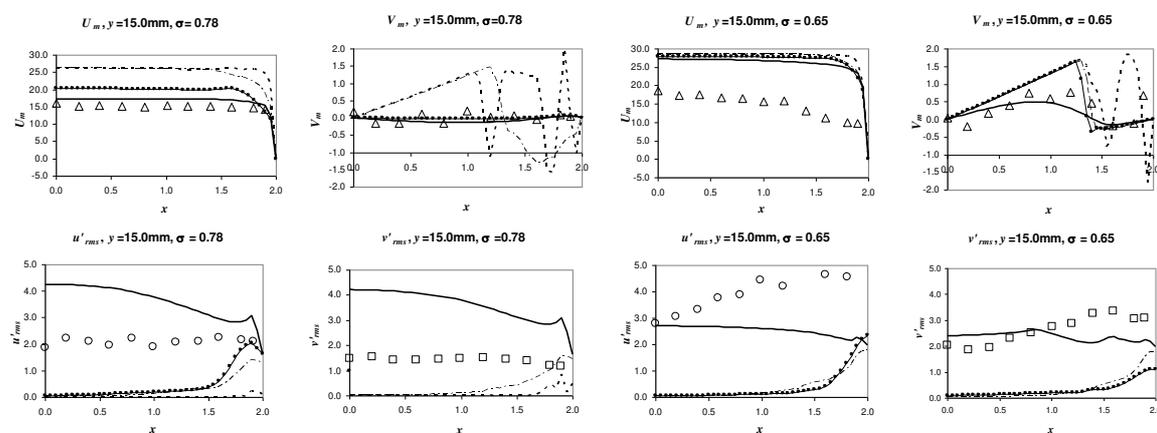


Fig.3: CFD comparisons for the velocity profiles at 0.5 mm from the nozzle inlet ( $y=0.5\text{mm}$ ) and depending on the  $\sigma$  value. **Top:** Streamwise component,  $U_m$ ; **Medium:** Streamwise fluctuating component,  $u'_{rms}$ ; **Bottom:** Lateral fluctuating component,  $v'_{rms}$ . **Notation:** Exp. data, Sou et al, 2006 ( $\Delta$ ,  $U_m$ ;  $O$ ,  $u'_{rms}$ ;  $\square$ ,  $v'_{rms}$ ). CFD results: —, Standard  $k-\epsilon$ ; - - -, SST  $k-\omega$ ; ·····, SA; —●—, RSM.

In general, near the nozzle inlet the fluctuations were strongly underpredicted for all turbulence models except for the Std  $k-\epsilon$ , Fig.3. At the center of the jet the fluctuations predicted tend to zero but experiments show that  $u'_{rms}$  has a value 0.5m/s. For each  $\sigma$ , the variations in the both fluctuations levels predicted are similar for all EVM models, showing

the intrinsic scalar character of the  $\mu_{t,m}$  (Boussinesq hypothesis). Instead, the RSM shows more sensitivity in the fluctuations predicted for the different cavitation stages and there are different variations in each fluctuating component.

At positions nearer the nozzle outlet only the Std  $k-\epsilon$  shows well predicted mean velocity values for quite developed cavitation stage, but the fluctuating velocities were overpredicted, Fig.4. All the other turbulence models overpredict the  $U_m$  velocity and underpredict the fluctuations. The  $V_m$  component is well predicted by the Std  $k-\epsilon$  and the RSM models. Notice that the SA and SST  $k-\omega$  showed strong oscillations in the lateral mean velocity  $V_m$ . This could be related with the overprediction of the cavity shape that becomes incompatible with the outlet boundary condition for the vapor fraction. Finally, for the full developed cavitation stage the prediction of the mean and fluctuating velocity fields get worse for all turbulence model tested. Only the Std  $k-\epsilon$  show predictions of the same order as experiments.



**Fig.4:** CFD comparisons for the velocity profiles at 1.0mm from the nozzle outlet ( $y=15\text{mm}$ ) and depending on the  $\sigma$  value. **Top:** Streamwise and lateral mean components ( $U_m, V_m$ ); **Bottom:** Streamwise and lateral fluctuating components ( $u'_{rms}, v'_{rms}$ ). **Notation:** Exp. data, [\*\*]Sou et al, 2006 ( $\Delta, U_m, V_m$ ; O,  $u'_{rms}$ ;  $\square, v'_{rms}$ ). CFD results: —, Standard  $k-\epsilon$ ; - - - SST  $k-\omega$ ; - · - · SA; —●—, RSM.

## 5 CONCLUSIONS

A set of CFD simulations for cavitating flow in a nozzle with a square section has been carried out by using combinations of several turbulence models (SA, Standard  $k-\epsilon$ , SST  $k-\omega$  and RSM) and one for cavitating flow (Singhal). The Standard  $k-\epsilon$  model shows a good behavior in these cases showing the best prediction of the mean values for velocities at the outlet in cases of square nozzles with low  $L/w$  ratios.

A comparison of the CFD results obtained against the experimental ones allows saying that despite that some clear differences among the mean velocity values at the outlet predicted by the turbulence models used, there are not big differences among the cavity structure predicted by all the turbulence models tested. For the incipient cavitation condition the cavity structure predicted by CFD is larger than the experimental one, being the results obtained by the Standard  $k-\epsilon$  the closest to the measured cavity length.

The experiments show that the  $u'_{rms}$  fluctuations become weaker near of the cavity when  $\sigma$  goes down, but the  $v'_{rms}$  ones do not change so much. The variation of both fluctuations along the nozzle width follows different trends. This behavior could be related to the tensorial character of the Reynolds stresses. Both fluctuations predicted for all EVM models are not so good compared against the experiments and are of similar level, pointing out the scalar character of the  $\mu_{t,m}$  predicted. This could be a reason for a not well predicted cavity when the

cavitation stage changes from incipient to developed. Surprisingly, the fluctuation velocities, related to the Reynolds stresses predicted by the RSM were not very different as the ones obtained by EVM.

It was also observed that an overprediction of the fluctuations level leads to a better fitting of the mean values of the velocity field. This fact could be taken into account for future EVM calibrations.

## 6 ACKNOWLEDGEMENTS

Current work was partially supported by the Universidad Tecnológica Nacional (UTN) within its own research programme (UTN/SCTyP). Authors would like to express their appreciation to the UTN for providing financial support for this study (research projects UTI3504TC, and UTI3543TC).

## REFERENCES

- Knapp R., Daily J. and Hammit F., 1970, "Cavitation," McGrawHill, New York, USA.
- Brennen C., 1995, "Cavitation and Bubble Dynamics," Oxford, New York, USA.
- Brennen C., 2005, "Fundamentals of Multiphase Flows," Cambridge, California, USA.
- Ardnt R., 2002, "Cavitation in Vortical Flows," Annual Rev. Fluid Mech, 34 pp 143–175.
- Ruiz, F., and He, L., 1999, "Turbulence Under Quasi-Cavitating Conditions: A New Species?," Atomization and Sprays, 9, pp. 419–429.
- Yuan, W., Sauer, J. and Schnerr, G., 2001, "Modeling and Computation of Unsteady Cavitation flows in Injection Nozzles," Mec. Ind. 2, pp. 383–394.
- Schnerr, G. and Sauer, J., 2001, "Physical and Numerical Modeling of Unsteady Cavitation Dynamics," Fourth Int. Conf. Multiph. Flow. New Orleans, USA.
- Singhal, A., Athavale, M., Li, H., and Jiang, Y., 2002, "Mathematical Basis and Validation of Full Cavitation Model," ASME J. Fluids Eng. 124(3), pp. 617–624.
- Coutier-Delgosha, O., Reboud, L. and Y. Delannoy, Y., 2003, "Numerical Simulation of the Unsteady Behaviour of cavitating Flows," Int. J. Numer. Meth. Fluids 42 pp 527–548.
- Habchi C., Dumont, N. and Simonin, O., 2003, "CAVIF: A 3D Code for the Modeling of Cavitating Flows in Diesel Injectors," Paper N°95, ICLASS Conf. Sorrento, Italy.
- Palau, G., and Frankel, S., 2004, "Numerical Modeling of Cavitation Using Fluent: Validation and Parametric Studies," AIAA Paper No. 2004-2642, 34th AIAA Fluid Dynamics Conference and Exhibit, Portland, Oregon, USA.
- Martynov, S., Mason, D., and Heikal, R., 2006, "Numerical Simulation of Cavitation Flows Based on Their Hydrodynamic Similarity," Int. J. Engine Res., 7(3), pp. 283–296.
- Giannadakis, E., Gavaises, M. and Arcoumanis, C., 2008, "Modelling of Cavitation in Diesel Injector Nozzles," Journal of Fluid Mechanics, 616, pp. 153–193.
- Li H., Kelecý F., Egelja-Maruszewski A. and Vasquez S., 2008, "Advanced Computational Modeling of Steady And Unsteady Cavitating Flows," Proceed. IMECE2008, ASME.
- Darbandi, M., and Sadeghi, H., 2010, "Numerical Simulation of Orifice Cavitating Flows Using Two-Fluid and Three-Fluid Cavitation Models," Num. Heat Transf., 58(6) pp.505–526.
- Gonçalves, E., Decaix, J., 2012, "Wall Model and Mesh Influence Study for Partial Cavities," Eur. J. Mech. B/Fluids 31(1) pp.12–29.
- Payri F., Payri R., Salvador F., and Martínez-López J., 2012, "A contribution to the understanding of cavitation effects in Diesel injector nozzles through a combined experimental and computational investigation," Computer&Fluids 58, pp 88-101.

- Zhang H., Han B., Yua X., and Ju D., 2013, "Numerical and Experimental Studies of Cavitation Behavior in Water-jet Cavitation Peening," *Process.ShockVibr.*20, pp 895–905.
- Duke, D., Kastengren, A., Swantek, A., Sovis, N., Fezzaa, K., Neroorkar, K. and Schmidt, D., 2014, "Comparing Simulations and X-ray Measurements of a Cavitating Nozzle," *Proc. ILASS-americas 26th Annual Conf. Liquid Atomiz. and Spray Syst.* Portland, USA.
- Sou, A., Biçer, B. and Tomiyama, A., 2014, "Numerical Simulation of Incipient Cavitation Flow in a Nozzle of Fuel Injector," *Comput. Fluid*, 103, pp. 42–48.
- Congedo, P., Goncalves, E. and Rodio, M., 2015, "About the Uncertainty Quantification of Turbulence and Cavitation Models in Cavitating Flows Simulations," *Eur. J. Mech. B/Fluids* 53 pp. 190–204.
- Naseri, H., Koukouvinis, P., and M. Gavaises, 2015, "Evaluation of Turbulence Models Performance in Predicting Incipient Cavitation in an Enlarged Step-Nozzle," *J. of Physics: Conf. Series. Vol. 656. No. 1. IOP Publ.*
- Coussirat M., Moll F. and Fontanals A., 2016, "Capability of The Present Cavitating and Turbulence Models For Confined Flow Simulations," *Proc.ENIEF 2016, Mecánica Computacional Vol XXXIV*, pp. 1989-2007.
- Coussirat, M., Moll, F., Cappa, F., and Fontanals A., 2016 "Study of Available Turbulence and Cavitation Models to Reproduce Flow Patterns in Confined Flows," *J. Fluids Eng.*, 138(9).
- Koukouvinis P., Naseri, H. and Gavaises M., 2016, "Performance of Turbulence and Cavitation Models in Prediction of Incipient and Developed Cavitation," *International Journal of Engine Research*, 2016.
- Coussirat, M., Moll, F., and Fontanals, A., "Reproduction of the Cavitating Flows Patterns in Several Nozzles Geometries by Using Calibrated Turbulence And Cavitation Models," *Proc. ENIEF 2017, Mecánica Computacional XXXV*, págs. 819-841.
- Ghorbani, M., Sadaghiani, A., Yidiz, M., and Kosar, A., 2017, "Experimental and Numerical Investigations on Spray Structure Under the Effect of Cavitation Phenomenon in a Microchannel," *J. Mechanical Science and Technology* 31(1), pp. 235–247
- Tseng, C. and Wang, L., 2014, "Investigations of Empirical Coefficients of Cavitation and Turbulence Model Through Steady and Unsteady Turbulent Cavitating Flows," *Comput. & Fluids* 103, pp. 262–274.
- Nurick, W., 1976, "Orifice Cavitation and Its Effect on Spray Mixing," *J. Fluids Eng.*, 98(4), pp. 681–687.
- Nurick, W. H., Ohanian, T., Talley, D.G., Strakey, P. A., 2008, "The Impact of Manifold-To-Orifice Turning Angle on Sharp-Edge Orifice Flow Characteristics in Both Cavitation and Non-Cavitation Turbulent Flow Regimes," *J. Fluids Eng.* 130(12).
- Nurick, W., 2011, "Flow Characteristics and Status of CFD Hydrodynamic Model Development in Sudden Contraction Manifold/Orifice Configurations," *Tech.Report*, Nurick and Associates Camarillo Ca.
- Sou A, Tomiyama A, Hosokawa S, Nigorikawa S and Maeda T, Cavitation in a Two-Dimensional Nozzle and Liquid Jet Atomization (LDV Measurement of Liquid Velocity in a Nozzle), *JSME International Journal Series B*, Vol. 49, No. 4, 2006.
- Sou, A., Maulana M., Hosokawa, S. and Tomiyama, A., 2008, "Ligament Formation Induced by Cavitation in a Cylindrical Nozzle," *J. Fluid Scie. and Tech.* 3(5) pp. 633–644.
- Sou, A., Maulana M., Isozaki K., Hosokawa, S. and Tomiyama, A., 2008, "Effects of Nozzle Geometry on Cavitation in Nozzles of Pressure Atomizers," *J. Fluid Scie.Tech.* 3(5) pp. 622–632.
- Ansys/Fluent Software, 2018, <http://www.ansys.com/Industries/Academic/Tools/>