STUDY OF TWO PHASE FLOW INSIDE GDI INJECTORS

[1]MOHAMMAD REZAEIMOGHADDAM, [2]AYSE EYLUL SENTURK

[1,2]Mechanical Engineering Department,Middle East Technical University(METU),Ankara, Turkey
E-mail: [1]mohammad.rezaeimoghaddam@metu.edu.tr, [2]senturk.eylul@metu.edu.tr

Abstract- The flow inside high pressure swirl injector has been simulated in two different ways. First, the two-dimensional axisymmetric Navier-Stokes equations coupled with the Volume-of-Fluid (VOF) method were employed for simulation of the formation mechanism of the liquid film inside the swirl chamber and the orifice hole of the pressure swirl atomizer. Second by using One Dimensional simulation approach the single phase flow inside GDI has been simulated by considering air cavity as a wall and two important characteristics of the flow has been calculated, discharge coefficient and cone angle. To validate both, the results for base injector were compared in the steady state operation with those of available experiments in the literature. In four different horizontal slices the mean values of swirl velocity, axial velocity and mean pressure has been compared. The cost and benefits are two approaches has been examined Through extensive simulations.

Keywords- GDI Injector, Volume of Fluid, One Dimensional Model, Spray Cone Angle, Discharge Coefficient

I. INTRODUCTION

In the recent years, many studies on swirl injectors have been carried out. The GDI engine is one of the most desirable internal combustion engines due to low fuel consumption while maintaining low emissions. They are often used in GDI engines because they allow fine fuel spray at relatively low injection pressure. The swirl atomizers are made of several main parts; tangential slots, a swirl chamber, a needle and a discharge orifice. Figure 1 shows a schematic view of a high pressure swirl injector.

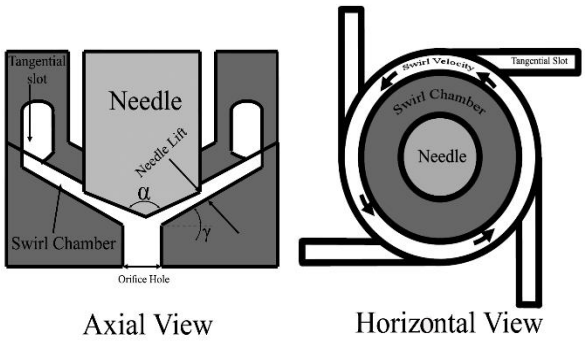


Fig.1: Schematic view of high pressure swirl injector

During the injection process, pressurized liquid is forced to flow through tangential slots into the swirl chamber which results in developing a strong swirling motion of the liquid in the chamber. The flow is accelerated through the swirl chamber and then enters the orifice hole. The swirl motion of the liquid pushes it close to the wall and creates a zone of low pressure along the center line which results in back flow of air in the injector. The liquid emerges from the orifice as a conical sheet that spreads radially outwards due to centrifugal force. The conical liquid sheet becomes unstable and undergoes a complex process of breakup to form a spray of droplets. As it can be seen the geometry of high pressure swirl atomizer is fairly which leads to coating delamination

As the mass flow rate through the atomizer is increased from zero, the performance parameters change sharply at first, but eventually at high mass flow rate, the discharge coefficient, the film thickness, and the spray cone angle become steady to the variations in flow Reynolds number. This indicates that discharge coefficient, cone angle and air core radius do not depend on injection pressure and keep constant values [2]. Two important performance parameters in high pressure swirl injectors are calculated in steady state operation; the discharge coefficient and the cone angle. The discharge coefficient is the ratio of the actual to the maximum theoretical flow rate that is determined from the measured pressure drop across the atomizer.

$$C_d = \frac{\dot{M}_L}{\rho_L A_o \sqrt{2\Delta P/\rho_L}} \quad (1)$$

where $\dot{M}_L$, ρ_L, A_o and ΔP are the mass flow rate of injection liquid, the density of injection liquid, orifice area and the pressure drop across the injector respectively. The spray cone angle is calculated by:

$$\theta = 2\tan^{-1}\left(\frac{\overline{W_e}}{\overline{U_e}}\right) \quad (2)$$

where $\overline{W_e}$ and $\overline{U_e}$ are the average swirl and axial velocities at the orifice exit.

Due to the difficulties outlined earlier, the initial investigations of pressure swirl atomizers modeled the liquid as inviscid and the flow irrotational [3, 4]. Taylor [5] gave the most valid and pioneering theoretical treatment for potential flow in a swirl nozzle and predicted that the air core diameter and spray cone angle were inverse functions. of inlet area to the product of swirl chamber diameter and exit orifice diameter. The flow characteristics within a pressure-swirl injector are of a highly complex nature due to creation Air-Cavity inside the injector. In the

recent years, many studies on swirl injectors have been carried out. Most of the current knowledge is empirical. In 1953 the first studies published by Doumas and Laster [2]. They have developed zero dimensional models where fluids were assumed inviscid. Dumouchel et al [3] studied the two-dimensional viscous flow inside a pressure-swirl injector by numerically solving the streamfunction and vorticity equations. The first three-dimensional computational analysis on internal flow in the high pressure swirl injector was carried out by Ren et al [4]. This approach has been completed and the needle movement inside the injector calculated in transient flows in high pressure swirl injectors by using the FIRE commercial code [5]. Rezaeimoghaddam and et al [7 and 8] used VOF method for finding the optimized pressure swirl and GDI injector. The more recent studies as those conducted by Arcoumanis et al [5 and 6] take into account both fuel and air flow. Cousin et al [4] investigated zero and one dimensional models in order to find a tool that is not time consuming. The two-dimensional non-swirl computational analysis was carried out by Moon et al [10] with CFD-ACE+ commercial code in order to investigate an optimized injector. Kub et al [10] simulated the flow inside high pressure swirl injector with viscous model with Star-CD commercial code.

In this paper two different approaches have employed to calculate spray cone angle and discharge coefficient and flow properties in four different sections of the solution. Simulations were performed for one commercial injector in 4.5Mpa Injection pressure with N-heptane as the injected liquid. First, a transient two-dimensional axisymmetric Navier-Stokes numerical model based on Volume-of-Fluid (VOF) technique is employed for accounting the liquid-gas surface interaction. A modified VOF technique based on Youngs' PLIC (Piecewise Linear Interface Construction) algorithm is employed to simulate the two-phase flow through the injector chamber. Second, by using empirical relations first the air core of the spray calculated and by considering it as a wall the geometry has been meshed in horizontal.

A one dimensional momentum equation in y direction has been used to calculate axial velocity by implementing Bernoulli's equation to calculate swirl-velocity in each y-steps. A guessed pressure is used to calculate the momentum equation in the first step and satisfies the boundary condition in the last step of the solution which should equals tozero. The friction loss can be calculated from moody diagram. The comparison between two proposed solutions has been carried out and the mean values of swirl velocity, axial velocity and pressure has been compared for both cases. The good agreement achieved for both cases and it turned out by using one dimensional model we can predict the injector properties more faster than VOF method.

II. GOVERNING EQUATIONS

2.1 Volume of Fluid method

Numerical simulations of the unsteady two-phase flow field in high pressure swirl injector are governed by the continuity equation and Navier-Stokes equations. The continuity equation is given by:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{u}) = 0 \tag{3}$$

Due to the full axisymmetric flow field through the injector chamber, swirl velocity inside the injector is assumed negligible and the momentum equation for two-dimensional flows written as follows:

$$\frac{\partial(\rho \vec{u})}{\partial t} + \nabla(\rho \vec{u}\vec{u}) = -\nabla p + \nabla\left[\mu\left(\nabla \vec{u} + \nabla \vec{u}^{T}\right)\right] + \rho \vec{g} + F_{vol} \tag{4}$$

In spite of the presence of the swirl chamber that creates a three-dimensional flow configuration, in this paper due to long time that required for three-dimensional simulation, the two-dimensional axisymmetric swirl model was used for compute flow through the injector. Due to important role of the swirl velocity in flow inside the injector the tangential momentum equation for two-dimensional swirling flows added to the momentum equations written as follow:

$$\frac{\partial}{\partial t}(\rho w) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho u w) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v w) = \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial w}{\partial x}\right] + \frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{w}{r}\right)\right] - \rho\frac{vw}{r} \tag{5}$$

where x the axial is coordinate, r is the radial coordinate, u is the axial velocity, v is the radial velocity, and w is the swirl velocity.

The phase change boundary is defined by Volume-of-Fluid (VOF) method where a scalar field is defined whose value is equal to zero in the gas phase and one in the liquid. When a cell is partially filled with liquid, f has a value between zero and one. The discontinuity in f is propagating through computational domain according to:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \bar{V}.\bar{\nabla} f = S \tag{6}$$

where S is the appropriate mass transfer source or sink term. Due to neglecting the cavitation phenomenon in this study" S "is considered to be equal to zero. The $k-\varepsilon$ renormalization group (RNG) model was also used in order to calculate turbulence effect. The RNG-based $k-\varepsilon$ turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique. Transport Equations for the RNG $k-\varepsilon$ Model are as follows:

$$\frac{\partial}{\partial t}(\rho k)-\frac{\partial}{\partial x_i}(\rho k u_i)= \quad (7)$$
$$\frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M+S_k$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_t)=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right)+ \quad (8)$$
$$C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon+S_\varepsilon$$

where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε, respectively. S_k and S_ε are source terms. The model constants are $C_{1\varepsilon}$ and $C_{2\varepsilon}$ and they were assumed 1.42 and 1.68 respectively [20]. Using the RNG model causes to better handle low-Reynolds-number and near-wall flows.Turbulence, in general, is affected by swirl in the mean flow. The RNG model provides an option to account for the effects of swirl or rotation by modifying the turbulent viscosity appropriately.

2.2 One Dimensional Model

The one dimensional simulation contains several steps. First by introducing injector constant by the following relation

$$A=\frac{r_o R_s}{r_p^2} \quad (9)$$

Where r_o is orifice radius and R_S is the diameter and r_p is the radius of the inlet port. Then from the following equation:

$$\frac{A^2}{2}\alpha^3-\alpha^2+2\alpha-1=0 \quad (10)$$

Where α is a core radius of air. By using Newton Raphson method α Can be calculated and the simulation can be conducted for a single phase by considering air core as wall without friction(Fig. 2).

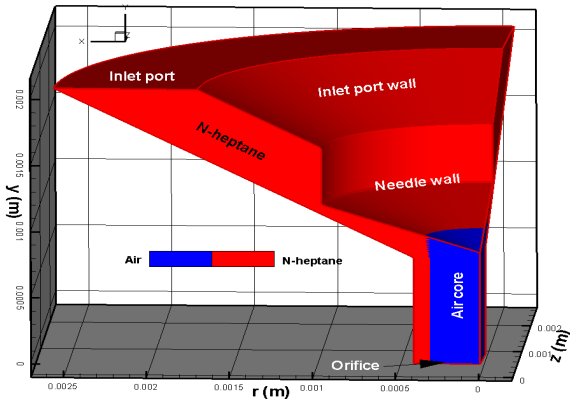


Fig.2:Three dimensional schematic view of numerical domain by considering air cavity as wall with no friction

Then by using one dimensional momentum equation in y direction and employing Bernoulli's equation for solving swirl component of velocity the flow inside GDI has been calculated. For the first step by guessing the pressure at the top the calculation, n=1, the calculation is started to satisfies the condition in the exit orifice by setting the pressure equals to zero. The calculation is continuing until we reached to this condition. In the calculation we neglect the radial velocity since its very small. The axial velocity has been calculated by the following:

$$\overline{U}_i=\frac{Q_v}{\pi\left(R_{out,i}^{\ 2}-R_{in,i}^{\ 2}\right)} \quad (11)$$

The tangential velocity of supplied at the inlet is calculated as following Muscelknautz:

$$\overline{W}_{inlet}=\frac{Q_V}{A_p}.\frac{D_s-d_p}{D_s} \quad (12)$$

The tangential velocity in each steps is calculated

$$\overline{W}_i=\frac{\overline{W}_{i-1}\cdot\overline{r}_{i-1}}{\overline{r}_i\ .\left(1+\Delta H_{i-1,i}^{\text{tangential}}\right)} \quad (13)$$

Where ΔH is the energy head loss of the flow which is calculated by using moody diagramThe discharge coefficient can be found by the following relation

$$C_d=\frac{4Ae\overline{V}_1}{\pi d_0^2}\sqrt{\frac{\rho}{2P_{inj}}} \quad (14)$$

And the spray cone angle is also found by:

$$\theta=2CArc\tan\left(\frac{\overline{W}_n}{\overline{U}_n}\right) \quad (15)$$

The schematic view of the computational domain is shown in Fig. 3

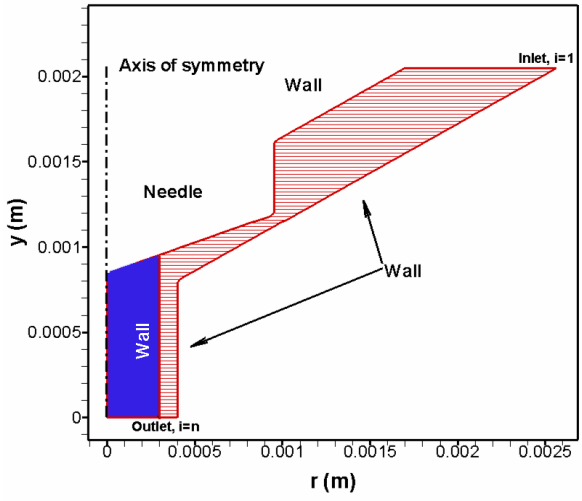


fig.3. schematic view of Computational grid for One dimensional model

III. VALIDATION

It is essential to validate the code with grid independence computational domain. For this reason,for VOF method several number of grid nodes (5500, 11800 and 16000) were selected and mesh study was carried out. To ensure grid independence of results, the values of discharge coefficient and spray cone angle were compared in different grid numbers.

With 11800 and 16000 cells, the discharge coefficient changed from 0.21095 to 0.212, and the spray angle remained unchanged at 87.99 The differences in the results using the two grids are very small and this indicates that 11800 cell grid is sufficient to get grid-independent results(see Fig. 3). For One dimensional model the computation domain is same everywhere except the inlet point for adjusting the solution, the inlet point considered the end point of the inlet channel (see Fig. 4). The mesh independency also has been carried out for the One dimensional model and the results shows that by using 240 meshes the calculation reaches the same answers for discharge coefficient and spray cone angle and above.

The The N-heptane (C_7H_{16}) was the injected liquid with density of 684 kg/m3, viscosity of 4.09×10-4 kg/m.s and surface tension of 0.02036 N/m. Calculations were carried out at 4.5MPa injection pressure and 300K constant temperature. The needle lift keeps constant values for all simulations. Liquid with uniform axial, radial and swirl velocity is assumed to enter in to the injector from the upper corner of swirl chamber.

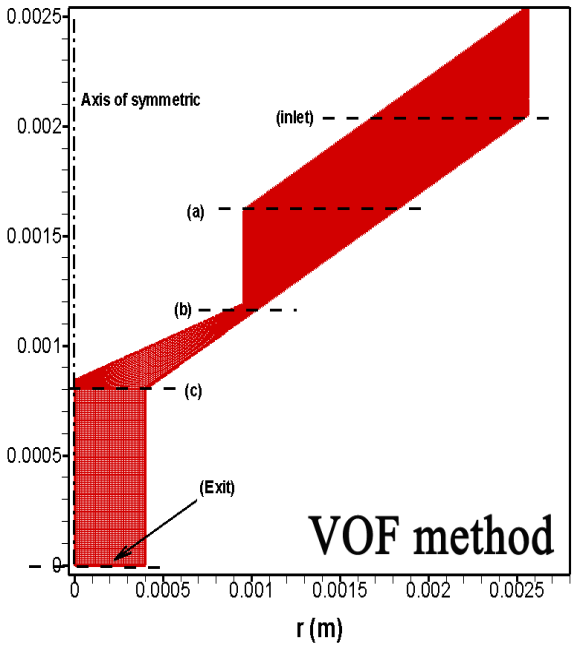


Fig.3. Computational domain for VOF method

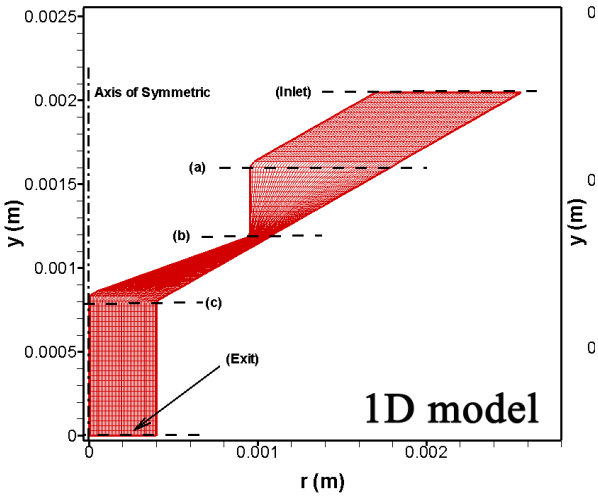


Fig.4. Domain used for One Dimensional Model

Brief comparisons between numerical and those of available experiments for characteristic parameters of the injector such as discharge coefficient and spray angle have been shown in table 1 whereas the mass flow rate inside the injector obtained from the steady state operation. The numerical results shown that the differences between the VOF method prediction and experimental data were within -4.3% for discharge coefficient and +1.8% for spray cone angle respectively. The results show a good agreement between VOF simulation and the experimental data. This verification demonstrates VOF model is a reliable method.

Table 3:Comparison between Numerical and experimental data

Method	C_d	error	θ (deg)	error
(VOF)	0.21	-4.3%	87.99	+1.8%
One Dimensional	0.216	-1.8%	86.2	-0.01%
Experimental[10]	0.22	-	86.4	-

IV. RESULTS AND DISCUSSION

The volumetric and axial velocity contour for VOF and One dimensional model is shown in Fig. 4. Velocity magnitude has been suddenly changed when inter into the orifice hole and expels as a high velocity thin film with conical shape. The two-phase interface is also clearly predicting in velocity contour when high gradient of velocity magnitude between gas and liquid is calculated. The liquid domain is also demonstrating a detailed view of spread flow through the chamber. Effect of swirl chamber is precisely simulated as an accelerator developing strong swirling motion of the liquid. As it can be since it is a one dimensional model the domain is started just

after inlet port. The predicted air core is matched to the VOF solution at the exit orifice. In the Fig. 5, the contours of swirl velocity and pressure has been shown. It can be seen that even by using the One dimensional model solution which is more simple the One dimensional model has a good accuracy to predict flow regime. Due to better understanding of the flow properties in 4 different sections the mean value of these quantities has been calculated and showed in fig, 6 and 7. The mean values of swirl velocity for both VOF and One dimensional model are very close in a, b, c and d sections.Although it has a little difference in section "a" and "b" but in the last two sections these values are reaching very close together for both VOF and One dimensional model. For the Axial velocity this mean values are much closer to each other and one dimensional model has become accurate to solve like VOF method. The Pressure mean values are not following the same procedure specially for the "a "section which is due to the fact that the domain are not the same and the effect of geometry has a great impact on this mean value. But when the calculation domain are much similar to each other like "b" section the mean values are getting closer to each other. The good agreement comes at the orifice which is the mean values of the pressure are the same for two method.

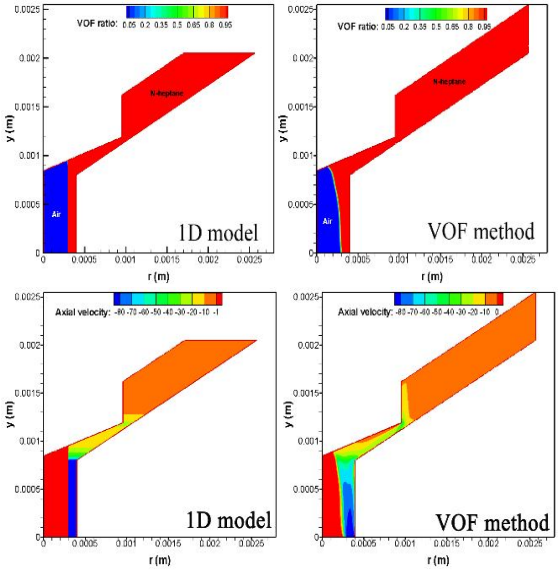


Fig.4. Volume of fraction and axial velocity contour for VOF method and One Dimensional calculation

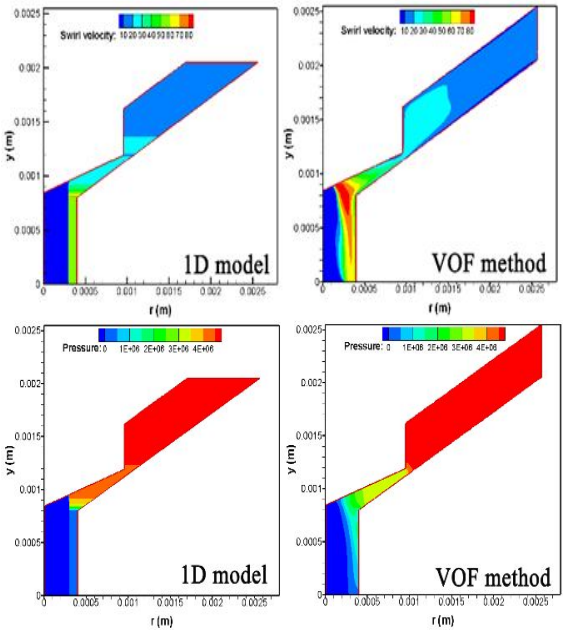


Fig.5. swirl velocity and pressure contour for VOF method and One Dimensional calculation

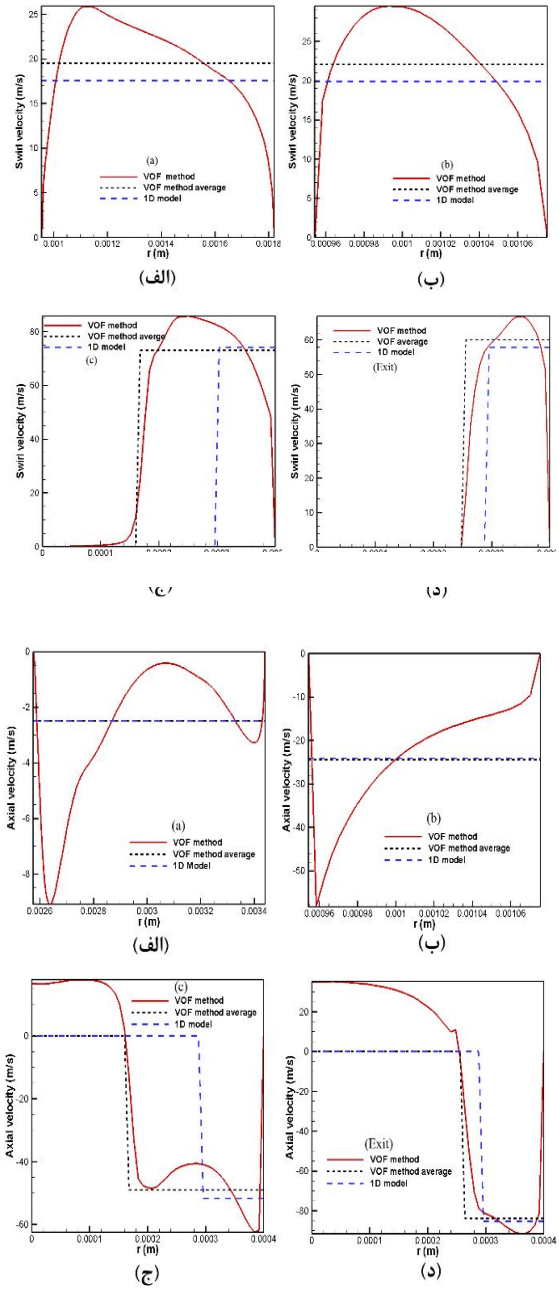


Fig.6. The mean value of swirl velocity and axial velocity in different sections for VOF and One dimensional model

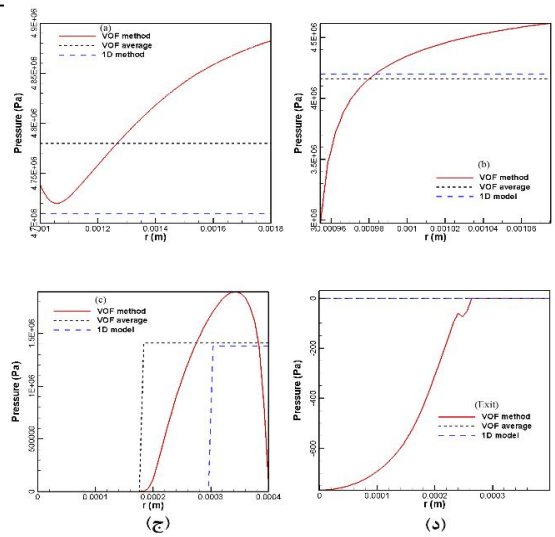


Fig.7. The mean pressure in different sections for VOF and One dimensional model

CONCLUSIONS

The two-phase flow inside high pressure swirl injectors was simulated. First, the VOF method was employed to track the surface between the fluid and air through the injector consisting of fuel and air that form the Air-Cavity. The proposed one dimensional model has been carried out to simulate the single phase of the flow inside GDI by considering air cavity as the wall, Numerical results have shown that the critical properties of the injector design such as cone angle and discharge coefficient can be calculated with good accuracy compared with the accurate VOF method and experimental data. The VOF method predicts accurate the air-cavity inside the injector and it leads to obtain a more accurate estimate size of the spray droplet that expels from the orifice hole of the injector. And it contribute the results which obtained by One Dimensional Model.

REFERENCES

[1]. A. H. Lefebvre, Atomization and Spray, Hemisphere Publishing Co, 1989.

[2]. M. Doumas, R. Laster,"Liquid-Film Properties for Centrifugal Spray Nozzles"1953.

[3]. C. Dumouchel, M. Bloor, N. Dombrowski, D. Ingham, ,"Viscous Flow in a Swirl Atomizer" Chemical Engineering Science, vol. 48, no. 1, pp. 81- 87.1993.

[4]. W. Ren, J. Cousin, S. Nally,"Transient Flows in High Pressure Swirl Injectors" SAE Technical Paper No. 980499, pp.213-222,1998.

[5]. C. Arcoumanis, M. Gavaises, "Modeling of Pressure Swirl Atomizers for GDI Engines", SAE Technical paper, pp. 516-532, Paper No 1999-01- 0500,1999.

[6]. C. Arcoumanis, M. Gavaises, "Pressure Swirl Atomizers for DISI Engines: Further Modeling and Experiments", SAE Technical paper, pp.1225-1240 ,Paper No 2000-01-1044,2000.

[7]. M. Rezaeimoghaddam , M. R. Modarres Razavi , B, Ayani, and R. Elahi, " Computational Modeling of Flow inside High Pressure Swirl Injectors by Considering Air-Cavity", Isme 2010 conference, Sharif university of technology, 11-13 May 2010.

[8]. M.Rezaeimoghaddam, Elahi, R., Ayani, M. B. and Modarres Razavi, M. R., "Modeling of non-Newtonian Fluid Flow within Simplex Atomizers", ASME 2010, 10th Biennial Conference on Engineering Systems Design and Analysis (ESDA 2010), July 12- 14, 2010.

[9]. S. Moon, C. Bae, C.Lee," Optimized Design of A New Gasoline Direct Swirl Injector" Journal of Numerical Heat Transfer Part A,pp.157-167,2001.

[10]. M. Kubo, A. Sakakida,A. Liyama," Technique for Analyzing Swirl Injectors of Direct-Injection Gasoline Engines" SAE Technical paper No 2001-01-0964, pp,2001.

★★★